SpatialAnalyzer

User Manual

NEW RIVER KINEMATICS

NRK METROLOGY INSTITUTE

SpatialAnalyzer

User Manual

2014.06.16

NEW RIVER KINEMATICS

NRK METROLOGY INSTITUTE

SPATIALANALYZER USER MANUAL

Every effort has been made to ensure that the information in this manual is accurate. NRK is not responsible for printing or clerical errors.

Because NRK frequently releases new versions and updates to its applications and Internet sites, images shown in this book may be slightly different from what you see on your screen.

New River Kinematics
436 McLaws Circle
Williasmburg, VA 23185
+1 (757) 565-1500
www.kinematics.com

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

CHAPTER 1 Welcome

Congratulations on your purchase of SpatialAnalyzer, the world's leading portable metrology software platform. SpatialAnalyzer is a traceable, graphical metrology software package that can simultaneously communicate to virtually any number and type of dimensional measurement systems and perform complex analysis tasks with ease.

SA is used in many high-profile applications throughout a wide array of industries, from manufacturing to science and research. Some of these industries include:

> **Note:** This user manual is currently under development.

- Aerospace (aircraft and spacecraft)
- Shipbuilding
- Particle Accelerators
- Nuclear Energy
- Automotive
- Heavy Equipment
- Machining

You are about to embark on learning how to adeptly use the most powerful portable metrology software ever created. We think this learning period will prove to be remarkably short. Most users are able to confidently acquire and analyze data after only one day. In fact, it's quite possible that you can start using SpatialAnalyzer simply by performing a few trial and error "test drives". The software is robust, and in general, it will politely inform you if you try something impossible.

We suggest that you may ultimately save time and grief by taking the time to read through this manual. Don't be too particular about memorizing anything—after all, this document is available from within SpatialAnalyzer. We hope you will obtain an understanding of the philosophy behind SpatialAnalyzer by reading this manual. If you are unable to read the manual, it also serves as an excellent reference when you have questions.

Above all else, the goal of our work has been to develop a software package where users of all levels of expertise may actually enjoy taking and analyzing measurements and may have confidence in the numbers they report.

About This Manual

New River Kinematics (NRK) regularly updates this manual to incorporate the most recent updates and changes to SA. However, due to the fact that this manual is currently under development, and due to the rapid pace of NRK's software development, you may encounter out-of-date material. NRK is aware of this and is working to bring content up-to-date.

SpatialAnalyzer comes in several packages. In cases where it may be ambiguous, we try to make it clear if a capability is not available in a specific SA package.

If you have any comments or suggestions for this manual, or if you notice any errata, please report them to support@kinematics.com.

Conventions

Note: An important note that accompanies the text appears in a box like this.

A standard set of conventions have been established to help ensure that this document is easy to read. The established standards are described here:

The titles of windows and dialogs will appear like this: ***Best Fit Transformation***.

Warning: A warning box, like this, will appear anytime there is information considered to be of a critical nature for either data integrity or measurement data accuracy. Failure to heed warnings could potentially lead to loss of data or incorrect results.

Menu choices appear like this: **Construct>Points>Enter**.

Checkboxes, dropdowns, and other control choices appear like this: *Show Scale Indicator in View*.

Button titles appear like this: Apply.

Keyboard combinations appear like this: **`Ctrl+D`**.

Named fields or boxes that subdivide areas of the user interface will appear like this: *Degrees of Freedom*.

Tip: Tips or recommendations that include best practices or time-saving techniques will appear in a box like this.

Filenames appear like this: `License10734.met`.

Text that you should type in will appear like this: **`Nominal`**.

New terms will look like this: *Relationship Minimization*.

Hyperlinks or links to other locations in this document look like this: support@kinematics.com.

A Closer Look

Additional information that is not critical, but may be of interest to curious readers, is printed in an box like this.

This Page Intentionally Left Blank.

What's New In SA

CHAPTER 2

One of the advantages of SpatialAnalyzer is that development occurs at a brisk pace. New feature requests, bug fixes, and changes are implemented quickly, giving you the opportunity to start taking advantage of newly implemented features in a very short period of time.

SA 2014.06.17

Slots

Support for both rounded and square slots has been added.

Construction

Use **Construct > Slots > Enter** to construct round or square slots relative to the working coordinate frame.

From Surface Faces

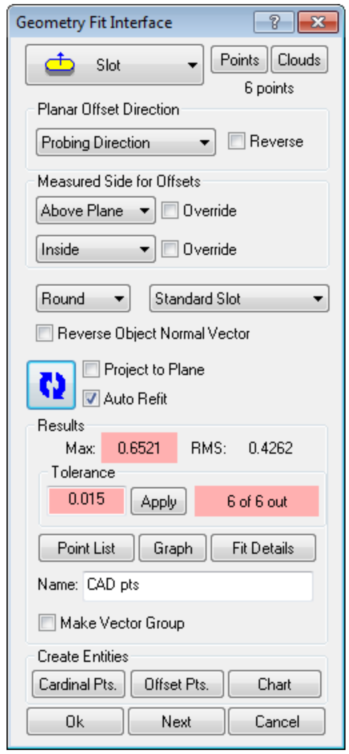
Given a merged CAD surface, you can extract a slot by using the new command **Construct > Slots > From Surface Faces**. Simply click on the interior wall of the slot and the nominal slot will be created.

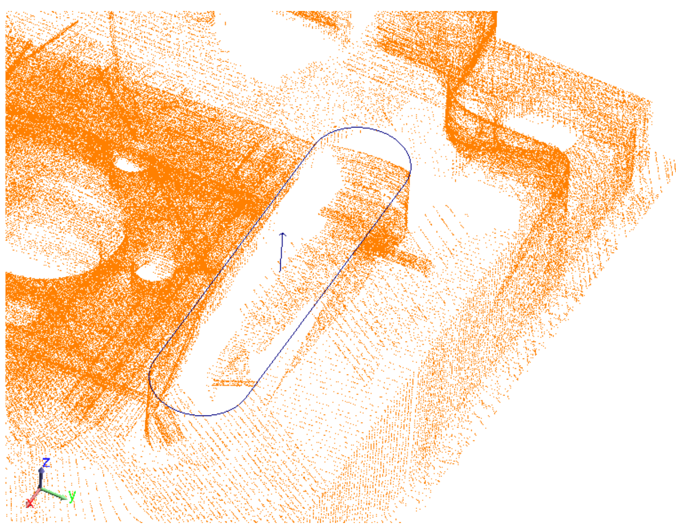
Geometry Fitting

Geometry fit now supports rounded and square slots. Use **Construct > Slots > Fit to Points** or select the *Slot* geometry type from the ***Geometry Fit*** dialog.

You can construct standard (best fit) slots (rounded or square), as well as max inscribed and min circumscribed slots.

For best results, measure at least 7 points for a slot, spread along both ends and the straight segments.



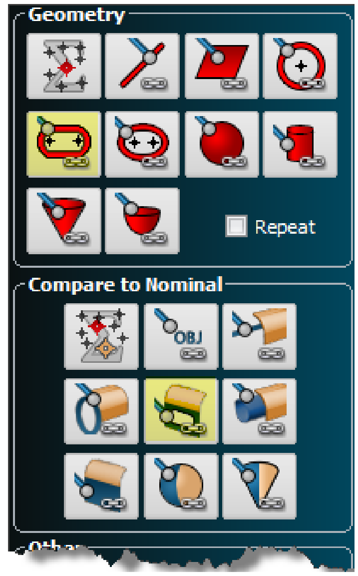


Slot Centers

To create a line along the center of one or more slots, use the new command **Construct > Lines > Center of Slot(s)**.

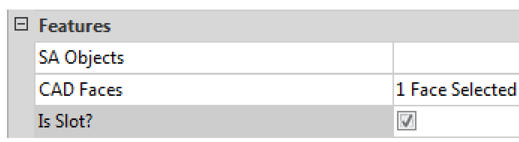
Geometry Relationships

You can now create geometry relationships for slots. Icons have been added in the Relationships tab in the Toolkit:



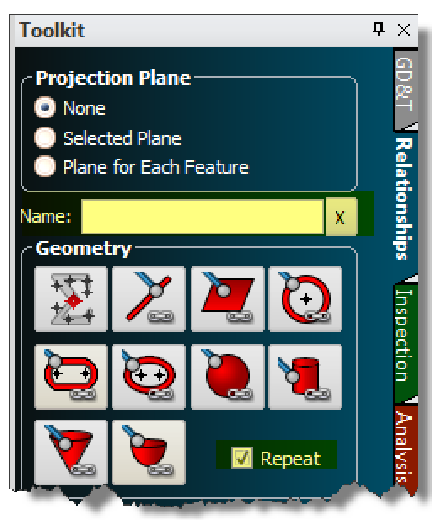
GD&T

GD&T now supports both SA Objects and CAD slot features. When using slots from CAD faces, it is necessary to enable the *Is Slot?* option in the *Features* section of the ***Annotation Properties*** dialog in order for the selected face(s) to be recognized as part of that feature.



Relationship Naming/Deletion

Renaming a relationship will cause the objects it created to also be renamed. Deleting a relationship will cause the objects it created to also be deleted.



A *Name* field has been added to the Relationships tab in the Toolkit which allows you to control the name of the relationship you create. If left blank, then the default name will be used.

Toolkit Repeat Feature

A *Repeat* checkbox has been added to the relationships tab of the toolkit (see previous image). When enabled, advancing forward in the Inspection tab or completing a feature with a measurement count limit will automatically create a new feature with the same properties as the last created feature.

This is particularly useful if you have to measure a large number of similar features, such as 100 holes. Simply define the first hole, enable the *Repeat* option, and as you advance past the first hole, new circle features will be created for you.

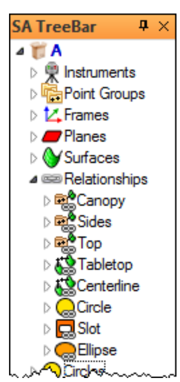
Single Point Circle Geometry Relationships

Support has been added for single point circles in a geometry relationship. As a reminder, single point circles allow you to lay a probe in a hole (assuming the hole diameter is smaller than the probe diameter) and create a circle with a single measurement, based on how far the probe sinks into the hole.

To create a single point circle geometry relationship, create a circle relationship as usual, and enable a projection plane. If the relationship has exactly one point, a single point circle will be created. If you continue measuring additional points, a regular circle relationship will be created instead. Note that the measured point must be on the positive side of the projection plane.

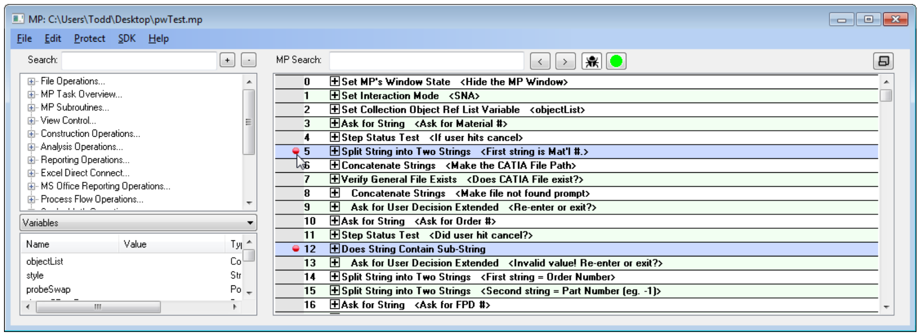
New Relationship Tree Icons

Several new relationship tree icons have been added to help you distinguish among different types of relationships in the tree:



Rewritten MP Editor

A new, entirely-rewritten MP Editor is available in this release.



The new editor brings a huge number of improvements over the previous script editor interface, making script development faster, easier, and requiring less mouse clicks. For detailed information on using this editor, refer to the chapter "Measurement Plans" in the SA User Manual.

To open the new editor, select **Scripts > Create/Edit Measurement Plan**, then click ⊟ to open the new editor.

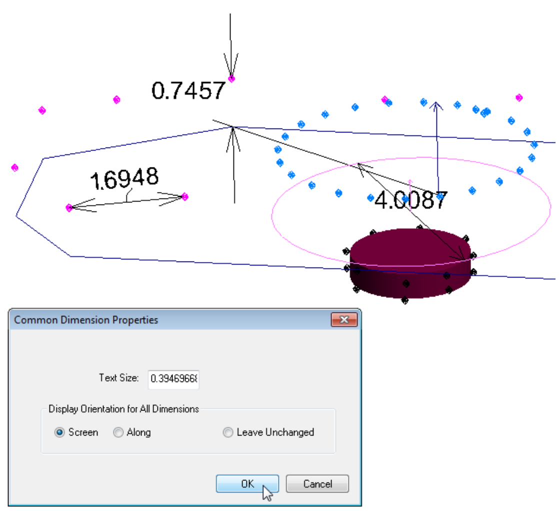
Note: The new MP Editor is still in beta testing mode—exercise caution if using it with production scripts.

New Measurement Plan Command

- **Construct Frames from Point Measurement Probing Frames.** Creates frames from points that have ijk vectors stored (supported by certain instruments).

Dimensions

Dimensions can now be oriented to always face the screen (*Screen*) in addition to the original setting to have them oriented along the dimension (*Along*).



This setting can be changed in the properties of a dimension, or you can right-click on the **`Dimensions`** category in the tree and select

Common Properties from the context menu to adjust the text size or display orientation for all dimensions in a collection.

Right-click selection in the graphical view has been implemented for dimensions as well.

Google Glass

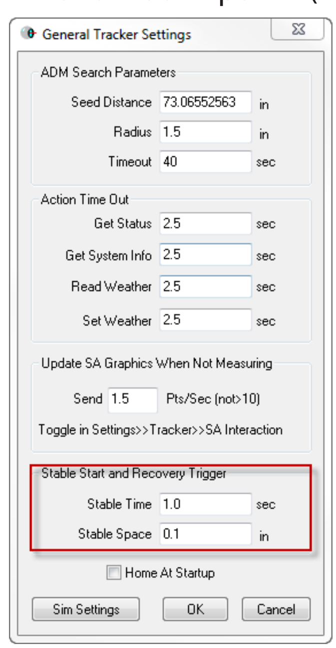
A Google Glass app is now available which displays watch window data on the glass, allowing it to perform as a Heads-Up Display.

The app is not yet available on the Google app store and requires manual installation. For more information, refer to the chapter "Watch Windows" in the SA User Manual or contact support@kinematics.com for download and installation instructions.

Laser Tracker Interface Enhancements

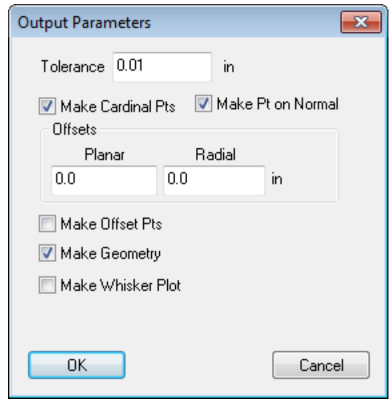
Stable Start and Recovery Triggers

You may now edit the parameters for the stable start and recovery triggers by clicking on the General Settings icon. This applies to any measurement's stable start trigger, stable recovery after beam break, measurement restart, and stable start for measurement modes such as projection planes and hidden points (vector bars).



Cardinal Point Offsets

For specialized applications, you can now specify the offsets for cardinal points created through the laser tracker interface or in Measurement Plans through Geometry Fit Profiles.



This control is found in the ***Output Parameters*** dialog of the tracker interface fit dialog or in the geometry fit profile settings. Both offsets default to 0.0 to preserve existing behavior.

User Interaction Mode

Laser trackers are now aware of SA's User Interaction mode. If set to *Silent*, the interface will suppress most error messages that require user interaction. This is meant for automation, where user interaction is not desired.

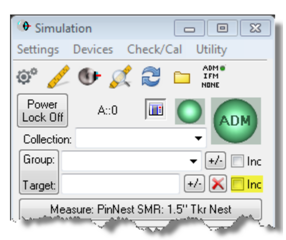
High Point

The planar offset for all High Point measurements is now set to zero for better representation, since offset compensation is applied to the high point. The radial offset is still that of the target specified for the measurement.

For *along axis* high point, the point is shifted in the negative direction of the axis.

For *from axis* high point, the point is shifted toward the axis.

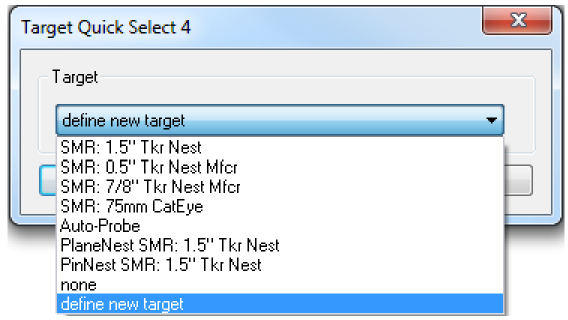
Auto-Incrementing Point Names



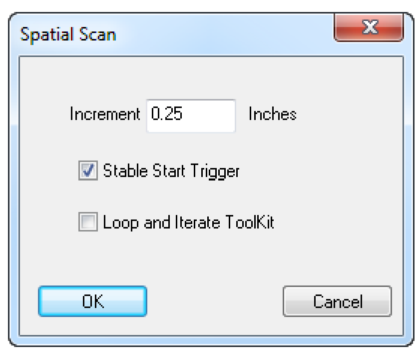
A check box option to auto-increment point names has been added. This setting *does not* persist. Each time the interface starts, it will be enabled to increment the point name (consistent with existing behavior). The primary purpose of this feature is to take multiple observations for the same point. It does not apply to spatial or temporal scans, or to measurements initiated from SA (such as auto-measure).

Tracker Toolbar

- **Reflectors and Targets.** You can now access the ***Reflectors and Targets*** dialog from the Toolbar. Right-click a target quick select and choose **Define New Target**.



- **Loop and Iterate Toolkit.** *Loop and Iterate Toolkit* has been added as an optional parameter for Spatial Scan in the tracker toolbar. This allows you to measure a set of circles defined in the Toolkit without returning to the computer between each iteration. When checked, this option sets the spatial scan to a stable start trigger, loop stop trigger and profile iterations to zero. It also iterates toolkit trapping to the next circle after each loop. When unchecked, this option sets the spatial scan to beam break stop trigger, one iteration, and no longer iterates the trapping from the toolkit. To navigate to this option, simply right-click on the spatial scan icon in the toolbar.

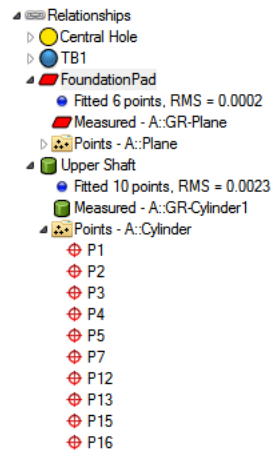


For additional fixes and improvements, refer to the SA Readme file.

SA 2014.04.15

Relationship Links

As a convenience, points or objects associated with a relationship are now displayed in the tree under the relationship:



This allows you to identify the associated entities quickly, and to access the properties of the associated objects directly under the relationship.

Relationships also list the points included in the relationship directly, as well as any cardinal points created by the relationship or projection planes used by the relationship. The geometry relationship icon now matches the object type being fitted as well.

You can interact with these tree entries just as you would elsewhere in the tree.

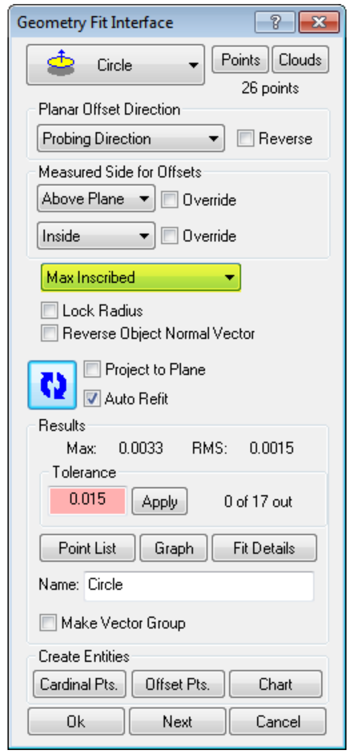
This capability is currently supported for Point to Point, Point to Object, Frame to Frame, Object to Object Direction, Geometry, and Average Point relationships.

Max Inscribed/Min Circumscribed Circle Fits

You can now create Maximum Inscribed and Minimum Circumscribed circle fits in the geometry fit, tracker, and geometry relationship interfaces:

(For geometry relationships, the setting is found by clicking the Fit Settings button). These options will yield the largest circle that fits inside the available data, or the smallest circle that will encompass the available data, respectively.

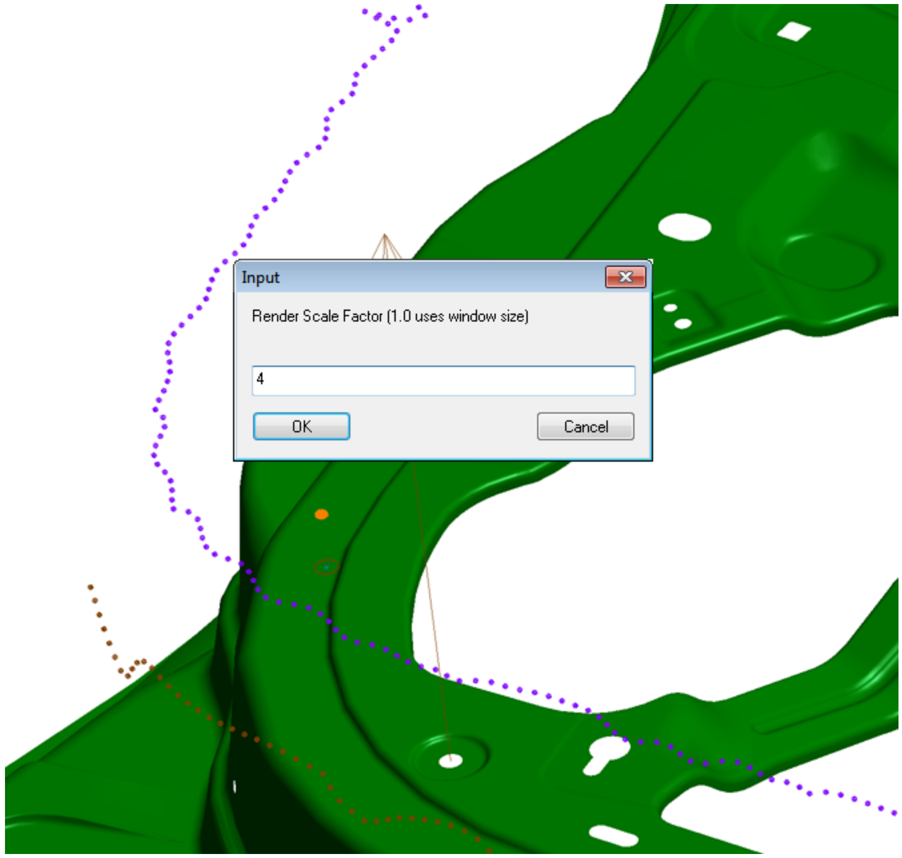
Note that these fit algorithms are sensitive to coverage—to avoid a fit error, try to ensure you have at least 180 degrees of circle coverage.



The fit mode (least squares, maximum inscribed, or minimum circumscribed) is recorded in the fit event and reported by the relationship as well.

High Resolution Graphic Export

When saving the graphical view to a file via **File > Capture Graphics > To File (BMP/JPG/PNG/GIF/TIFF)**, you can now specify a render scale factor, enabling high resolution image exports of anything in the graphical view. When prompted, enter the desired scale factor. A value of `2.0`, for example, will render an image that is twice the resolution of the graphical view.



LAP CAD-Pro Laser Projector: New Interface

A new interface has been added for the LAP CAD-Pro Laser Projector.

This interface supports creating alignment (calibration) files from SA.

To align the projector, select **Instrument > Laser Projector > Alignment** from the menus. After selecting the instrument and alignment points, a dialog will guide you through driving the projector to each point in sequence with the mouse.

Once you've driven to all the points, the projector will calibrate and a file with the alignment target positions and galvo angles will be created.

You can use this file to register the projector with a single click after the one time manual drive, as long as the projector is not moved.

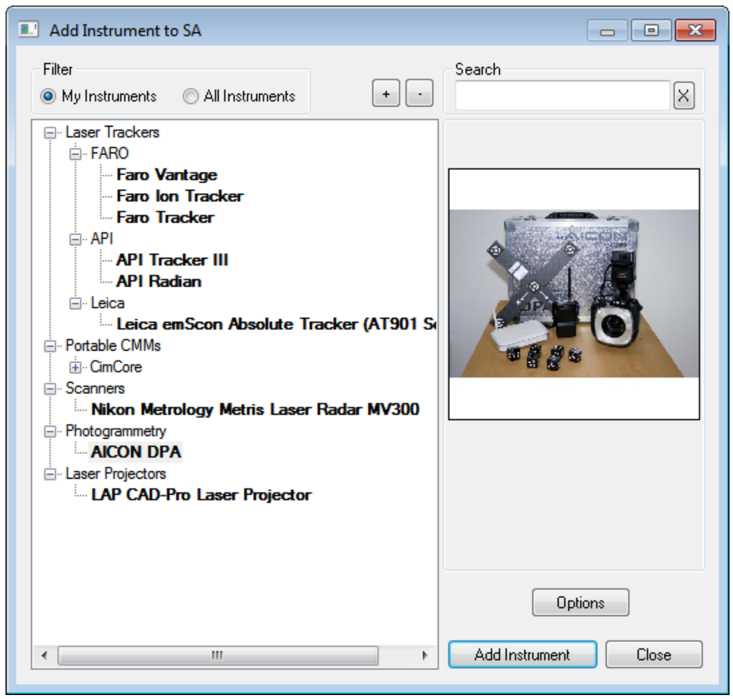
To project objects, select **Instrument > Laser Projector > Project Objects**. Select the objects to project. SA will project the objects and create a projection file using the name of the first object.

Once you have created multiple projections, you can switch among them using the Previous, Current, and Next buttons in the instrument interface.

Measurement Plan support has been added to control the current projection as well. Refer to the "Instrument Operational Check" command in the MP Command Reference.

AICON DPA: New Interface

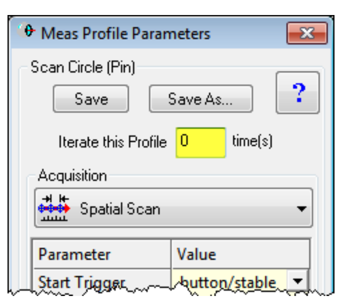
An instrument interface has been introduced to support the AICON DPA system.



This interface supports the import of points from the DPA system.

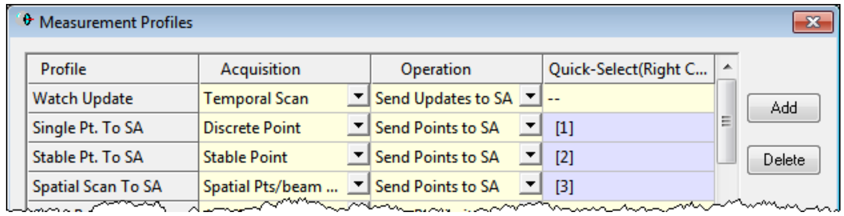
Laser Tracker Interface Enhancements

- You can now iterate measurement profiles indefinitely (until you abort the profile) by setting the *Iterate This Profile* option to **0** iterations:

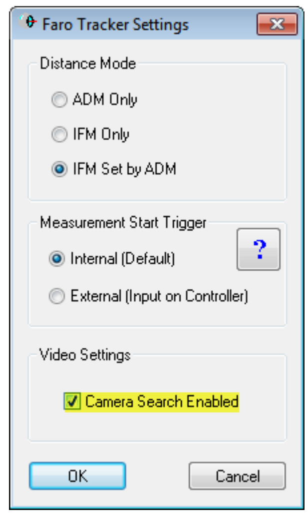


- Clicking on a measurement profile name in the ***Measurement***

Profiles dialog (below) will copy the name to the clipboard, for easy use in Measurement Plan commands:



- You can now enable or disable use of a Faro tracker's camera during a search operation. This is independent of whether the camera is used for catching the beam outside of a search operation. To toggle this setting, click the gear icon in the tracker interface, then click the Faro button. In the ***Faro Tracker Settings*** dialog, toggle the *Camera Search Enabled* option:



New Measurement Plan Commands

- **Get Instrument Model.** Retrieves the name of the specified instrument.
- **Create Point to Point Dimension.** Creates a dimension between two points.
- **Create Point to Object Dimension.** Creates a dimension between a point and an object.
- **Create Object to Object Dimension.** Creates a dimension between two objects.
- **Create Diameter Dimension.** Creates a diametrical dimension for a circle, sphere, or cylinder.
- **Create Radius Dimension.** Creates a radial dimension for a circle, sphere, or cylinder.
- **Set OPC DA Tag Value Double.** Creates or overwrites a double

value in the OPC server's address space with the specified tag.

- **Get OPC DA Tag Value Double.** Retrieves the double value associated with the specified tag from the OPC server's address space.
- **Set OPC DA Tag Value Integer.** Creates or overwrites an integer value in the OPC server's address space.
- **Get OPC DA Tag Value Integer.** Retrieves the integer value associated with the specified tag from the OPC server's address space.

The "Instrument Operational Check" command has a number of new commands for instrument control, ranging from Innovo teach and measure controls to LAP CAD-Pro projector control.

The "Dock Instrument Interface" command now also supports the Sokkia Net1 and Leica total station interfaces, and the Leica Automation Interface.

For more information, refer to the MP Command Reference.

OPC Data Server

SA now has the capability of acting as an OPC Data Access server for broadcasting real-time data to other applications or hardware devices. OPC stands for "Open Platform Communications" (originally "OLE for Process Control") and is a specification establishing standards for communication of real-time plant data between control devices and software from different manufacturers.

The server can also be accessed via Measurement Plans. For more information, refer to "The OPC Interface" in the SA User Manual.

What's New In SA

You can now access the What's New document (this document) directly from SA using **Help>What's New in SA**.

For additional fixes and improvements, refer to the SA Readme file.

SA 2014.02.04

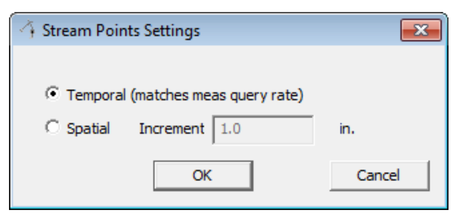
New Instrument Interface: Leica ScanStation P20

Support for the Leica Geosystems ScanStation P20 room scanner has been added. You must have a firmware license to activate the *API Data Access* toggle on the P20. You can toggle this on the scanner in **Status>System Information>Options**. The setting must be explicitly activated (it is deactivated by default).

Contact your Leica Geosystems representative for more information.

Arm Toolbar: Stream Points Settings

When working in the arm toolbar, you can now right-click the Stream Points button to set temporal or spatial scanning mode.



Shift Pts Along Vector

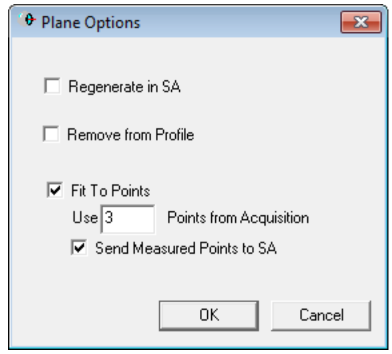
A new laser tracker measurement operation called "Shift Pts Along Vector" has been added. Along with it, a new measurement profile called "Scan Shifted Pts" or "Measure Shifted Pts" has been added.

With this operation, you define or measure a plane (as with projection planes) and define an offset (positive or negative). As you measure, each point is shifted by the specified amount.

Note that to reduce potential for confusion, the shifted points will retain offsets. Therefore, offset compensation should be considered when deciding on your shift offset. (In other words, if you want to create the point exactly 2" above the center of a 1.5" SMR, use 2" for the offset value—not 1.25").

Sending Measured Plane Points

Laser tracker measurement operations that provide the option of measuring a projection or reference plane in situ now also provide an option to send measured points, so that the points defining the plane now appear in SA as well.

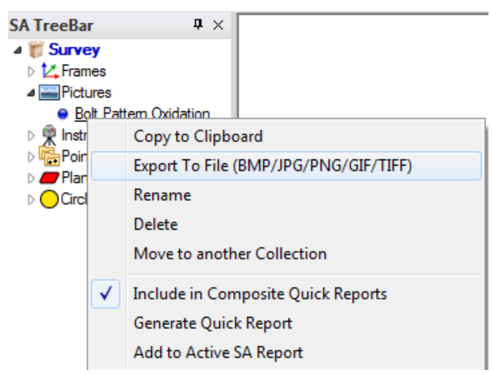


Report Image Drag and Drop

Supported images can now be dragged from the desktop directly into an SA Report. The file will be automatically embedded into the open job file's tree.

Image Export From the Tree

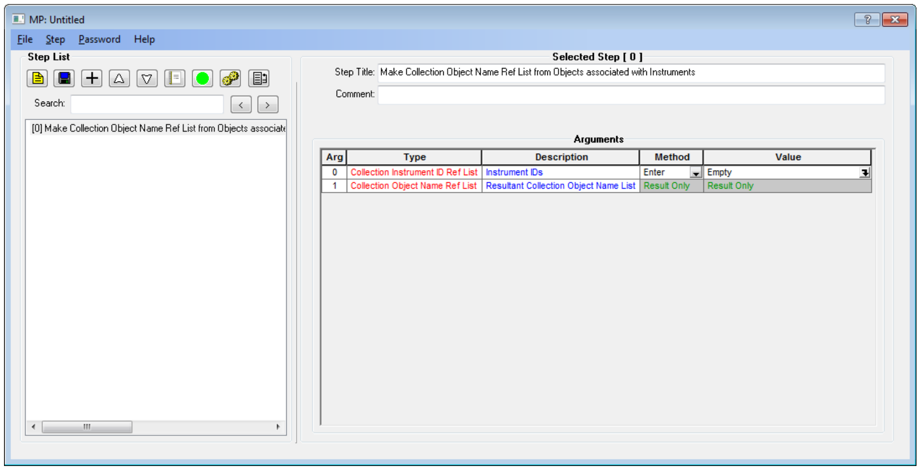
Any embedded image in the tree can now be exported to an external image file by right-clicking the picture in the tree and selecting **Export To File (BMP/JPG/PNG/GIF/TIFF)**.



MP Editor Improvements

The ***MP Editor*** has been modified to make resizing easier. In older versions, resizing the MP Editor required resizing multiple nested tables. Now, the window holds just one nested table which automatically resizes with the window.

Also, MP commands with very long argument lists are now displayed in their entirety, without requiring you to scroll down in a nested table.

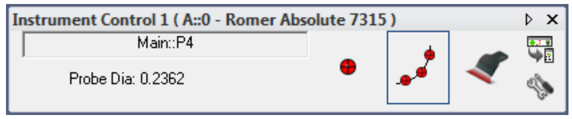


New MP Commands

- **Query Points to Single Point.** Performs a query from multiple points to a single point, creating a vector group.
- **Make Collection Object Name Ref List from Objects associated with Instruments.** Creates a list of all objects associated with one or more instruments.
- **Get Folders by Wildcard.** Obtains a list of folders that match a set of wildcard criteria.
- **Get Folder Notes.** Retrieves the notes associated with a folder.
- **Set Folder Notes.** Sets the notes associated with a folder.
- **Get Folder Collections.** Retrieves a list of collections inside a specified folder.
- **Output SA Report to Excel.** Sends an SA report to an Excel file.
- **Make a Double List.** Creates a list of double values.
- **Add Double to Double List.** Adds a double value to an existing list.
- **Get Double List Max/Min.** Retrieves the maximum and minimum values from a list of double values.
- MP control has been added for major theodolite functionality. Refer to the **Instrument Operational Check** command in the MP Command Reference for details.
- MP control has been added for Powerlock, SmartFind, and I-Vision. Refer to the **Instrument Operational Check** command in the MP Command Reference for details.

SA 2013.12.10

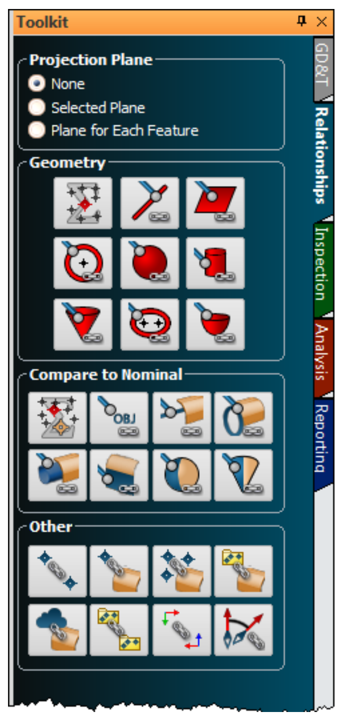
Arm Instrument Toolbar



A new instrument toolbar has been added for portable CMM arm operation. The arm toolbar is a greatly simplified interface for taking simple measurements with an arm. It is perfect for taking simple measurements or measuring an inspection routine with the SA Toolkit.

To access this new toolbar, click the Switch to Instrument Toolbar button in the arm interface. For details on using the new toolbar, refer to "Measuring with Arms" in the SA User Manual.

New Relationship Types: Average Point



Two new relationship types have been added:

- **Average Point.** Creates a point that is the average of all associated points.
- **Average Point to Nominal.** Creates a point that is the average of all associated points, and compares it to a nominal point.

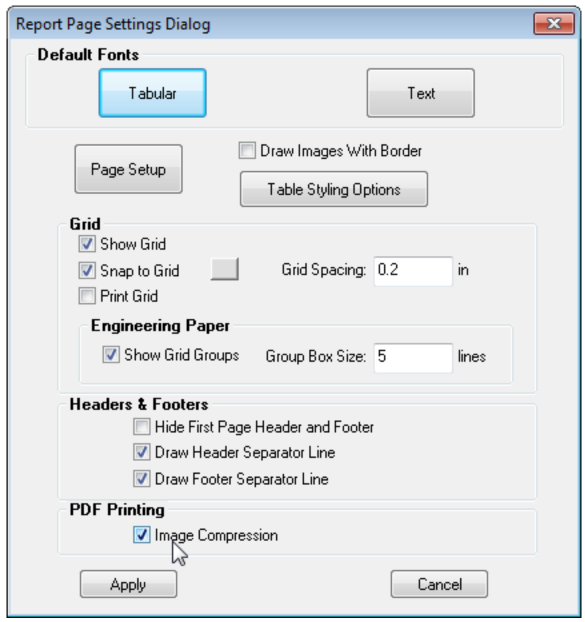
The points can be projected to a plane before averaging.

These relationships are accessible in the menus via **Relationships>Average Point>Average Point** and **Relationships>Average Point>Average Point to Nominal**. They are also available as new icons in the Relationships tab of the SA Toolkit.

PDF Image Compression Control

You can now disable image compression when outputting a report to PDF format. This will yield a higher quality report image, at the expense of larger file size.

To disable compression, uncheck the *Image Compression* option in the *PDF Printing* section of the SA report settings dialog.



Toolkit/Toolbar Keyboard Shortcuts

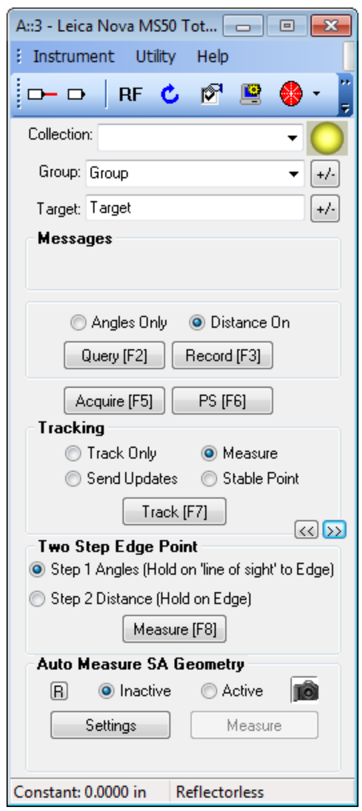
Keyboard shortcuts and T-Probe/4-button remote functions have been added for the SA Toolkit and Instrument Toolbars:

- **F6 ('A' button).** Toggle measurement in the instrument toolbar.
- **F7 ('B' button).** Go to the next feature in the toolkit.
- **Ctrl + F7.** Go to the previous feature in the toolkit.
- **F8 ('C' button).** Remove last point in the toolbar.
- **Ctrl + F8.** Stop trapping in the toolkit.
- **F9 ('D' button).** Iterate the measurement mode in the toolbar.
- **F10.** Iterate the active target in the toolbar.
- **F12.** Find the reflector (target search) in the toolbar.

New Instrument Supported: Leica Nova Series MultiStations

Support has been added for the new Leica Nova series multiStations. These instruments support traditional total station behavior, but also incorporate perimeter scanning technology.

This includes both scanning and camera capture. Remote video can be watched with VLC player, which can be downloaded at http://www.videolan.org/vlc/. Once installed, a 'Video' button will appear on the interface camera dialog.



Leica B-Probe Support

SA now supports the new B-Probe from Leica, which works with the AT402 laser tracker.

Note: Both the B-Probe and the 402 must have compatible firmware versions for proper operation. Refer to Leica for more information.

Leica AT 402 Enhancements

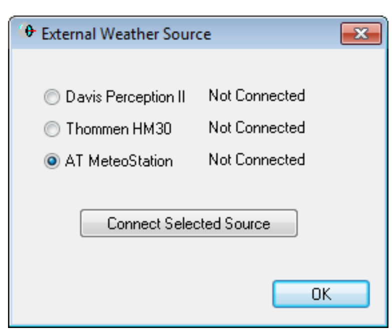
- **Discovery Support Added.** The connection dialog for AT402s now supports IP discovery. .NET Framework v4.5 or later is required.
- **Color Overview Camera Support.** Support for color imagery from the overview camera has been added. This requires a new OCX file to be installed, available at ftp://ftp.kinematics.com//pub/SA/Install/Driver%20Downloads/Laser%20Trackers/Leica/EmScon/Camera/.

Leica EmScon Enhancements

- **Brightness & Contrast Controls.** Controls for brightness and contrast of the overview camera have been added to the video window.
- **SensorError SDK Support.** Error conditions are now reported with more detail and potentially with proposed solutions.
- **PowerLock button.** The PowerLock toggle button has been added back by popular request.

MeteoStation As External Weather Source

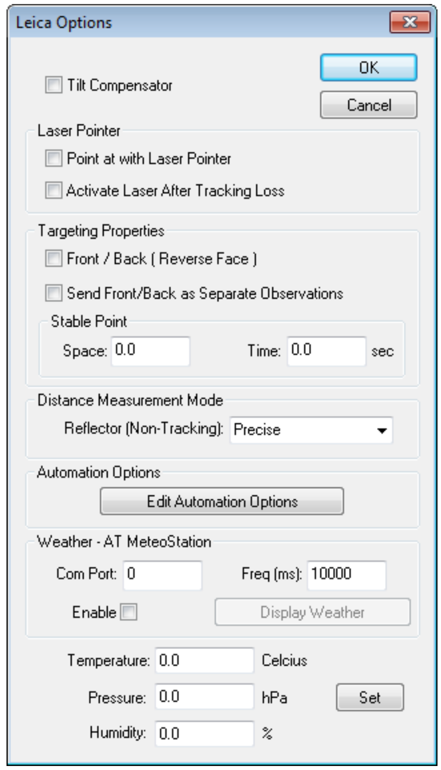
AT MeteoStations can now be connected directly to your computer and used as an external weather source for the laser tracker interface. If connected to an EmScon controller, it should still be set as an internal weather source.



Note that this may require installation of a USB to COM driver so that the MeteoStation creates a COM port when plugged into the computer (available from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/Laser%20Trackers/Leica/EmScon/Driver%20USBtoCOM%20for%20ATMeteo%20as%20External%20Weather%20Source/).

Leica Total Station Manual Weather Input

Manual weather data can now be entered in the options dialog for Leica total stations.



New MP Commands

- **Import SAT File.** Imports a .SAT file into the current job.
- **Import MP File as Embedded MP.** Imports an MP file into the current job as an embedded MP.
- **Delete from ODBC Database.** Deletes records from an ODBC database.
- **Show Items in Tree.** Shows specified items in the tree, optionally collapsing all other tree categories.
- **Construct Point Clouds from Existing Clouds - Uniform Spacing.** Creates a point cloud as a uniformly-spaced subset of existing clouds.
- **Make a Datum Ref List from a Collection.** Creates a list of all datums in a collection.
- **Get i-th String From String Ref List (Iterator).** Iterates

through a list of strings, retrieving a string on each iteration.

- **Get Number of Datums in Datum Ref List.** Returns the number of datums in a list of datums.
- **Get i-th Datum From Datum Ref List.** Retrieves a specific datum from a list of datums.
- **Get i-th Datum From Datum Ref List (Iterator).** Iterates through a list of datums, retrieving one datum on each iteration.
- **Refresh Datums/Feature Checks From Annotations.** Updates the datums/feature checks associated with the specified annotations.
- **Set Relationship Position Fit Constraints (Vector Type).** Sets positional constraints for a frame to frame relationship.
- **Set Relationship Orientation Fit Constraints (Vector Type).** Sets orientational constraints for a frame to frame relationship.
- **Start Robot/Machine Interface.** Starts the interface for a robot/machine.
- **Stop Robot/Machine Interface.** Stops the interface for a robot/machine.

SA 2013.10.02

Tracker Instrument Toolbar

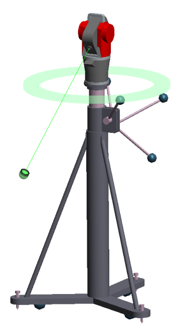
This release has a newly-developed interface for efficient measurement with laser trackers—the Instrument Toolbar.

This new interface is greatly simplified for more streamlined interaction with laser trackers. Less-common detailed settings have been stripped away, leaving just the essential elements of day-to-day measurement, particularly suited to measurement using the Inspection tab of the SA toolkit. It can be used for single point, stable point, spatial scan, tooling ball, and 4 additional user-defined measurement modes. You will likely find that this new streamlined interface covers a large proportion of your measurement needs, unless finer control is desired (in which case the traditional tracker interface can be used).

To access this new toolbar, click the Switch to Instrument Toolbar button  in the laser tracker interface. For much more on the new tracker instrument toolbar, refer to "Measuring with Laser Trackers" in the SA User Manual.

Instrument Connection Indicator

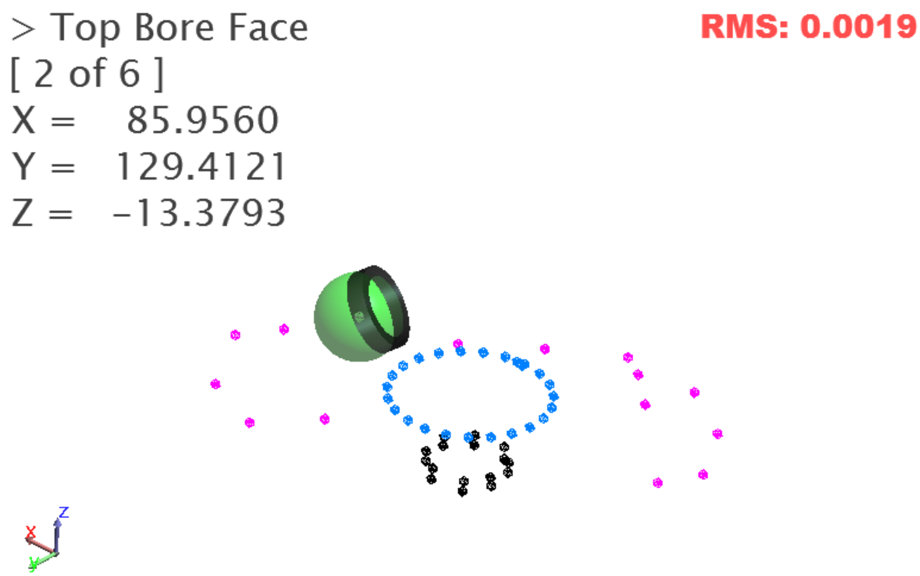
Connected instruments are now clearly indicated in the graphical view by a new Instrument Connection Indicator, a translucent green ring that hovers around the base of the instrument.



The display and size of this indicator can be controlled in the *User Options > Display* tab.

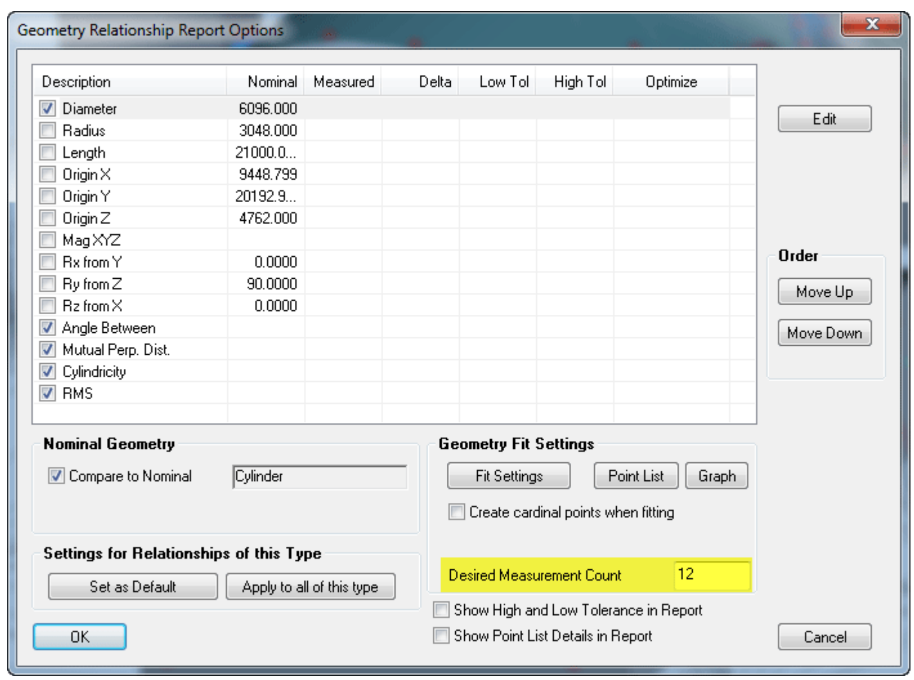
Heads-Up Display

The graphical view is now equipped with a heads-up display (HUD) that includes feature name (if trapping to a feature), number of total points measured (if applicable), target name, measured coordinates, and RMS (if applicable). The HUD's font, color, transparency, size, and dwell time (how long it persists before clearing from the screen) are all adjustable.



Measurement Count

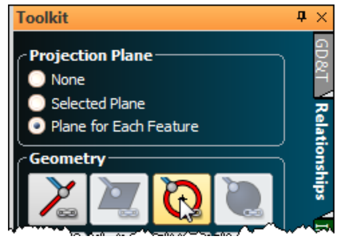
For measurement trapping (both to relationships and GD&T Datums and Feature Checks) you can now prescribe a specific number of points to be measured for a feature, and measurement trapping will automatically advance to the next feature when all measurements have been received. The current count is also displayed in the HUD.



You can also set the default number of points to be used for different feature types on the *User Options > Analysis* tab. Use 0 to indicate that you do not want a prescribed limit on the number of points.

SA Toolkit Feature Projection Planes

The projection plane options have been improved in the Relationships tab of the SA Toolkit for projection planes.

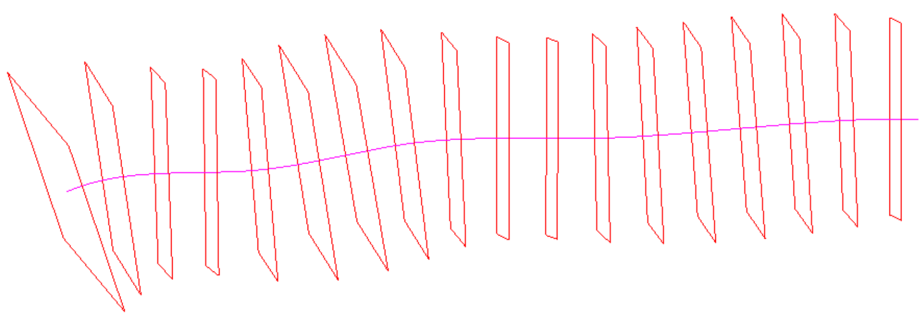


There are now three options:

- **None.** Does not use a projection plane for the feature.
- **Selected Plane.** Identical to the "Selected Plane" checkbox of previous versions. You choose a specific plane which will be used as a projection plane.
- **Plane for Each Feature.** When selected, each additional feature added to the inspection routine will also have a projection plane feature added, so that you first measure the projection plane, then the feature to project to it.

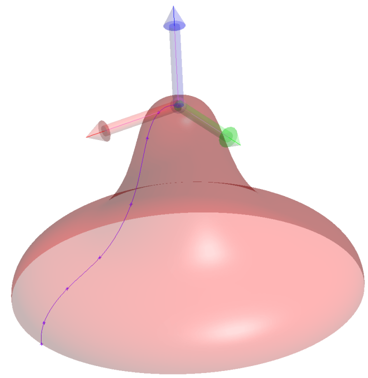
Layout Planes on a Curve

A new command (**Construct > Planes > Layout on a Curve Spaced at a Distance**) has been added to lay planes along a B-Spline spaced at a specified interval. The planes will be oriented perpendicular to the curve.



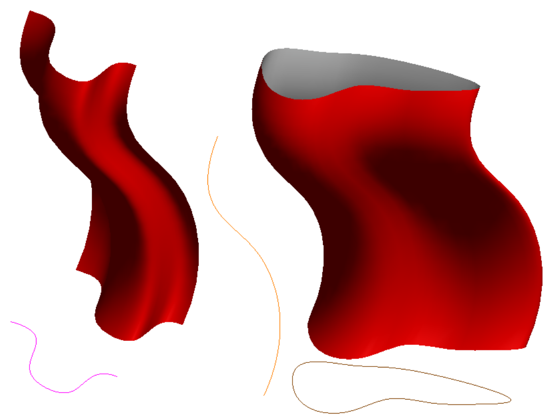
Revolved Surfaces

You can now create a revolved surface by revolving a B-Spline about an object's axis using **Construct > Surfaces > From B-Spline Rotated about an Object**. Select the B-Spline, the object, and then choose the axis about which to rotate, and the revolved surface will be created.



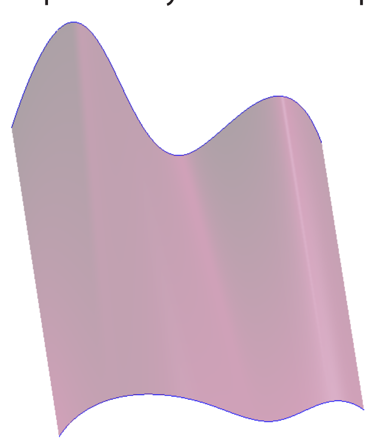
Swept Surfaces

Swept surfaces can now be created using the new **Construct > Surfaces > From B-Spline Swept along another B-Spline** command. Chose a B-Spline for the profile (can be open or closed), a B-Spline for the sweep path (can be open or closed), and indicate whether the sweep should be performed using pure translation (no rotation) or whether the profile spline should be permitted to rotate through the sweep.



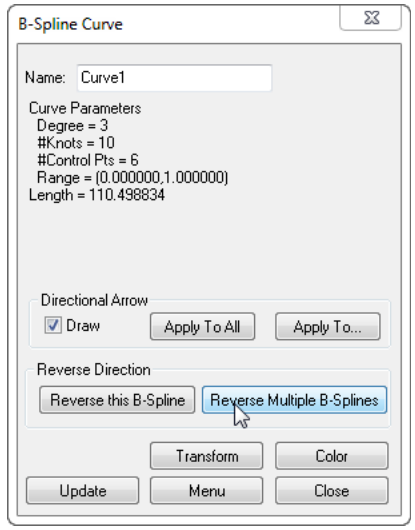
Surfaces From B-Splines or Point Groups

Using **Construct > Surfaces > From B-Splines** or **Construct > Surfaces > From Point Groups**, surfaces can now be created from as little as two B-Splines or point groups, where previously four were required.



B-Spline Reversal

Multiple B-Splines can now be reversed simultaneously. In a B-Spline's properties dialog, click the Reverse Multiple B-Splines button.

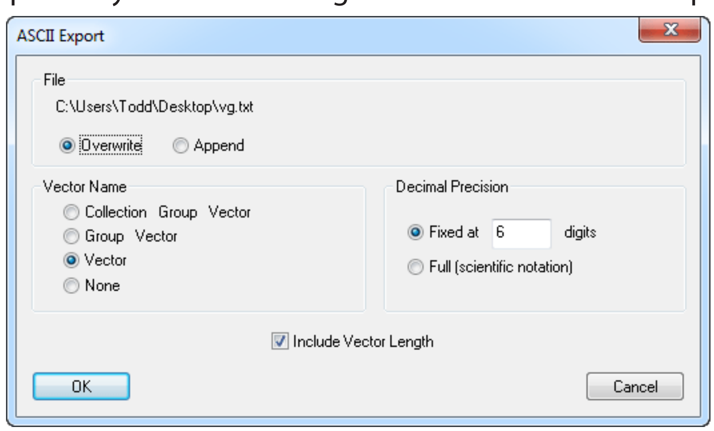


Vector Group Export

Settings for finer control over vector group exporting have been added to the **File > Export > Vector Group** command.

With these new options, you can:

- Overwrite or append to an existing file
- Specify the vector name format
- Control decimal precision
- Optionally include the length of the vector in the output.



Improved GD&T Surface Datum Handling

In previous versions, general surface features used as datums did not preserve primary/secondary/tertiary relationships. With this improvement, you can now use general surfaces as datums while preserving primary/secondary/tertiary datum priorities.

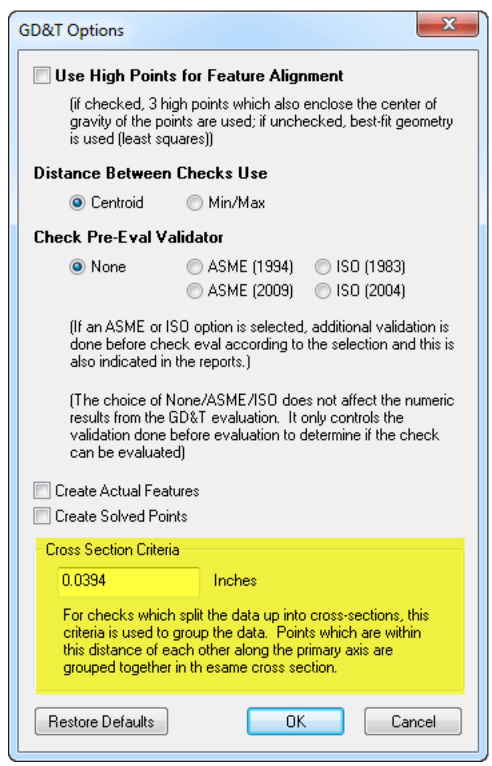
In order to do this, each general surface datum must be associated with an "offset" line or plane which defines the degrees of freedom that will be locked once the alignment to that datum surface has completed.

This is best explained with an example. Suppose a mostly-horizontal surface with some curvature is desired to be used as a primary datum, and a protruding vertical surface is intended to be the secondary datum. The primary surface can be defined with a horizontal offset plane. This implies that after the data is aligned to the mostly-horizontal surface using all degrees of freedom, the secondary datum is used for alignment with translation along the vertical direction and rotation about the horizontal axes disallowed.

The offset plane or line must be associated with the annotation before the datum is created. Also, a rough alignment using other methods (such as manual positioning, relationship fitting, Quick-Align, etc.) is recommended prior to evaluating general surface checks to maximize surface projection quality.

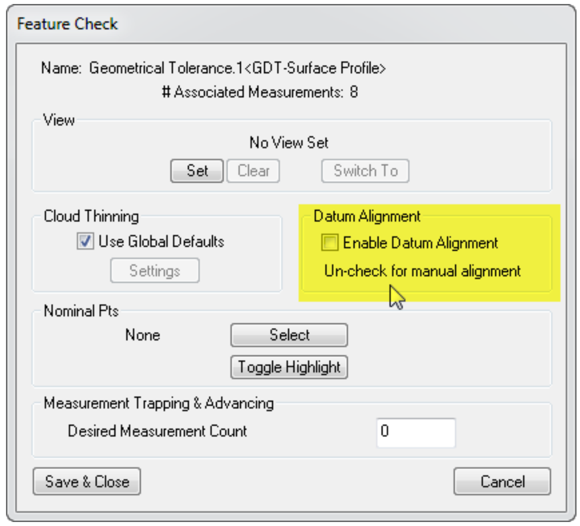
GD&T Cross Section Analysis Control

For GD&T checks which involve analysis of cross-sections, you can now control the distance between points (along the primary axis) used to group points together as a single cross section.



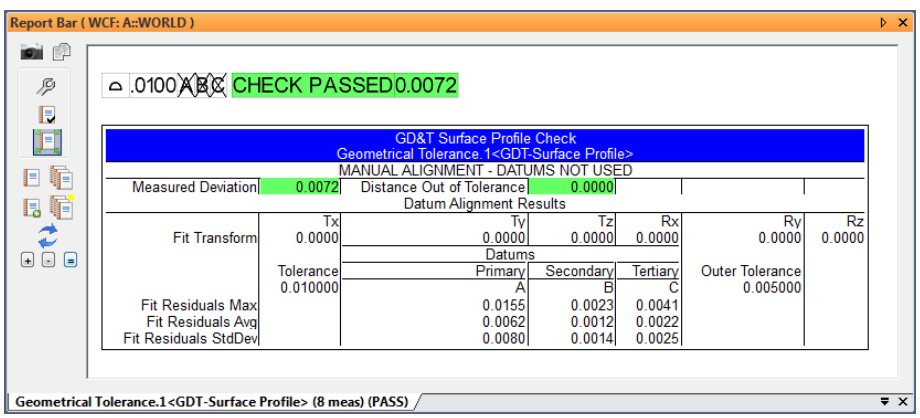
Manual GD&T Alignment

You can now evaluate GD&T feature checks by disabling datum alignment completely—that is, by manually aligning before performing the check.



In the feature check's ***Properties*** dialog, disable the *Enable Datum Alignment* checkbox. Doing so implies that the feature check will be evaluated using the current alignment. Note that this generally will not adhere to the GD&T standard.

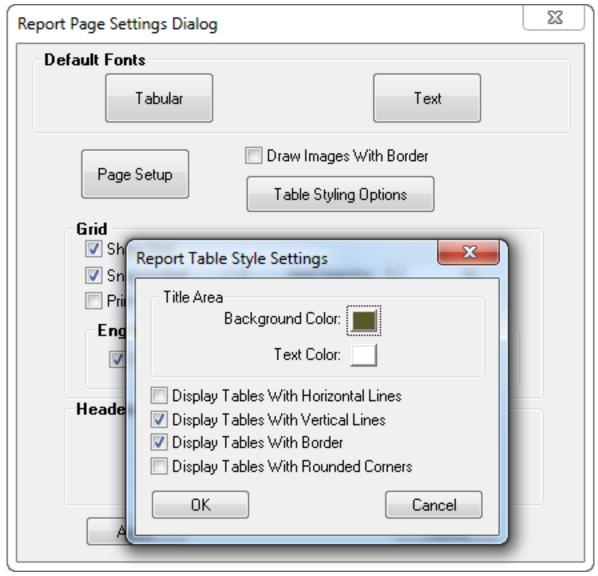
When evaluated this way, the feature check has its datums crossed out and the text **`MANUAL ALIGNMENT - DATUMS NOT USED`** will appear in the feature check's summary table in the report bar.



Reporting Improvements

Table Styling

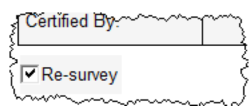
The new ***Report Table Style Settings*** dialog—accessible by clicking the Table Styling Options button below—holds the existing table formatting options, and additionally now provides access to control of table header background colors and the text color.



Plane 336A::B-Spline Plane			
A	B	C	D
0.280556 Proj. Ang. (deg.)	-0.163636 Rx from Y -99.8159	-0.945786 Ry from Z 163.4776	0.155644 Rz from X -30.2531

Checkboxes

Checkboxes can now be added to reports.



To add a checkbox, right-click the report and select **Add New CheckBox**.

Repeated Paste

Items can now be pasted multiple times into reports.

Field/Combo Box Excel Support

Fields and combo boxes are now supported for Excel export.

Measurement Plans

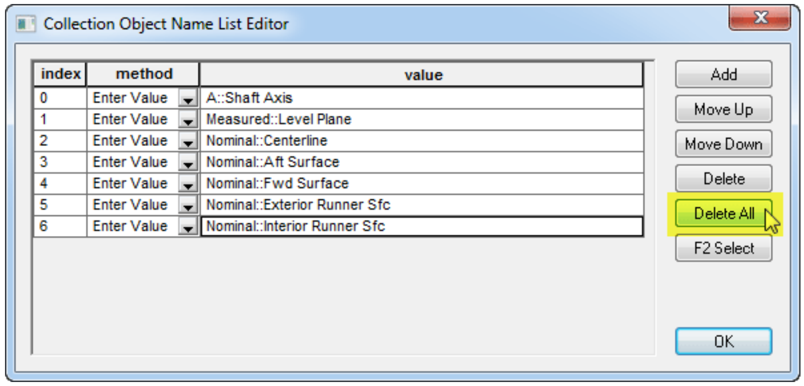
Automatic Backups

Automatic backups are now available for the MP Editor. The *User Options > Machine Configuration* tab has settings for enabling backups and setting the backup interval.

When enabled, the MP Editor will save backups of the script being edited to your backup directory, assuming changes have been made since the last save. Backups are saved to the same location for both embedded and external scripts.

Reference List Editing: Delete All

When editing reference lists, you can now click Delete All to remove all items from the list at one time.



25 New Commands

- **Get Instrument Target Status.** Returns information on the current instrument target.
- **Open XML File.** Opens an arbitrary XML file for reading and writing.
- **Set XML Attribute.** Modifies or creates one or more XML attributes for a specified node.
- **Get XML Attribute.** Reads one or more attributes from a specified node.
- **Close XML File.** Closes the XML file.
- **Add Item to SA Report at Location.** Adds an item to a report with an optional specified page number and position on the page.
- **Get Custom Table Cell String.** Retrieves a string from a table's cell.
- **Get Custom Table Cell Double.** Retrieves a double from a table's cell.
- **Make a String from a String Ref List.** Converts a string reference list into a single concatenated string.
- **Construct Points at Projection on Surfaces - Parallel to WCF Axis.** Creates points by projecting them to a surface along a working coordinate frame axis.
- **Construct Points at Projection on Surfaces - Radial from WCF Axis.** Projects points radially to a surface from an axis of the working coordinate frame. This essentially creates a cylindrical projection.
- **Construct Points at Projection on Surfaces - Spherical from WCF Origin.** Projects points spherically outward from the origin of the working coordinate frame's origin.

- **Get Collection Notes.** Retrieves the notes for a collection.
- **Set Collection Notes.** Sets the notes for a collection.
- **Get Object Notes.** Retrieves the notes for an object.
- **Set Object Notes.** Sets the notes for an object.
- **Get Point Notes.** Retrieves the notes for a specific point.
- **Set Point Notes.** Sets the notes for a specific point.
- **Set Relationship Associated Data.** Associates data with the specified relationship.
- **Get Relationship Associated Data.** Retrieves the data associated with the specified relationship.
- **Get Vector Group Colorization Options.** Retrieves the colorization options for the specified vector group.
- **Get Vector Group Display Attributes.** Retrieves the display attributes associated with a specified set of colorization options.
- **Construct Point Clouds from Existing Cloud Points - Runtime Select.** Constructs a point cloud from cloud points selected by the user at runtime.
- **Enable/Disable Alignment for Feature Check.** Enables or disables GD&T alignment for a given feature check. If disabled, the alignment in place at runtime is used for the feature check evaluation.
- **Show/Hide Annotations For Datums.** Permits showing or hiding, as well as highlighting, of annotations associated with specified datums.

For other improvements, changes, and fixes, please refer to the SA Readme file.

SA 2013.08.02

Expanded File Import/Export Support

The command **File>Export>Other>Surface Wireframe Curve File(s) (.crv)** has been added. This command writes a wireframe representation of selected surfaces to a text file containing 6 comma-separated values defining two endpoints of a line in Cartesian coordinates: Ax, Ay, Az, Bx, By, Bz.

The command **File>Import>Custom Formats>Leica ADF (.adf)** has been added to support the import of Leica ADF files.

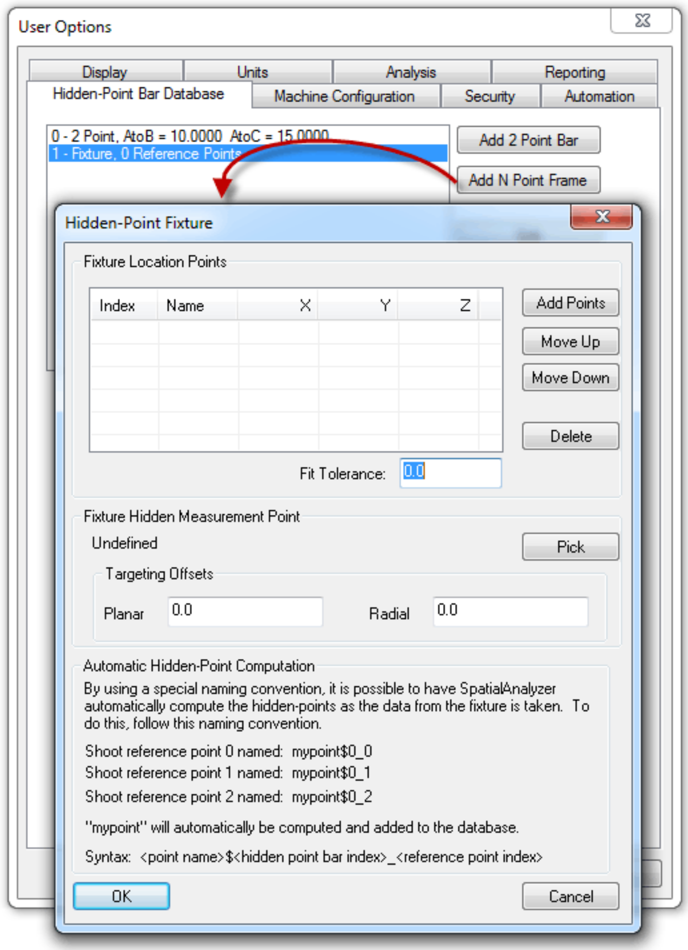
Direct CAD Access import support has been added for the following file formats:

- ACIS v23 (.SAT, .SAB)
- Autodesk Inventor 2014
- CATIA V5-6R2013 (R23)
- Adobe 3D PDF (.PRC)
- Rhinoceros (.3DM)
- Siemens NX 8.5

Hidden Point Fixtures

In the **User Options>Hidden Point-Bar Database** tab, you can now define a hidden point fixture. This allows you to use three or more points to define another point on a fixture. Also, hidden points now support target offsets for the computed point.

To use this new feature, first measure all points on the fixture and construct the hidden point manually. Then, in the Hidden Point-Bar Database tab, click to add an N Point frame. Add the points defining the fixture's orientation, then choose to pick the hidden point, and enter offsets for the hidden point if desired.



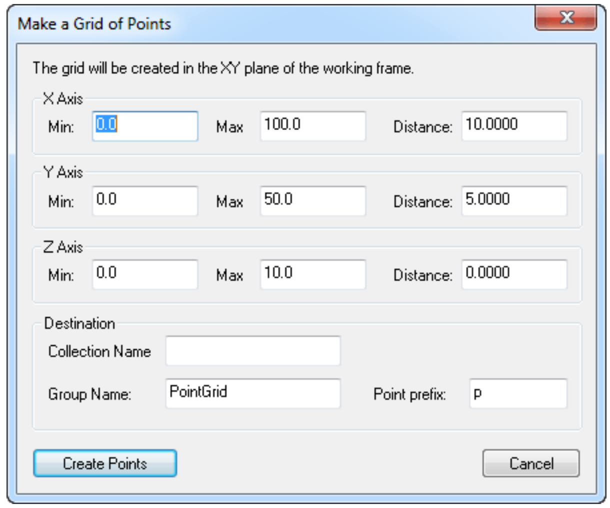
Remove Measurements By Instrument

The new command **Analysis>Measurement Simulations>Remove Measurements By Instrument** will remove all measurements associated with a specific instrument, leaving non-measured points behind.

Grid By Distance

In addition to the existing command to lay out points on a grid based on a count (which has been renamed to **Construct>Point(s)>Layout>Grid by Count**), a new command has been added to lay points out on a grid based on a spacing (**Construct>Point(s)>Layout>Grid by Distance**).

This command maintains the specified spacing, laying off points inside the specified boundary.



New Deletion Commands

Two new commands for deleting relationships and pictures have been added: **Edit>Delete>Delete Relationships** and **Edit>Delete>Delete Pictures**. These allow the deletion of one or more relationships or pictures in one command.

Tree Improvements

Several new commands have been added to the context menu in the tree:

- **Hide All.** Hides everything in the SA job.
- **Show All.** Shows everything in the SA job.
- **Delete Empty Folders.** Deletes all empty folders or folders containing only empty folders from the SA job.
- **Delete Empty Collections.** Deletes all empty collections from the SA job.

To access these new menu items, right-click in an empty area of the tree.

Also, `(Working)` now appears next to the working coordinate frame in the tree.

Italian Language Support

Italian is now included as a stock language in SA. Stock language translations are now available in the SA Viewer as well.

XML Import

The XML import and export format has been published for the benefit of all users. For more information, refer to "Importing XML Files" in the chapter "Working With Files".

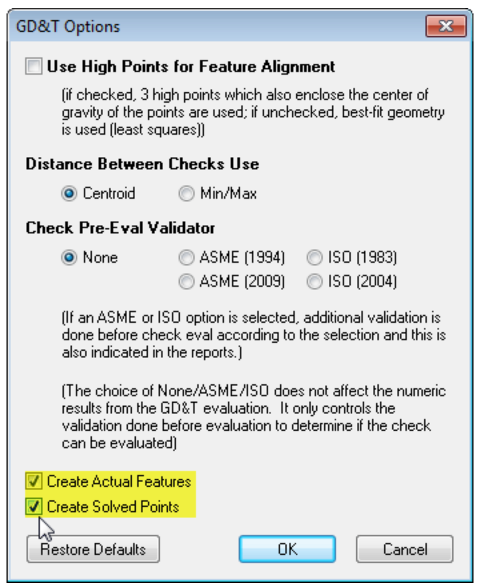
GD&T: Line Profile Check

A line profile check has been added to GD&T. Line profile checks require objects with directional information ("direction objects") to be associated with the annotation; these are used to determine how the measurements are grouped into cross-sections. Direction objects can be associated in the annotation properties dialog or when using the GD&T toolbar through an additional prompt during creation.

All measurements within 1 mm of each other in the specified direction are grouped together and analyzed as a cross-section. If multiple direction objects are used, the direction is obtained by averaging the individual object directions.

GD&T Actual Features & Solved Points

The ***GD&T Options*** dialog has two new options: *Create Actual Features* and *Create Solved Points*.



- **Create Actual Features.** Creates actual measured geometric features for GD&T checks. For instance, if performing a flatness check, the plane representing the orientation of the plane used for the flatness check is created. This allows you to query your measured points to this plane to determine high and low points, for instance.
- **Create Solved Points.** This creates the center points used for true position analysis when determining if a feature is out of tolerance.

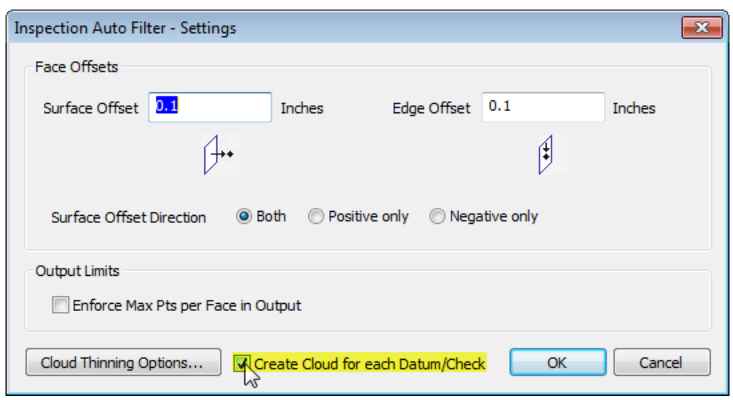
GD&T Actual Diameter Override

For cases in which a single feature check has been applied to multiple cylindrical features , you can you now create size checks for individual features and enable *Actual Diameter Override* to apply an actual diameter value for that feature without explicitly measuring it.

For instance, in the case of a true position check at MMC, any hole larger in diameter than MMC will result in a bonus tolerance. To apply this bonus tolerance to the true position check without physically measuring the size of the hole, you would create a diameter check for the hole, enable its *Actual Diameter Override* option, and type in an actual diameter. This will result in a bonus tolerance being calculated and applied, but without requiring you to measure the actual size of the hole.

Feature Inspection Auto Filter Cloud Organization

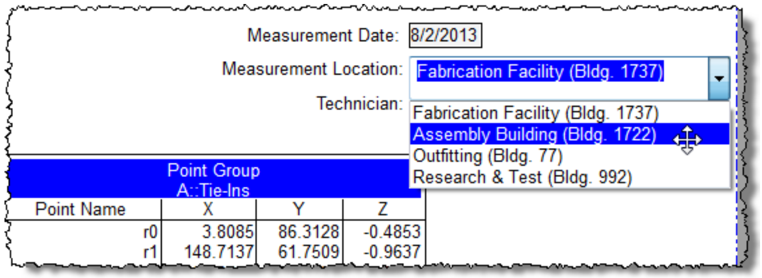
The **GD&T>Feature Inspection Auto Filter** command has a new *Create Cloud for each Datum/Check* option which will create a separate point cloud for each datum or feature check. Keep in mind that this may create multiple copies of a given cloud point, since one point may be associated with multiple datums or feature checks.



Reporting Improvements

Fields

A new ComboBox widget is now available within reports. Combo boxes allow you to present a drop down list to a user in a report. The list items may be populated from report tags or static text. Simply right-click in the field, select **Cell Options>Make ComboBox**.



New Tag

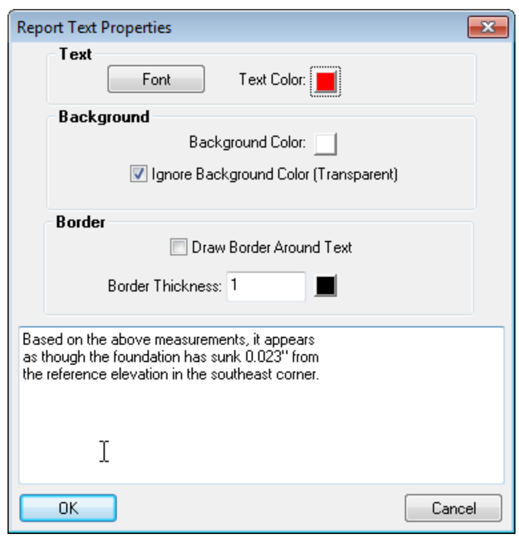
A new system tag has been added, `<<Filename Short>>`, which lists the current filename without the full path.

Borderless Tables

Borderless tables can now be created in reports by right-clicking a cell and selecting **Cell Options>Disable All Cell Borders**. Select **Cell Options>Enable All Cell Borders** to re-enable them.

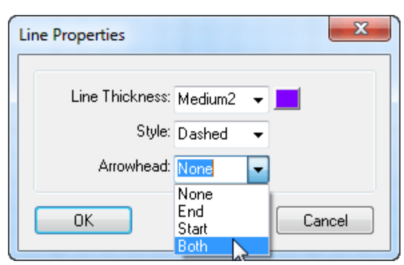
Text Properties

The properties of a text block now contain controls for font, background color, text color, border color, and thickness.



Arrowhead Ends

Line arrowheads can now be added to just the start, just the end, or both ends of a line.



Vector Group TCB Option

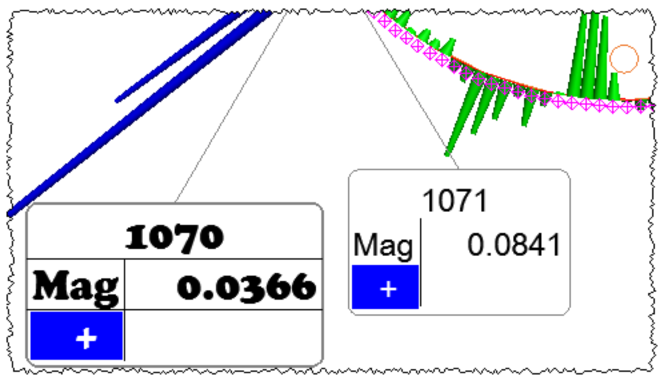
The report options for vector groups now has a *TCB* checkbox which allows disabling the display of the tolerance color box column in its table on a report.

USMN Max Instruments Per Point

The **User Options>Reporting** tab has a new *USMN Max Instruments Per Point* option in the *Event Reporting* box, which controls the maximum number of instruments listed per point within the event details table.

Callout Styling

Individual callouts can now have their own leader/border thickness/color and fonts. To modify an individual callout's settings, right-click the callout and select **Callout Styling**.

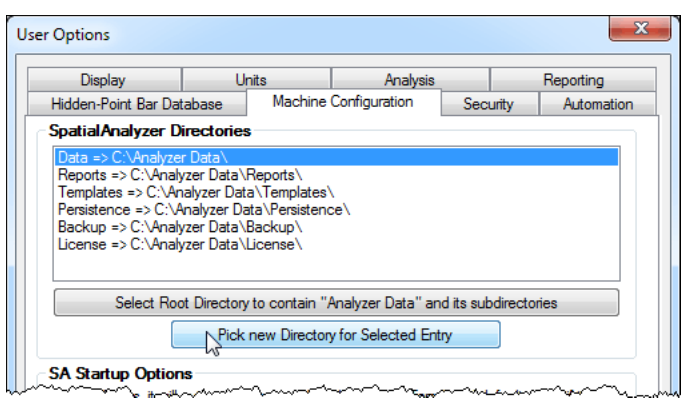


Callout Snapping

Callouts with leader lines can now be constrained along 45 degree angle increments. Simply hold down the **Shift** key while dragging a callout with a leader line to snap the callout and its leader line.

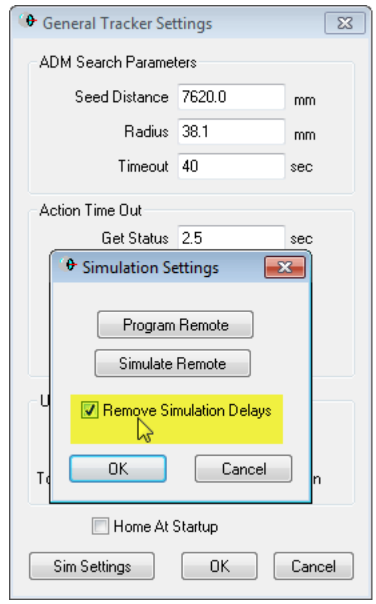
SpatialAnalyzer Directories

The **User Options>Machine Configuration** tab now allows you to select specific directories for each of the standard SA directories, avoiding the need to edit the registry to make such changes.



Tracker Interface: Remove Simulation Delays

The laser tracker interface has a new option remove simulation delays when connected to a simulated interface. This allows for faster testing when simulated measurements are being performed.



When enabled, the "Point-At" command (e.g. double right-clicking a point in SA) will now simulate a beam lock immediately.

New Interface: API Axxis 7-100 7-DOF Probe Arm

A new interface has been added for the API Axxis 7-100 7-DOF probe arm. This arm runs under the API Baces interface, similar to the 6-100 arm. It supports the new style 7-100 arms with the probe only (no scanner) attached.

New Interface: OmniTrak 2

A new interface has been added for the API OmniTrak 2 laser tracker.

This interface uses the API Device Interface, and has as of yet been untested with hardware.

New Interface: Leica AT4xx

This interface has been added to support the Leica AT402. The AT4xx designation is for support of future models. Note that the AT401 can be run via this interface, but the initial AT401 interface and instrument model in SA are preserved for continuity.

New Interface: Assembly Guidance Laser Guide Projector

A new interface has been added for the Assembly Guidance Laser Guide projector.

This interface supports creating calibration (registration) files from SA. Simply select **Instrument>Laser Projector>Alignment**. You will be prompted to select the projector and the group of points you will use to align the projector. In the interface, the UI will be displayed to guide you through driving the projector to each point in sequence with the mouse. Once you have driven to all the points, the projector will register and a file with the registration targets will be created. You may use this file to register the projector with a single click after the one-time manual drive, so long as the projector is not moved.

This interface supports projections from SA or from pre-created files. You are able to project objects from SA by selecting **Instrument>Laser Projector>Project Objects**. The selected object will be projected and a file will be created using the name of the first object. You have the capability to set the interface to add objects to an existing projection file, or to create a new file with each new projection. This way, you may create your own database of projection files simply by writing them to a common folder on your hard drive.

Once you have created a projection database, you can use the Previous, Current, and Next buttons in the interface to run through a given folder of projection files. You must install LASERGUIDESDK_3_03_11.exe (or later) before running the interface. This may be downloaded from the NRK FTP site: ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/AssemblyGuidance%20LaserProjector/.

This FTP location also has batch files which need to be executed to register the projector DLL in the SA install.

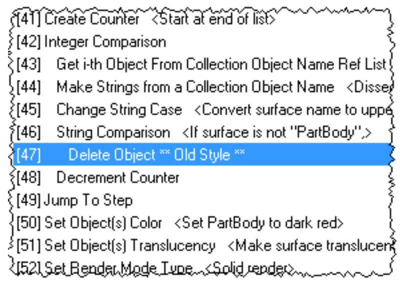
Automation

SDK Code Generation & Commented Lines

SDK code generation now ignores MP lines which have been commented out.

MP Indentation

Indentation in the MP editor can now be controlled using **Tab** and **Shift+Tab**. Select one or more lines, then press these keystrokes to add or remove indentation, respectively.



New MP Commands

33(!) new MP commands have been added in this release. They are fully documented in the MP Command Reference, but are briefly listed here for convenience:

- **Reverse Plane Normals.** Reverses a plane's normals.
- **Get Feature Check Cylinder Eval Options.** Retrieves cylinder evaluation options for a cylindrical feature check.
- **Set Feature Check Cylinder Eval Options.** Sets cylinder evaluation options for a cylindrical feature check.
- **Evaluate Feature Checks.** Calculates one or more feature checks.
- **Start/Stop Feature Check Trapping.** Starts or stops trapping to an individual feature check.
- **Get GD&T Options.** Retrieves the job file's GD&T options.
- **Set GD&T Options.** Sets the job file's GD&T options.
- **Start/Stop Relationship Trapping.** Starts or stops trapping to a specific relationship.
- **Get Number of Pictures in Picture Name Ref List.** Obtains the number of pictures in a picture list.
- **Get i-th Picture From Picture Name Ref List.** Retrieves a specific picture from a picture list.
- **Add a Picture to Picture Name Ref List.** Adds a picture to a picture list.

- **Construct Surface from Point Groups.** Fits a surface to points arranged by point group.
- **Construct Frame at Robot Link.** Creates a frame on a robot's linkage.
- **Make Axis Identifier from String.** Converts a string to an "Axis Identifier" for use in a frame construction command.
- **Make a Collection Vector Group Name Ref List - Runtime Select.** Prompts the user to select one or more vector groups.
- **Make a Point Name Ref List - Wildcard Select.** Creates a list of points based on a wildcard string.
- **Make a Report Ref List - Runtime Select.** Makes a list of reports from a user selection.
- **Make a Report Ref List from a Collection.** Makes a list of all reports in a collection.
- **Make a Picture Name Ref List.** Creates a list of pictures.
- **Make a Picture Name Ref List - Runtime Select.** Makes a list of pictures from a user selection.
- **Make a Point Name - Ensure Unique.** Creates a unique point name guaranteed to not be used in the file.
- **Make a Collection Object Name - Ensure Unique.** Creates a unique object name guaranteed to not be used in the file.
- **Import Nominals from XML File.** Imports nominal points from an XML file.
- **Merge Measurements into XML File.** Merges measured points into an XML file.
- **Rename Picture.** Renames a picture.
- **Cloud Display Control.** Sets the size and thinning for cloud points.
- **Find Files in Directory.** Retrieves a list of files in a given directory.
- **Get Directory and Filename from Path.** Splits a path into a directory and a filename.
- **UDP Send String.** Sends a UTF-8 string via UDP over the network.
- **UDP Receive String.** Receives a UTF-8 string via UDP over the network.
- **Ask for User Decision from Image.** Displays an image that has clickable regions to get input from the user.

- **Get Datum Measurements.** Retrieves the points/clouds associated with a specific datum.
- **Set Datum Measurements.** Sets the points/clouds associated with a specific datum.

For other improvements, changes, and fixes, please refer to the SA Readme file.

SA 2013.03.22

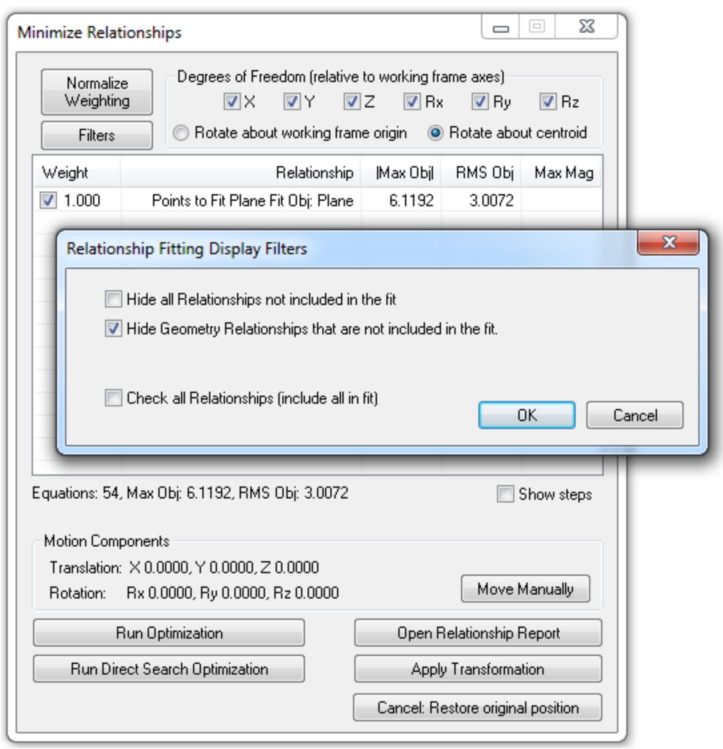
Closest Points between Two Surfaces

This command finds the closest points between two surfaces, which is particularly useful for determining clearance/interference for virtual fit-up and other alignment applications. From the menu, select **Construct>Points>Closest Points between 2 Surfaces**. Pick two surfaces and this creates two points, one on each surface, minimizing the distance between the two.

Relationship Fit Filters

A series of filters have been added to the ***Minimize Relationships*** dialog (see image below). This filter provides three options:

- **Hide all Relationships not included in the fit.** This will remove any unchecked (excluded) relationships from the ***Minimize Relationships*** dialog.
- **Hide Geometry Relationships that are not included in the fit.** Only geometry relationships that are unchecked (excluded) will be filtered out of the ***Minimize Relationships*** dialog.
- **Check all Relationships (include all in fit).** Selects (checks) all relationships in the collection for inclusion in the ***Minimize Relationships*** dialog.



Stock Language Translation-French

French is now included as a stock language in SA.

Geometry Relationship Storage

The results of geometry relationships are now stored in the SA file instead of being recalculated on open. This avoids potentially lengthy file open commands in certain cases which required a large number of geometry relationships to be recalculated.

SA Remote

You now have the ability to browse for available tracker interfaces, removing the need to manually key in an IP address.

Support for Wavefront Objects File (.obj)

Files formatted in Wavefront's `.obj` file format may now be imported via the **File>Import>Custom Formats>Wavefront Objects File (.obj)** menu command.

SolidWorks 2013 Support

SolidWorks 2013 file formats are now supported through the Direct CAD Access import option.

DXF Export

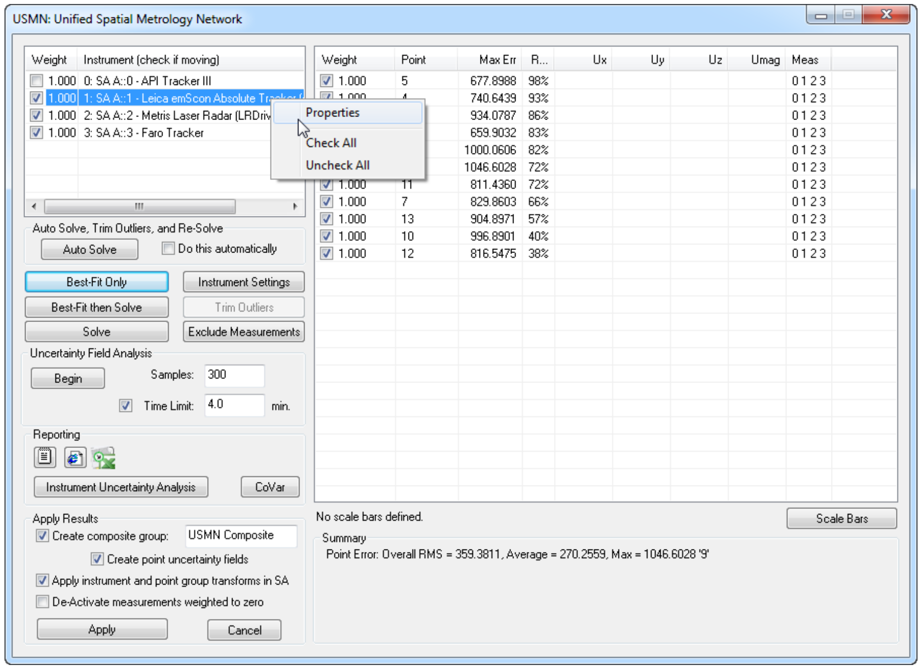
When exporting a `.DXF` file via **File>Export>DXF File**, there is now added support for exporting point clouds, as well as controlling whether or not point labels are included.

Direct CAD Access Settings: Reset to Defaults

In the ***Direct CAD Access Settings*** dialog, a new Reset to Defaults button restores the settings to factory defaults.

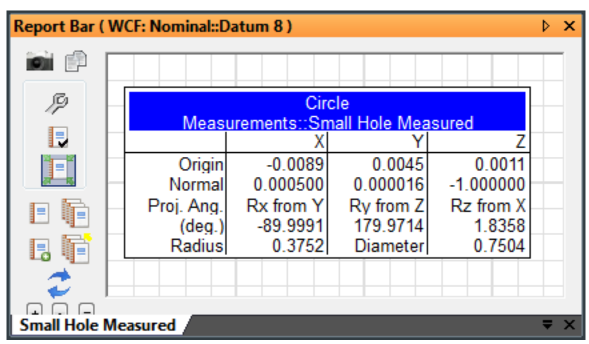
USMN Dialog

A number of minor improvements have been made to the USMN dialog. You can now right-click any instrument in the instrument list of the USMN dialog to access its properties or to check/uncheck all instruments. The point list has checkboxes for easily removing points from the USMN solution, and you can right-click any point in the point list to access its properties or to check/uncheck all points from the solution. Right click menu support has been added to the properties for the instrument list as well as the point list. Simply right click and you now have the option to select/unselect all. Check boxes have also been added to determine active points.



Working Frame in Report Bar

The report bar now displays the name and collection of the working coordinate frame (see below).



Remove Last Point Assignment

Remove Last Point can now be assigned to a function key in the instrument interface. This also means that this function can be assigned to a programmable remote, such as a T-Probe or RF Remote.

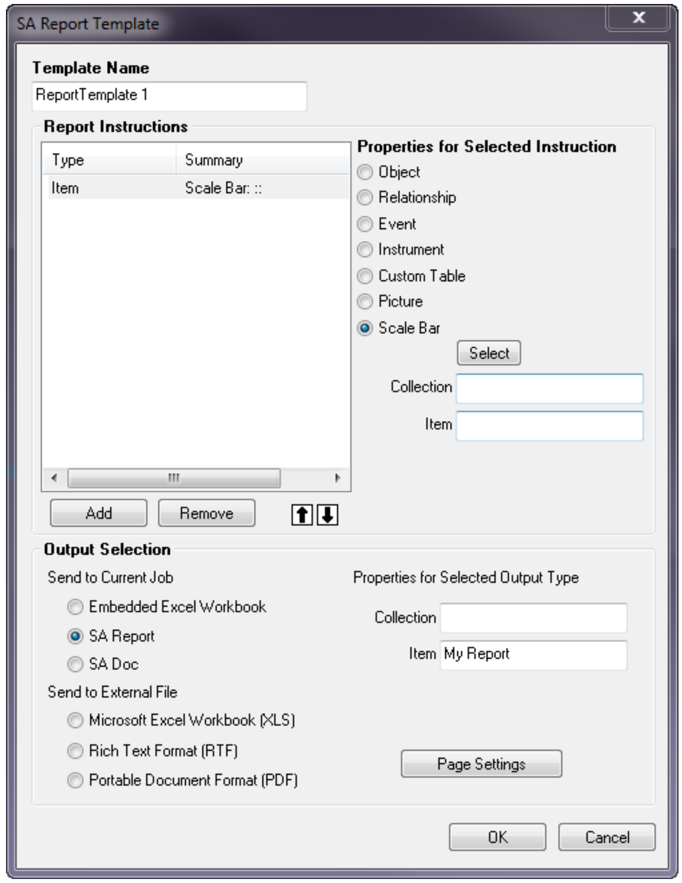
Reporting Improvements

Pictures

Pictures within reports may now be rotated clockwise in 90 degree increments. Simply right-click an image and select **Rotate 90** from the context menu.

Templates

Report templates now support scale bars, so they can be added to a template (see below).



Movement Constraints

Object movement in a report can be constrained to the horizontal or vertical direction by holding the **SHIFT** key down while dragging an object. This permits more precise alignment and positioning on reports.

Copy/Paste

Report items can be copied/pasted using the new **Copy** and **Paste** context menu items (see below).

Export to Excel

Single and multiple items can be exported from an SA report to an Excel spreadsheet using the new **Export to Excel** context menu (see below).

Measurement Plan Improvements

In the **Utility Operations>Delay for Specified Time** command, you can now suppress the display of the progress dialog using the argument *Show progress dialog?*

The **Reporting Operations>Set Report Options for Object** command now supports relationships.

The **Instrument Operations>Instrument Operational Check** command supports several new operations:

- **"Show Big Group/Target Window".** Displays the resizable group/target window, which remembers its size and position onscreen. This will return SUCCESS if the dialog is already showing.
- **"Close Big Group/Target Window".** Closes the resizable Group/Target window if it is open.
- **"API DI Enable iVision Single SMR".** Enables iVision Single SMR mode.
- **"API DI Enable iVision Multi SMR".** Enables iVision Multi-SMR mode.
- **"API DI Disable iVision".** Disables iVision functionality.

A number of new commands have been added to Measurement Plans. These include:

- **Variables>Set Collection Object Name Variable**. This creates a variable of type collection object name.
- **Variables>Get Collection Object Name Variable**. This retrieves the value for a variable of type collection object name.
- **Construction Operations>Callouts>Create Vector Callout**. Creates a vector callout in a callout view.
- **Construction Operations>Callouts>Create Point Comparison Callout**. Creates a point comparison callout in a callout view.
- **Construction Operations>Callouts>Create Relationship Callout**. Creates a relationship callout in a callout view.
- **Construction Operations>Callouts>Delete Callout View**. Deletes a callout view.
- **Construction Operations>Callouts>Rename Callout View**. Renames a callout view.
- **Utility Operations>Create Robot Calibration**. Creates a robot calibration.
- **Utility Operations>Delete Robot Calibration**. Deletes a robot calibra-

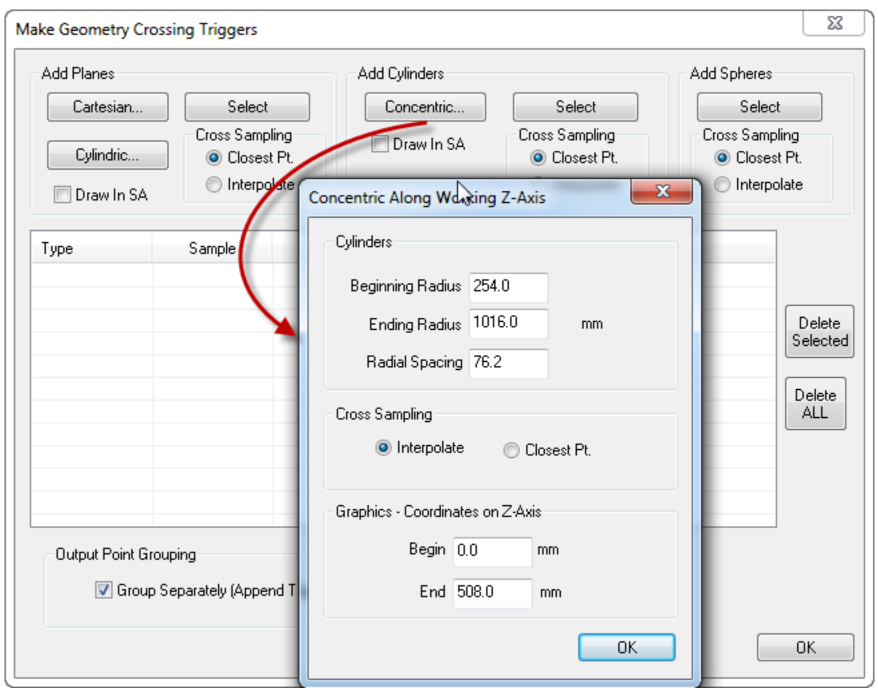
tion.

- **Utility Operations>Import Poses Match to Measurements**. Imports a set of poses and associates them with measurements.
- **Utility Operations>Perform Robot Calibration**. Performs a robot calibration procedure.
- **Utility Operations>Start/Stop Robot Calibration Trapping**. Starts or stops measurement trapping for a robot calibration.
- **Utility Operations>Make a Collection Instrument ID from a Collection and an Integer**. Creates a value of type collection instrument ID from a collection and integer.
- **Utility Operations>Make a Collection Machine ID from a Collection and an Integer**. Creates a value of type collection machine ID from a collection and an integer.
- **Utility Operations>Make a Collection Instrument ID - Runtime Select**. Prompts the user to select a single instrument, from which a collection instrument ID is retrieved.
- **Utility Operations>Set Active Robot Calibration**. Sets a specific robot calibration to be the active calibration.

Geometry Triggers: Concentric Cylinders

Concentric cylinders can now be used as geometry triggers in laser tracker, portable arm, and NDI OptoTrak instrument interfaces. Set *Closest Point* or *Interpolate* at the time of concentric cylinder creation.

`



API Device Interface Upgrade

The API Device Interface has been updated to version 4.6.1.0. This version provides support for new Innovo camera functions, namely the ability to enable/disable the camera in single or multi-SMR mode.

TTL triggered data can also now be buffered as well. The buffer size can be set in the DI settings dialog.

All emScon Trackers

Updated the TPWizard SDK to v.2.0.0.4978 for AT-401 and AT-901 trackers. Results from all checks/compensations will be added to the instrument History in SA (right-click on the instrument in the tree, and select **History** from the context menu).

Deeper support for external triggering has been incorporated into the instrument interface, such as the ability to set the Minimal Time Delay, Clock Transition (positive or negative edge), and Trigger Start Signal (high or low active). For faster triggering (External Source and Event Trigger Mode), you need to run a measurement profile with a Temporal Scan acquisition, and be sure to note what the operation is sending to SA. For more information on external triggering, see the Laser Tracker notes in the Readme file for SA 2011.12.22.

Leica AT-401

Automeasure now uses a routine that improves speed of an automeasure operation.

A new outdoor measurement mode is supported, and can be toggled from the **Settings>Leica 4xx** menu. When on, this measurement mode overrides the FAST, STANDARD, and PRECISE modes, no matter which is set. This mode is used for all measurements until toggled off. (The Power Lock Mode settings—*Indoor*, *Outdoor <80m*, and *Outdoor >80m*—have been deprecated by this new Outdoor Measure Mode.

For other improvements, changes, and fixes, please refer to the SA Readme file.

SA 2013.01.09

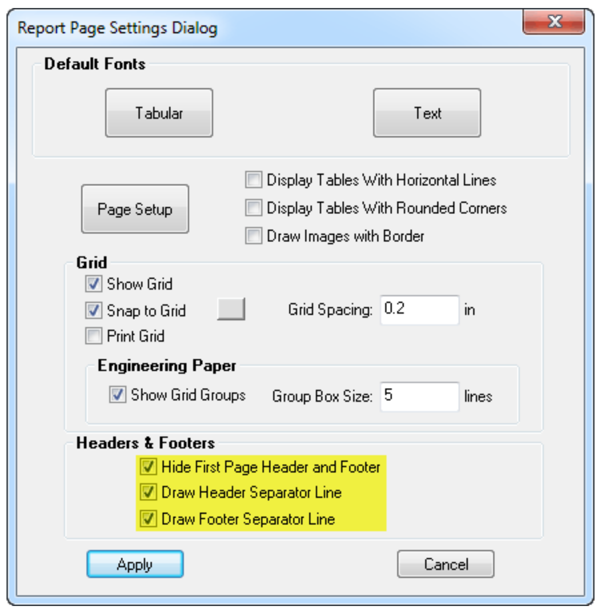
Jump To Views

You can now jump back and forth through previous views using **View>View Control>Go to Prev View** and >**Go to Next View** or using the **Ctrl+Alt+←** and **Ctrl+Alt+→** keyboard shortcuts. This allows you to jump to a previous (or next) view on which you remained for at least 1 second.

SA Report Header and Footer Improvements

The SA ***Report Page Settings*** dialog has new header and footer settings. A new *Hide First Page Header and Footer* option will prevent the header and footer from displaying on the first page so that you can include a customized first page such as a title page.

There are also two new options which, when enabled, will draw the separator line below the header or above the footer: *Draw Header Separator Line* and *Draw Footer Separator Line*.



Relationship Trapping by Selection

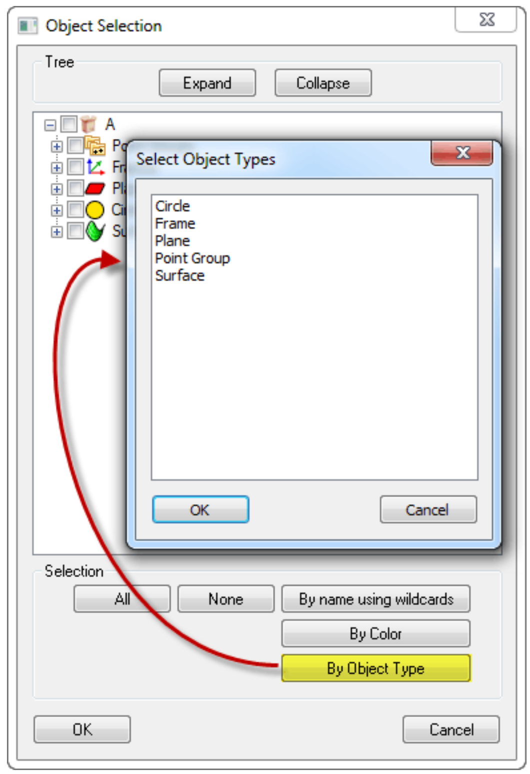
You can now trap points to a relationship by selection. For example, with a circle geometry relationship, if you want to add points to the circle fit, you can individually click points to add to the fit without having to reselect all of the points.

To trap additional points by selection, right-click a relationship in the tree and select **Trap Selections** from the context menu. Then click points to add to the relationship. When finished, press **ESC** or **Enter**.

F2 Selection by Object Type

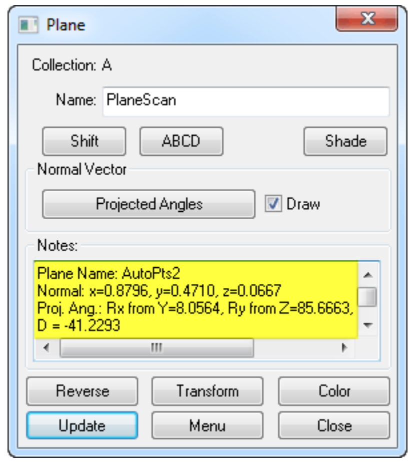
In object selection commands (such as **Delete Object**), you can now F2-select objects by type. For instance, you can quickly select all circles in the ***Object Selection*** dialog by clicking the new By Object Type button. When the ***By Object Types*** window appears, select one or more object types from your job, then click OK. All objects of that type will be selected.

The ***By Object Types*** window allows toggle selection, so that multiple types can be selected simultaneously. Simply click a row to toggle its selection state.



Fit Results from Instrument Interface

When geometry is created via a measurement profile, the results of the fit (residuals, weighting, etc.) are now stored in both the geometry's notes and the instrument history. Data is stored for up to 500 points to control file size.



Steinbichler AC File Format Support

Files formatted in the .AC file format that originated from a Steinbichler scanner (such as a T-Scan) can now be imported via the **File>Import>Custom Formats>Steinbichler AC File** menu.

T-Mac Tracker Pilot Support

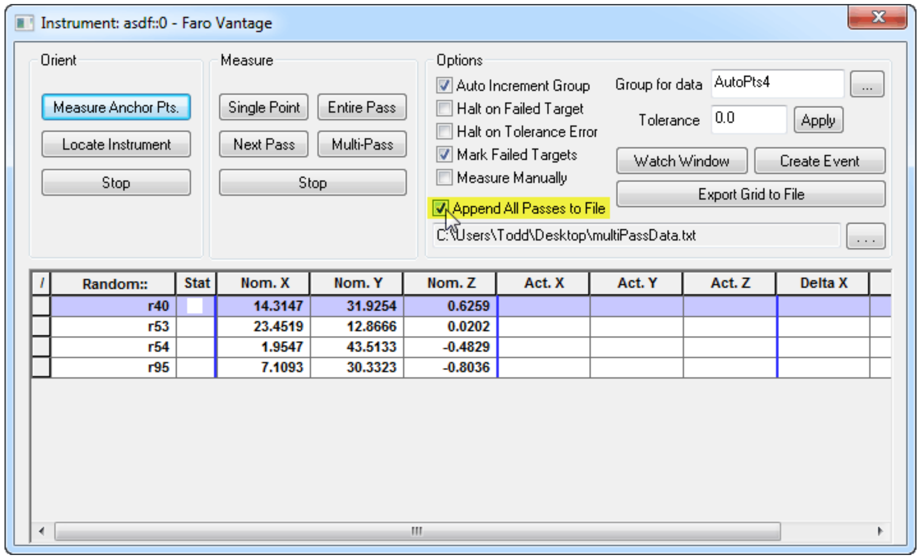
Support has been added for Leica's Tracker Pilot T-Mac to run manufacturer checks and compensations. If you have emScon v.3.8 or higher, you will need to install Leica Tracker Pilot T-Mac. This augments the check/configure/compensate functions found in the emScon Web App Server.

To run these new routines, select **Check/Cal>Tkr Specific (Mfcr)** from the tracker interface. You will be given the choice of running the T-Mac Tracker Pilot functions, or the Web App. As with the 401, the interface will disconnect from the tracker before running TP commands, and reconnect after.

If you are running an emScon version lower than 3.8, the Web App will be launched immediately, as always.

Multi-Pass Auto-Measure File Export

While performing automatic measurement using the **Instrument>Automatic Measurement>Auto Measure** command, a new *Append All Passes to File* option allows you to specify a file for export. SA will then append each pass of automatic measurement to a file.

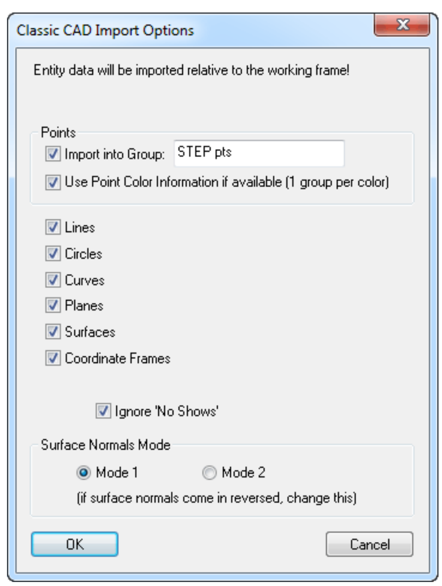


Relocating Pictures to Another Collection

Pictures can now be moved to different collections. You can move the whole collection's pictures by right-clicking the `Pictures` category in the tree and selecting **Move all to another Collection**, or an individual picture by right-clicking the picture and selecting **Move to another Collection** from the context menu.

Standard CAD Import: Match Point Color

When importing a standard CAD file via **File>Import>Standard CAD Formats**, a new *Use Point Color Information if available (1 group per color)* option will set the group of an imported point to match the same color as the source CAD file's point, if that color is available. Of course, since a point group can only have one color, this option will create a new point group for each color, and organize the points into different groups by color.



LPT Laser Projector Enhancements

A new Browse button has been added next to the online projector part. This allows you to upload parts from the current job's part database, and fully enables 2-way communication with the parts database. Plies are still added to the current part in the database as you project from SA, as always.

At startup, the interface looks for the current projector part in the LPT database. If it is found, the part is automatically uploaded from the database to ensure synchronicity. This is also true for the MP "Set Part Name" command.

Units are now supported passively. The current job's units of measure are queried at start up time, and everything written to the database is written in those units. The job name and its units are now displayed in a new connection status indicator at the top of the main interface window.

RDS v3.6 Support

Support for v3.6 of Hexagon's RDS has been added.

New MP Commands

For details on these new MP commands, see the MP Command Reference.

- **Move Robot/Machine to Joint Pose (6DOF).** Permits you to explicitly set the joint values for 6DOF robots and machines.
- **Simulate Robot/Machine Path, Output CSV File.** This command will simulate a robot's path through a set of frames and export the joint values to a text file.
- **Set Vector Group Report Options.** Sets all of the report options for a vector group.
- **Set Relationship Report Options.** Sets all of the report options for a relationship (group to group, points to objects, clouds to objects, and point to point).
- **Set Point Delta Report Options.** Creates report options which can then be referenced by the *Set Vector Group Report Options* or *Set Relationship Report Options* commands.

For additional improvements, changes, and fixes, please refer to the SA Readme file.

SA 2012.12.06

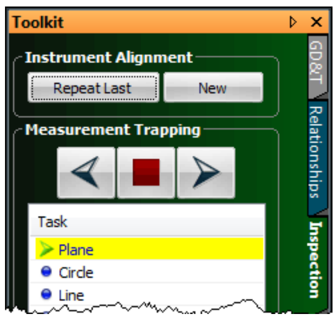
Repeat Alignments

Any instrument's alignment can now be repeated, making alignments as part of a repeatable process easier to set up.

Whenever an instrument is aligned using Quick Align, Measure Nominal Points, or the Frame Wizard, the alignment parameters are stored with the instrument. This alignment can then be repeated at any time.

To Repeat an Instrument's Alignment:

- In the *Inspection* tab of the SA Toolkit, click the Repeat Last button while connected to an instrument.

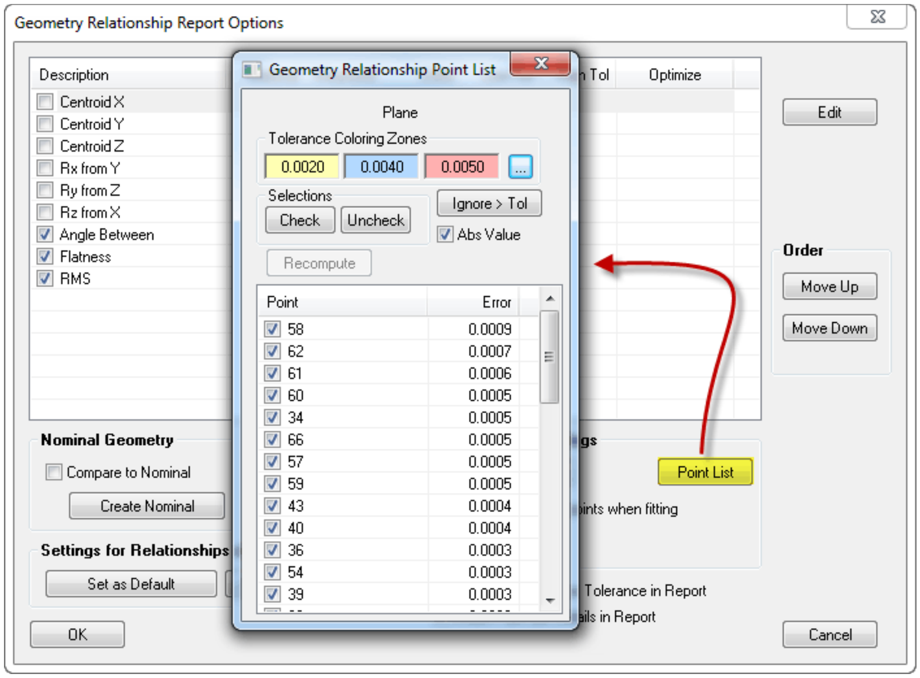


This will automatically recall the previously saved alignment parameters, which can then be modified or committed.

Clicking the New button is a convenient way to perform a new alignment on the connected instrument.

Geometry Relationship Point Lists

You now have access to and control over point lists for geometry relationships. Simply click the Point List button in the ***Geometry Relationship Report Options*** dialog. You can also access the point list by right-clicking a geometry relationship in the tree and selecting **View Point List**. Just as in the ***Geometry Fitting*** dialog, you can deactivate points, deactivate points exceeding a critical threshold, and sort by name or error.



A new option has been added to the ***Geometry Relationship Report Options*** dialog to permit the display of these points on a report. Simply select the *Show Point List Details In Report* option, and when the geometry relationship is added to a report, the point list will be included.

Stock Language Translation

Five stock languages are now included with SA:

- English
- German
- Spanish
- Russian
- Chinese

Set Viewpoint From Object

The new **View>View Control>Set Viewpoint From Object** command aligns the viewpoint down the -Z axis of the selected object.

Set Viewpoint From Surface Point

You can now align the view to look directly down the normal to a surface at a specific point. Select **View>View Control>Set Viewpoint From Surface Point**, then click on a surface. The view will realign to look directly down the normal of the surface at the selected point.

Reporting Improvements

Insert/Delete Page

Pages can now be inserted or deleted in an SA report. To do so, right-click in the report and choose **Insert Page** or **Delete Page**. These menu items are only available if the action is possible from the right-click location.

- Pages can only be inserted if there are no report items intersecting the horizontal line where the click was initiated.
- Pages can only be deleted if the page does not contain any report items.

Disabled Table Truncation

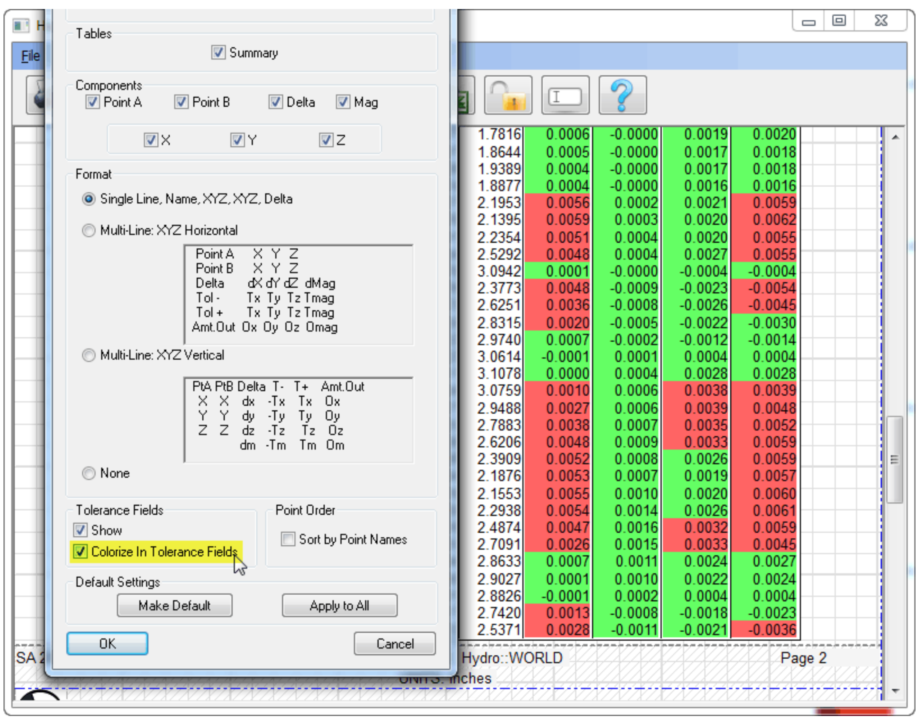
You can disable truncation on a per-table basis (regardless of the *Maximum Number of Rows Per Table* setting in the User Options *Reporting* tab) by right-clicking the table and selecting **Do Not Truncate** from the context menu.

Remove Trailing Empty Pages

In the Report Designer, there is a new option to remove all empty pages at the end of a report. Simply select **Edit>Remove Trailing Empty Pages**.

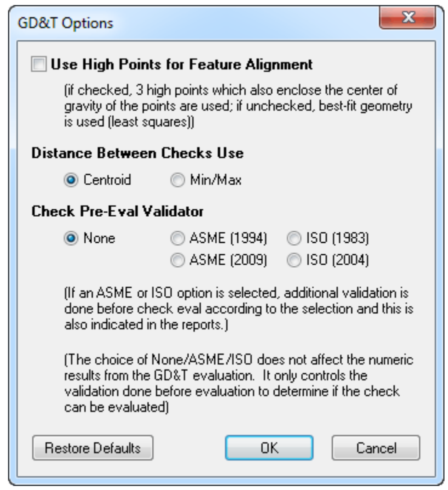
Colorization of In-Tolerance Fields

A new report option has been added for coloring in-tolerance delta fields green (see below).



GD&T Improvement: Check Pre-Eval Validator

In the User Options *Analysis* tab, in the ***GD&T Options*** dialog (accessible by clicking the GD&T Options button), a check Pre-Eval Validator has been added. This performs standard-specific validation checks on measurement data prior to a GD&T evaluation.



The GD&T evaluation results are equivalent regardless of setting, since the standards in that area do not differ. However, depending on which standard is active (ASME 1994, ASME 2009, ISO 1983, or ISO 2004), different checks may be performed on the initial data. This may result in a check failing even though it may have passed in an earlier version of SA.

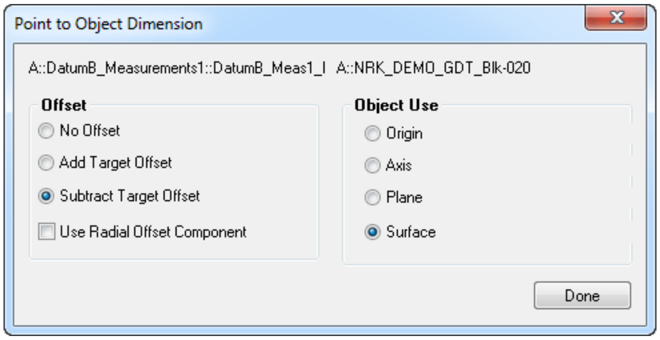
Note that the *None* option maintains the equivalent behavior of previous versions of SA. This setting is saved with the job file.

GD&T Improvement: Evaluate All Feature Checks

A new option has been added to force a recalculation of all feature checks in an entire job. Simply select **GD&T > Evaluate All Feature Checks**.

Point to Object Dimensions: Radial Offsets

Point to object dimensions can now compensate for radial offsets in addition to planar offsets. Simply check the *Use Radial Offset Component* option. This is useful, for instance, when determining the distance to an edge that was measured with a pin nest.

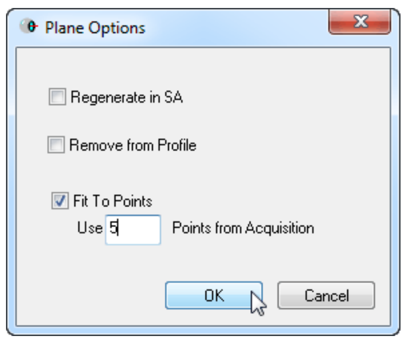


Laser Trackers: Projection Plane Settings

New settings have been established for measurement operations that accept projection planes, including:

- Circle
- Patch Point
- Single Point Circle

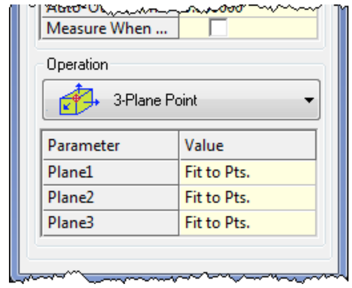
Left-click the projection or intersection plane option in the operation's parameters to select an existing plane to use. Right-click to specify options for automatically fitting the points while measuring:



- **Regenerate in SA.** This option automatically creates the construction plane after the profile completes.
- **Remove from Profile.** Clears an existing selected plane from the profile.
- **Fit To Points.** When checked SA will automatically fit a plane to the measured points. Specify the number of points to measure here.

Laser Trackers: 3-Plane Point

This new measurement operation in the laser tracker interface automates the process of measuring a "corner point"—a point that is constructed by measuring 3 intersecting planes. Many different acquisition modes can be used with this operation, including discrete point, stable point, and spatial scan.

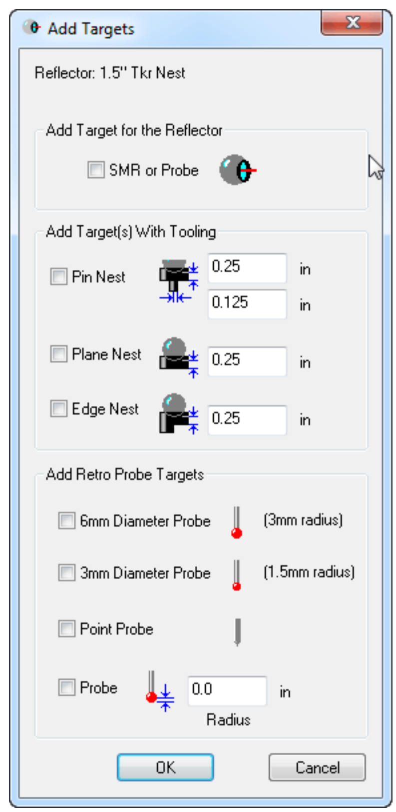


To define or measure the three planes, use the instructions in "Laser Trackers: Projection Plane Settings" on page 66.

Laser Trackers: Adding Targets with Custom Radial Offsets

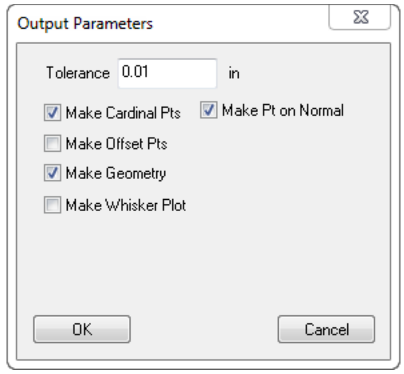
When adding defined targets in the laser tracker interface using the Add: From Selected Reflector button, you can now specify both the planar and radial offsets. This allows you to set these offsets in this dialog instead of the *Targets* table.

The default values for these fields will change depending on the size of the selected reflector and your current active units.



Laser Trackers: Circle Fit Point on Normal Option

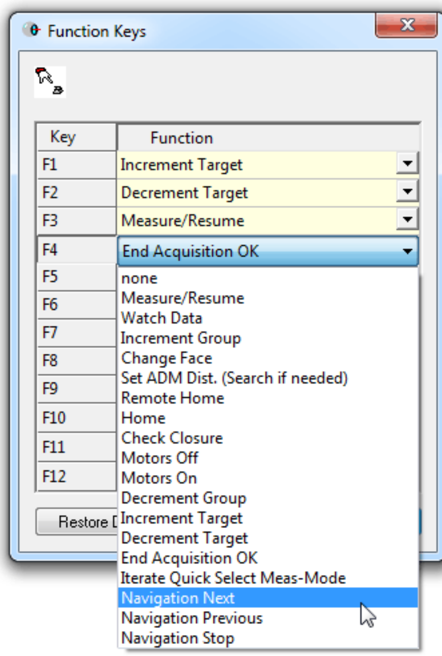
When measuring a circle in the laser tracker interface or fitting a circle using Measurement Plans, the circle fit output parameters now have a *Make Pt on Normal* option.



When checked (if *Make Cardinal Pts* is enabled), the fit will create a cardinal point on the circle's normal (existing behavior). You can now turn this option off to create a cardinal point only on the circle center.

Inspection Routine Navigation Enhancements

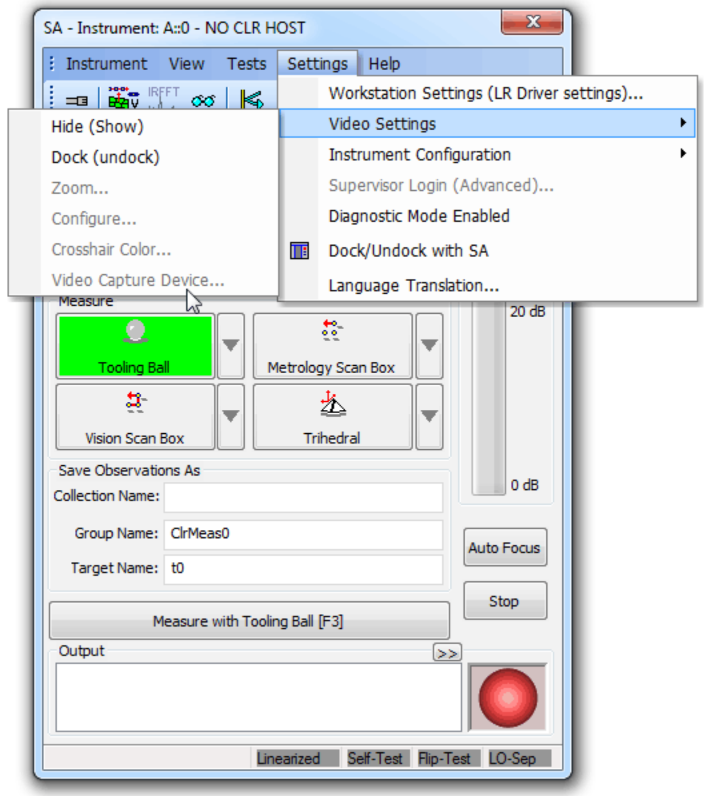
You can now program function keys (or remotes that emulate function keys) for "Next", "Previous" and "Stop" navigation operations, primarily to allow easier progression through an inspection routine in the SA Toolkit.



You can now also program CMM arm buttons for Next/Previous to correspond to a short/long button press.

Laser Radar: Video Source Selection

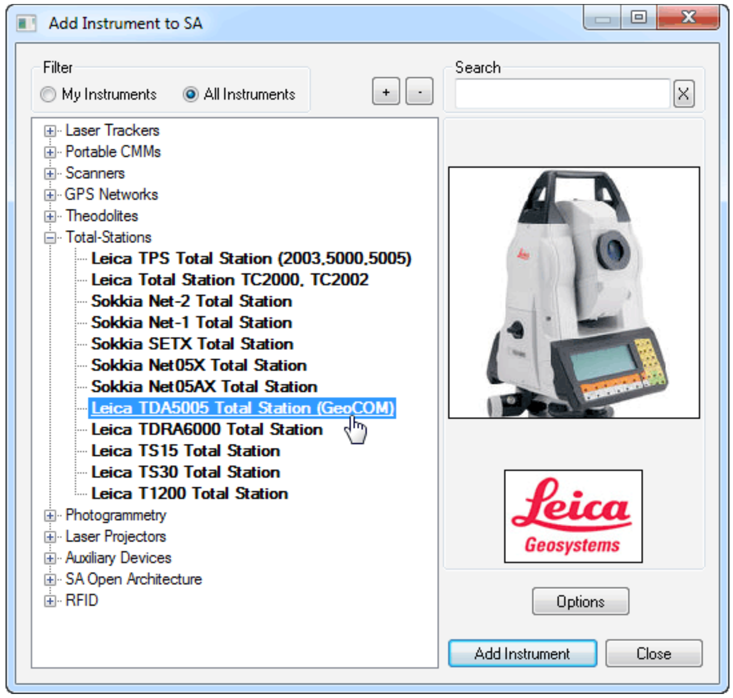
In the Laser Radar interface, you can now explicitly choose the video source. From the interface, select **Settings>Video Settings>Video Capture Device.**



Leica TDA5005 GeoCOM Interface Added

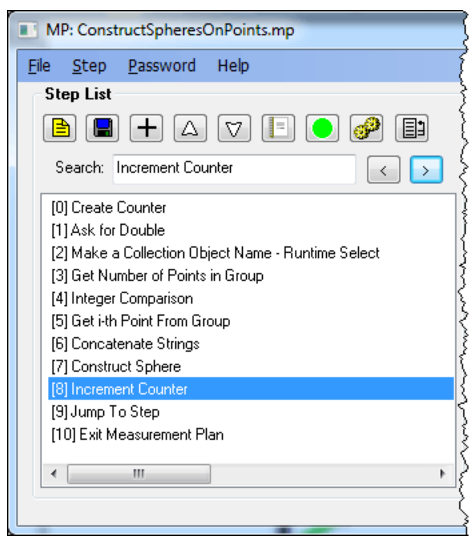
A GeoCOM-based interface has been added for the Leica TDA5005 that does not require the use of the NRK embedded SpatialAnalyzer application. This interface looks similar to the TDRA6000 interface. For this to function properly, the instrument must be added as a 'Leica TDA5005 Total Station (GeoCOM)' type (see below).

For the GeoCOM mode to work properly, the instrument must be in the 'MEAS' program, running from its top level menu. Entering lower-level menu items within the program will cause the device to cease GeoCOM communication until returning to the top level menu. The TDA5005 should be configured with GSI Communication: 9600/N/8/1/No Protocol and GeoCOM Communication: 9600/N/8/1/GeoCOM Protocol.



MP Script Searching

The MP Editor now has a *Search* field which allows you to search the current script to find a command or comment (below).



This can save a lot of time when working on a lengthy script.

Measurement Plan Commands

A number of new commands have been added to Measurement Plans. These include:

- **View Control>Show/Hide Instrument Probe Tip.** Shows or hides the probe tip in the graphical view.

- **View Control>Show/Hide Points.** Shows or hides individual points.
- **Construction Operations>Points and Groups>Copy Group Excluding Obscured Points.** Creates a new group of points by excluding points from source groups that are not visible to a specific instrument (because they are obscured by surfaces).
- **Analysis Operations>Get Cone Properties.** Obtains the properties of a cone.
- **Instrument Operations>Delete Measurements.** Deletes a set of specified measurements from an instrument.
- **Instrument Operations> Advanced Instrument Operations> Issue Instrument Actuator Command.** asfd
- **Instrument Operations> Advanced Instrument Operations> Set Instrument Axes.**
- **Instrument Operations> Advanced Instrument Operations> Set Alignment Projector.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Hardware Connect.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Hardware Disconnect.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Verify Hardware Connection.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Set Red Laser Intensity.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Self Test.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Self Test - Linearization.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Self Test - Flip Test.**
- **Instrument Operations> Nikon Metrology - Metris Laser Radar (LR)> LR Self Test - LO Sep.**
- **Instrument Operations> Start Instrument Interface.** Added a "Run in Simulation" argument to force immediate simulation mode, useful for testing scripts.
- **Instrument Operations> Locate Instruments (USMN).** Added arguments for "point groups to exclude" and "exclude points measured by only one instrument."
- **Relationship Operations>Make Group to Group Relation-**

ship. Added arguments for placing tolerances and constraints on the relationship.

- **File Operations>File Import>Import Leica GSI File.** Imports the Leica GSI file format.
- **View Control>Show Labels.** Independent control is now provided for turning point labels and object labels on or off.
- **Process Flow Operations>Ask For ...** Font control is now provided in the "Ask For ..." commands.

For additional improvements, changes, and fixes, please refer to the SA Readme file.

SA 2012.09.14

Threaded Projection

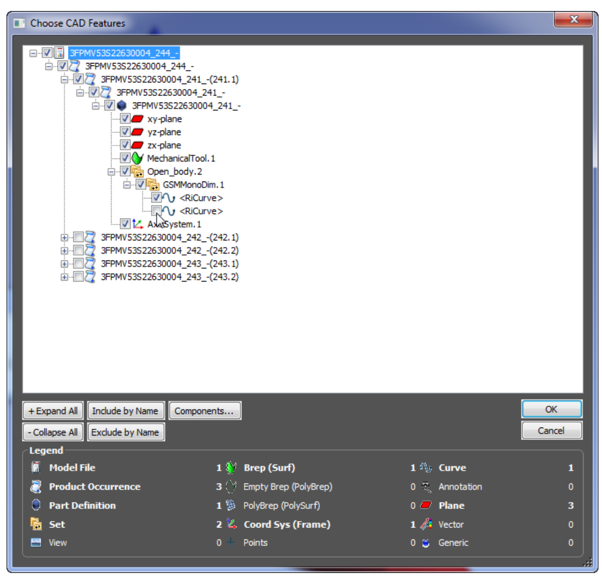
Multi-threaded processing has been added for surface projection operations. Any operation that involves projection to surfaces is now up to several times faster! This includes operations such as:

- Relationship fitting
- Queries and vector group creation
- Feature Inspection Auto-Filter
- Auto-Filter to Faces

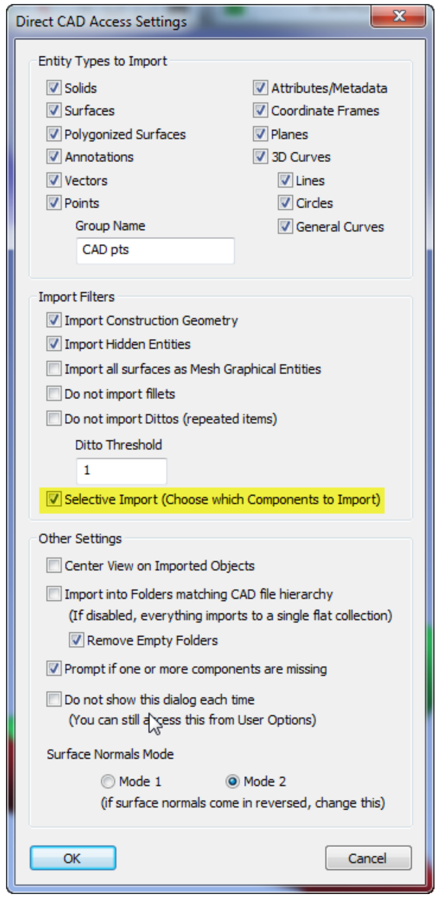
If your machine has multiple processor cores, you will see a speed improvement unless multi-processing is disabled in the SA user options (it's enabled by default).

Selective CAD Import

For CAD formats that support part assemblies (such as CATIA's `.CatProduct` files), you can now choose which individual components in the assembly to import, instead of making an attempt to import the entire assembly. Moreover, the contents of any CAD file being imported are displayed before import so that you can choose which specific items to include in the operation.

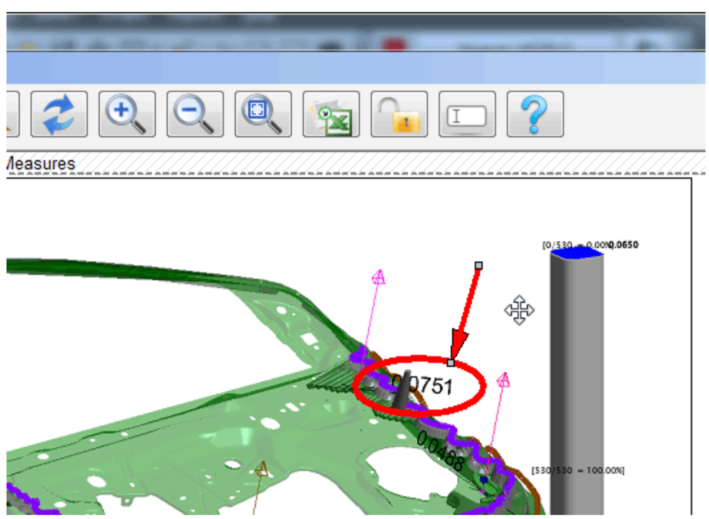


To access this capability, import a CAD file using Direct CAD Access. In the ***Direct CAD Access*** dialog, ensure that the *Selective Import (Choose which Components to Import)* option is checked (see next page).



Report Shapes

To help with annotating your reports, you can now draw boxes, ellipses, and lines (or arrows) directly on reports. Draw in any color or using several different line thicknesses or line styles.



Report Object Locking

Individual items on reports can now be locked, which "freeze-dries" values and prevents those items from being modified in the report designer. Simply right-click an object on a report and select **Locked Object** from the context menu to toggle the object's locked state.

All Vectors Summary: Vector Group
Measured::Auto Vectors: Points to TopSideSurface

Statistic	dX	dY	dZ	Mag
Min	-0.0038	-0.0363	-0.0211	-0.0558
Max	0.0456	0.0337	0.0538	0.0337
Average	0.0030	0.0000	0.0022	-0.0037
StdDev from Avg	0.0079	0.0057	0.0118	0.0153
StdDev from Zero	0.0084	0.0057	0.0120	0.0158
RMS	0.0084	0.0057	0.0120	0.0157
Tol Range				-0.0650 0.0650
In Tol				530 (100.0%)
Out Tol				0 (0.0%)
Count	530			

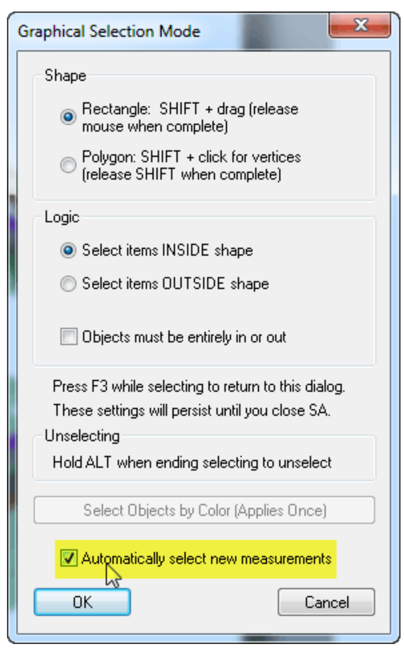
Report Undo

Undo support (**Edit>Undo** or **Ctrl+Z**) has been added to the report designer.

Automatic Measurement Selection

Note: Even with this mode active, selection via the graphical interface still works normally.

The ***Graphical Selection Mode*** dialog (**View>Graphical Selection Mode**) has a new option, *Automatically Select New Measurements*. When enabled, incoming point measurements are automatically applied to the current selection. For example, with this option enabled, if you'd like to construct a line between two points, you can select **Construct>Line>2 Points** from the menu, them measure two points with an instrument, and the line will be created between them. When this mode is active, an `[M]` will appear in the command prompt and the feature navigation buttons will accept a prompt (next) or cancel/back up a prompt (previous).



New Commands

A number of new commands have been added, including the following:

- **Construct>B-Splines>From Circles.** Creates a B-Spline from a circle.
- **Edit>Delete Cloud Points>By Radial Distance From Points.** Enables you to quickly delete cloud points inside of (or outside of) a specified distance from a set of reference points. This is particularly useful for deleting registration sphere data from a scan once sphere centers have been extracted.
- **Relationships>Geometry Comparison>Select Nominal Geometries.** Creates a geometry relationship with the specified nominal geometry. (This command supports **F2** selection).
- **Construct>Point Clouds>From Existing Point Clouds Uniform Spacing.** Takes source point clouds and creates a new cloud from them with as close to even cloud point spacing as possible (you can specify the desired spacing).
- **Construct>Points>From Cloud Points>Pick Clouds.** Creates point from source cloud points using a desired subsampling value.
- **Query>Point to>Point along a Curve.** Provides the distance between two points along a specified curve.

New Measurement Plan Commands & Features

- **Analysis Operations>Query Point to Point Along Curve.** Provides the distance between two points along a specified curve.
- **Construction Operations>Points and Groups>Construct Points Auto-Correspond 2 groups Proximity.** Copies a group of points to be renamed and renames the copied points based on proximity to reference points.
- **Construction Operations>Points and Groups>Construct Points Auto-Correspond 2 groups Inter-Point Distance.** Copies a group of points to be renamed and renames the copied points based on the inter-point distance method.
- **Other MP Types>Make Projection Options.** Creates projection options for use with vector groups.
- **Other MP Types>Make a Transform from Doubles (Matrix Elements).** Creates a transform data type from 16 source matrix values.

- **Other MP Types>Decompose Transform into Doubles (Matrix Elements).** Extracts the 16 transform matrix values from a transform data type.
- **Reporting Operations>Delete Picture.** Deletes a picture from the SA file.

The MP Editor now supports drag and drop loading of MPs from external files. Simply drag an MP file onto the MP editor, and it will be opened.

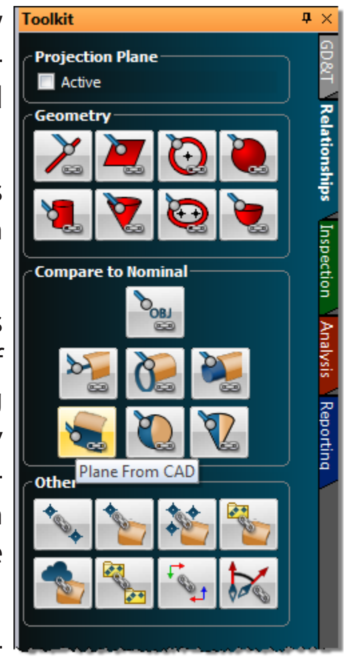
A new button, Show Step References, has been added which shows the steps that reference the selected step.

For a list of dozens of other updates and changes, refer to the SA Readme file.

SA 2012.07.09

SA Toolkit Bar

The SA ***Toolkit*** bar provides a set of new tabs for quick access to functionality related to GD&T, part inspection, analysis, and reporting.



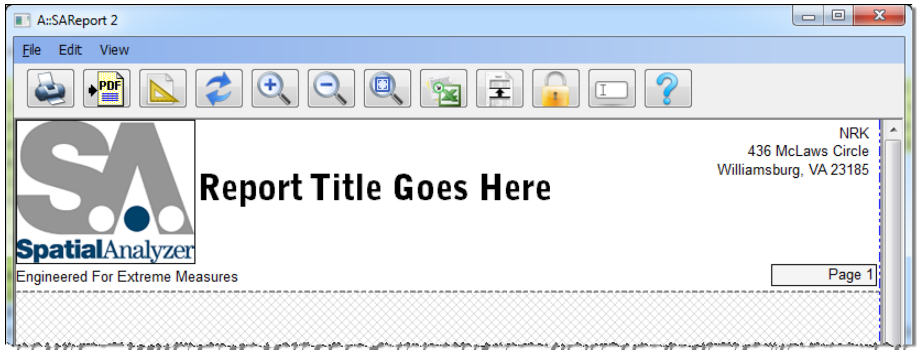
- **GD&T.** What was formerly known as the GD&T bar has been brought on as a tab in the Toolkit bar.
- **Relationships.** The relationships tab enables one-click definition of features (defined by CAD or using primitive geometry) primed for easy inspection using geometry relationships, with or without projection planes. Other relationships can be easily defined as well.
- **Inspection.** The inspection tab presents all trappable features from all collections—relationships, datums, and feature checks—in a single list. This allows you to easily advance through an inspection routine, trapping measurements for each feature, and instantly providing inspection results against nominal data.
- **Analysis.** Some of the more commonly-used analysis tools, such as the frame wizard and geometry fitting, are presented together in the analysis tab.
- **Reporting.** Functionality for creating reports, making dimensions, and creating/annotating callout views is gathered together in the reporting tab.

To display the Toolkit bar, right-click on the menu/toolbar/status bar area of the SA interface and select **Toolkit**.

Reporting Enhancements

Headers/Footers & Templates

You can now define custom headers and footers for your own reports, including elements that update automatically (such as SA version, page number, and date). Headers and footers can contain anything a report body can contain: images, text, tables, and fields—simply drag elements as you would anywhere else on the report (see following image). While in the header/footer edit mode (accessible by double-clicking a header/footer, right-clicking a header/footer or by selecting **Edit>Modify Header/Footer** from the report window), you can resize the header/footer by dragging its divider.



When you're happy with your header and footer, exit the mode by selecting **Edit>Modify Header/Footer** again. This will lock the header and footer in place, although dynamic fields will update as usual. If you'd like to set the report as a template for future reports, right-click the SA report in the tree and choose **Default Template**, or select **File>Make Default Template** from the report window. This can also be done through the **User Options>Reporting** tab. Any additional reports created in the file will match the template's format.

To use the template in other files, you can import the SA file containing the report template into the other file.

Keyboard Zooming

You can now zoom in and out in the report window using the + and - keys. Zooming to 100% can be accomplished by pressing `Enter`.

Jump to Page

Quickly jump to any page in your report by pressing `Ctrl-G` (for Go) while in the report window.

Image Border Control

Disable the display of borders around images from the report's page settings options by clearing the *Draw Images with Border* checkbox.

Page Settings for Report Templates

Page settings for report templates can now be controlled from the ***SA Report Template*** properties dialog. In that dialog, click on the Page Settings button to define the parameters for the final generated report.

Assigned Reporting Frames

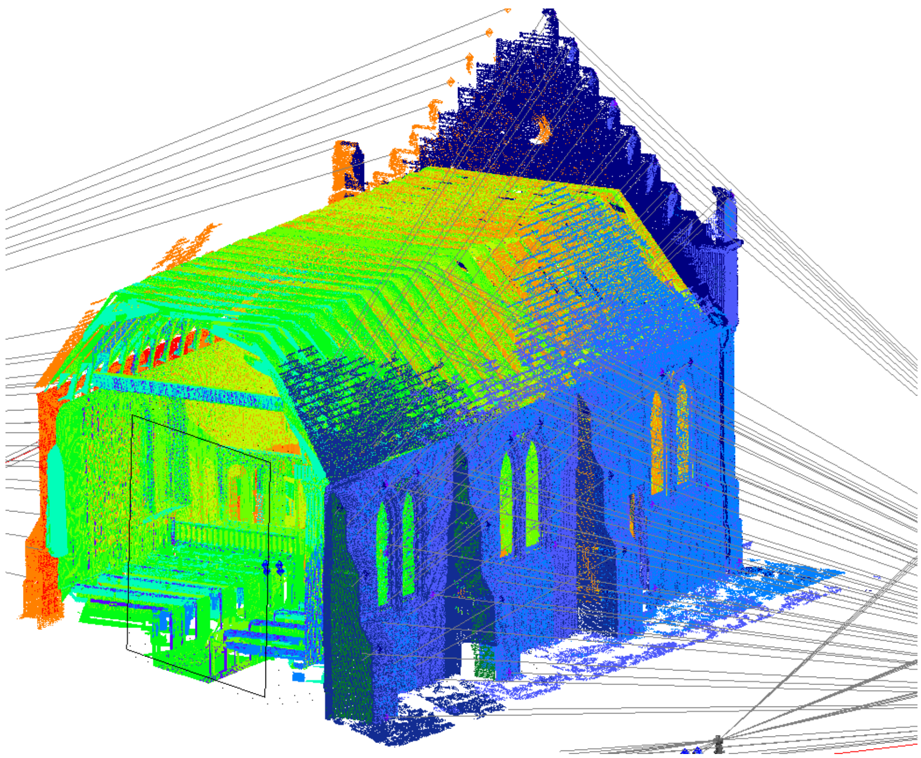
Individual objects can now be reported/displayed with respect to a specified coordinate frame—not necessarily the active frame. This allows you to have any coordinate frame active while always viewing a given object relative to its assigned coordinate frame.

To assign a specific reporting frame to an object, right-click the object and select **Reporting Frame** from the context menu.

Be aware that the reporting frame applies to reported values in the report bar and reports only. For example, when double-clicking a point to view its coordinates, you will always see the coordinates relative to the active coordinate frame, regardless of the point group's assigned reporting frame.

Clipping Planes

Support for full clipping planes has been added to the graphical view. This is a great option when you want to display a cutaway or cross-section of your data. You can define as many clipping planes as you'd like, and any of them can be active at any one time.



Any number of objects can define the position and orientation of clipping cubes, and each clipping cube can have up to six clipping planes. You can interactively drag and resize the clipping box to reposition the clipping planes.

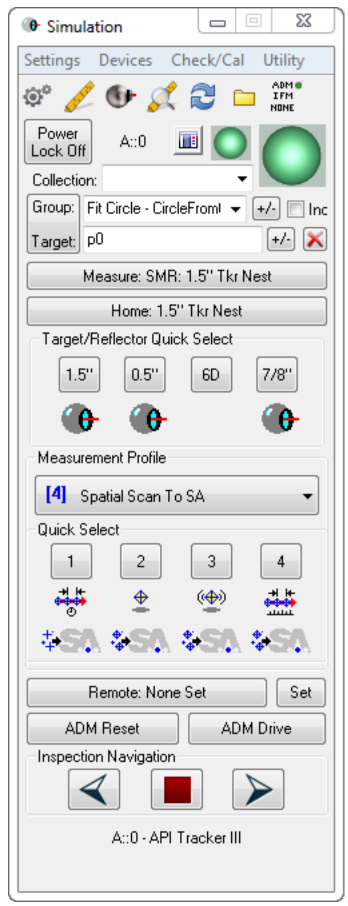
GD&T: Fit to Points

A new creation option has been added to GD&T: *SA Objects (fit to points)*. This enables you to create a form or orientation GD&T check directly from measurements, instead of creating nominal geometry first.

For example, to perform a circularity check on a set of measured points, ensure that *SA Objects (fit to points)* is selected. Click the circularity button, select the measurements, then press **Enter**. The circularity results will be calculated immediately.

Tracker Interface

A number of enhancements have been made to the tracker interface in order to better communicate current target and measurement profile settings and provide access to common functions, as displayed below.



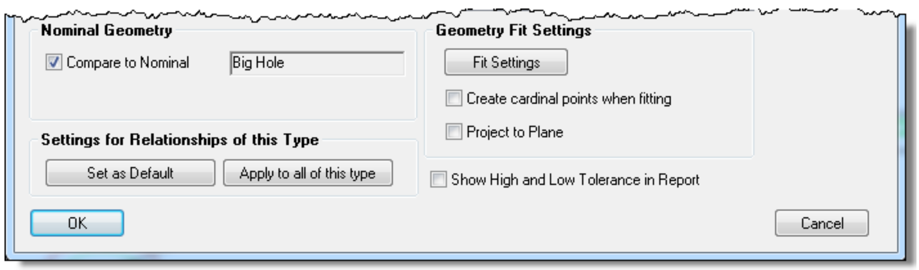
- The *Target/Reflector Quick Select* box now displays text on the four pre-select buttons indicating the current target instead.
- An icon indicative of the current tooling setting is displayed underneath the quick-select buttons. These images indicate if the assigned setting is just for an SMR or is defined for use with additional tooling (such as a flat nest or pin nest).
- The four measurement profile quick select buttons now display icons underneath them indicating both the acquisition and operation modes for the profile assigned to each button.
- With all quick select buttons, you can hover over them to dis-

play the full name of the assigned tooling definition/measurement profile.

- For trackers that support Leica's PowerLock, a new toggle button has been added to the left of the interface docking icon to enable you to quickly activate or deactivate this functionality.
- A new Inspection Navigation box has been added to the bottom of the interface which enables easy navigation through trappable features. Easily advance forward or backward to the next trappable feature, or stop trapping altogether directly from the tracker interface.

Geometry Relationship Features

Several changes have been made to geometry relationships that improve flexibility and functionality.



- **Projection planes can now be defined for line, circle, and ellipse geometry relationships.** This can be accomplished by either choosing the *Project to Plane* option in the relationship's properties (below), or clicking the *Projection Plane: Active* checkbox in the Relationships tab of the SA Toolkit bar. In the latter case, the selected projection plane will remain active and apply to all following geometry relationships until the checkbox is cleared.
- **Geometry Relationships can now be switched between "fit only" and "fit and compare" types.** By selecting the *Compare to Nominal* option (above), you can select nominal geometry to use for "fit and compare" relationships. If you simply want a basic geometry fit relationship, simply clear the checkbox.
- **Nominal Geometry can be created from the relationship properties.** If you have an existing "fit only" relationship, you can create nominal geometry for it and convert it to a "fit and compare" relationship by clicking the Create Nominal button in the relationship's properties.
- **New menu item for fit and compare relationships.** A new menu item (**Relationships>Geometry Comparison>Select Nominal Geometries**) has been added, along with a button on the SA Toolkit's new *Relationships* toolbar, that creates fit and compare

relationships from SA objects such as planes and circles. This is similar to **Relationships>Geometry Comparison>Fit and Compare to Nominal**, but doesn't prompt for existing measured points.

- **New option to clear point associations.** Point associations for any type of relationship can now be cleared by selecting the new **Clear Point Associations** context menu item. This is available for both individual relationships and the parent relationship category in the tree. (From the SA Toolkit's *Relationships* tab, you can also choose to delete the cleared points from the file).
- **Access to fit settings prior to measurement.** Geometry relationships now allow access to fit settings prior to assigning points, so you can now prepare the relationship in a template file prior to measurement.

Quick Align and Best-Fit Transformation with Cloud Points

Quick Align and classic best-fit transformation can now be performed directly with cloud points. This is intended to be used to align cloud data to CAD surfaces after measurement, and has been implemented to avoid the need to first convert clouds to points for use with the existing commands. Three new menu items have been added to support this: **Analysis>Best-Fit Transformation>Point Clouds to Surfaces/Objects>Quick Align Clouds to CAD**, **Analysis>Best-Fit Transformation>Point Clouds to Surfaces/Objects>Point Clouds to Objects**, and **Instrument>Locate>Quick Align Clouds to CAD**.

Frames from Probe Measurements

Frames can now be constructed from probe measurements (from devices such as the Intelliprobe 360 or T-Probe) if the measurements have probing direction information. Use the new command **Construct>Frames>From Point Measurement Probing Frames**.

Show Points

Selected points can now be shown in one command using the new **View>Show Points** command.

New MP Commands

For details on these new MP commands, see the MP Command Reference.

- **Reporting Operations>Report Bar>Add Custom Tables to Report Bar.** Adds one or more custom tables as tabs in the Report Bar.
- **Reporting Operations>Custom Report Tables>Set Custom**

Table Cell Font. Sets the font for a specific cell in a custom table.

- **Reporting Operations>Custom Report Tables>Delete Custom Table.** Deletes a custom table from the tree.
- **Reporting Operations>Set Scale for Picture.** Assigns a scale value to a picture to control its size when added to a report.
- **Reporting Operations>Append Items to SA Report.** Sequentially adds one or more items to an SA report.
- **Reporting Operations>Make New SA Report.** Creates a new, empty SA report (using a defined template, if one exists).
- **Reporting Operations>Output SA Report to PDF.** Exports an SA report to a PDF file.
- **Utility Operations>Copy Directory.** Copies a directory from one location on the file system to another, optionally replacing the destination folder's contents.
- **Utility Operations>Set Notification Cancel Override.** When enabled, the Cancel button and "X" in the corner of the window are not displayed in dialogs resulting from any "notification" command (such as Notify User String).
- **Utility Operations>Set Automatic Backup State.** Enables or disables full file and measurement automatic backups.
- **Utility Operations>Move Robot/Machine through Path.** Commands a robot/machine to pass through a sequential set of coordinate frames, optionally interpolating between those frames as linear segments or as a smooth spline.
- **View Control>Set View Clipping Plane.** Creates a clipping plane on a specified object.
- **Construction Operations>Frames>Construct Frame - Average of Other Object Frames.** Creates a frame with the average position and orientation of the supplied frames.
- **Construction Operations>Other MP Types>Decompose Transform into Vectors (Origin and Axes).** Converts a transform value into four vectors: one for the origin and one for each of the three axes defining the transform.
- **Construction Operations>Other MP Types>Make a World Transform Operator (from Transform and Scale).** Converts a separately supplied transform and scale factor in to a single world transform operator.
- **Process Flow Operations>Ask for User Decision Extended.** Provide a question or prompt and a series of choices to appear

in buttons. Clicking a button will jump to a specified step. Similar to "Ask for User Decision", except allows many more than three potential answers.

- **Analysis Operations>Query Frame to Frame.** Queries the delta between two coordinate frames.
- **Analysis Operations>Query Point to Objects.** Compares a point to one or more objects and returns the resulting deviation.
- **Analysis Operations>Remove i-th Object From Collection Object Name Ref List.** Removes the specified object from a collection object name reference list.
- **Analysis Operations>Remove i-th Point Name From Point Name Ref List.** Removes the specified point from a point name reference list.
- **Analysis Operations>Auto Filter Points/Groups/Clouds to Surface Faces.** Filters a set of points, groups, and/or clouds to a series of provided surfaces. Equivalent to the **Construct>...>Auto Filter to Faces** commands.
- **Instrument Operations>Dock Instrument Interface.** Docks or undocks an instrument interface to the SA user interface.

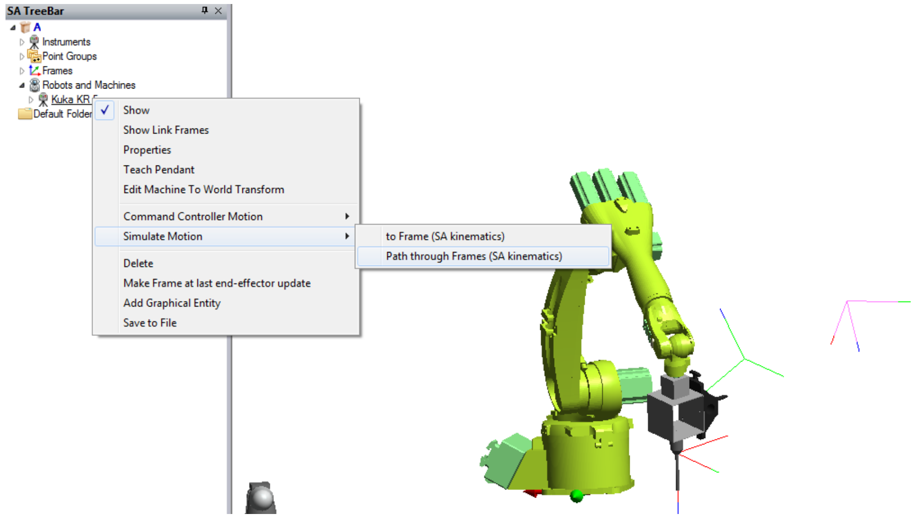
New MP Editor Features

- **Recent Files.** A list of recent files are now displayed in the MP Editor.
- **Moving Blocks of Steps.** An entire block of MP steps can now be moved up or down in the MP Editor. Select the steps you'd like to move, then click the up or down arrow to move the set of steps as a unit.

New Robot/Machine Features

- **Simulate Motion Through Frames.** Right-click a robot or machine and select **Simulate Motion>Path Through Frames** to run the inverse kinematics calculation and display the robot's pose at each of the selected frames.
- **Command Motion Through Frames.** Right-click a robot or machine and select **Command Controller Motion>Run Path Through Frames** to use manipulator kinematics to send frames directly to the controller, or have SA run a kinematics calculation and send joint sets to the controller for a calibrated path. Paths can be linearly interpolated between frames or via controller-created splines.

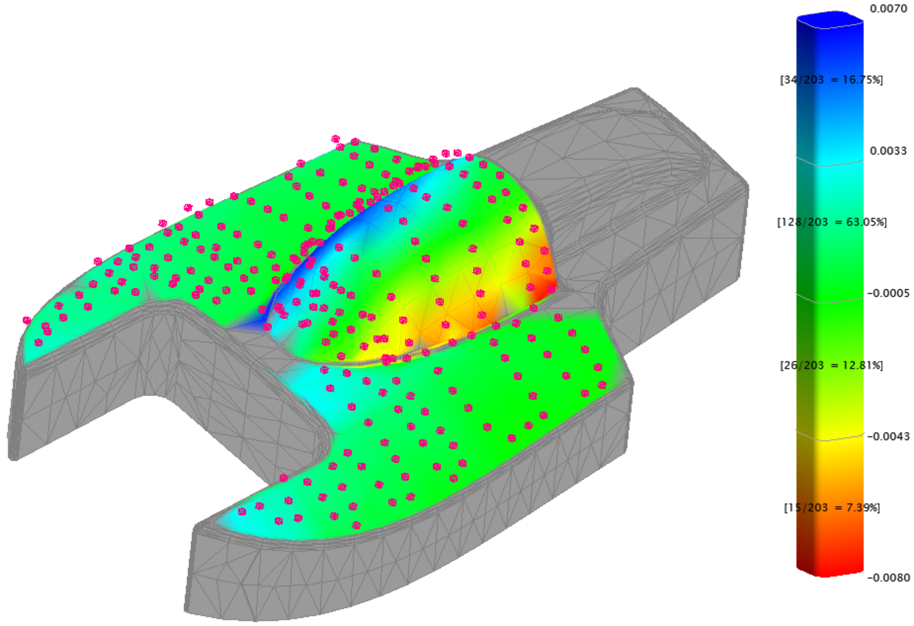
- **Imported Joint Sets.** For defining machine/robot poses, you can now import joint sets and match them with frames in SA instead of just importing points. To do this, right-click the calibration and select **Import Joint Poses and Match to Frames**.



For a list of dozens of other updates and changes, refer to the SA Readme file.

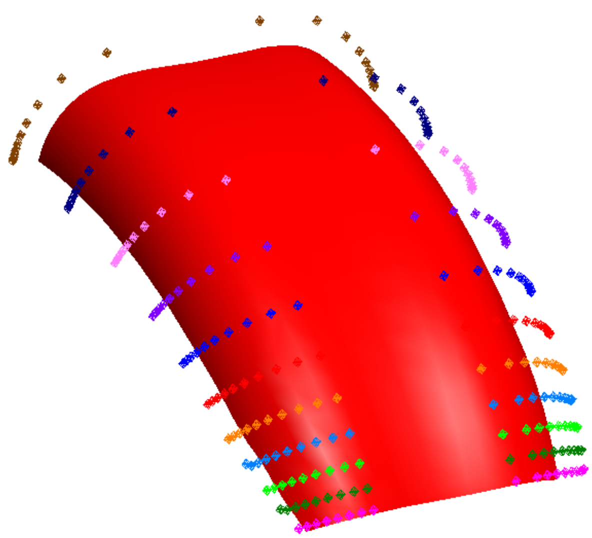
SA 2012.03.13

Surface Colorization



You can now generate colorized deviation maps based off of deviation of points from surfaces. This is similar to the view for a vector group, in which measurements indicating excess material are blue and measurements indicating too little material are red, except that the colors are interpolated and "painted" across the mesh. This is intended for use as another tool in visually reporting measurement deviation. This is available via **Construct>Colorized Graphical Mesh>From Point Proximity**, and replaces the **Construct>Polygonized Mesh>Colorized Mesh from Nominal Face and Actual Data** command from previous versions.

Offset Surfaces



Using the new **Construct>Surfaces>From Point Groups** command, surfaces will now be created taking into account measurement offsets. This allows you to create offset surfaces using any desired offset. This new command replaces the former **Construct>Surfaces>From Point Groups (Interpolate)** and **Construct>Surfaces>From Point Groups (Approximate)** commands.

Cylindrical/Spherical Report Options

All Vectors Summary: Vector Group
Hydro::Boat Error Vectors Blotches

Statistic	dR	dTheta	dZ	Mag
Min	-0.0067	-0.1056	-0.0033	-0.0075
Max	0.0008	0.1223	0.0039	0.0065
Average	-0.0008	-0.0071	0.0010	0.0011
StdDev from Avg	0.0016	0.0238	0.0016	0.0026
StdDev from Zero	0.0018	0.0248	0.0019	0.0028
RMS	0.0018	0.0248	0.0019	0.0028
Count	245			

Vector group and relationship reports now support output in Cylindrical and Spherical coordinates in addition to Cartesian coordinates.

File Integrity Verification

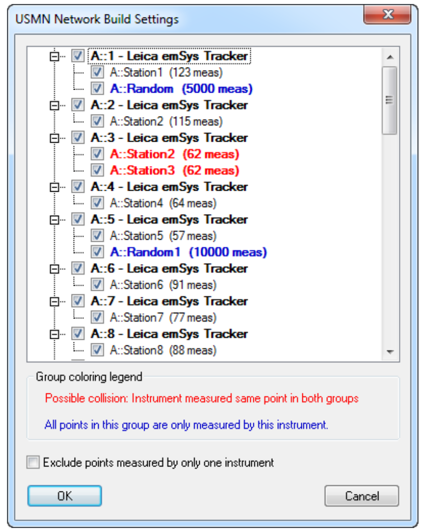
When saving an `.xit` or `.xit64` file, SA will now calculate and store a checksum with the file. This checksum is unique to the file—somewhat like a fingerprint. Upon loading the file at a later time, SA will automatically examine this checksum value and use it to verify that the contents of the file have not changed. This means that SA will provide an indicator as to whether the file's contents have changed or become corrupt since the file was saved.

A new option in the *User Options>Machine Configuration* tab, ***Verify File Checksums after Saving***, verifies the contents of a file immediately after it has been saved, and ensures that the file contents match the current SA file. This option, when checked, provides early and immediate warning to file corruption, giving you an opportunity to re-save without losing any data. This feature is useful for all users, but is particularly aimed at those working with temporary storage media (such as thumb drives) which have been known to experience file corruption issues in the past.

USMN Filtering

A number of improvements have been made to the USMN dialog that allow for faster selection filtering of points and a dramatic speed-up in the main USMN dialog when dealing with networks with very large numbers of points.

After initially selecting the instruments to be a part of the USMN dialog, you are now presented with a ***USMN Network Build Settings*** dialog. This dialog allows you to remove entire instruments or groups from the USMN process. For example, if you have 8000 scanned measurements associated with an instrument but which are not involved in the USMN network, you can choose to exclude this group to minimize the number of points displayed in the USMN dialog. (This is equivalent to the Exclude Measurements button in the USMN dialog—the functionality is just available earlier in the process).



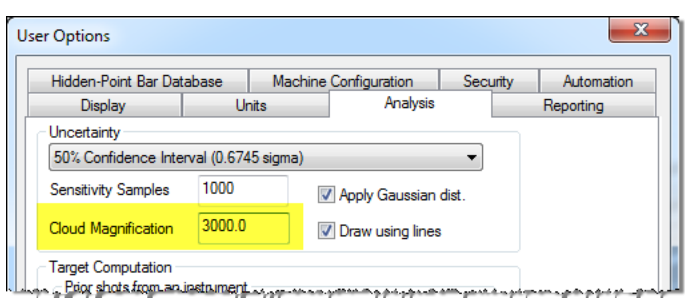
Note: Any points for which you would like to calculate uncertainties must be included in the USMN dialog. As a result, do not use *Exclude points measured by only one instrument* if you would like to calculate uncertainties for those points.

The dialog will also highlight red any pairs of point groups that have one or more measured points with the same name and are measured by the same instrument (to aid in detecting point name collisions).

Any point groups that contain point names that are not repeated in any other point groups will be highlighted blue, to indicate that the groups may not be involved in the USMN network and that you may not want to include them in the USMN solution (of course, those points will still be transformed along with the instrument (ie. "come along for the ride") in the USMN solution.

Finally, there is an *Exclude points measured by only one instrument* option to automatically exclude points from the USMN dialog that are not measured by other instruments. These points are not involved in the USMN network and therefore there is no need to explicitly show them in the USMN dialog.

Uncertainty Cloud Scale Control



The scale of rendered uncertainty clouds is now controlled by a more intuitive *Cloud Magnification* field, instead of the old *Cloud Radius Relative to Target Radius* setting. Like vector groups, this allows you to now directly scale uncertainty clouds such that a Cloud Magnification of 3000 will render clouds 3000 times their true size. In previous versions, the scale was indirectly determined based on the target radius.

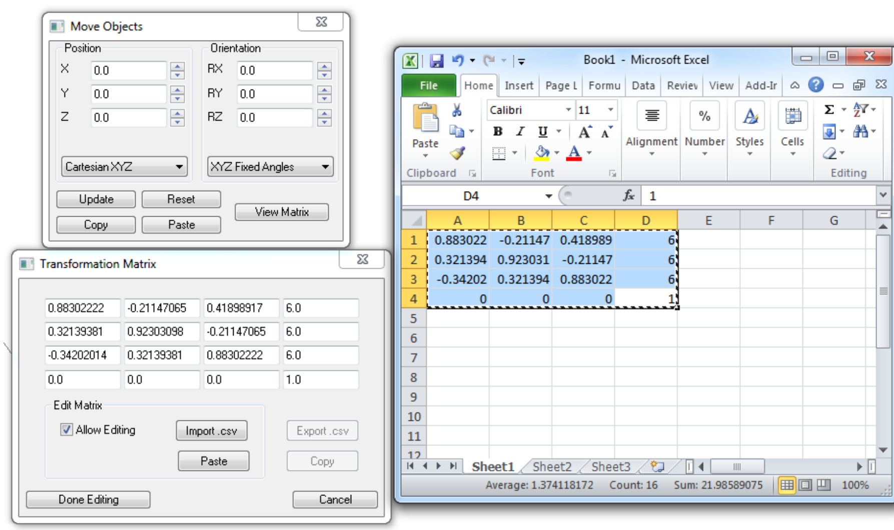
Underconstrained Feature Checks

When underconstrained feature checks are evaluated (that is, feature checks that have more than one solution), the evaluation will stop at the last iteration and present a message that the solution is not unique. Previously, the check simply produced an error message.

Transformation Matrix Copy/Paste

Note: The last row in the pasted transformation matrix should be (0, 0, 0, 1).

You can now paste 4x4 transformation matrices from the clipboard into any object's transformation matrix in SA. To do this, copy the 4x4 values to the clipboard (in row-major order, ie. row 1 column 1, then row 1 column 2...row 4 column 3, then row 4 column 4). Then, in any object's transformation properties, click the View Matrix button. Check the *Allow Editing* checkbox, then click the Paste button.



You can also copy an object's existing transformation matrix to the clipboard so that it can be pasted to a text file or Excel. With the *Allow Editing* checkbox deselected, click the Copy button when viewing any object's transformation matrix. It can then be pasted from the clipboard to another application.

Vector Group Construction Enhancements

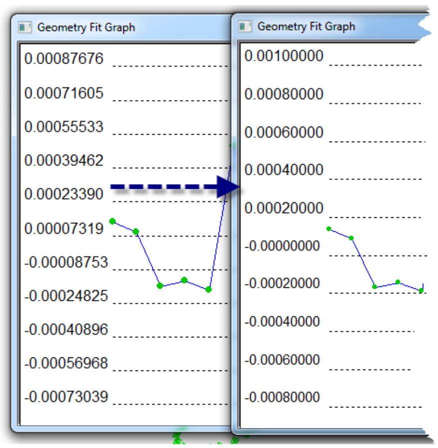
The **Construct>Vector Group>From Lines** command has been enhanced to support general objects with direction (such as planes, circles, cylinders, etc). The command has been renamed to **Construct>Vector Group>From Object Directions**.

Frame Export Options

When exporting frames via **File>Export>Frames (and object's frames)**, you now have the option of exporting either fixed values (XYZRxRyRz) or full transformation matrices.

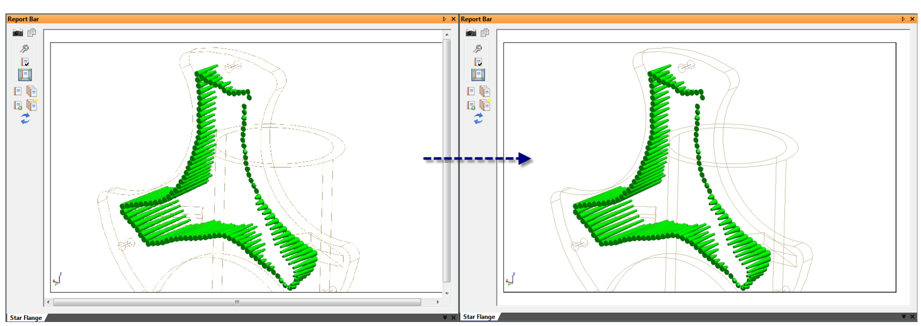
Geometry Fit Graph: Tick Spacing

The tick spacing on geometry fit graphs has been improved to land on more even numbers and to also display the zero-error point in the graph, as depicted below.



Improved Reporting Image Quality

The quality of images such as screenshots, charts, and imported images displayed in the Report Bar and SA Reports has been greatly improved.



ASCII Import: Points With Asymmetric Tolerances

A new format has been added for ASCII import which allows for the import of points with asymmetric tolerances: *PointName X Y Z THx TLx THy TLy THz TLz THd TLd (Point Tolerance)*.

New Wildcard Deletion Commands

Two new edit commands have been added: **Edit>Delete Folders by Wildcard** and **Edit>Delete Collections by Wildcard**. These commands allow you to delete folders and collections from the tree using a wildcard search criteria.

New MP Commands

For details on these new MP commands, see the MP Command Reference.

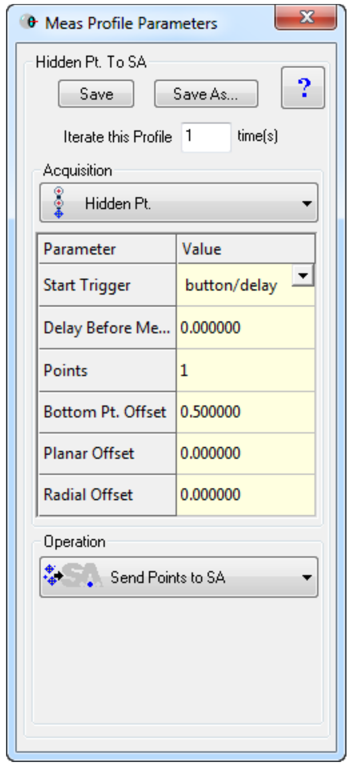
- **Construction Operations>Points and Groups>Construct Points Layout on Grid.** Creates a point group by laying out points on a grid.
- **Construction Operations>Collections>Delete Collections by Wildcard.** Deletes a set of collections based on a wildcard search criteria.
- **Construction Operations>Folders>Delete Folders by Wildcard.** Deletes a set of folders based on a wildcard search criteria.
- **Construction Operations>GD&T>Delete Feature Checks.** Deletes a list of GD&T feature checks.
- **Construction Operations>Other MP Types>Make an Event Reference List-Wildcard Selection.** Makes a list of reportable events based on wildcard selection criteria.
- **Analysis Operations>Events>Get Number of Events in Event Ref List.** Obtains the number of events in an event reference list.
- **Analysis Operations>Events>Get i-th Event From Event Ref List.** Obtains the event at the specified index of an event reference list.
- **Analysis Operations>Events>Get i-th Event From Event Ref List (Iterator).** Iterates through an event reference list, retrieving successive events in the list.
- **Analysis Operations>Events>Rename Event.** Renames an event.
- **Analysis Operations>Events>Delete Event.** Deletes an event.
- **Reporting Operations>Report Bar>Add Events to Report Bar.** Adds one or more events (in order) as separate tabs in the Report Bar.
- **Utility Operations>Set View Idle Update Frequency.** Sets

the frequency at which the graphical view is updated.

- **Reporting Operations>Combine SA Reports.** Combines separate SA reports into a single report.
- **Reporting Operations>Custom Report Tables>Make Custom Table.** Creates a custom table in the tree for later addition to a report.
- **Reporting Operations>Custom Report Tables>Clear Custom Table.** Clears a custom table.
- **Reporting Operations>Custom Report Tables>Set Custom Table Title.** Sets the title for a custom table.
- **Reporting Operations>Custom Report Tables>Set Custom Table Header Cell.** Sets a header for one or more consecutive columns in a table.
- **Reporting Operations>Custom Report Tables>Set Custom Table Cell String.** Places the specified string into a custom table.
- **Reporting Operations>Custom Report Tables>Set Custom Table Cell Double.** Places the specified double into a custom table.
- **Reporting Operations>Custom Report Tables>Set Custom Table Cell Color.** Sets a cell's foreground and background color in a custom table.
- **Reporting Operations>Custom Report Tables>Add Custom Table to SA Report.** Adds a custom table to an existing SA report.

New Tracker Acquisition Mode: Hidden Point

A new acquisition mode has been added to the laser tracker interface: Hidden Point. This mode measures two points, then automatically creates a hidden point based on the offsets specified in the acquisition mode's parameters. This acquisition mode is part of a new Hidden Point measurement profile.



New Tracker Operation: Patch Points

This operation projects points to a defined plane as they're measured—appropriate for measuring patch points.

RMS Monitor Improvements

The laser tracker interface's *RMS Monitor* now persists its state between interface usage. If the RMS Monitor is open when the tracker interface is closed, then it will remain open when the interface is later reopened. Also, the RMS Monitor now also indicates stable point state (Waiting for Move, Waiting for Stable, etc.) which can be resized for easier viewing from a distance.

New Instrument Interface: Steinbichler/Leica Probe Scanner for T-Probe

Support for the Steinbichler/Leica Probe Scanner has been added to SA. After starting the emScon tracker interface, the scanner is connected to a T-Probe and data is collected through ProbeScan, in the same way that the T-Scan interface runs through T-Scan Collect.

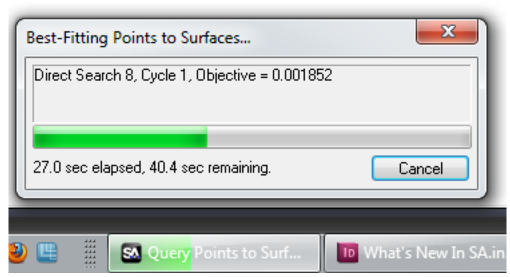
The settings are very similar to the T-Scan interface, with the exception of a few options. Shutter time teaching is supported (using button C or D on the T-Probe), and success or failure is reported in the interface's status window.

Portable CMM Arms

A new measurement mode, *Single Pt Hole*, has been added to the Portable CMM interface. This mode functions identically to the tracker mode of the same name. the probe radius and depth sunk beneath the measured plane determine the radius of the measured hole.

Windows 7 Taskbar Progress Support

For users running Windows 7, support has been added to display progress bars in the taskbar.



For more improvements and updates, refer to the SA Readme file.

SA 2011.12.22

SA 64-bit

The 64-bit version of SA is here!

SA 64-bit provides significant performance improvements (20% or more in some cases), and enables handling enormous data sets and CAD files. For example, 500 million cloud points can be handled comfortably in 64-bit SA on a system with 16 GB of RAM.

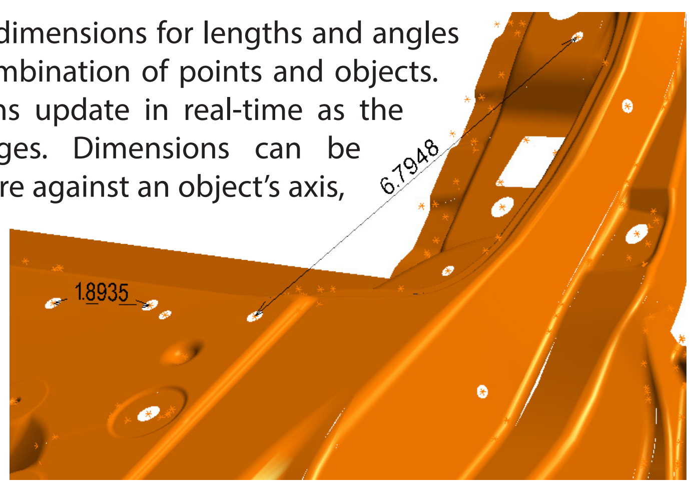
From the 64-bit version of SA, you can now save .xit files into two formats: `.xit`, and `.xit64`. The traditional `.xit` files can be opened by both 32- and 64-bit versions of SA. The `.xit64` format can only be opened by 64-bit SA, but will maintain all data when saving extremely large files.

The SA install now contains both a 32-bit version of SA (for compatibility with existing 32-bit systems), and a 64-bit version of SA. Both versions are installed by default.

To run the 64-bit version of SA, select it from the Windows Start menu.

Dimensioning

Create dynamic dimensions for lengths and angles between any combination of points and objects. These dimensions update in real-time as the geometry changes. Dimensions can be placed to compare against an object's axis, origin, plane, or surface, and allows you to add offsets (for inside measurements) or subtract them (for outside measurements).

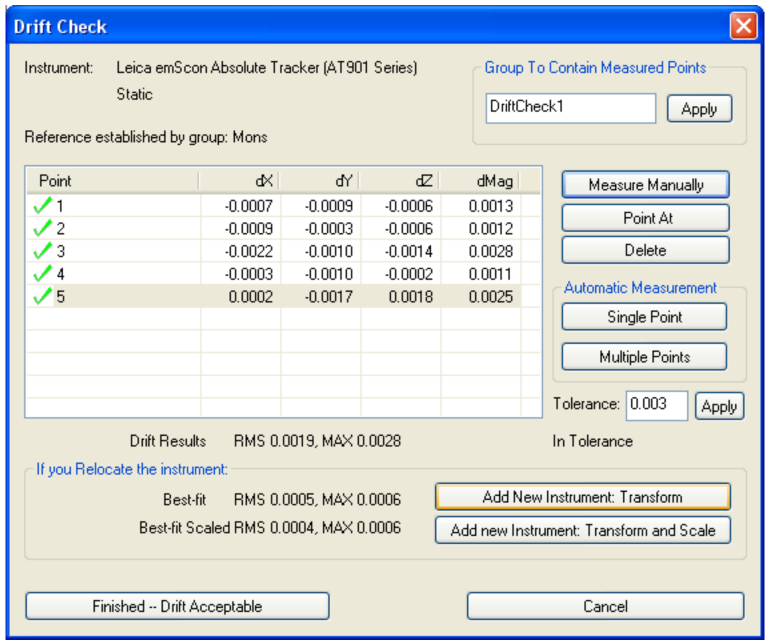


SA Remote Update

With the 2011.12.13 update to SA Remote, you can now record measurements while in a watch update mode.

Automatic Drift Check Refitting

The **Instrument>Drift Check** command now automatically shows what your deltas would look like if a refit were performed both keeping scale constant and allowing scale to vary.



If a best fit with fixed scale significantly improves the results, that indicates that the instrument has moved relative to your monuments. If a best fit with scaling is required, then your reference system has scaled, and therefore the part has likely scaled as well.

Two new buttons: Add New Instrument: Transform and Add new Instrument: Transform and Scale will conveniently add another instrument, activate its interface, and place you into a completed Measure Nominals dialog, allowing you to apply a best fit or best fit with varying scale without measuring any additional points.

Set Viewpoint from Frame

The viewpoint can now be directly set from an existing frame using **View>View Control>Set Viewpoint From Frame** and **View>View Control>Set Viewpoint From Frame>Clip Behind XY-Plane**. The second of the two new commands will set up a clipping plane such that everything closer than the XY plane is removed from the view. This yields a cross-sectional cut through a surface.

New MP Commands

For details on these new MP commands, see the MP Command Reference.

- **Instrument Operations>Get Instruments with Observations on Target.** Returns the list of instruments that have observations on (measurements of) the specified target.
- **View Control>Set Point of View from Frame.** Sets the point of view to match the orientation of a given coordinate frame, optionally clipping the view in front of the frame's XY plane.
- **Analysis Operations>Query Clouds to Objects.** Queries one or more point clouds to one or more objects, creating point groups or vector groups in the process.
- **Analysis Operations>Get Geom Relationship Criteria.** Retrieves the nominal, measured, delta, low tolerance, and high tolerance values (as applicable) for a specific geometry relationship parameter.
- **Analysis Operations>Reverse Surface Normals.** Reverses the normals of specified surfaces.
- **Construction Operations>Construct Objects from Surface Faces-Runtime Select.** Creates a set of primitive geometric shapes (planes, cylinders, spheres, cones, lines, points, and circles) from CAD surfaces selected by the user at runtime.
- **Construction Operations>Points and Groups>Construct Points from Surface Faces-Runtime Select.** Creates points from CAD surfaces selected by the user at runtime.
- **Construction Operations>Planes>Construct Planes from Surface Faces-Runtime Select.** Creates planes from CAD surfaces selected by the user at runtime.
- **Construction Operations>Cylinders>Construct Cylinders from Surface Faces-Runtime Select.** Creates cylinders from CAD surfaces selected by the user at runtime.
- **Construction Operations>Spheres>Construct Spheres from Surface Faces-Runtime Select.** Creates spheres from CAD surfaces selected by the user at runtime.
- **Construction Operations>Lines>Construct Lines from Surface Faces-Runtime Select.** Creates lines from CAD surfaces selected by the user at runtime.
- **Construction Operations>Cones>Construct Cones from Surface Faces-Runtime Select.** Creates cones from CAD surfaces selected by the user at runtime.

- **Construction Operations>Circles>Construct Circles from Surface Faces-Runtime Select.** Creates circles from CAD surfaces selected by the user at runtime.
- **Construction Operations>Polygonized Surfaces>Construct Polygonized Surface from Point Clouds.** Creates a polygonized surface (mesh) from a set of input point clouds. This command is the MP equivalent to the **Construct>Polygonized Mesh>From Point Clouds** menu command.
- **Construction Operations>Cones>Construct Cone.** Creates a cone.
- **Excel Direct Connect>Run Macro.** Runs a macro stored in an Excel workbook.
- **Excel Direct Connect>Write>Write Picture.** Places a picture stored in the SA tree into an Excel workbook.
- **Utility Operations>Get Tick Count.** Returns an approximately millisecond-accurate tick count (timer value) of the computer clock, in seconds.
- **Utility Operations>Scale Objects.** Scales a list of objects about the working coordinate frame.
- **Analysis Operations> Relationship Operations> Relationship Attributes> Make Vector Tolerance.** Creates a vector tolerance that can be fed into another command, such as Make Point to Point Relationship.
- **Analysis Operations> Relationship Operations> Relationship Attributes> Make Vector Constraint.** Creates a vector fit constraint that can be fed into another command, such as Make Point to Point Relationship.

Independent Decimal Precision Settings

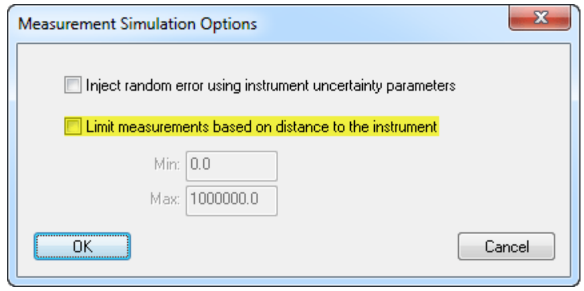
You can now set independent decimal precision settings for the following different types of values:

- Lengths
- Angles
- Scales
- Unit Vectors
- Weights

Angle Representations

You can now control whether angles are represented in the -180° to 180° range, or in the 0° to 360° range.

Measurement Simulation Ranges



When fabricating measurements for a simulation, you can now specify a distance envelope, measured from the instrument. Any points outside of this envelope will not have measurements fabricated.

Vector Groups: Continuous (Entire Range) Color Ranging Method

A new Continuous (Entire Range) color ranging method has been added to Vector Groups. This option examines the max and min deviations in the vector group and uses this range to span the entire color range.

Multiple Frame Sizes

Frames can now be assigned a custom size separate from the default frame size in the User Options. This is useful when different sized features have separate coordinate frames, and you want the coordinate frames to be similar to the scale of the features.

Printing Invisibility

Right-click an object in an SA Report and toggle the *Printing Invisibility* option to prevent the item from appearing in printed reports.

Relationship Reporting Details

Relationship summaries in the tree, report bar, and SA reports now contain additional reporting details, including:

- Subsampling
- Constraints
- Outlier rejection
- Ignored edge projections
- Overridden target offsets
- Added material thickness

Relationship Minimization Context Menu

You can now minimize relationships by right-clicking any relationship category in the tree and selecting the desired command from the context menu.

MP Watcher

An option has been added to automatically show the MP Watcher when an MP breakpoint is hit. This option can be enabled in the *User Options>Automation* tab.

Geometry Fitting: Object Association

If all measurements used in a fit are associated with a single instrument, then the resulting geometry is now also associated with the instrument (does not apply to cloud points).

Geometry Relationships: Cardinal Points

Fit only and Fit & Compare to Nominal Relationships now have a setting to enable the creation of dynamically updated cardinal points for the fit.

Hexagon SE Absolute Arms with CMS and Perceptron v5 Shark Scanners

Support has been added for Perceptron v5 Shark and CMS scanners for use with Hexagon Absolute SE arms. Once set up in RDS Control Panel, both will act like integrated scanners. Just mount the scanner, switch it on, and go.

For additional improvements, changes, and fixes, refer to the SA Readme file.

This Page Intentionally Left Blank.

Getting Started

CHAPTER 3

You are about to start your journey toward learning the world's most powerful portable metrology software package. But you can't take the first step until the software's installed!

SA Packages

SA is available in three forms: SA Professional, SA Ultimate, and SA Machine. The details of the functionality provided in each of these packages in provided below.

SA Professional

SA Professional with Native CAD is the professional measurement, alignment, inspection, analysis, and reporting software for all portable metrology instruments. In addition to all essential measurement needs, it includes several key inspection features, geometry inspection, and measurement automation.

MEASUREMENT

SA Professional offers 100% traceability from measurement to reporting. All measurement data is stored alongside time stamps, instrument information, weather data, and other measurement parameters. Log files track user actions, fit results, and more, allowing you to see the entire history of the job file.

ALIGNMENT

An essential part of the measurement process is the alignment of measurements to a known coordinate system. SA Professional includes a variety of alignment methods ranging from traditional 3-2-1 alignments to more advanced surface fits.

BUILD

The Relationship functionality offers an easy way to calculate and observe part deviation. Relationships are dynamic in nature and update automatically if part alignment or data changes.

EVALUATION & ANALYSIS

A user-friendly interface permits both graphical and numerical depiction of measurement uncertainty, enhancing your perspective of measurement quality.

REPORTING

SA Professional offers quick, user-friendly reporting functionality. Quick Reports are ideal for on-location reporting, report templates are perfect for repetitive work, and the Report Designer allows you to drag and drop items to develop custom reports using tables, charts, and graphics.

GD&T INSPECTION

GD&T allows you to import CAD with GD&T annotations, create annotations manually, and inspect to GD&T standards with real-time reporting.

GEOMETRY INSPECTION

Geometry inspection enables you to define design-based inspection routines from a CAD model or primitive geometry.

NATIVE CAD

Native CAD provides import support for such major applications as CATIA V4, V5, Pro/Engineer, SolidWorks, Unigraphics, and other CAD formats.

SA Ultimate

SA Ultimate is the premier measurement, optimization, analysis, reporting, and automation software suite for all portable instruments. It includes everything contained in SA Professional, plus the following features:

REAL-TIME ALIGNMENT

Transformation Tracking allows you to track moving parts in real-time so that you can monitor a part's position as it is guided into place.

ADVANCED FIT OPTIMIZATION

Relationship minimization provides the power necessary for advanced alignments and can help bring an out-of-tolerance part back within tolerance. You can also define fit envelopes to satisfy multiple constraints/requirements.

COMPLEX INSTRUMENT NETWORKS

USMN is an extremely powerful feature that leverages the uncertainty characteristics of different instruments to provide a much more accurate instrument network than that of traditional alignment methods. It is used by many metrology groups to solve large-scale networks such as accelerator rings, full submarine surveys, and large machinery analysis.

AUTOMATION

The Measurement Plan and SA SDK functions can add a significant layer of automation to your processes.

Generating simple to complex scripts can greatly improve workflow and productivity, reducing analysis time from days to minutes while eliminating errors and saving significant resources.

PIPE FITTING

The Pipe Fitting function is used in large piping applications when precision cuts must be determined.

Precision measurements and optimization greatly improve accuracy, ultimately reducing the cost of rework.

DATABASE OUTPUT

Database output is an additional reporting feature that gives you the ability to transfer SA data to databases and track projects/data long-term.

SA Machine

SA Machine with Native CAD contains all features of SA Ultimate, but also allows users to interface with robots and CNC machines. This permits calibration and real-time compensation of robots and large-volume CNC machines to high degrees of accuracy. It also enables robotic scripting, teach pendant behavior, and linear or joint space robotic control.

Feature Summary

Features	SA Professional	SA Ultimate	SA Machine
Measurement:			
Traceability	✓	✓	✓
Simultaneous Instrument Interfacing	✓	✓	✓
Over 120 Supported Instruments	✓	✓	✓
Alignment:			
Best Fit	✓	✓	✓
Feature Based	✓	✓	✓
Points to Surfaces	✓	✓	✓
Standard Instrument Network Bundle	✓	✓	✓
Relationship Fitting		✓	✓
Advanced Network Optimization (USMN)		✓	✓
Inspection:			
Geometric Dimensioning & Tolerancing	✓	✓	✓
Geometry Inspection	✓	✓	✓
Build:			
Relationships	✓	✓	✓
Watch Windows	✓	✓	✓
Transformation Tracking & Guiding		✓	✓
Evaluation & Analysis:			
Queries	✓	✓	✓
Spatial Transformations	✓	✓	✓
Robust Fit Algorithms (NIST & PTB Certified)	✓	✓	✓
Measurement Uncertainty Analysis	✓	✓	✓
Relationship Optimization (independently moving objects)		✓	✓
Reverse Engineering		✓	✓
CAD Interface:			
Standard CAD Exchange (IGES, STEP, etc.)	✓	✓	✓
Native CAD Access (CATIA, ProE, NX, etc.)	✓	✓	✓
Automation:			
Automatic Measurement	✓	✓	✓
Measurement Plan Scripting		✓	✓
Software Development Kit (SDK)		✓	✓
SA Reporting:			
Standard Output (Excel, Word, PDF, Text)	✓	✓	✓
Graphical Callout Annotations	✓	✓	✓

Features	SA Professional	SA Ultimate	SA Machine
Quick Reports	✓	✓	✓
Drag & Drop Reporting	✓	✓	✓
Database Output (ODBC)		✓	✓
Pipe Fitting:			
Pipe Cut Optimization		✓	✓
Robotics:			
Kinematic Modeling			✓
Kinematic Calibration			✓
Direct Motion Control			✓
Graphical Simulation			✓

System Requirements

SpatialAnalyzer is capable of running on most modern computing platforms, both 32 and 64-bit.

Minimum Requirements

Microsoft Windows XP™ SP3 or later.

512 MB system RAM

560 MB free disk space for installation (more for your data files)

1024 x 768 screen resolution

Microsoft .NET 4.0 Runtime (if using the Laser Radar Interface)

Recommendations for Optimal Performance

3 GHz or faster Dual-Core 64-bit processor

Windows 7™ (64-bit)

8 GB system RAM (64-bit systems)

1280x1024 or higher screen resolution

Hardware-accelerated graphics card supporting OpenGL®

1 GB of free hard disk space

Supported Auxiliary Input Devices

3DConnexion® SpaceNavigator™ (USB)

3DConnexion® SpaceExplorer™ (USB)

3DConnexion® SpacePilot™ (USB)

3DConnexion® SpacePilot™ PRO (USB)

Check My Computer

SA provides a utility named *Check My Computer* that detects certain system configuration issues that may result in errors or problems while running SA. While this tool does not detect all issues—and while some checks may fail even though you never experience problems in SA—this is a useful diagnostic tool to use when you *do* encounter a problem. This tool does the following:

- Checks for certain problematic software known to interfere with certain aspects of SA operation.
- Verifies that a default printer is installed.
- Checks for TCP/IP availability (a network interface).
- Checks firewall settings.
- Ensures that your computer's network interface provides support for UDP broadcasting.
- Ensures that your computer's network interface provides support for socket communication and that it is not blocked.
- Checks your User Account Control settings (Windows 7 and later) to ensure they're not set too high.
- Provides convenient access to system information and network configuration settings.

If you encounter an issue running SA, it is recommended to run this utility to check for problems and to contact support if necessary to find a resolution.

To Run the Check My Computer Utility

- From SA, Select **Help>Check My Computer**, or
- From the Windows Start menu, in the SA install folder, select the **Check My Computer** entry.
- In the Main toolbar, click the SA Network icon.

Installation

Installing SA

The SA Installer will guide you through installing the following:

- SpatialAnalyzer™ Application
- Documentation (this document)
- Hardware lock driver

The installer is included with your provided USB flash drive and is also available for download at www.kinematics.com/download.

To Install SpatialAnalyzer:

1. Double-click `SpatialAnalyzer 2014.XX.XX-Installer.exe`.
2. Choose a destination folder (the default folder is usually preferable), then click Next.
3. Leave both components (hardware driver & SA application) checked unless you have a specific reason to deselect a component. (For instance, if you are an OEM running a software lock, you would not need to install the Sentinel Hardware Driver). Click Next.
4. Choose additional install options. The `Check My Computer` utility will check your system for some common configuration issues that often cause problems with instrument communication in SA. When done selecting options, click Install.

Note: Updated SA versions may require a new version of the hardware lock driver, so it is best to leave that option checked. Older drivers will be updated, if necessary.

SA is continuously updated with improvements and new features, and new beta releases are posted frequently. If your maintenance supports upgrading, you are encouraged to download the latest version and explore the newest functionality.

Installer Command-Line Options

The SA Installer supports several command-line options. These are typically most useful for IT departments that wish to do a silent or automated install. The most command-line options can be combined and are as follows:

- **/?.** Lists the command line options.
- **/S.** Performs a silent install (this must be a CAPTIAL S). The installer will operate in the background with no user intervention.
- **/CREATESHORTCUT.** Creates a desktop shortcut when run in silent mode.
- **/TEMPUNITS=[Celsius, Fahrenheit].** Sets the default temperature units for the install. If omitted, Fahrenheit will be used as a default, unless SA was installed previously, in which the temperature units will not change. Ex. usage: **`/TEMPUNITS=Celsius`**.
- **/LENGTHUNITS=[Inches, Feet, Millimeters, Centimeters, Meters].** Sets the default length units for the install. If omitted, inches will be used as a default, unless SA was installed previously, in which the length units will not change. Ex. usage: **`/LENGTHUNITS=Millimeters`**.

To Install SA with Command-Line Options:

- At the command line, follow the executable name with the options. For instance, the following installs SpatialAnalyzer silently in the background while also creating a desktop shortcut and setting the length units to millimeters: `SpatialAnalyzer 2014.XX.XX-Installer.exe /S /CREATESHORTCUT /LENGTHUNITS=Millimeters`

To Check for SA Updates:

- Select **Help>Check for SA Updates**,
- Choose **Help>SpatialAnalyzer License Management>View Maintenance Support and Subscription Status**, then click the Check for Updates button, or
- Navigate to http://www.kinematics.com/products/spatialanalyzer/download-software.html.

Instrument Drivers

Most portable measurement devices (instruments) require a driver to be installed in order to properly communicate with SpatialAnalyzer. When configuring a new instrument, it is always best to follow the instrument manufacturer's recommended installation procedure and to install their software components/drivers as provided by the hardware manufacturer. However, the minimum drivers required for communicating with SA for most supported instruments are provided in a single bundled instrument driver installer provided by NRK. This installer is included with the provided USB flash drive and is also available for download at www.kinematics.com/download.

You can also locate drivers for your instrument on NRK's FTP site at ftp://ftp.kinematics.com/pub/SA/Install/Driver Downloads/.

To Install Instrument Drivers using the Bundled Instrument Driver Installer:

1. Double-click `SA Instrument Mfg Drivers.exe`.
2. Select the desired instrument drivers to install and click Next.
3. You will be guided through the installation for the selected instruments. Note that the installers are usually provided by the hardware manufacturers, and each one is different. Contact NRK if you have problems while installing.

Licensing

Your SpatialAnalyzer license is controlled by a USB hardware lock. The hardware lock and driver must be installed before using SpatialAnalyzer. The driver is installed by default as part of the SA install.

Meta-key File Installation

Meta-key files (extension `.met`) are used to upgrade or extend a license. They are small encrypted files that the software opens and uses to enable additional capability in your license. Normally, a new user with a hardware lock will have no need for a meta-key. However, if software licensing is extended or renewed, a user may receive a meta-key to extend the license for an additional period of time. By placing this meta-key file in the `Analyzer Data\License` directory, the hardware lock can be upgraded. When the file is present in the License folder, it supersedes the information that has been hard-coded onto the hardware lock.

Meta-keys have filenames like `License12345.met`, where the 5-digit number matches the serial number on the key.

To install an SA Meta-key File:

> **Note:** If a hardware lock is moved to a different computer, the meta-key file must be transferred with it by copying the file into the new computer's `Analyzer Data\License` folder. Failure to do this will result in the hardware lock reverting to its original license settings.

1. Double-click the `.mez` file you received from NRK and SA will automatically install it.
2. Ensure that the hardware lock with the matching serial number is installed in the USB port.
3. Run SpatialAnalyzer.
4. Choose the **Help>SpatialAnalyzer License Management>View Current License Information** menu option to verify your license.

Viewing Your License Information

Occasionally—particularly when contacting customer support—it may be necessary to examine the details of your SpatialAnalyzer license, such as the serial number, user and company, upgrade date, and available software components. You can view the details of the current license in use at any time.

To View the Current License Information

1. From the SA menu, select **Help>SpatialAnalyzer License Management>View Current License Information**.
2. A window will be displayed showing your license details, and this text will be automatically copied to the Windows clipboard.

> ***A Closer Look***
>
> When SA starts up, it first looks for a hardware lock. It retrieves the license information and serial number from the hardware lock, then checks your `Analyzer Data\License` folder to see if there is a meta-key file matching the hardware key's serial number. If found, it uses the meta-key file to update the license information from the lock.

You may also wish to view your maintenance subscription information, which will indicate the date software maintenance will expire.

To View Your Maintenance Subscription Information:

- From the menu, select **Help > SpatialAnalyzer License Management > View Maintenance and Support Subscription Status**.

To Request Renewal for Your Maintenance and Support Subscription:

- Select **Help > SpatialAnalyzer License Management > Request Maintenance and Support Renewal**, or
- Choose **Help > SpatialAnalyzer License Management > View Maintenance and Support Subscription Status**, then click the Request Upgrade/Renewal button.

Certification

SpatialAnalyzer has been certified by the National Institute of Standards and Technology (NIST), the Physikalisch-Technische Bundesanstalt (PTB), and the National Physical Laboratory (NPL).

How to Get Help

Documentation

If you need help, the easiest source of information is the SA User Manual (this document). This document is in PDF format and is fully searchable.

To Access the SpatialAnalyzer User Manual:

Do one of the following:

Click the User Manual icon in the *Main Toolbar*,

Select **Help>User Manual**, or

In the Windows Start Menu, navigate to the SA install folder, then the `Documentation` subfolder, and select the `SpatialAnalyzer User Manual` link.

Every SA Release has a complete Readme file documenting every change to the software, organized by software version. This is often a useful source of information, particularly with determining instrument driver compatibility with a given SA version.

To View the SA Readme File:

- Select **Help>View the SA Read Me File**, or
- From the Windows Start Menu, navigate to the SA install folder, then the `Documentation` subfolder, and select the `Release`

`Notes` link.

Customer Support

If you need help setting up or using SpatialAnalyzer, there are several ways that you can reach customer support:

> **Tip:** We recommend against contacting support personnel directly via their e-mail address or phone number. Doing so may result in a delayed support response, as the support individual may be out of the office or otherwise unreachable.

- **E-mail.** support@kinematics.com
- **Telephone.** Dial +1 (757) 565-1500 and choose the support option (option 1). Be aware that this will transfer you directly to a voice mailbox, but your call will be returned promptly.
- **Web.** Navigate to www.kinematics.com and choose the support link, or from within SA, select **Help>Visit NRK Website!**

Regardless of how you contact us, we try to respond to all support requests in less than 24 hours.

Context-Sensitive Help

SA has a context-sensitive help system wherein you can click on menu items, interface components, or select dialogs in order to be directed to that topic in this User Manual.

To Initiate Context-Sensitive Help:

1. In SA, press **`Shift + F1`**. The mouse cursor icon will change to indicate that you are now in context-sensitive help mode.
2. Click on a menu item, toolbar item, or interface item to be directed to the topic in the user manual.

Some dialogs have a question mark in their upper-right corner. Click this question mark to be directed to the appropriate topic in the documentation.

Tutorials & Sample Files

SA comes with a variety of sample files to demonstrate some of the key features of SA. Most of these sample files are used in the Tutorials chapter toward the end of this manual.

To Open a Sample File:

From the menu, select **Help>Open Sample SA Files**.

This Page Intentionally Left Blank.

SA Fundamentals

CHAPTER 4

It is important to have a good understanding of SA's "modus operandi". Having a good fundamental understanding of how SA operates will prove to be extremely useful and will make you a more effective user of the software. In this section, we'll describe some of the fundamental principles surrounding SA and its use.

Architecture

SA has been designed from the ground up with a very decentralized concept of operation. That is, the software's various parts do not necessarily need to be running on the same computer. Because of its network architecture, SA is able to communicate with its various instrument interfaces via the TCP/IP network protocol, which permits a number of interesting capabilities. In addition, instruments themselves can communicate to SA through a wide array of protocols/interfaces, including TCP/IP, Serial RS-232, Bluetooth, and USB. Each method requires that the settings be configured to allow proper communication between SA and the instrument(s).

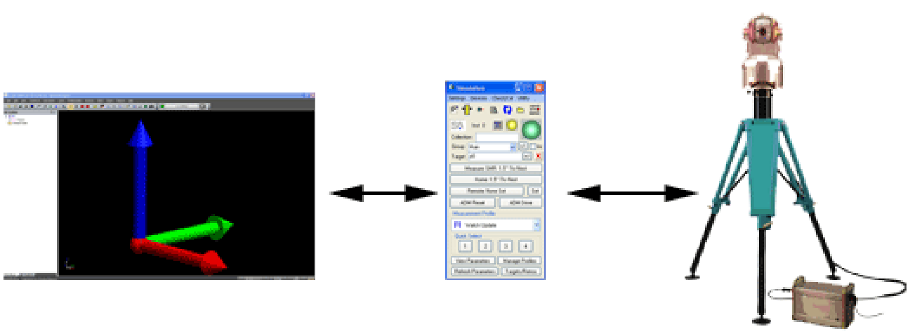


Figure 4-1. The components involved in measuring with an instrument in SA. From left to right: the Host (SpatialAnalyzer itself), the Interface, and the Instrument.

There are three main components to an SA configuration: The host

SpatialAnalyzer session, the instrument interface, and the instrument (see Figure 4-1).

Host SpatialAnalyzer Session

The host SpatialAnalyzer session is an instance of SA running on a computer that will be the destination for measured data. While it is usually the same computer as the one running the instrument interface (see Figure 4-2), it can be any computer that is connected via TCP/IP on the local network to the computer running the instrument interface (see Figure 4-3).

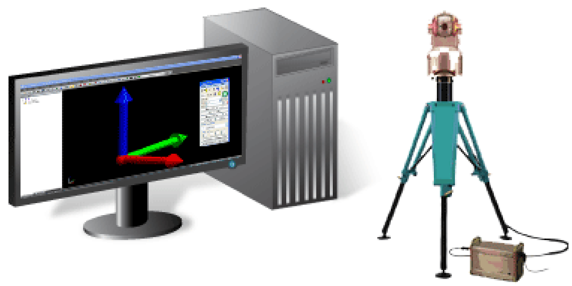


Figure 4-2. A host SA session and an instrument interface running on the same computer.

The flexibility provided by the separation of host, interface, and instrument means that you can run multiple instruments and interfaces all in the same SA job.

Figure 4-3. A host SA session and an instrument interface running on different computers.

Instrument Interface

The instrument interface is the application that directly connects with an instrument, "speaks the instrument's language", and directs measured data to the host SA session. When starting the instrument interface, you must choose which SA session will be the destination for measured data--if multiple computers on a network are all running SA, any one of them could have measured data sent to it. If not on the same computer, the host SA session should be located on the same network as the instrument interface. You must also define how the interface should connect with the instrument. (For example, which COM port, or using which IP address?)

There are two primary requirements in order to start up an instrument interface:

- In order for a host SA session to be a destination for measured data, you must add the appropriate instrument to the job file on the SA host, and it must be discoverable on the network.
- In order to connect to the instrument, the interface must be able to communicate with it: Either directly via an interface such as USB or Bluetooth, or via a network using TCP/IP.

A single instrument interface can connect to only one SA host session and only one instrument at a time. However, any one computer may run multiple instrument interfaces (connected to different instruments) at the same time.

The instrument interface is the mediator between the SA host and the instrument. The interface sends commands to the instrument to control its behavior, and the instrument sends data and state information back to the interface. The interface then grooms this data as necessary and sends it on to the SA host.

Instruments

Each instrument must be able to communicate with its respective instrument interface, and therefore must be set up correctly to talk to the computer running the interface. This is handled in a number of ways depending on the method that the instrument uses to connect to the computer. Detailed information on the proper setup for instruments using each communication protocol is found in the Instrument Interfaces section.

The following graphics illustrate just a few of the many possible ways to connect instruments with SpatialAnalyzer.

Figure 4-4. A single computer connected directly to an instrument via TCP/IP. This is the most common setup.

In Figure 4-4, since the instrument is connected directly to the host computer via a network cable and the TCP/IP protocol, no other communication hardware is required. The interface runs on the same computer as the host SA session. This is the most common way to connect in SA.

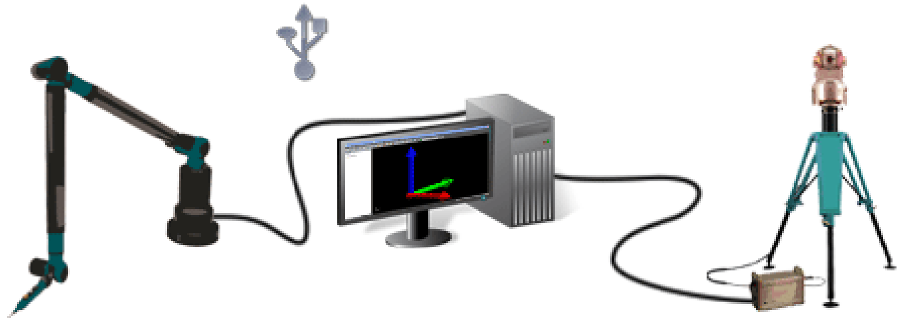


Figure 4-5. Two computers connected directly to the host computer: One via USB, the other via TCP/IP.

In Figure 4-5, one instrument is connected via USB, and the other through TCP/IP. The instruments are connected directly to the same computer on which the instrument interfaces and the host SA session are running. Only one instrument might be running at a time, or two instrument interfaces might be running simultaneously in a single session of SA.

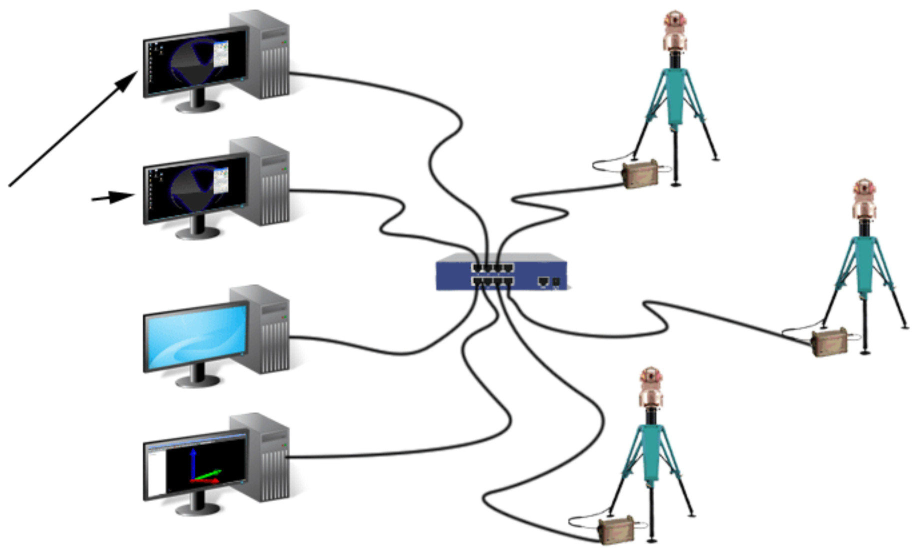


Figure 4-6. Multiple instruments and computers on a local network connected via TCP/IP and an ethernet switch. The interfaces may be running on a different computer than the host.

With this configuration (Figure 4-7), a connection to the internet is not used. Instead, one or more instruments and one or more computers are connected to a local network through a network hub or switch (and the TCP/IP protocol). One SA host computer may be running instrument interfaces for all instruments, or the interfaces may be run on separate networked computers. Or, three instances of SA might be running on three different computers to control the three different instruments.

While the configuration in Figure 4-7 is rare, it is technically possible. In this case, an instrument is connected via TCP/IP over the internet to an instrument interface, perhaps on the other side of the world. The interface then connects to the SA host via the local network. As long as you can ping the desired instrument from the computer running the instrument interface, you can connect to it.

As you can see, the networked architecture of SA lends immense flexibility to the way that you measure and operate on data. With this setup, it is possible for an operator in a metrology office to control a laser radar hanging from a gantry crane in a facility on the other side of the world. Or, one workstation might be set up to receive measurement data, and three operators with laptops might measure data with their own instruments and send it to the receiving workstation.

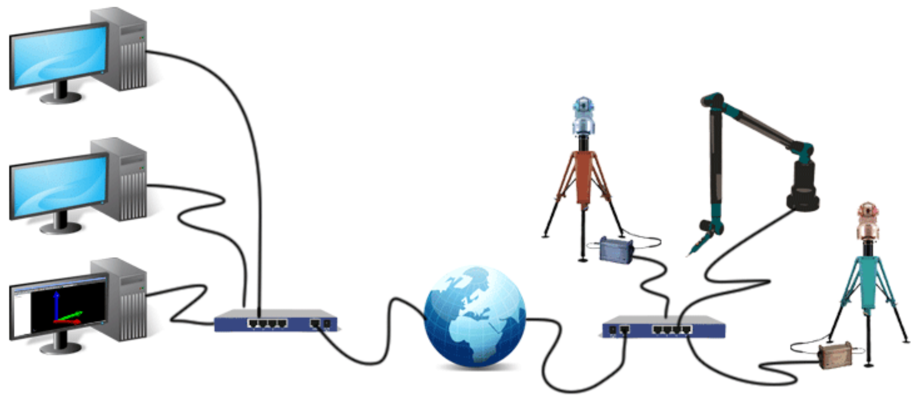


Figure 4-7. Networked configuration. Instrument and interface connected via TCP/IP over the internet, and the host SA session is connected to the interface via a local network.

While there are many different configurations and possibilities, most users end up with one of the first three configurations illustrated above.

Selection Filtering

In any SA file, there are four main types of entities. Three of them have a represented position in space (that is—they have a position and, with the exception of points, an orientation), and the fourth encompasses everything else. These entity types are described below:

- **Points.** This includes measured points (recorded by an instrument) and constructed points (created via any other method).
- **Instruments.** This includes only measurement instruments.
- **Objects.** Anything that has a position and orientation in space that is not a point or instrument is an object. This includes entities such as lines, point groups, vector groups, cylinders, spheres, planes, surfaces, and point clouds.
- **Other.** Anything that is not a point, object, or instrument. For example, relationships, Measurement Plans, scale bars, charts, and vector groups.

Knowing these four entity types is important, because most commands in SA are filtered for the proper entity type. For instance, the **Edit>Delete Objects** command only allows selection of objects—attempting to select a point will choose its parent point group instead (because a point is not an object, but a point group is).

Double Colon Format

When referring to an entity in your SA file, it is important to know where it is organized in your job file. For instance, in what collection is a specific surface located?

You will notice a specific syntax used in SA in various dialogs to describe the organizational location of an item in SA. A specific point is referred to by listing its containing collection, enclosing point group, and target name separated by pairs of colons, which act as separators:

`Measured::Tie-Ins::3`

This is an unambiguous reference to the point named **`3`** in the point group named **`Tie-Ins`**, which is inside the collection named **`Measured`**.

Objects and other entities have a similar syntax, but only the collection and entity name are used:

`Nominal::Centerline`

This name refers to an object named **`Centerline`** inside the **`Nominal`** collection.

Instruments are referred to by their instrument index number. An instrument index number is a unique number that is assigned to each instrument in a given collection. The first instrument added to a collection becomes instrument index number **`0`**, the second is **`1`**, and so on. When an instrument is moved from one collection to another, its index number changes. If the fourth instrument in collection **`A`** is moved to collection **`B`**, and there are already 2 instruments in collection **`B`**, the instrument will change from index number **`3`** in the old collection to index number **`2`** in the new collection.

The syntax for instruments is similar to that for objects, except instead of using an object name, the instrument index number is used instead:

`ThisCollection::1`

This name refers to the second instrument in the **`ThisCollection`** collection. Sometimes, you will also see this syntax with the instrument name at the end, such as

`ThisCollection::1 - Zeiss ETh 2 Theodolite`

The double colon syntax is used in several places in SA—for example, when displaying full point names or instrument names in the graphical view, when viewing an instrument interface, or when working in Measurement Plan scripts.

The Wire Model

One of the most important things to keep in mind when working in SA is that measurements are linked and tied directly to the instrument that measured them. If you move an instrument (using **Instrument>Drag Instruments Graphically**) from one area of the SA workspace (Figure 4-8) to another (Figure 4-9), you will find that all of the

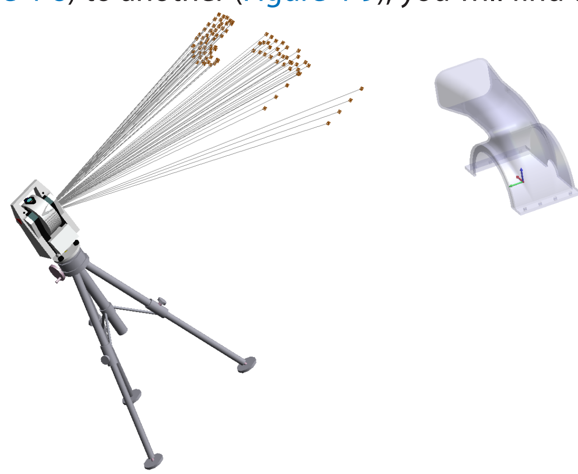


Figure 4-8. An instrument has measured the surface of this part. Prior to alignment, the measured points are not yet located to the CAD surfaces.

targets that were measured by that instrument also move along with it. You can think of each of the measured points as having a straight piece of wire between the instrument and the measurement—the instrument and its measurements move together as a rigid body.

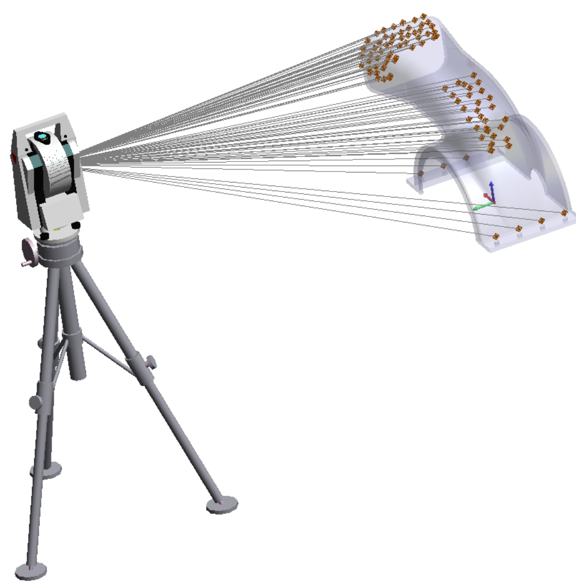


Figure 4-9. After alignment, the points are aligned to the CAD surface. Here, the instrument was moved. The points just "came along for the ride".

This behavior has a number of important benefits, and if you stop to think about it, the behavior most accurately reflects reality. When a measurement is recorded, the instrument provides to SA the position of the probe relative to the instrument's reference system. For spherical measurement devices such as total stations, laser trackers, and laser radars, this consists of a horizontal and vertical angle and a distance component. These three values indicate where the point is located relative to the instrument's reference--usually its optical center. The only thing the instrument knows about the measurement is the location of that measurement relative to itself—it (typically) knows nothing about where the measurement is located in the real world.

The link between the position of the instrument and the position of the measurement (relative to the instrument) should really never be broken, because if the two were to become disassociated relative to each other, then the value of the measurement is lost. That is, its position (and therefore the measurement) becomes meaningless to us. Therefore, it makes sense that SA maintains and keeps track of this relationship.

The Alignment Process

Tied to the association between the instrument and its measurements is the idea of alignment. Alignment is really the process of positioning the digital instrument in the SA file until it ends up in the same corresponding position as the instrument in the real world. For example, Figure 4-8 shows an instrument and its measurements prior to alignment, and Figure 4-9 shows that same instrument after alignment. In both cases, the only thing that was moved was the instrument—the measurements merely moved as a result of being rigidly tied to the instrument. Put another way, the measurements "came along for the ride".

As long as a reference system is measured to properly perform an alignment at some later time, alignment is not required to be performed as one of the first steps in the measurement process. If alignment to CAD surfaces or other instruments is not required, then there is no need to perform any alignment at all.

> **Note:** If you are performing a build operation or need on-the-spot comparison of measurements to CAD or to your reference system, then the alignment must be performed at that time.

You can measure features on a part, measure a reference system, move the instrument, measure more features and the reference system, and then measure again from a third perspective, all without once performing an alignment. Only when the analysis is performed are the instruments required to be aligned (so that you can extract meaningful information from the data). As a result, you can measure everything on the shop floor and perform the alignment at your desk at the end of the day.

Feature Measurement

One important general rule to follow in SA is that each feature should be measured into its own point group when possible (see Figure 4-10). One intended meaning for the term *feature* in this case is a specific portion of a measured part. For instance, measurements on the inside of a hole should go into one group, measurements of the top surface of the part should go into another, and measurements of the side of the part should go into a third. But another intended meaning of feature in this context is to refer to any set of related points. So, for instance, all points in a drift check should be measured into their own point group, and the reference points (fiducials) should be measured into another.

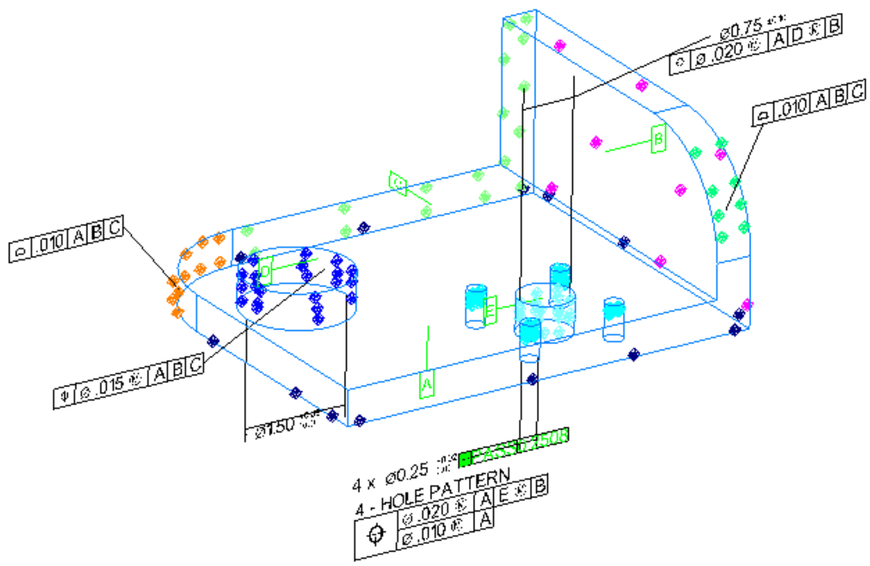


Figure 4-10. Proper feature-based organization of point groups. Measurements for each individual feature on this part are organized into separate point groups.

SA is designed around the assumption that features are measured into their own individual point groups. If you follow this rule, you will find that working in SA becomes much easier and more intuitive. The only time when this rule does not apply is when measuring with theodolites. Since theodolite measurement requires at least two observations to calculate a target position, each observation of a given target should be measured under the same point.

Offset Compensation

When storing measurements, SA always records the point representing the center of the probe (SMR, ruby tip, etc). Unless non-contact or zero-offset measurement is being performed (such as measurement with a laser radar, scanner, point probe, or photogrammetry), the recorded point will not represent the point where the probe touched the measured surface, but instead will be at its center (see Figure

4-11). (For zero-offset measurement, the recorded point is at the point of contact). What this means, of course, is that the distance between the measured point and the measured surface is not zero, but is instead equal to the probe radius (also known as the planar offset).

In order to properly compensate for probe offsets, two values must be known: the probe offset, and the surface direction. These two properties are discussed in the following sections.

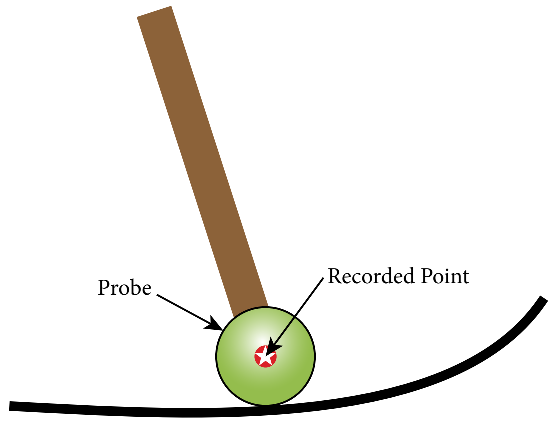


Figure 4-11. The recorded point for any measurement is always at the probe center.

Probe Offsets

When a measurement is recorded, SA stores not only the probe center point, but also the corresponding offsets. There are actually two offsets used: a planar offset and a radial offset. The planar offset is an offset perpendicular (or normal) to the resulting geometry, whereas the radial offset is an offset in the plane of the geometry.

The planar offset is used to offset the measured point toward a surface, line, or curve. The radial offset is used to offset measurements inward or outward when measuring or fitting circles and ellipses using tooling that introduces a radial offset, such as a pin nest (see Figure 4-12).

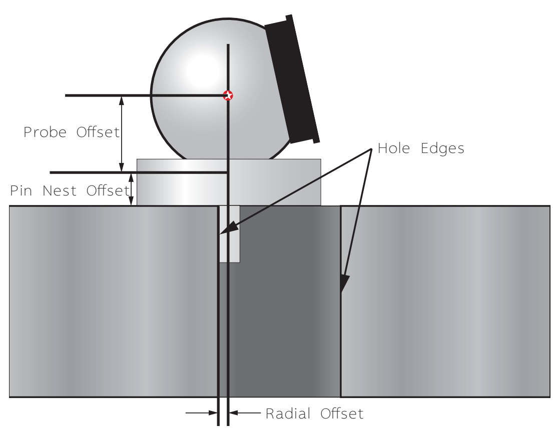


Figure 4-12. The radial offset is used for shifting measurements inward or outward when using a pin nest.

If just a spherical probe is used (such as a ruby tip or an SMR), the radial offset is ignored (it is never used). If additional target tooling is used—such as the pin nest in Figure 4-12, the offsets attributable to the additional tooling are included in the point's offset. For instance, the pin nest pictured introduces 0.25″ of additional planar offset and 0.125″ of radial offset on top of the SMR's 0.75″ planar offset. This yields a total planar offset of 1.0″ and a total radial offset of 0.125″.

Whenever a measured point is compared to an object such as a CAD surface, line, or curve, probe offset compensation is automatically applied, and the reported deviation is actually the deviation between the point of contact and the corresponding closest point on the object of interest. Likewise, whenever geometry is created from measured points, the resulting geometry is automatically offset from the measured points, and the resulting geometry represents the surface actually measured.

Obviously, in order for SA to compensate for offsets correctly, the correct offsets must be stored with the measured points. Offsets should be set in the appropriate instrument interface whenever target tooling is changed, and as a result measured points should have the correct offsets as they are measured. However, if you forget to set the proper offsets when measuring, you can always correct them after the fact by changing the offsets on measured points. (More information on this can be found in the Measurement section).

Surface Normal Direction

In addition to having the proper stored offsets, one additional factor is required to be correct in order to perform a properly compensated analysis: it is critical that the direction of the surface normals being used in any comparison are facing the proper direction. Consider the example in Figure 4-13, in which four measurements were taken with an SMR at four positions on a measured surface (the blue dashed surface in the illustration). Note that, for clarity, the as-built error in the measured surface is greatly exaggerated here.

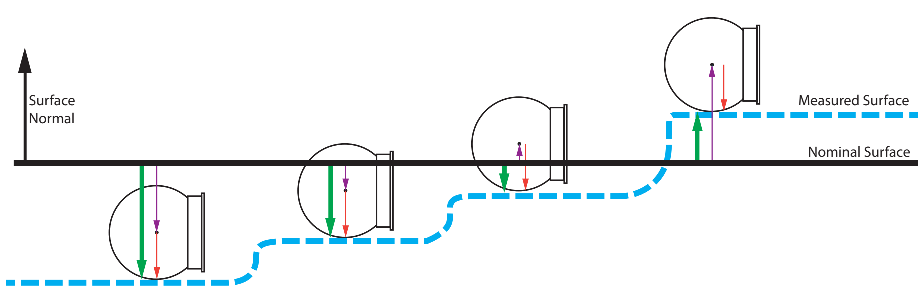


Figure 4-13. Proper offset compensation is applied when the surface normal faces the side to be measured.

Suppose we wish to compare our measurements to the nominal CAD part in order to determine whether the as-built part is "high" or "low" of the ideal design. When comparing the measured point to the nominal surface, SA first calculates the deviation from the nominal surface to the recorded point (the purple vector), and then adds the probe offset in the direction opposite to the surface normal (the red vector in the illustration). The resulting vector (green) indicates the deviation between the measured surface and the nominal surface. In this example, the first three measurements are low of nominal, and the last measurement is high of nominal.

To visualize how an incorrect surface normal direction can result in incorrect results, suppose that the surface normal in Figure 4-13 is accidentally reversed and is now facing the wrong direction (Figure 4-14), with all else being the same. The deviation from the nominal surface to the measured point stays the same (purple vector). Again, SA adds the probe offset in the direction opposite to the surface normal, but since the surface normal is facing the wrong direction, the offset compensation is performed in the wrong direction. As a result (green vectors), the reported deviation is off by exactly the probe diameter—a very large error!

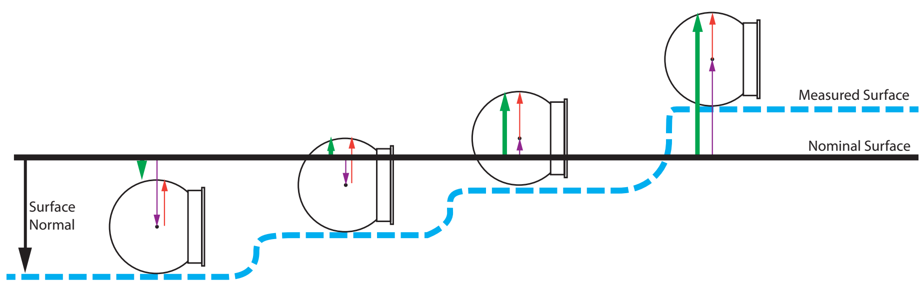


Figure 4-14. Offset compensation applied when a surface's normal is facing the wrong direction. This will result in incorrect results!

This behavior leads to a golden rule in SA—one that is referred to in various places in the SA dialogs. The rule is that in order for deviations to be reported properly, all surface normals must face the side being measured. Put another way, the surface normal direction for the surfaces being compared should point to the side from which you're approaching the part with the probe (the other direction should point into the part's "material").

Warning: If surface normals are facing the wrong direction, the calculated deviation will typically be in error by exactly the probe diameter.

There is one simple way to ensure that you comply with this rule: When working with surfaces, you want to ensure that the normals all point toward the "outside" of the object—that is, they should point away from the material of the part. If you imagine that your CAD part is submerged in the ocean, the surface normals should point to the side where the water is touching the surface. Upon importing CAD to be used in analysis, one of your first steps should be to ensure that the surface normal directions are all correct, and to correct any normals that point inward.

Warning: If stored target offsets are incorrect, the calculated deviation (or the location of geometry resulting from a geometry fit) will be incorrect.

For information on identifying surfaces whose normals are facing the wrong direction, see the Surface Backsides section in "The Graphical View".

Comparing Measurements to Points and Objects

The fact that SA automatically compensates for probe offsets results in a few rules that should be kept in mind when performing comparisons between measured points, objects, and surfaces:

- When comparisons between points are performed, the comparison is always between centers. Point to point deltas and best-fits, for example, always compare center to center. This leads to a corollary that if two separate points are to represent the same point in space (such as for a reference monument), their centers should be coincident. This might seem obvious, but if the same point is measured twice with probes that have different offsets, a physical target adapter is required to make one of the offsets match the other.
- When comparisons between points and objects or surfaces are performed, the comparison automatically compensates for probe offset.

There are a few cases where you will want to override this behavior and ignore offsets, however.

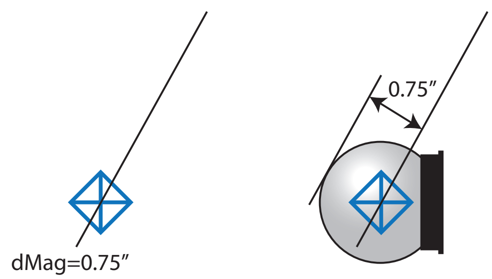


Figure 4-15. Comparing a measured point with 3/4″ offset to a line.

For example, suppose a measured point with a 3/4″ offset lies directly on a line. This would appear in SA as indicated at left in Figure 4-15. A normal comparison between the point and the line will indicate a deviation of 0.75″ as depicted at right, because offset compensation is being applied. That is, you are 0.75″ from touching the line. Typically, however, in this situation you want to compare the center of the probe to the axis. For instance, perhaps you've set the probe in a hole to take a measurement, and you want to compare the measurement to the nominal hole center. In that case, you'll want to ignore the offset so that this situation would indicate a deviation of zero. There are several ways to do this in SA, including overriding target offsets to zero or using commands that specifically ignore offsets.

Measured Points vs. Cloud Points

It is important to understand the distinction between measured points and cloud points, and to know when it is more appropriate to use one instead of the other when taking measurements. Both points and clouds are discussed in more detail in later chapters—however in this section we will discuss the broad differences between the two entities.

Measured points carry with them a relatively large amount of overhead. They are not simply an XYZ coordinate with a name, but actually store quite a bit of information. As described in "Traceability" on page 130, measured points store metadata in addition to the measurement itself and its offsets. Weather parameters (temperature, pressure, and humidity), RMS data, measurement mode information, timestamps, and other information are all stored with a point for traceability and documentation purposes. While this information is useful and great to have, it also means that a measured point puts a relatively large strain on the data processing and computing capabilities of the machine. Storing and handling that wealth of data can become relatively stressful on a computer's resources if there is a very large number of points.

Note: All points in a cloud should be measured with the same offset. If you change probes and the offset changes, put those points into a different cloud.

Cloud points, on the other hand, are designed for very large quantities of measurements and are optimized to be much more lightweight from a computer resources perspective. Cloud points only store an XYZ value. They have no metadata and no individual names or offsets, and therefore use the smallest possible computer resources. Instead, the point cloud itself (which is the container for the cloud points) stores the offsets for the measurements. Although cloud points (collectively known as a cloud) are typically created from scanning devices, they can be created by discrete measurements from instruments like laser trackers and portable arms as well.

Tip: For best performance, measure into a cloud whenever recording a large number of points (>10,000).

So the question to answer is: when should a measurement be taken as a regular measured point, and when should it be recorded as a cloud point? The general rule of thumb is that if you are performing any type of surface scanning consisting of a large (>10,000) number of points, you should use point clouds. Note that nothing in this rule of thumb refers to the type of instrument being used. That is—whether or not you are using an actual scanner is of no consequence. If you are "scrubbing" a surface with an SMR and scanning with a laser tracker, for instance, that is considered scanning and point clouds should be used.

Unbounded Geometry

There are several primitive geometry types in SA that are unbounded. This means that the geometry extends infinitely without bound. For an example of this, refer to Figure 4-16.

Although a plane in SA is depicted in the graphical view with a boundary (blue line), this is only so that the plane is visible in the graphical view. From an analysis perspective, the plane extends infinitely in two dimensions, just as it might be defined in an academic math class. Keep in mind that any time you compare a point to unbounded primitive geometry, the geometry will behave as if it extends infinitely along its unbounded edges. The following primitive geometry types are unbounded:

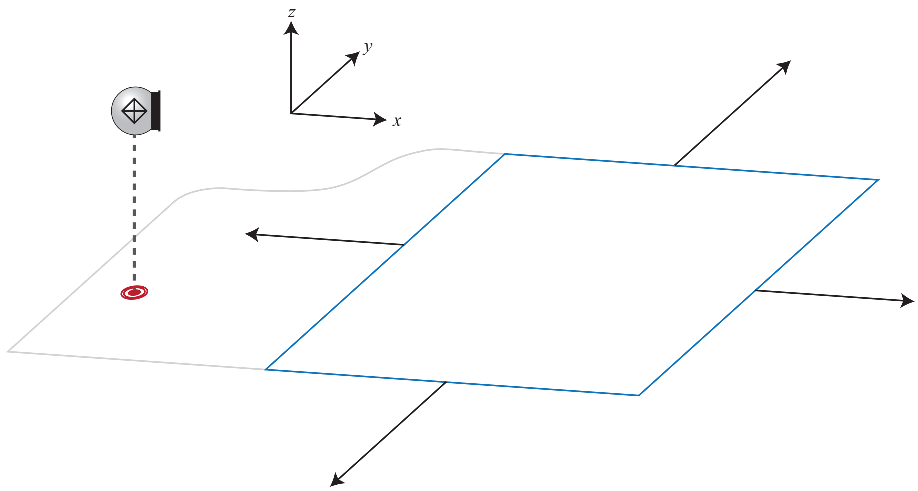


Figure 4-16. A plane in SA has a graphical depiction (blue lines) so that it can be seen, but from an analysis perspective it extends infinitely in two dimensions. When compared to a point beyond its graphical depiction, a plane is extended infinitely. The closest point to this measurement on the depicted plane is shown as a red dot above.

- Planes
- Lines
- Cylinders (lengthwise)
- Paraboloids
- Cones

If you would like the geometry to be bounded for analysis purposes, it must be converted to a surface.

The Working Frame

SA has a concept known as the *working frame*. The working frame is a coordinate frame through which all data is represented or mapped. Put another way, it is the active coordinate frame. Any coordinate frame can be made the working frame, and when a frame is working, all data (point coordinates, delta values, rotations) is represented in SA with respect to that frame unless specified otherwise.

It is important to recognize that changing a coordinate frame does not physically move anything in space—it merely results in a change in the reference system used to represent that same position and orientation. This allows you to take measurements in any arbitrary default coordinate system and represent them in a system that better describes the data.

A Closer Look

You can assign a frame of reference to a watch window in order to see deviations represented in a given frame regardless of the working frame. You can also specify a reporting frame for an object in order to view that object's properties on a report with respect to a specific coordinate frame. (Those values will still be represented outside of reports with respect to the working coordinate frame. That is, if you assign a reporting frame to a point group, the coordinates will be represented in the active coordinate frame unless viewing them in a report).

The Default Collection

Collections are organizational elements used to contain entities in a hierarchy in SA. Just as there is always exactly one working frame, there is always exactly one default collection. The default collection (colloquially referred to as the working collection) is the organizational container inside which all newly created entities are placed, unless specified otherwise. For example, if a circle is created as the result of a geometry fitting operation, that circle will be placed into the default collection.

Traceability

One of the unique features of SA is its traceability. Traceability refers to maintaining an unbroken chain of comparisons relating measurements back to a defined standard (Figure 4-17). National measurement institutes such as the National Institute of Standards and Technology (NIST-USA), the National Physical Laboratory (NPL-UK), and the Physikalisch-Technische Bundesanstalt (PTB-Germany) all maintain national standards for weights and measures for their respective countries. Properly calibrated measurement instruments are evaluated and certified to this standard, which establishes the first link in this chain.

The chain is continued through measurement in SA. Each measured point in SA contains an array of metadata that describes auxiliary information about the measurement, such as the instrument (and serial number) that measured it, the time and date it was measured, who measured it, weather factors such as temperature/pressure/humidity, and more (see Figure 4-17). This information all helps maintain a complete record of the data.

In addition, actions performed in a given job file are logged, and entities themselves have historical logs that indicate how they have been modified throughout their existence. Entities even maintain information about how they were created in the first place. All of this information helps provide a complete picture of traceability for your measurements. If a specific instrument begins to show anomalies, you can look at any previous measurements, perhaps from weeks or months earlier, and identify which instrument measured it. This might give you cause to question those measurements as well—but without this traceability metadata, such a correlation may not have been possible.

NIST further maintains that "Traceability of measurement requires the establishment of an unbroken chain of comparisons to stated references, each with a stated uncertainty".[1] Measurement uncertainty information can also be calculated for any measurement in SA, providing compliance with this policy but also providing a number of additional benefits as well (such as USMN—Unified Spatial Metrology Network).

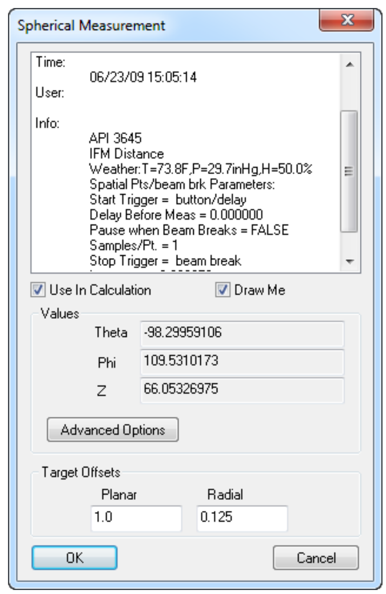


Figure 4-17. Some of the traceability metadata stored with a laser tracker measurement.

1 NIST Policy on Metrological Traceability.

This Page Intentionally Left Blank.

Exploring The User Interface

CHAPTER 5

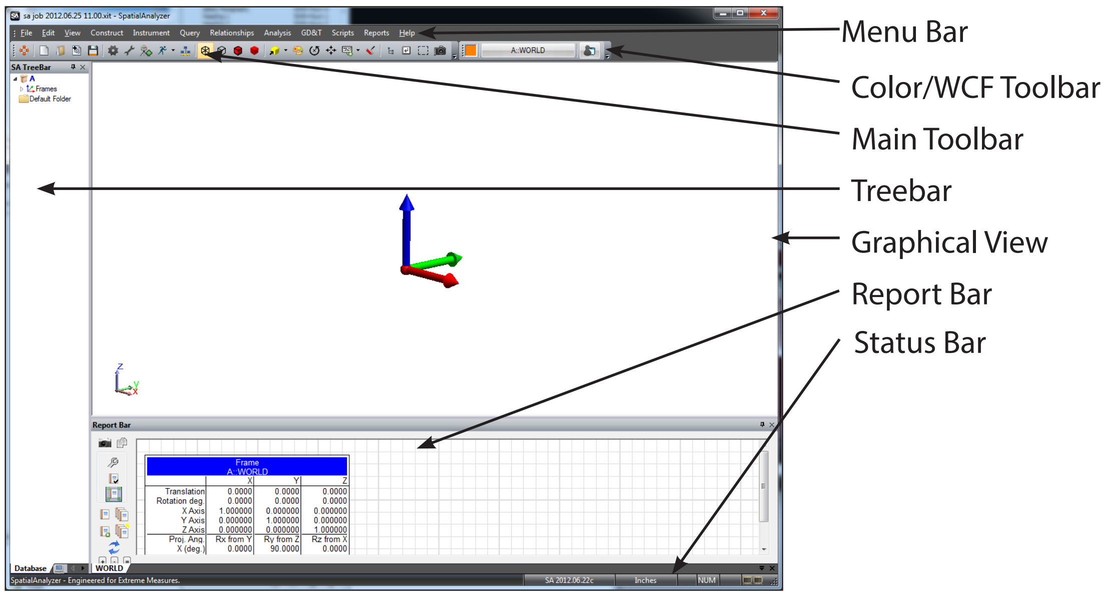
SpatialAnalyzer provides a user-friendly interface to access all of your measurement data. There are often several ways to perform a given task, enabling you to choose a preferred method of interacting with the application.

There are several components to the SpatialAnalyzer interface. Each component comes in a separate window which can be docked to the edge of the screen or to the edge of another window. The components can be closed independently to allow you to maximize your screen space and to enable you to customize the user interface for the specific task at hand.

As depicted on the next page, the various components of the SA interface include:

- **Menu Bar.** A list of menu commands as seen in nearly all Windows applications.
- **Toolbars.** Strips of icons that provide convenient graphical access to common commands. By default, SA provides the Color/WCF Toolbar and the Main Toolbar, although you can create additional custom toolbars.
- **Treebar.** A hierarchical view of all entities in the open job. For brevity this will typically be referred to simply as the tree.
- **Graphical View.** A three-dimensional rendered view of the current job.
- **Report Bar.** Displays important information about the selected item and a convenient way to generate reports.

- **Status Bar.** Displays the software version number, non-critical messages, and includes a few other minor indicators.



As Figure 5-1 indicates, additional components may be visible (but aren't displayed by default):

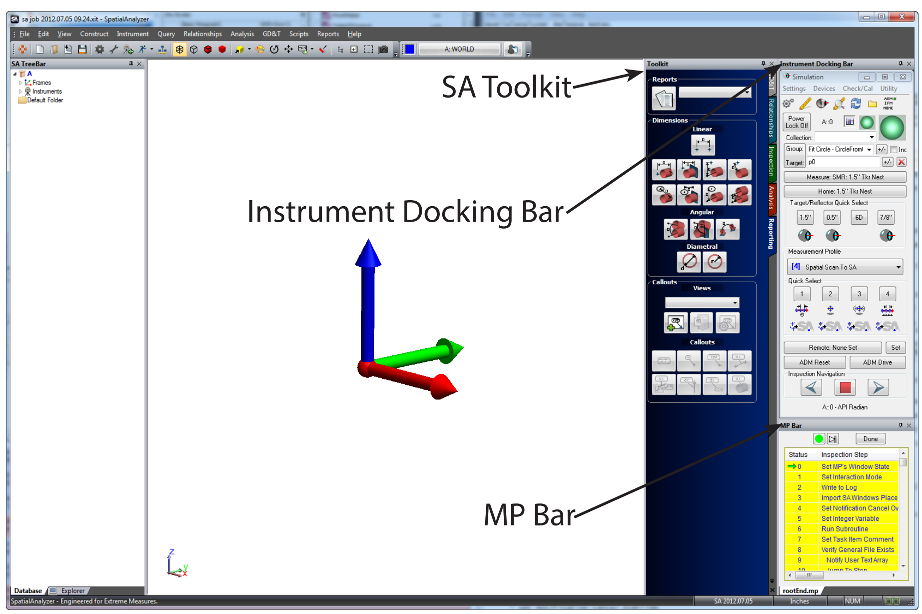


Figure 5-1. Additional User Interface windows.

- **MP Bar.** Displays Measurement Plan commands as they execute (requires SA Ultimate or greater).
- **Instrument Docking Bar.** Houses instrument interfaces when they are docked inside the SA window.
- **SA Toolkit.** Provides access to Geometric Dimensioning & Tolerancing as well as convenient access to tools for inspection, relationships, analysis, and reporting.

To Hide an Interface Component:

- Click the icon in the corner of the window, or
- Right-click on the menu bar, toolbar, or a window title bar and uncheck the component you wish to hide. Check it to show it again.

Interface components can be set to auto-hide, so that they collapse into a tab at the edge of the screen when not active. This provides for more space to work in other interface windows.

To Set an Interface Component to Auto-Hide:

Click the icon. When the mouse is moved away from the window, it will collapse into a tab (after a short delay). Roll over the tab and click the icon to disable the auto-hide behavior.

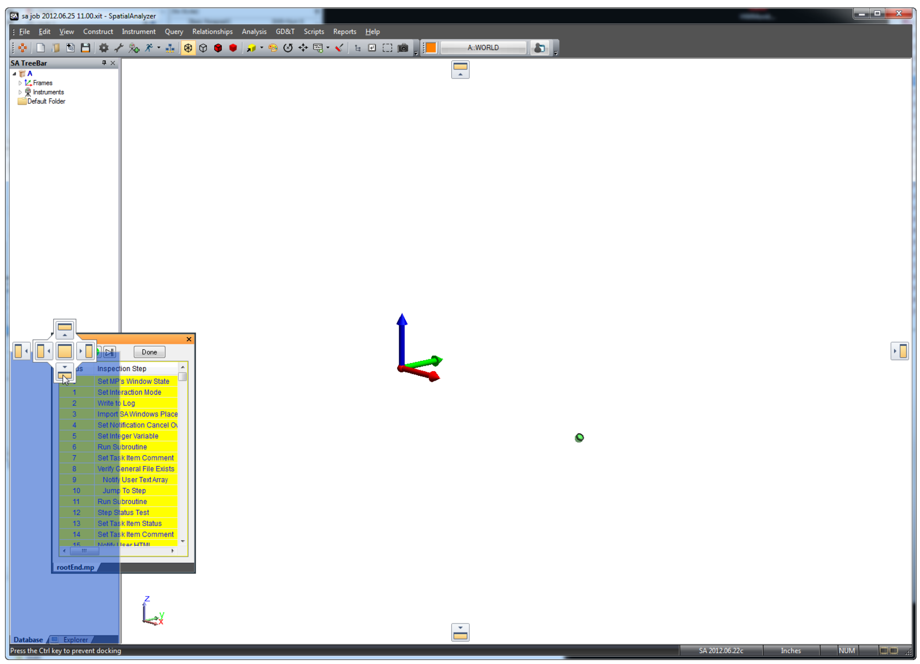


Figure 5-2. Docking the MP Bar to the bottom of the tree.

You can rearrange the positioning of the different user interface components using docking. Interface components can be undocked, docked to the edge of the application screen, or docked along the edge of another interface component.

To Undock an Interface Component:

- Click and drag the title bar for the component away from its current position.

To Dock an Interface Component to the Edge of the Screen or Another Component:

- Click and drag the title bar for the component. As you drag, a series of boxes representing locations to dock the window will appear. Drag over an icon to dock to a specific position (see Figure 5-2).

You can customize the user interface, and then save this customization to a User Interface Profile. For more information, see "Custom UI Profiles" on page 144.

Toolbars

Toolbars provide icons for quick access to frequently-used functionality and are organized into related groups. There are two factory-default toolbars that ship with SpatialAnalyzer: the *Main Toolbar* and the *Color/WCF Toolbar*.

Main Toolbar

Figure 5-3. The Main Toolbar.

User Manual. Opens the manual (this document) for SpatialAnalyzer.

New. Starts a new job file.

Open. Opens a saved job file.

Auto Import. Imports a file automatically based on its file type.

Save. Saves the current job file.

User Options. Opens the user options dialog.

Object Properties. Displays the properties of an object after clicking the object in the graphical view.

Add Instrument. Adds an instrument to the current job.

Run Instrument Interface. Starts an instrument's interface.

SA Network. Opens the SA Network Browser.

Rendering Settings. Switches between Wireframe, Hidden Line Removed, Solid+Edges, and Solid rendering modes.

Views. Selects, edits, and saves preset views.

Background. Changes the background and highlight colors.

View Rotation Center. Defines a pivot point for view rotations.

Autoscale. Automatically scales the graphical view to fit all visible entities.

Callout View. Adds callout views to the active collection.

Color Objects. Changes the color of selected objects in the graphical view.

F2 - Select from a List. Opens the F2 selection dialog.

Enter - End Multiple Selection. Equivalent to pressing the `Enter` key. Ends multiple-select mode.

Graphical Selection Mode. Changes the method for graphically selecting items with the mouse.

Capture Graphics Picture. Captures the current graphics view as an image for use in reports or other documents.

Color/WCF Toolbar

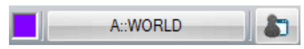


Figure 5-4. The Color/WCF Toolbar.

Working Color. Changes the working color.

A::WORLD Working Frame. Displays the current working frame and allows you to pick a new one.

User Interface Profiles. Used to define and activate custom user interface profiles. For more information, see "Custom UI Profiles" on page 144.

Toolbars can be rearranged by dragging on their toolbar grips(see Figure 5-5). You can completely undock a toolbar from the menus, or simply rearrange them. To dock a toolbar back into its original location, drag the grip to slide the toolbar back into place.

Figure 5-5. Rearranging a toolbar by grabbing its grip.

You can create your own custom toolbars. For more information, see "Custom UI Profiles" on page 144.

Treebar

Database Tab

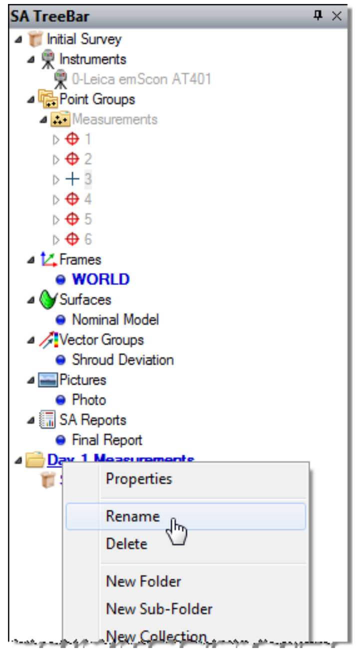


Figure 5-6. The database tab.

The Database tab of the tree displays a hierarchical view of all entities in the current job (see Figure 5-6). It's called the Database tab because it reveals all of the entities in SA's underlying measurement database. This provides quick access to items of interest and enables simple but effective organization and management of properties. The tree contains items such as instruments, point groups, reports, embedded files, and geometric objects (lines, planes, etc). Headings or parent items within the tree can be expanded by clicking on the ⊞ button to expose child data (on some machines, this appears as an ▷ icon). Clicking the ⊟ button will collapse these items again. Right-clicking on an entry will display a context menu of commands that relate to the selected item.

The hierarchical tree displays a number of different entities depending on what is present in the current job file:

Folders. Like folders in Windows Explorer, you can sort collections into folders to organize the SA file.

Collections. An organized collection of entities (objects, instruments, reports, etc) that can be managed as a set in the tree.

Frames. Coordinate frames.

Instruments. Measurement instruments—laser trackers, portable CMMs, etc.

Point Groups. A series of points (measured or constructed) that are conveniently treated as a single object.

Lines. Geometric lines with a begin/end point and a defined direction.

B-Splines. Curves that have a mathematical definition and a defined direction along the curve.

Planes. Planar geometry that, while drawn with a finite size in the graphical view, extends infinitely in two dimensions.

Circles. Circular geometry.

Ellipses. Elliptical geometry.

Cylinders. Cylindrical geometry.

Spheres. Spherical geometry.

Cones. Conical geometry.

Paraboloids. Parabolic geometry.

Polygonized Surfaces. Unlike regular surfaces, polygonized surfaces are faceted and have no true mathematical definition for the surface (except for its planar facets).

Surfaces. Surfaces are parametrically defined and have a complete mathematical definition for every point on the surface. They are exact representations of a given surface.

Vector Groups. Vector groups represent the deviation between two entities as one or more vectors.

Point Clouds. Groups of points typically gathered from non-contact measurement devices (such as a laser line scanner or coherent laser radar).

1-D Data. One-dimensional (scalar) data that has been measured by an auxiliary device (such as a digital micrometer).

Relationships. Dynamic (continuously updated) comparisons between entities.

Perimeters. Boundaries that are used to define automated scan areas.

Callouts. Callout views enable you to add captions, labels, and callouts in the graphical view.

Charts. A number of different statistical charts can be created as a result of various operations in SA.

SA Docs. An RTF (Rich-Text File) based reporting format that can be generated in SA.

SA Report Templates. A template defined for a report in advance, which enables generation of that report with updated data at any time.

Measurement Plans. Scripted programs that can automate measurement, analysis, and reporting operations.

Embedded Files. Any file can be embedded into an SA file for quick access or to conveniently link a file with the SA job.

Graphical Entities. Geometric shapes (such as boxes and pyramids) that act as purely visual elements and cannot participate in any analysis.

Crib Sheets. Macros to help automate the naming of points and the way in which they're measured.

SA Docs. A legacy reporting format that is no longer used, but may be used in older .xit files.

Annotations. Annotations associated with a feature for use with SA's GD&T functionality.

Datums. Datums for use with SA's GD&T functionality.

Feature Checks. Analysis results for use with SA's GD&T functionality.

Pictures. Imported images or graphical view screenshots that can be dragged into SA Reports.

SA Reports. One of several types of reports that can be created in SA.

Events. Easily-reportable records of significant events (such as fitting geometry, minimizing relationships, or importing a CAD file) that have taken place in the file.

Robots and Machines. The robots or CNC machines in the current job file.

Scale Bars. Any defined scale bars.

Dimensions. Defined dimensions between any two entities.

Custom Report Tables. Custom tables generated via scripting that can be used in reports.

All of the above items (with the exception of Folders and Collections) are actually categories of items. The tree organizes items in a collection by their category--the individual items of that category are listed under the category. For instance, in the Instruments category, there might be three actual instruments.

Explorer Tab

The Explorer tab displays the file and directory structure of the local computer. This is a convenient way of navigating the disk to find files to import into the current SA job.

Report Bar

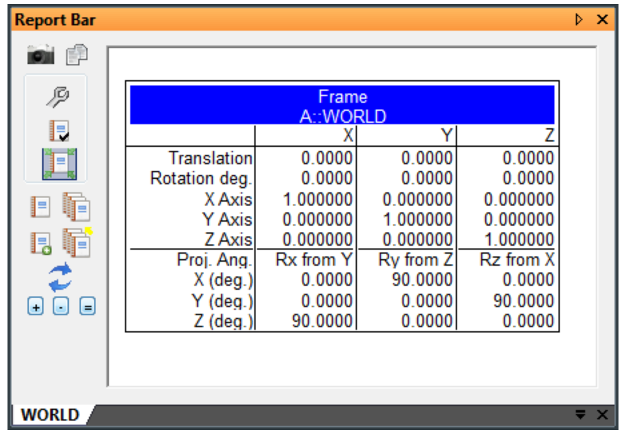


Figure 5-7. The Report Bar showing properties for a coordinate frame.

The Report Bar (Figure 5-7) displays the detailed properties of the currently selected item in the tree. It can also serve as a report generator by allowing multiple tabs of information to be compiled into a single report. For details on the report bar and reporting, refer to the Reporting section.

To Toggle Display of the Report Bar:

- Right-click in the menus, toolbars, or status bar area of the interface and select **Report Bar** from the context menu, or
- From the menus, select **Reports>Report Bar Visible**.

Status Bar

The Status Bar (Figure 5-8) is a small bar at the bottom of the interface that has a number of minor, auxiliary displays. The left side of the bar will display any minor, low-importance messages that may arise during the course of your work (for example, when copying data to the clipboard from the Report Bar).

Figure 5-8. The Status Bar.

To the right of this message area is the SA version. In Figure 5-8, SA 2012.07.05 (a beta version) is running. Next to the version number is the current length unit setting. The three slots to the right of the units display indicate whether `Caps Lock`, `Num Lock`, or `Scroll Lock` are active on the keyboard (respectively). Two small processor icons on the right side of the status bar light up when SA is in a multi-threaded operation.

MP Bar

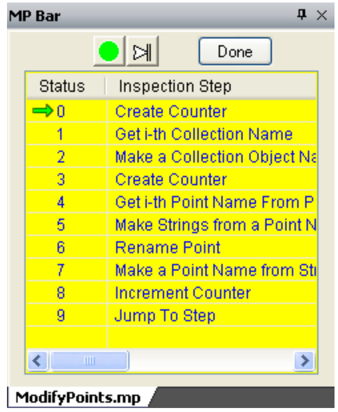


Figure 5-9. The MP Bar with an MP loaded.

The MP Bar (Figure 5-9) is normally only visible when a scripted Measurement Plan has been loaded for execution. The main window lists each command of the MP, as well as a pointer indicating the command which will be executed next, and the status (success, partial success, or failure) of each step. The MP Bar has a button to start executing the current MP , execute the next step in the MP , and terminate/close the current MP . The tabs at the bottom of the MP Bar display the primary MP and any MP subroutines currently executing.

Instrument Docking Bar

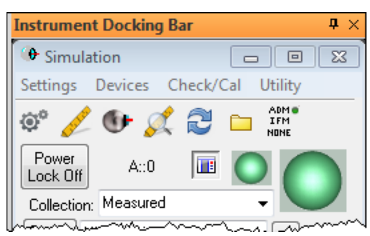


Figure 5-10. The tracker interface inside the Instrument Docking Bar.

The Instrument Docking Bar (Figure 5-10) is a window that houses the currently docked instrument interface. It enables the instrument interface, which is actually a separate application from SA, to be docked onto one edge of the SpatialAnalyzer interface.

SA Toolkit

The SA Toolkit (Figure 5-11) contains five tabs that provide convenient access to a number of useful SA tools. These tabs include GD&T, Relationships, Inspection, Analysis, and Reporting.

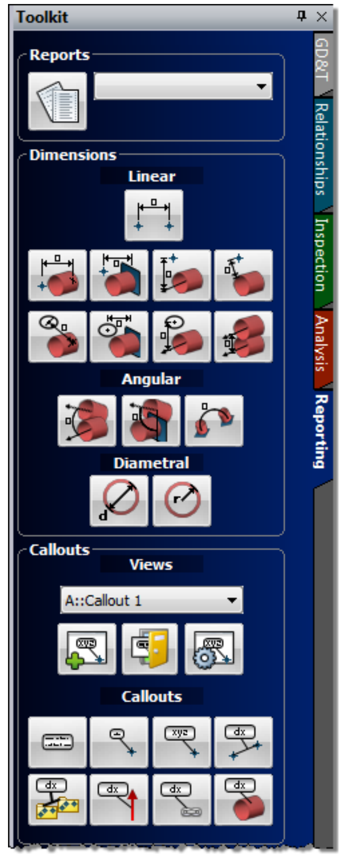


Figure 5-11. The SA Toolkit.

- **GD&T.** Tools for creating GD&T checks from CAD surfaces, primitive objects, or directly from points.
- **Relationships.** Tools for quickly creating various types of relationships.
- **Inspection.** Provides access to a list of inspection-ready items with buttons for advancing through the inspection.
- **Analysis.** One-click access to many analysis tools.
- **Reporting.** Buttons for creating reports, dimensions, and callouts.

UI Themes

A number of standard user interface themes are available to change the look of the SA interface. Three of the 14 interface styles are pictured below:

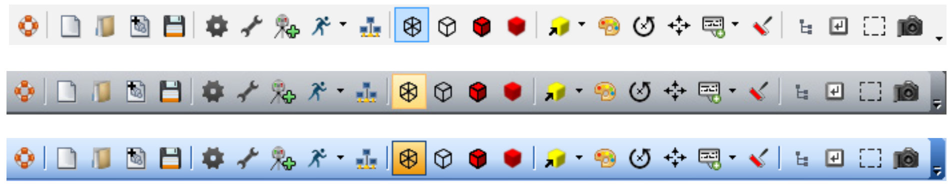


Figure 5-12. Top to bottom: MS Office XP, MS Office 2007 (black), and MS Office 2003 themes.

To Change the Look of the SA Interface:

Note: The user interface theme is saved as a machine setting.

- In the **View>User Interface Features>UI Theme** menu, choose a theme from the list. The interface will update immediately.

Custom UI Profiles

A large portion of the SA user interface is customizable. With user interface customization you can:

- Add, rearrange, delete, and rename menu items,
- Create custom toolbars and menus,
- Save menu and toolbar docking locations,
- Create custom keyboard shortcuts, and
- Create custom icons.

This allows you to modify an interface to display only the commands you use while hiding unused features and leaving more room to work.

User interface settings are saved as profiles, with an `.saprofile` extension in the `Analyzer Data\Templates` directory. Once saved, they can later be recalled, or easily transferred to another computer by copying the file into the same directory on another machine. The selected profile becomes the default used when SA starts, and is saved as a machine setting.

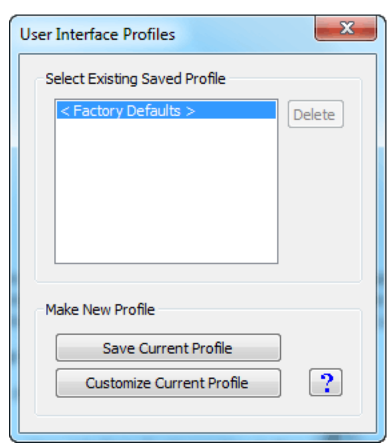


Figure 5-13. The User Interface Profiles dialog.

To Create a Custom User Interface Profile:

1. Do one of the following:

- Click the User Interface Customization button in the *Color/WCF Toolbar*, or
- Select **View>User Interface Customization**.

2. Click the Customize Current Profile button (Figure 5-13). This places you into Customize mode, which allows you to edit the properties of toolbar and menu items instead of performing their equivalent operations. (For details on customization, read the following sections). Customize mode ends when you return to the ***User Interface Profiles*** dialog.

> **Warning:** User Interface profiles are not forward-compatible. Once a profile file is saved, it cannot be opened by a previous version of SA.

3. When finished customizing the profile, click the Save Current Profile button. Enter a profile name, and click OK.

To Delete a Custom User Interface Profile:

1. Do one of the following:
   - Click the User Interface Customization button in the *Color/WCF Toolbar*, or
   - Select **View>User Interface Customization**.
2. Select the profile in the list that you'd like to delete (Figure 5-13).
3. Click the Delete button. The profile will be deleted from the list and from the `Analyzer Data\Templates` directory.

To Reset to the Factory Default User Interface Profile:

- In the ***User Interface Profiles*** dialog (Figure 5-13), choose **`<Factory Defaults>`** from the list, or
- From the menus, choose **View>User Interface Features>Reset GUI Toolbar Layout**.

Customizing Toolbars & Menus

The Toolbars tab (Figure 5-14) allows you to show and hide toolbars, as well as create your own. Custom toolbars are treated just like standard toolbars and act identically. Checked toolbars are displayed, while unchecked toolbars are hidden. The following assumes you are in customize mode (see "To Create a Custom User Interface Profile:" on page 144).

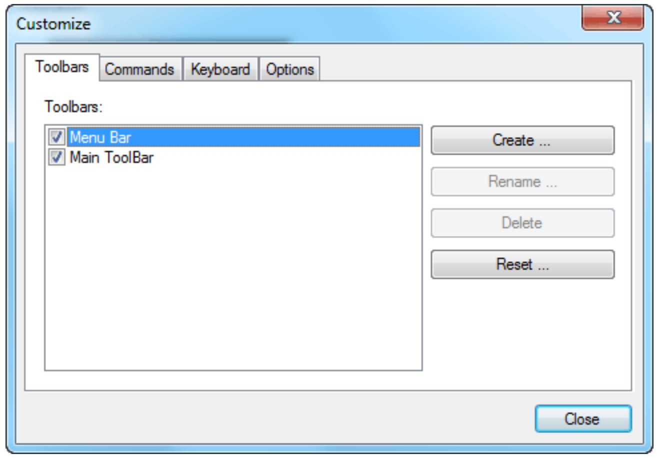


Figure 5-14. The Toolbars tab of the Customize dialog.

To Create a Custom Toolbar:

1. In the *Toolbars* tab, click the Create button.
2. Give the toolbar a unique, descriptive name.
3. The new toolbar will appear in the list, and an empty toolbar will appear in the user interface (Figure 5-15).

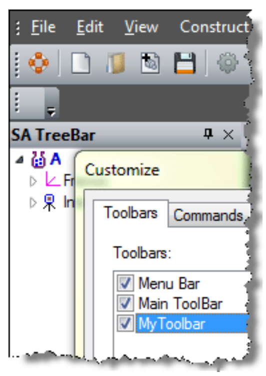


Figure 5-15. A new custom toolbar named MyToolbar has been added. Notice the empty toolbar below the main toolbar.

While in Customize mode (see "To Create a Custom User Interface Profile:" on page 144), you can modify toolbars as well as menu items. You can drag items to create new menus, drag them into current menus, or drag them into toolbars.

To Move an Item:

- Drag a menu, menu item, or toolbar button to the desired position. Drag it back to reverse the process.

To Add a Separator Between Items in a Custom Toolbar or Menu:

- Right-click an item and choose **Begin a Group**. The separator will be placed before the selected item.

Figure 5-16. A group separator has been added after the Scripts menu.

To Delete an Item from a Custom Toolbar or Menu:

- Drag the item into an "empty" area of the screen (not over any menu or toolbar). The mouse cursor will change to , signifying that the item will be deleted, or
- Right-click the item and choose **Delete**.

To Add a New Item to a Custom Toolbar or Menu:

- From the Commands tab, choose a menu category from the list at left. Then drag a command from the list at right to the toolbar or menu.

The appearance of items in a toolbar or menu can be changed. You can view an item as an icon only, text only, or both. You can even modify the icon for an item.

To Change the Appearance of a Toolbar Item:

- Right-click the item and select from either **Default Style**, **Text Only (Always)**, **Text Only (In Menus)**, or **Image and Text**.
- **Default Style.** This will display the icon if one is available. Otherwise, it will display text.
- **Text Only (Always).** This will display the item as text, regardless of whether it is on a toolbar or in a menu.
- **Text Only (In Menus).** This will display the item as an icon when in a toolbar (if an icon is available), and as text when in a menu.
- **Image and Text.** This will display both the icon (if available) and the text for the item, regardless of whether it is in a menu or a custom toolbar.

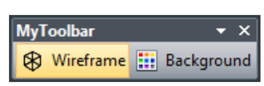


Figure 5-17. Two icons displayed with the *Image and Text* option.

To Change the Text Associated with an Item:

- In Customize mode, right-click the item and click in the *Name* text field. Enter a new name for the item. Note that this will only be displayed if the current appearance setting displays text.

To Reset an Item to its Default Appearance:

- Right-click the item and choose **Reset**.

To Change a Toolbar Icon's Appearance:

- In Customize mode, right-click the item and choose Edit Button Image. This will open a simple editor to allow you to modify the icon, as depicted in Figure 5-18:

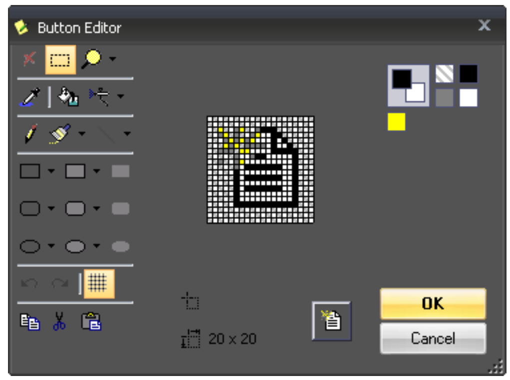


Figure 5-18. The Button Editor.

To Reset a Button's Image Back to Defaults:

- In Customize mode, right-click the item and choose **Reset Button Image**.

To Copy or Paste a Button Image:

- In Customize mode, right-click the item and choose **Copy Button Image** or **Paste Button Image**.

To Assign a Supplied Image to an Icon:

Note: If a command does not come with a default icon, a custom icon cannot be added.

- In Customize mode, right-click the item and choose **Change Button Image**, then choose an image from the list.

Keyboard Shortcuts

You can create custom keyboard shortcuts for virtually every command available to you in SA. Keyboard shortcuts are defined in the *Keyboard* tab of the Customization window. These keyboard shortcuts are saved as part of the custom user interface profile.

To Define a Keyboard Shortcut for a Command:

1. In the Keyboard tab (Figure 5-19), choose the desired category and command from the list.

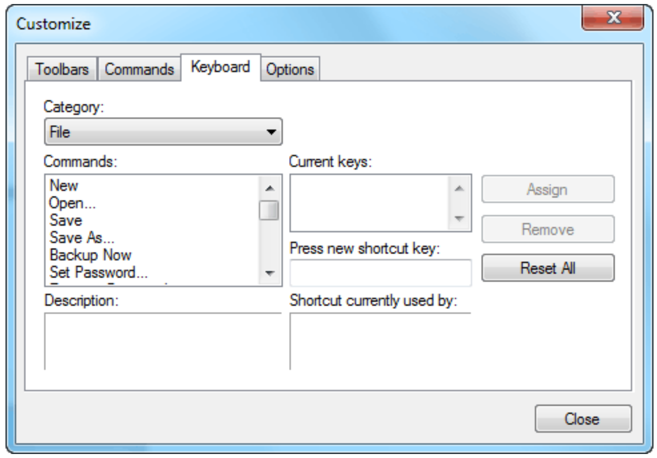


Figure 5-19. Assigning custom keyboard shortcuts.

Tip: You can assign a keyboard shortcut to a main menu item by placing an ampersand before the letter that you'd like to become the shortcut—for example, &GD&&T. This will allow you to open the GD&T menu by typing Alt+G. To place a single ampersand in a menu item, the name field must have two consecutive ampersands (to distinguish it from the keyboard shortcut assignment), as illustrated in this example.

2. Click in the Press new shortcut key field, then enter the desired shortcut.
3. If the shortcut is currently assigned to another command, it will appear in the *Shortcut currently used by:* field as a warning.
4. Click the Assign button to assign the new shortcut.

To Remove a Keyboard Shortcut:

- Select the Category and Command from the *Keyboard* tab, then choose the shortcut from the *Current keys:* box and click Remove.

To Reset Keyboard Shortcuts to the Defaults:

- In the Keyboard tab, click the Reset All button.

Options

The ***Customize*** dialog provides several additional options for controlling the display and behavior of the interface (Figure 5-20).

- **Always Show Full Menus.** If disabled, menu items which are used rarely are not displayed. As a result, only an abbreviated menu is displayed. Click the down-arrow in the menu listing to view all commands for that menu.
- **Show Full Menus After Short Delay.** If selected, the abbreviated menu will be displayed for a moment, followed by the full menu. (Disabled if *Always Show Full Menus* is selected).
- **Animate Menu When Expanding.** Animates the menu as it expands from the abbreviated menu to the full menu. (Disabled if *Always Show Full Menus* is selected).
- **Highlight Rarely Used Menu Items.** Menu items which are not used often will be highlighted.
- **Display Menus With Shadows.** Renders a shadow around menus.

- **Large Icons in Toolbars.** When selected, large icons will be rendered in the toolbars. This is useful for very high resolution displays (or poor resolution eyeballs!)
- **Large Icons in Menus.** When selected, icons displayed along menu items will be drawn in large size.
- **Show Screen Tips on Toolbars.** When selected, displays tooltip information for toolbar icons.
- **Show Screen Tips on Menus.** When selected, displays tooltip information for menu items.
- **Show Shortcut Keys in Screen Tips.** When selected, displays the assigned shortcut key for a toolbar item as part of its tooltip information.
- **Menu Animations.** Controls the transition used to display menus as they expand.

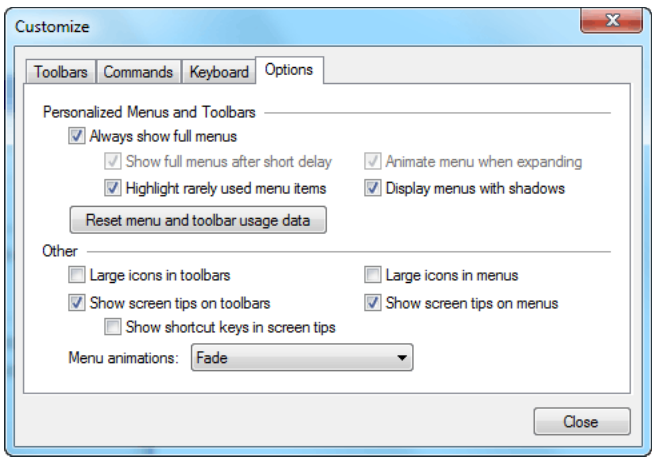


Figure 5-20. Changing UI options in the Options tab.

To Reset the Historical Usage Information for Menus and Toolbars:

- Click the Reset menu and toolbar usage data button.

Language Translation

SpatialAnalyzer comes with seven stock languages:

- English
- Chinese
- French
- German
- Italian
- Russian
- Spanish

If you would like to add an additional language, SA has a language translation tool that can be used to translate menu items, dialogs, and other user interface elements. Language translation files are stored as `.lan` files which can be transferred to different computers and loaded into the interface. Several SA resellers also offer ready-made language translation files to customers.

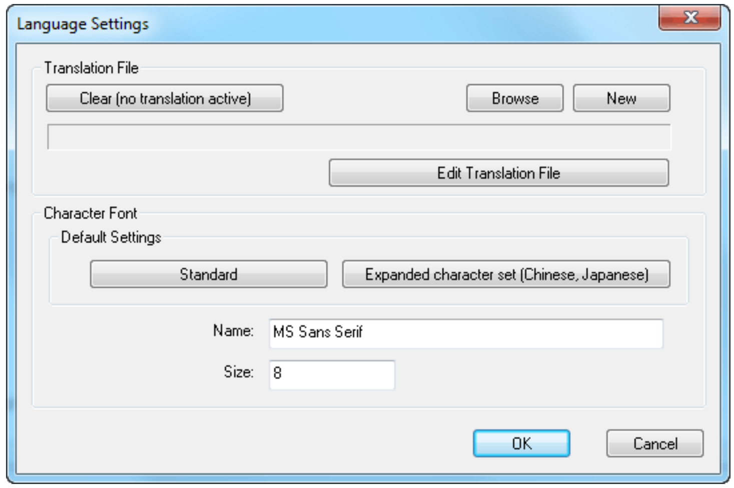


Figure 5-21. The Language Settings dialog.

> **Note:** Language settings are saved with custom user profiles. If you attempt to create a new language translation file while a custom user profile is active, the user interface will be returned to the default.

The language translation interface (Figure 5-21) works by storing translations from English to a specified language. If a certain phrase has not been translated, it will remain in the default English language.

To Make A Stock Language Translation Active:

> **Note:** After changing the active language, you may need to restart any running instrument interfaces for the new language to take effect.

1. Select **File>Language Translation**.
2. From the ***Language Translation*** dialog, select the desired language, then click the Close button. If the window indicates that a custom translation is active, it must be deactivated first before choosing a stock language.

The active stock language will also apply to instrument interfaces unless a specific language has been loaded into that interface.

You cannot modify a stock language—it is read-only. However, you can create a new translation based off of a stock language, as described below.

All custom translation is performed in the ***Language Settings*** dialog. To access this dialog, click the User Options icon, and in the *Display* tab, click the Custom Language File button.

To Create a New Language Translation:

1. In the ***Language Settings*** dialog, click the New button.
2. In the dropdown, choose **Empty Language File** to create a new language file, or **Language File based on** to create a custom language file based on one of the stock languages.
3. Assign a filename for the translation file and click OK.

SA will build a database of phrases to translate and present the NRK Language Translation Interface. If the translation is based off of a stock language file, you will find that many of the phrases are already translated.

To Load an Existing Language Translation File:

- In the **Language Settings** dialog, click the Browse button, and select a language translation file. The translations will immediately be applied to the user interface.

To Remove All Translations and Return to the Default Language of English:

- Click the Clear (no translation active) button.

To Edit the Active Translation File:

- Click the Edit Translation File button.

Character Sets

Certain languages (Asian languages in particular) do not use a traditional Latin character set (standard). To account for this, they may require an expanded character set in order to be displayed properly.

To Modify the Character Set Used for Dialogs:

- Click the Standard or Expanded character set (Chinese, Japanese) buttons.

To Modify the Font Type and Size Used for Dialogs:

- Enter a new font name or size into the dialogs.

Creating a Translation

The language translation interface (Figure 5-22) allows any phrase that is used in SA to be selected and translated. Once a translation has been added for a particular phrase, it will appear translated in the interface. The top list contains phrases that have not been translated. The bottom list contains phrases that have been translated, and lists both the original English and the translated phrase.

To Assign a Name for the Language:

- Type a language name in to the *Language Name* field, then click Apply.

To Search for an Untranslated Phrase:

> **Note:** The search field allows you to enter phrases with wildcard characters (***** and **?**). For example, to search for the phrases "Password Protection" and "Projection", enter `Pro?ection` in the search field.

1. In the top search field, type in a phrase. As you type, the first matching item will be highlighted.
2. To find the next result, click the Next Missing Phrase button again.

To Save the Current State of the Language File and Apply Changes:

- Click the Save button.

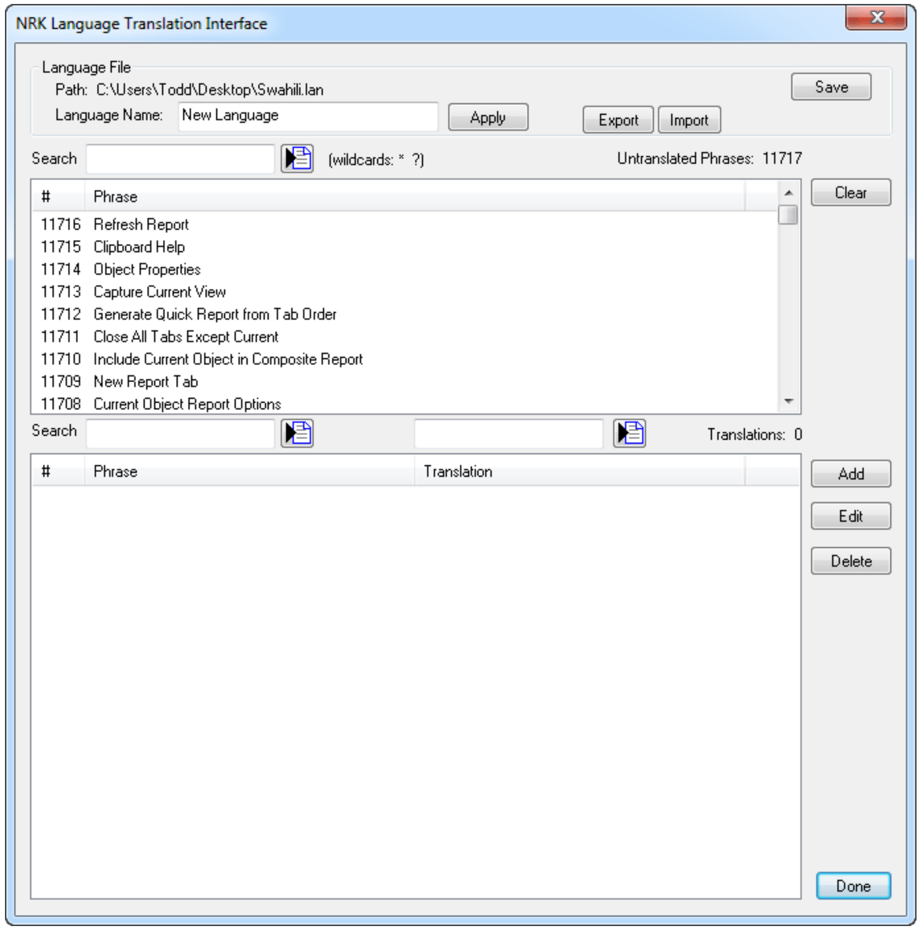


Figure 5-22. The Language Translation Interface.

To Apply a Translation to a Phrase:

1. Select the phrase from the top list of untranslated phrases.
2. Click the **Add** button.
3. In the Translation Entry window, enter the translated phrase. To automatically assign an **ALT** keyboard shortcut to a menu item, precede the desired letter with an ampersand. To enter an ampersand in the phrase, use two consecutive ampersands (**&&**).
4. When finished, click OK. The translated phrase will appear in the list at the bottom, and the phrase will disappear from the top list of untranslated phrases.

To Edit or Delete a Translated Phrase:

- Select the phrase in the bottom list of translated phrases and click Edit or Delete.

To Search for an Original Phrase (English):

1. Enter the search text in the lower-left search field. As you type, the first matching phrase will be highlighted. Wildcard characters (* and ?) may be used in this search field.
2. To find the next matching phrase, click the Next English Phrase button.

To Search for a Translated Phrase (Translated Language):

1. Enter the search text in the lower-right search field. As you type, the first matching phrase will be highlighted. Wildcard characters (* and ?) may be used in this search field.
2. To find the next matching phrase, click the Next Foreign Phrase button.

To Clear Untranslated Phrases From The Dialog:

- Click the Clear button. All untranslated phrases will be removed from the list.

Merging Language Files

Multiple language files can be merged into a single language file. Consider translating an instrument interface, which is performed in the instrument interface itself. This translation is saved into a separate translation file. You can merge an instrument interface translation file into the SA translation file so that a separate translation does not need to be applied to the instrument interface. (This is how the stock language files work).

To Merge Language Files:

1. From the ***NRK Language Translation Interface*** dialog, click the Import button.
2. In the file filter dropdown, change the file type to *Language File (*.lan)*.
3. Select the desired language file to merge, then click Open.
4. Confirm the merge procedure. The translated phrases from the language file will be merged into the current file.

Using .CSV Files For Translation

Once translations have been applied, converting one language to another is made easier with the ability to use .CSV files. Translated phrases (those appearing in the bottom list) may be exported to a two-column .CSV format, which allows you to simply change a translation to another language using a spreadsheet program such as Excel. This way, you can avoid having to click the Edit button for each translated phrase.

To Export Translated Phrases to .CSV Format

1. Ensure the desired phrases to translate already have some sort of translation applied to them (so that they appear in the bottom list).
2. Click the Export button, and choose a file name for the .CSV file.

To Edit an Exported .CSV Translation File

1. In a program capable of working with the CSV format (such as Microsoft Excel or Notepad), open the .CSV file.
2. Leave the first column (*English Phrases*) alone, but change the second column (*Translated phrases*) to the desired language.
3. Save the .CSV file without changing its format.

To Import a .CSV Translation File

1. In the Language Translation Interface, click the Import button.
2. Choose the file to import and click OK. The new phrases will load into the translated phrases section of the window.

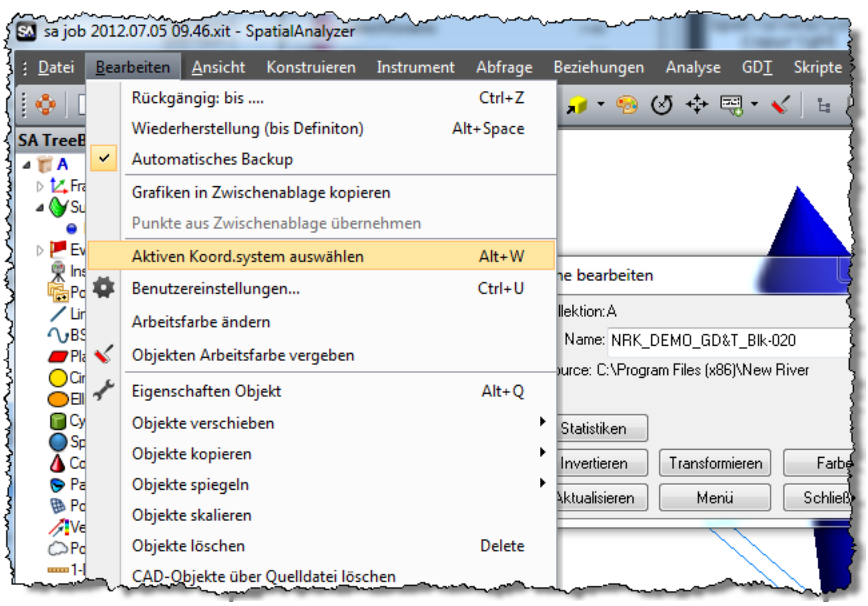


Figure 5-23. SA with a German language translation active.

This Page Intentionally Left Blank.

The Graphical View

CHAPTER 6

The graphical view is arguably the most important single display in all of SpatialAnalyzer. It allows for graphical selection, provides visual feedback, and allows you to visualize your measurements and data.

The graphical view is drawn as a parallel projection—that is, lines of perspective do not converge in the distance but are instead parallel. This is done for a very good reason: there is no "vanishing point", therefore objects appear the same size regardless of their distance from the viewpoint. From a practical perspective, this allows you to visually estimate size and position differences among objects without worrying about whether their distance from the viewpoint is affecting their apparent size.

Navigating the Graphical View

The graphical view provides a graphics-accelerated, three-dimensional rendered view of everything in your job. It allows you to inspect, select, and operate on objects graphically. In order to take advantage of this realistic view of your data, it's important to be able to navigate the view with ease.

Zoom

To Zoom In and Out:

- Hold down `Ctrl` and click-drag the mouse up or down, or
- Scroll the mouse wheel up/down, or

- Press the **PgUp**/**PgDn** key, or
- From the menus, choose **View>View Control>Zoom>In [Out]**, or
- Use the controls in the ***Adjust View Components*** window (see "Adjust View Components" on page 159).

To Zoom Into a Region of the View:

Hold down the left mouse button, drag the mouse to define a rectangle, and release the mouse button. The view will zoom into the defined region.

Autoscale

Autoscale will zoom and center the view so that all visible objects are displayed in the graphical view at once.

To Autoscale the View:

- In the main toolbar, click the Autoscale button, or
- In the menu, select **View>View Control>Zoom>Autoscale**, or
- Press **Alt+A**, the default keyboard shortcut.

Pan

To Pan the View:

- Hold down the **Shift** key and drag the mouse in the graphical view, or
- Use the keyboard arrow keys (**Up**, **Down**, **Left**, and **Right**), or
- Select **View>View Control>Pan>Up/Down/Left/Right**, or
- Use the controls in the ***Adjust View Components*** window (see "Adjust View Components" on page 159).

To Center the View on a Point:

- Middle-click anywhere in the graphical view.

Rotate

To Rotate the View (Freeform):

- Hold down the right mouse button and drag, or
- Use the controls in the ***Adjust View Components*** window (see "Adjust View Components" on page 159).

The viewpoint can be rotated about an axis perpendicular to the screen (this is referred to as spinning the view).

To Spin the View:

- Hold down the **Ctrl** key and right-click-drag the mouse.
- Normally, SA chooses the entity closest to the center of the

graphical view to become the pivot point for rotation. However, if desired you can select a specific point to serve as the rotation pivot point.

To Set a Specific Point as the Rotation Pivot Point:

1. On the Main Toolbar, click the View Rotation Center button.
2. Click a point to set the pivot point.

By default, SA uses the *Front of Sphere* view rotation mode. When rotating in this mode, the rotation center is considered to be at the center of a sphere, while the cursor is "stuck to" the front of the sphere and spinning it. This rotation sense can be reversed to *Back of Sphere* mode in the User options *Machine Configuration* tab.

To Reverse the View Rotation Sense:

In the User options *Machine Configuration* tab, change the *Front of Sphere Mode* setting.

Center In View

Tip: Centering the view on a point is an easy way to make that point the center of rotation.

If you have an item in the tree and are having trouble finding it in the graphical view, you can easily center the view on an object, point, instrument. (Does not apply to point groups, point clouds, vector groups, annotations, datums, feature checks, or dimensions).

To Center the View on an Item:

- Right-click the item in the tree or graphical view and select **Center In View** from the context menu.

Adjust View Components

If desired, you can control the viewpoint (zoom, rotation, and panning) through the ***Point of View*** dialog. To open the ***Point of View*** dialog, choose **View>View Control>Adjust View Components**.

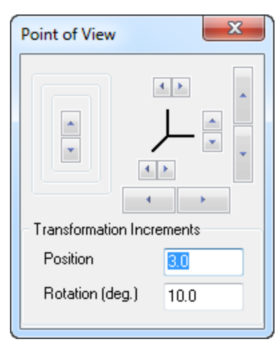


Figure 6-1. The ***Point of View*** dialog has controls for zoom, rotation, and panning, as well as transformation dialog increment fields.

Each of these buttons works similar to the manual view operations, except that the rotation buttons rotate about axes that are horizontal, vertical, or perpendicular to the screen.

This dialog also contains fields for controlling how much an object's position or orientation changes when an up/down arrow is clicked in a transformation dialog.

- **Position.** Controls the increment applied to positional values.
- **Rotation (deg).** Controls the angular increment applied to angular values.

Jumping to Previous/Next Views

It is often useful to jump back and forth among various recent views. As you navigate throughout the graphical view, if you remain at a given viewpoint for at least one second, SA will store the viewpoint. You can then quickly navigate back and forth between previous and next views.

These views are not saved with the SA file. If you manually change views while at an earlier view in the list, later views are removed from the list.

To Navigate Among Previous Views:

Note: The smoothness of the transition between views is controlled in the User Options>Display tab with the *View Animation Steps* option.

- To jump to a previous view, press **Ctrl+Alt+←** or select **View>Viewpoint Control>Go to Prev View** from the menus.
- To jump to the next view, press **Ctrl+Alt+→** or select **View>Viewpoint Control>Go to Next View** from the menus.

Set Viewpoint From Frame

The viewpoint can be set from any existing frame using the **View>View Control>Set Viewpoint From Frame** command. The resulting view will look down the +Z axis.

To Set a Viewpoint From a Frame:

1. Select **View>View Control>Set Viewpoint From Frame**.
2. At the prompt, select the frame you'd like to use for orienting the view.

Set Viewpoint From Instrument Updates

The viewpoint can be set by the current probe position/orientation for instruments that provide 6-DOF information, such as portable CMM arms or probes. This enables the capability to "fly around" a part using an instrument, or to update the viewpoint while measuring.

To Set the Viewpoint Based on Instrument Updates:

Note: For some instruments, you may need to put the instrument interface into a mode that sends updates to see an update in the view.

1. Select **View>View Control>Set Viewpoint From Instrument Updates**.
2. At the prompt (if more than one instrument is available), select the instrument to use for establishing the viewpoint.

Set Viewpoint From Object

Any object's orientation can be used as the basis for a viewpoint. The resulting view will look down the -Z axis of the object.

To Set a Viewpoint From an Object:

1. From the menu, select **View>View Control>Set Viewpoint From Object**.
2. At the prompt, select the object to use for establishing the viewpoint.

Set Viewpoint From Surface Point

You can also set the viewpoint to look at a surface along its normal.

To Set a Viewpoint From a Surface Point:

1. From the menus, select **View>View Control>Set Viewpoint From Surface Point**.
2. At the prompt, click on the surface where you'd like to look.

View Clipping

Circumstances might arise in which you want to clip portions of the graphical view. Two major examples include:

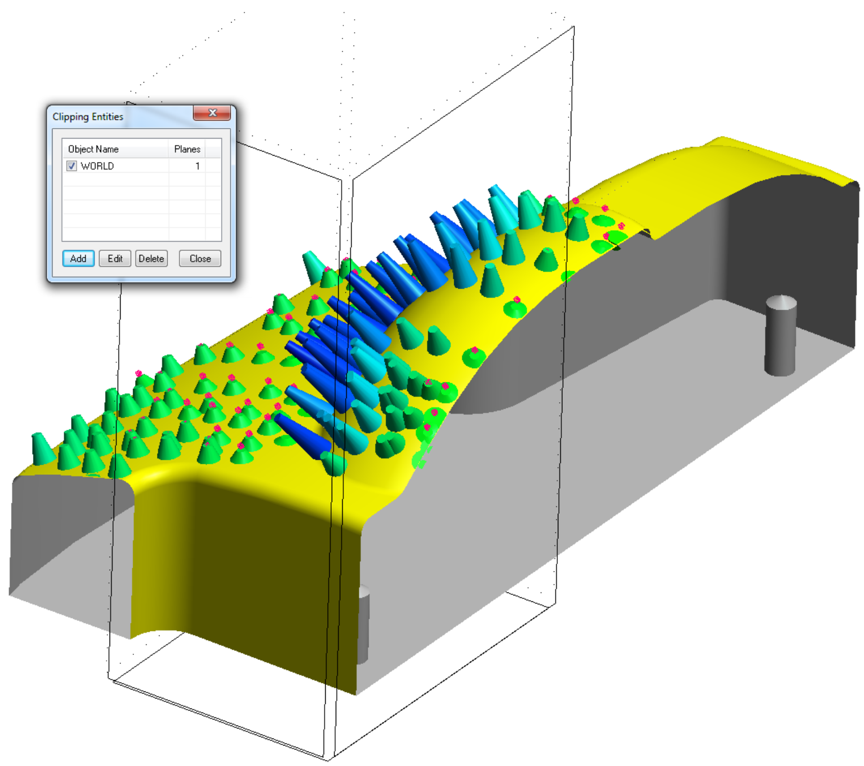


Figure 6-2. Viewing a model, measurements, and a vector group with an active clipping region.

- Removing data (such as point cloud data) outside of a region in order to simplify the view. This may be done for reporting reasons (to obtain a clear screenshot for a report) or just to be able to more easily work in the view. For example, if you have point cloud scans of the inside and outside of a building, you might clip away the near "wall" so that you can view the data on the inside of the building without also seeing the wall data.

- Displaying a "section cut". You might do this to "cut away" portions of your CAD or data in order to display only a region of interest, as in Figure 6-2.

A clipping region is defined by a box that can be moved or have any of its sides scaled along a coordinate direction. The orientation of the region (its rotation in space) can be defined by any object, and each of the six faces of the box can have clipping enabled or disabled.

To Set A Clipping Region:

1. From the menus, select **View>View Control>Clipping Planes** or press **Alt-X**.
2. In the Clipping Entities dialog, click the Add button.
3. Select any object to define the initial position and orientation of the clipping box.
4. Mouse over the center of a face (it will turn black) and click it to toggle that face's clipping plane.
5. Mouse over the center of any edge (an icon will appear) and drag to move the clipping region along that edge's direction.
6. Mouse over the end of any edge (an icon will appear) and drag to grow or shrink the region along that edge's direction.
7. When you're happy with your region, click the Close button, or leave the dialog open if desired.

Note: Clipping regions are tied to the object used to define the position and orientation of the clipping cube. If the object moves or rotates, the clipping planes will move or rotate along with it.

To Assign Numeric Boundaries to a Clipping Region:

1. Create the clipping region as described above in "To Set A Clipping Region:" on page 162.
2. In the ***Clipping Entities*** dialog, double-click on a clipping region or select it and click the Edit button.
3. In the resulting ***Clipping Volume Properties*** dialog, check the regions you'd like to be active, and enter desired values for each clipping plane.

To Delete a Clipping Region:

- In the ***Clipping Entities*** dialog, select the clipping region to delete, then click the Delete button.

You can define as many clipping regions as you'd like. Any of them can be activated or deactivated.

To Activate or Deactivate a Clipping Region:

- In the ***Clipping Entities*** dialog, check (to activate) or uncheck (to deactivate) the clipping region's checkbox.

Multiple Graphical Views

Multiple graphical views (see Figure 6-3) can be displayed by selecting **View>New Graphical View Window**. This allows you to simultaneously view and interact with your job from multiple perspectives.

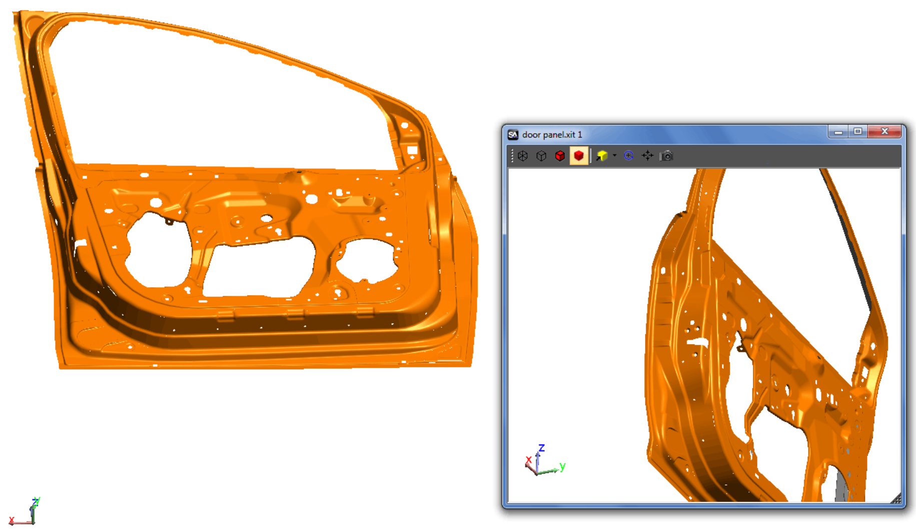


Figure 6-3. An auxiliary view has been opened to view and interact with this door panel from a different perspective.

Auxiliary graphical views can be rotated, positioned, and rendered independent of the primary graphical view, and they also support selection. Hold the **Ctrl** key while changing the render mode, autoscaling, or creating a snapshot of the screen to have the setting apply across all open graphical view windows.

You can add as many auxiliary views as you'd like, although more than two auxiliary views begins to become unmanageable without multiple computer displays.

Heads-Up Display

The graphical view has a Heads-Up Display (HUD) that displays a variety of measurement information (see Figure 6-4), including:

- Feature Name (if trapping measurements to a feature)
- Current Point Number (if trapping to a feature with a prescribed number of measurements)
- Point coordinates
- Point name (when not trapping)
- RMS error (if the RMS error of a sampled measurement exceeds

the warning threshold)

You can control whether the HUD is displayed, the font, color, translucency, and dwell time for the display all from the user options.

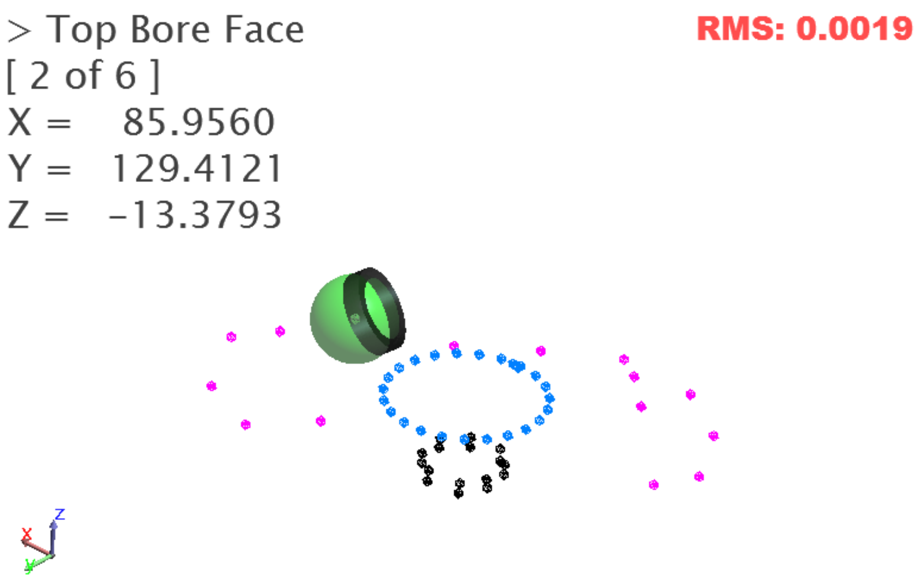


Figure 6-4. The Heads-Up Display.

To Modify Settings for the Heads-Up Display:

1. From the SA Toolbar, click the User Options icon.
2. In the *Display* tab, click the Heads Up Display Settings button (Figure 6-5).

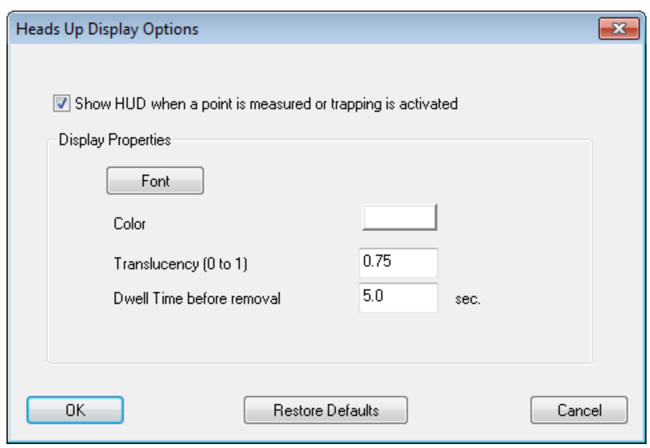


Figure 6-5. The ***Heads Up Display Options*** dialog.

To Enabled/Disable the Heads-Up Display:

- In the HUD options dialog, enable or disable the *Show HUD when a point is measured or trapping is activated* checkbox.

To Change the Font used by the Heads-Up Display:

- In the HUD options dialog, click the Font button, then choose a new font from the dialog.

To Change the Color of HUD Text:

- In the HUD options dialog, click the color swatch and choose a new color.

To Change the Translucency of the Heads-Up Display:

- In the HUD options dialog, set the *Translucency (0 to 1)* field to a value between 0 (fully transparent) and one (fully opaque).

To Change How Long the HUD Remains On-Screen:

- In the HUD options dialog, set the *Dwell Time before removal* option (in seconds).

Saved Viewpoints

You can save custom viewpoints in order to quickly return to a preset view orientation.

To Save a Custom Viewpoint:

1. Orient the view to the desired position.
2. Click the Views icon in the Main Toolbar, or select **View>View Control>Set Viewpoint**.
3. In the ***View List*** dialog, click Add Current, as depicted in Figure 6-6.

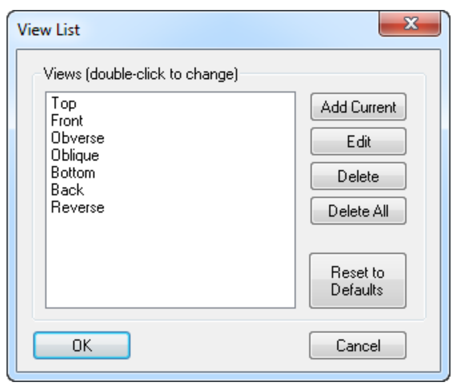


Figure 6-6. The View List dialog.

4. In the ***Preset View*** dialog, type a name for your custom view (Figure 6-7).

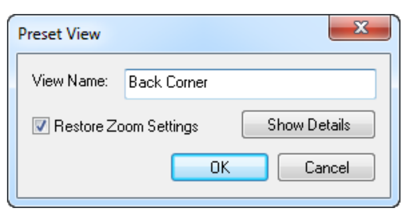


Figure 6-7. Entering the name for a custom view.

5. If you would like the view to restore the current zoom settings, select *Restore Zoom Settings*.
6. To view position and orientation information about the view, click the Show Details button.
7. Click OK. Your custom view will now be available in the ***View***

List:

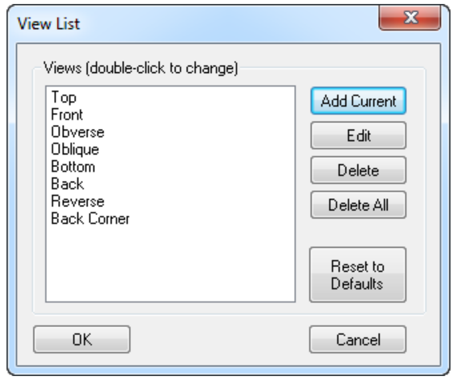


Figure 6-8. The custom view has been added to the view list.

To Restore a Custom Viewpoint:

- Click the down-arrow next to the Views icon and pick the view from the list, or
- In the ***View List*** dialog, double-click the desired view or select it and choose OK.

To Edit a Custom Viewpoint:

- In the ***View List*** dialog, select the view and choose Edit.

To Delete a Custom Viewpoint:

- In the ***View List*** dialog, select the view to delete and click Delete.

To Delete All Viewpoints:

- In the ***View List*** dialog, click Delete All.

To Restore the Default Viewpoints:

In the ***View List*** dialog, click Reset to Defaults.

By default, saved view directions are based on the orientation of the world coordinate frame. For example, if a view is saved looking along the world frame's Z-axis, then when the view is restored, it will again be looking along the Z-axis of the world frame, regardless of the active frame. However, you can choose to restore saved view directions relative to the working frame instead. By doing so, restoring the view described above would look along the Z-axis of the active frame.

To Restore a Viewpoint Based on the Working Frame:

- In the *User Options>Display* tab, check the ***View Relative to Working Frame*** option.

Rendering Modes

SA provides four rendering modes for viewing geometry: Wireframe, Hidden Line Removed, Solid+Edges, and Solid. The rendering modes are selected from the Main Toolbar.

- **Wireframe.** Geometry is displayed as edges only, and hidden edges are visible, as depicted in Figure 6-9.

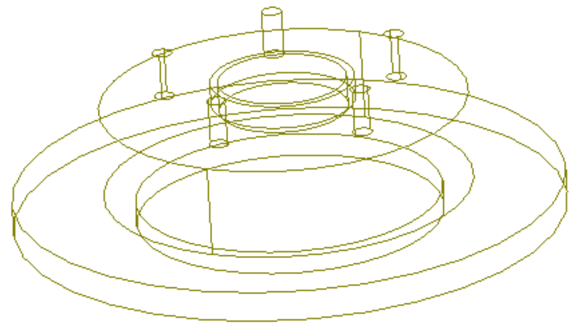


Figure 6-9. Wireframe rendering mode.

- **Hidden Line Removed.** Geometry is displayed as edges only, but hidden edges are not displayed, as depicted in Figure 6-10.

> **Note:** By default, silhouette edges are not rendered in Hidden Line Removed mode. To draw silhouettes, in the *User Options>Display* tab, check *Draw Silhouettes in Hidden Line Removed Mode*.

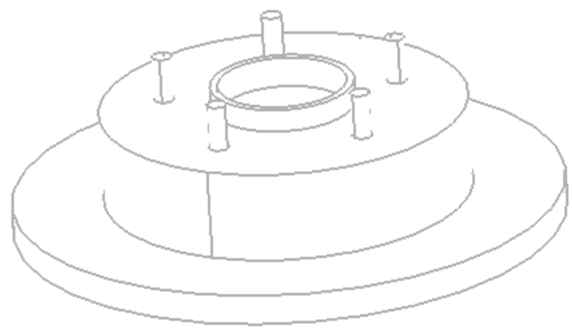


Figure 6-10. Hidden Line Removed rendering mode.

Here is the same view with silhouettes turned on:

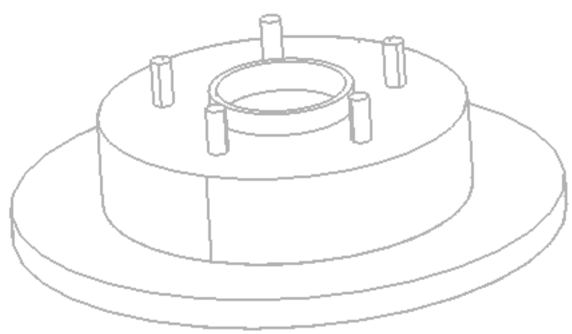


Figure 6-11. Silhouettes have been drawn in this hidden line view.

> **Note:** Because of the necessary computation time, each time the view is rotated, you need to click the Hidden Line Removed icon to redraw silhouettes if you want to see them. *The Draw Silhouettes in Hidden Line Removed Mode* setting is saved with the SA file.

- **Solid+Edges.** Solid surfaces are rendered in addition to edges, as shown in Figure 6-12.

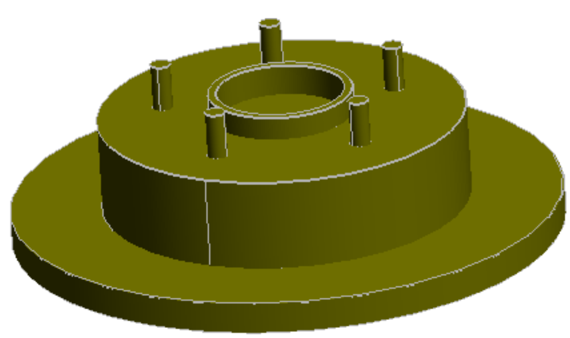


Figure 6-12. Solid+Edges rendering mode.

> **Note:** When a file is first opened, the rendering method always defaults to wireframe mode. This prevents a potentially lengthy rendering process from starting immediately after a file is loaded.

- **Solid.** Solid surfaces are rendered without edges, as depicted in Figure 6-13.

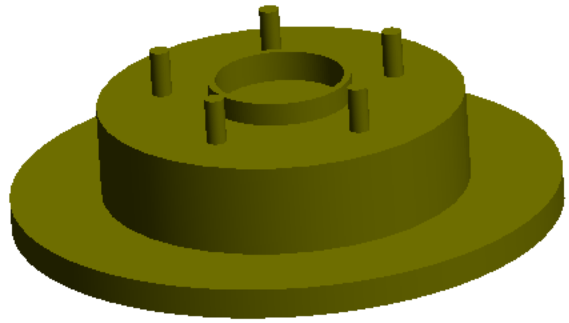


Figure 6-13. Solid rendered mode.

Surface Backsides

Every surface in SA (in addition to other entities, such as planes) has a positive and a negative side (see Figure 6-14). In order to properly account for a probe's target offsets, it is important that all comparisons be performed against the positive side of a surface. In order for you to determine the positive side, SA renders the negative side of surfaces in a special, user-definable color (a light gray by default).

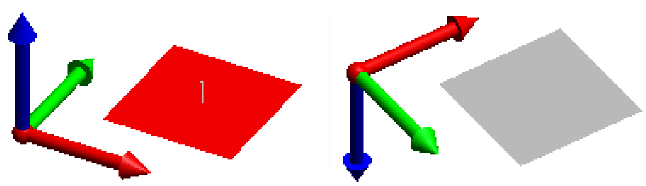


Figure 6-14. The same surface as viewed from the front (left) and back (right) with *Highlight Surface Back-sides* enabled.

Although the default gray is usually sufficient, you can choose a custom color to denote surface backsides. It is best to choose an unusual color which is not one of the typical working colors.

If desired, you can disable surface backside highlighting so that both the front and back side of surfaces appear in the same color.

To Disable Surface Back-Side Highlighting:

Note: The *Highlight Surface Back-sides* setting is saved as a machine setting.

- In the *User Options>Display* tab, uncheck the *Highlight Surface Back-sides* option.

To Change the Color of Surface Back-sides:

1. Ensure the *Highlight Surface Back-sides* option is selected.
2. Click the Color button, then choose a new color.

Background/Highlight Color

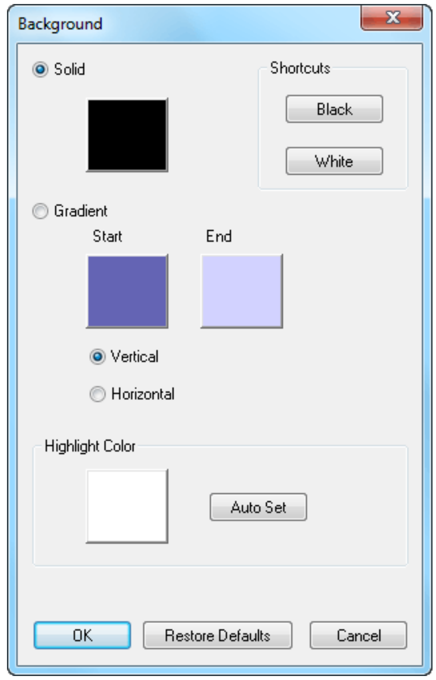


Figure 6-15. The Background dialog.

The background of the graphical view can be set to a solid color or a gradient, and you can also adjust the highlight color used in SA.

To Adjust Background Color Settings:

- Click the Background icon in the Main Toolbar.

To Change the Background Color to a Solid Color:

- Click the *Solid* radio button, then click the color swatch to change the color.
- Click the Black button to change the background color to black.
- Click the White button to change the background color to white.

To Change the Background Color to a Gradient:

Tip: For fastest rendering, set your background to a solid color. Solid backgrounds redraw faster than gradient backgrounds.

- Click the *Gradient* radio button, then choose a start and end color. Specify whether you'd like a horizontal or vertical gradient.

The highlight color is used when an object is selected or when it is explicitly highlighted using the **Highlight** command.

To Change the Highlight Color:

- Under the *Highlight Color* section, click the color swatch and choose a new color, or
- Click the Auto Set button, which will automatically select a highlight color that has high contrast when compared to the current background color.

To Restore the Factory Default Background/Highlight Colors:

- Click the Restore Defaults button.

Hardware Acceleration

If your graphics card supports it, SpatialAnalyzer provides OpenGL™ hardware acceleration for the graphical view. This option is enabled by default, and provides a significant performance increase—particularly with complex models and large amounts of geometry.

To Enable or Disable Hardware Acceleration:

1. Open the User Options dialog by clicking the icon.
2. On the *Display* tab, click the Performance Options button.
3. Check or uncheck the *Use Hardware Accelerated Rendering if Available* option (Figure 6-16).

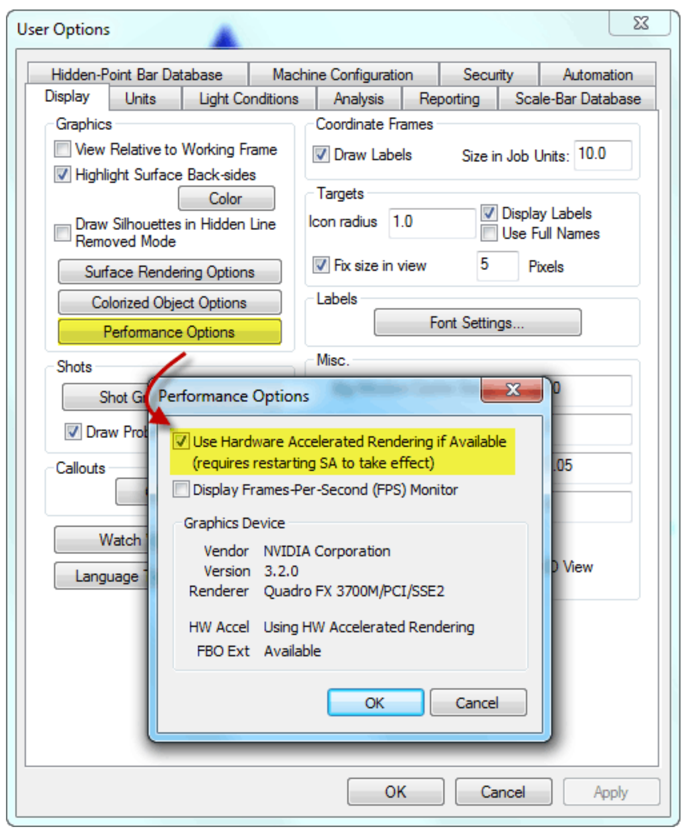


Figure 6-16. Enabling hardware acceleration for faster graphics performance.

> **Note:** Some graphics cards may cause visual artifacts with hardware-accelerated rendering enabled. If you see these artifacts, first ensure that you have the latest drivers for your graphics card. If the artifacts persist, disable hardware acceleration.

After changing the hardware acceleration option, it is necessary to exit SA and restart for the new setting to take effect.

SA has a built-in Frames-per-second (FPS) monitor that you can use to determine whether hardware acceleration is improving rendering performance in your job (see Figure 6-17). The FPS monitor displays a running average of the rendered frame rate. The progress bar under the frame rate shows how "full" the measurement buffer has become.

For the most accurate value, continue rotating the view until this buffer is nearly full.

To Test the Effectiveness of Hardware Acceleration on Your Machine:

Note: You will not see a framerate that exceeds the refresh rate of your display--typically around 60 fps.

1. Open the User Options dialog by clicking the ⚙ icon.
2. On the *Display* tab, click the Performance Options button.
3. Select the *Display Frames-Per-Second (FPS) Monitor* option.

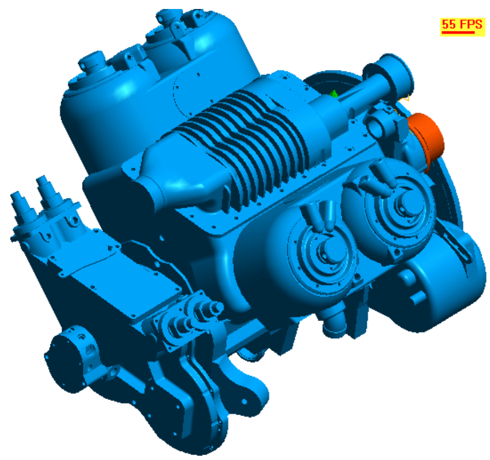


Figure 6-17. The Frames-per-second (FPS) monitor indicates how quickly the graphical view is being redrawn.

Surface Rendering Options

Note: Surface rendering options only affect the depiction of the CAD in the graphical view. In calculations, the exact surface is always used.

CAD surfaces (sometimes referred to as B-Reps or NURBS—Non-rational Uniform B-Splines) are mathematically defined and therefore have an “exact” representation. However, curved surfaces must be converted into a series of approximated flat faces in order to be rendered in the graphical view. The quality of this approximation is determined by the Surface Rendering Options (see Figure 6-18).

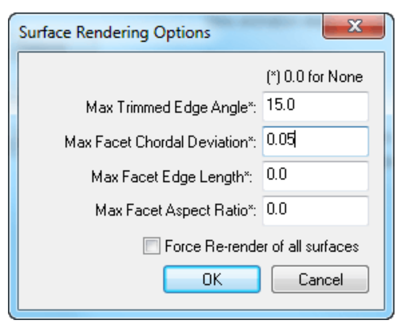


Figure 6-18. The Surface Rendering Options dialog.

If you find that your curves and surfaces are not being represented accurately enough for your liking, or if you see “tearing” in your CAD surfaces, you can modify the surface rendering options.

To Change the Surface Rendering Options:

- In the User Options>Display tab, click the Surface Rendering Options button. Choose the desired settings, then click OK.

The adjustable options that affect surface rendering are as follows:

- **Max Trimmed Edge Angle.** Specifies the maximum length of edge segments for trimmed edges. A lower value gives a higher quality representation. Use a value of 0 to default to a low-quality setting.

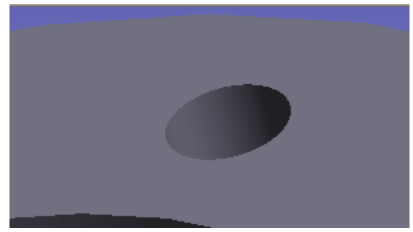



Figure 6-19. Max trimmed edge angle: 65 (left) and 15 (right).

Note that this setting also applies to spline curves:


Figure 6-20. Max trimmed edge angle: 125 (left) and 5 (right).

- **Max Facet Chordal Deviation.** Specifies the maximum allowable distance between a rendered edge and the surface with which it is associated. Use a value of 0 to default to a low-quality setting.

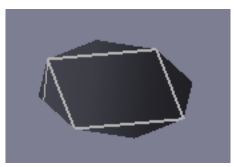
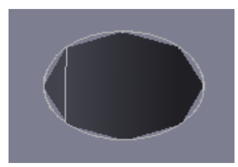



Figure 6-21. Max facet chordal deviation of 0.000 (left) and 0.001 (right).

When laying out points on an object's vertices, this setting affects how many points are constructed.

- **Max Facet Edge Length.** Specifies the maximum allowable length of each edge segment.
- **Max Facet Aspect Ratio.** Specifies the maximum aspect ratio (length vs. width) of each rendered facet.
- **Force Re-render of all surfaces.** Checking this option will force SA to recalculate surfaces using the latest settings in the window. If unchecked, and surfaces have already been rendered, then the settings will not take effect until the file is reopened.

Labels

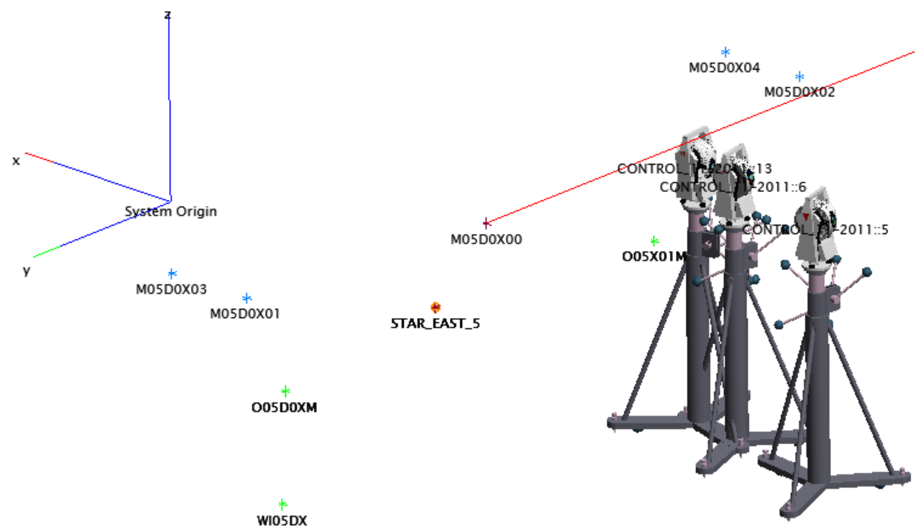


Figure 6-22. Point and object labels are visible in this view.

Labels can be turned on in the graphical view for displaying names of points, objects, and instruments (see Figure 6-22). This is often useful for visually distinguishing different objects, but can become overwhelming in complicated files—so you can toggle label visibility.

You can specify the font with which to display labels, as well as the text size.

To Change the Font Used for Labels:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, click the Font Settings... button in the *Labels* box.
3. Select the desired font, style, size, and color (the other options will not apply).

Labels can be controlled individually for points (which also displays instrument labels) or objects.

To Toggle the Display of Labels:

> **Tip:** Displaying a large number of labels simultaneously in the graphical view can significantly slow down display performance. Hide items not in use or turn labels off to speed up display speed.

- **Points and Instruments.** Toggle **View>Show Point Labels** or press `Alt+L` (default keyboard shortcut).
- **Objects.** Toggle **View>Show Object Labels**.

Point labels can display just the target name (such as **`STAR_EAST_5`** in Figure 6-22) or can display the entire collection, group, and target name (such as **`Controls::Network::STAR_EAST_5`**).

To Toggle the Display of Full Point Names:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, check or uncheck the *Use Full Names* check-

box.

To Display Instrument Labels Without Point Labels:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, uncheck the *Display Labels* checkbox.
3. Ensure that **View>Show Point Labels** is turned on.

Note: Hiding obstructed labels is computationally intensive and can slow down the graphical view considerably. For a more efficient method, simply hide the points.

When point labels are enabled, by default they are displayed even when the corresponding point is hidden behind a surface. To avoid confusion, you can set the graphical view to hide labels for points that are not visible from the current point of view.

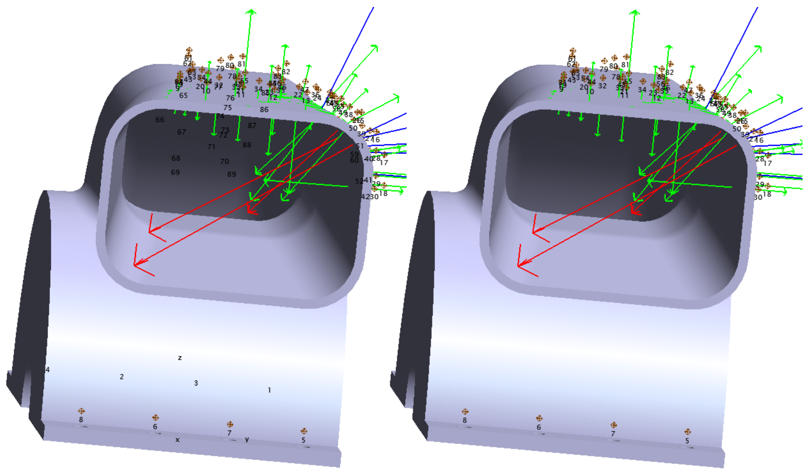


Figure 6-23. Hide obstructed labels disabled (left) and enabled (right).

To Hide Labels for Hidden (Obstructed) Points:

- Select **View>Hide Obstructed Labels**.

When point labels are displayed, by default SA will label the axes of coordinate frames as X, Y, and Z. This can be disabled.

To Disable Labeling of Frame Axes:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, uncheck the *Draw Labels* checkbox in the *Coordinate Frames* box.

The Hover Window

If you are looking at an item in the graphical view, finding that item's name—or where it is located in the tree—can be a difficult task, particularly if the file is complicated and has lots of objects. The ***Hover*** window is a floating dialog, so it can be left up and "floating" over the graphical view. When you hover over an item in the graphical view, the Hover window will indicate the collection, name, type, and location of the item in the tree, the coordinates of a point (in Cartesian, Cylindrical, or Spherical coordinates, the deltas for a vector, or the number of measurements associated with an instrument.

To Toggle Display of the Hover Window:

Do one of the following:

- From the menu, select **View>Hover (nearest point)**, or
- Press **Ctrl+H** (the default keyboard shortcut), or
- From the Toolkit's *Analysis* tab, click the Hover Window 

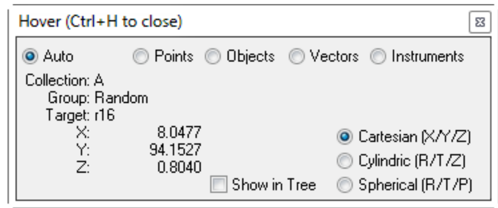


Figure 6-24. The ***Hover*** window showing the name and coordinates of a point.

The ***Hover*** window has a filter that you can apply to indicate what can be "hovered".

- **Auto.** No filter is applied—SA will provide information for points, objects, vectors, or instruments—whatever you roll over.
- **Points.** Information will only be displayed when the cursor is rolled over a point.
- **Objects.** Information will only be displayed when the cursor is rolled over an object.
- **Vectors.** Information will only be displayed when the cursor is rolled over a vector.
- **Instruments.** Information will only be displayed when the cursor is rolled over an instrument.

To Set the Hover Window Filter:

- With the ***Hover*** window displayed, choose either the *Auto*, *Points*, *Objects*, *Vectors*, or *Instruments* option.

To Find an Item in the Tree:

1. Activate the ***Hover*** window and ensure that the *Show In Tree* option is selected.
2. Hover the mouse cursor of the item of interest. The tree will be expanded and auto-scrolled to reveal the item, and it will be selected.
3. To keep the selection active and the tree expanded, turn off the hover window while hovering over an item by pressing **Ctrl+H**.

To Change the Coordinate System for Viewing Point Coordinates:

- In the ***Hover*** window, choose the *Cartesian*, *Cylindric*, or *Spherical* option.

Object-Specific Rendering Modes

It is often useful to make a specific object or instrument in the graphical view partially transparent (translucent) or to render it differently than other objects. For example, if a measured part has an area of insufficient material, its corresponding vector group will point into the CAD model. As a result, this vector will be nearly invisible when the CAD is rendered in Solid, Solid+Edges, or Hidden Line Removed mode. As another example, depending on viewing angle, sometimes the instrument's graphical representation, a surface, or the active coordinate frame will obstruct the work area that you're most interested in (see Figure 6-25). A third case might involve a surface that you want to see through. While there is always the option of hiding the obstructing items, often times it is desirable to keep them visible. In such circumstances, translucency allows you to avoid hiding the objects, but still be able to see through them to the work area.

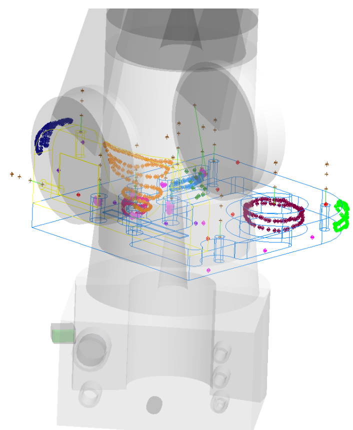


Figure 6-25. Viewing the work area through a translucent instrument.

To Toggle Instrument Translucency:

- Select **View>Translucent Instruments**.

To Toggle Frame Translucency:

- Select **View>Translucent Working Frame**.

Surfaces and primitive objects can be set to a specific level of translucency or can be independently set to render in solid or wireframe mode.

To Set the Rendering Mode of an Individual Surface or Primitive Object:

1. Right-click the object in the tree or graphical view.
2. Select **Translucency**.
3. Select a desired rendering mode. If *Translucent* is selected, choose an opacity level with the slider.

To Set the Rendering Mode of Multiple Surfaces or Primitive Objects:

1. Select **View>Object Visibility>Translucency>Individual Objects** or **>All**

Objects as desired.

2. If **>Individual Objects** was selected, select the objects of interest and press **Enter**.
3. Select a desired rendering mode. If *Translucent* is selected, choose an opacity level with the slider.

Visibility

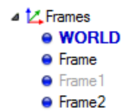


Figure 6-26. The frame named **Frame1** in this collection is hidden.

All of the items listed above can be shown or hidden. Hiding an item will remove it from the graphical view, and (unless it is a folder or collection) will dim the item's name in the tree to gray (see Figure 6-27).

Hiding items not in use has a number of benefits, chief of which are:

- The graphical view becomes less cluttered, and it's easier to work with the items of interest.
- If many items are being displayed at once, hiding those not in use will speed up the graphics considerably.
- Graphical selection becomes easier, because there are less items to pick incorrectly.

To Toggle An Individual Item's Visibility:

- Right-click the item in the tree or graphical view and toggle the **Show** option.

To Hide or Show All Items In a Container or Category:

- Right-click the container (collection or folder) or category in the tree and select **Hide All** or **Show All**.

To Hide or Show Individual Points:

- Right-click the point in the tree or graphical view and toggle the **Show** setting to toggle visibility of individual points, or

Select **View>Show Points** or **>Hide Points**, then select the points to show or hide.

To Hide or Show Multiple Objects:

- Select **View>Object Visibility>Hide Objects** or **>Show Objects**.

To Show All Objects:

- Select **View>Object Visibility>Display All**.

You can also filter the graphical view to display items only from a specific source. This source can be either SA itself, or an imported CAD file. This allows you to immediately hide objects imported from one or more CAD files, or to immediately hide native SA objects.

To Filter The View By CAD Source:

1. Select **View>Object Visibility>Filter By CAD Source**.
2. In the ***Show Selected Sources*** dialog, select the sources you'd like to show, then click OK.
3. The filter will apply until you change it again.

Miscellaneous Settings & Commands

View Animation Steps

Sometimes, switching from one saved viewpoint to another can be disorienting. If you find that switching from one saved view to another is too jarring, you can increase the number of animation steps as the view transitions from one saved viewpoint to another. This has the effect of smoothing out (and slowing down) the transition.

To Change the Number of View Animation Steps:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, change the *View animation steps* value. The value is the number of frames used to animate from one view to the other.

Arrowhead Sizes

You can change the size of arrowheads on vectors with the Line arrow size factor setting (see Figure 6-27).

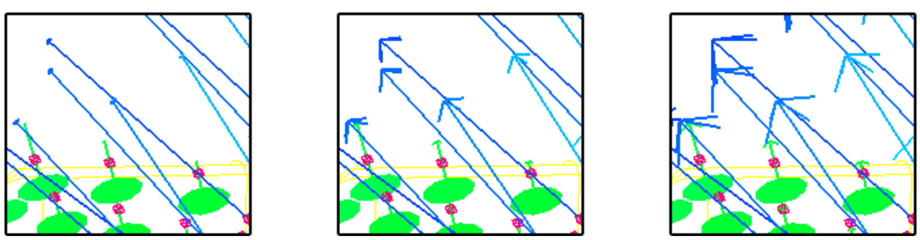


Figure 6-27. Line arrow size factors of 0.01 (left), 0.05 (middle), and 0.1 (right).

To Change the Size of Arrowheads on Vectors:

Note: Vector arrowheads also get larger as the vector gets longer.

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, change the *Line arrow size factor* setting to a different value. A small change goes a long way!

Scale Indicator

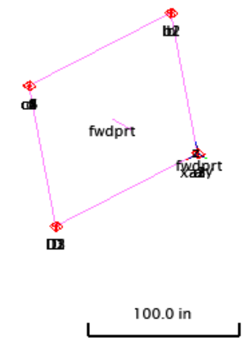


Figure 6-28. The scale indicator. These points are approximately 100 inches apart.

To get a better grasp on the scale of the current zoom setting, you can display a scale indicator—somewhat like a distance legend on a map—that indicates the size of distances in the graphical view (Figure 6-28). The scale indicator automatically updates and adjusts itself as you zoom in and out in the view.

To Display the Scale Indicator:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, check the *Show Scale Indicator in View* checkbox.

Connection Indicator

If you have more than one instrument in your job file, it may begin to get confusing as to which instrument is currently connected to the instrument interface (that is, which instrument in the SA file is currently live). The connection indicator (Figure 6-29) is designed to highlight the connected instrument and prevent the confusion.

Figure 6-29. The connection indicator.

You can control whether the indicator is displayed, and if so, how large it is drawn.

To Adjust the Connection Indicator:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, enable or disable the *Connection Indicator* checkbox.
3. To change the size of the indicator, change the *Radius* value.

Uncertainty Fields

As discussed in "Measurement Uncertainty", uncertainty fields allow you to visualize the uncertainty of a measurement as a cloud of points (Figure 6-30). Uncertainty clouds provide a reference for determining relative uncertainty magnitude, shape, and distribution when comparing measurements.

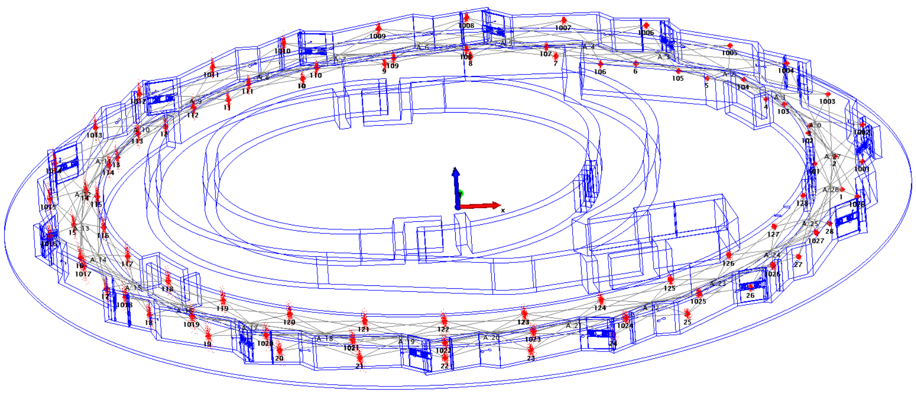


Figure 6-30. Viewing uncertainty fields for a set of measurements.

You can choose to display these uncertainty fields in the graphical view, or to hide them.

To Toggle Display of Uncertainty Fields:

- From the menu, select **View>Show Uncertainty Fields**.

Uncertainty fields are rendered in the graphical view at a certain magnification level so that they can be easily seen.

To Change the Size of Uncertainty Fields:

1. Click the User Options icon in the toolbar.
2. In the *Analysis* tab, change the *Cloud Magnification* value to the desired amount.

By default, uncertainty clouds are drawn as individual dots on the screen, each representing a sample of the overall uncertainty cloud. For ease of viewing, you may prefer to draw the cloud with lines instead of dots (Figure 6-31).

To Draw Uncertainty Clouds as Lines:

1. Click the User Options icon in the toolbar.
2. In the *Analysis* tab, enable the *Draw using lines* option.

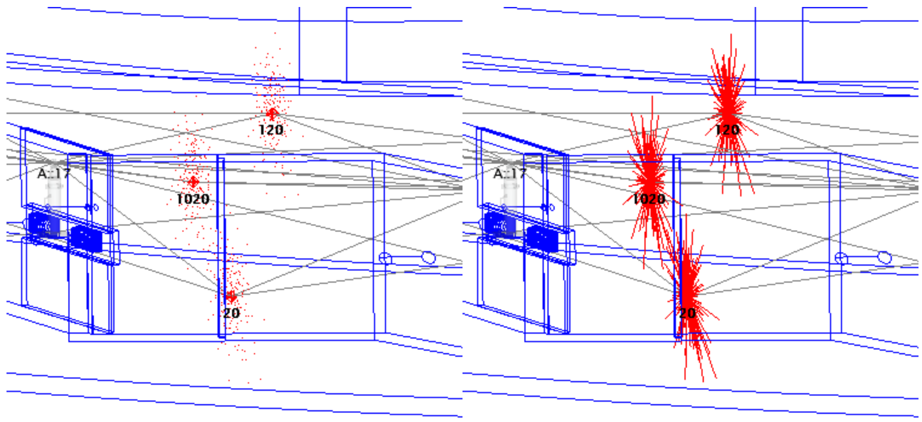


Figure 6-31. Uncertainty clouds drawn as dots (left) and lines (right).

Cloud Display Control

When working in scan data, it is common to be dealing with tens or hundreds of millions of cloud points. Rendering a large number of cloud points on-screen would lead to very sluggish graphical view performance.

Whenever you rotate/pan/zoom in the graphical view, any displayed point clouds are automatically thinned while the view adjustment operation is in-place. Once complete, the original point cloud is displayed.

You can further limit the number of cloud points rendered on-screen using the ***Cloud Display Control*** (Figure 6-32).

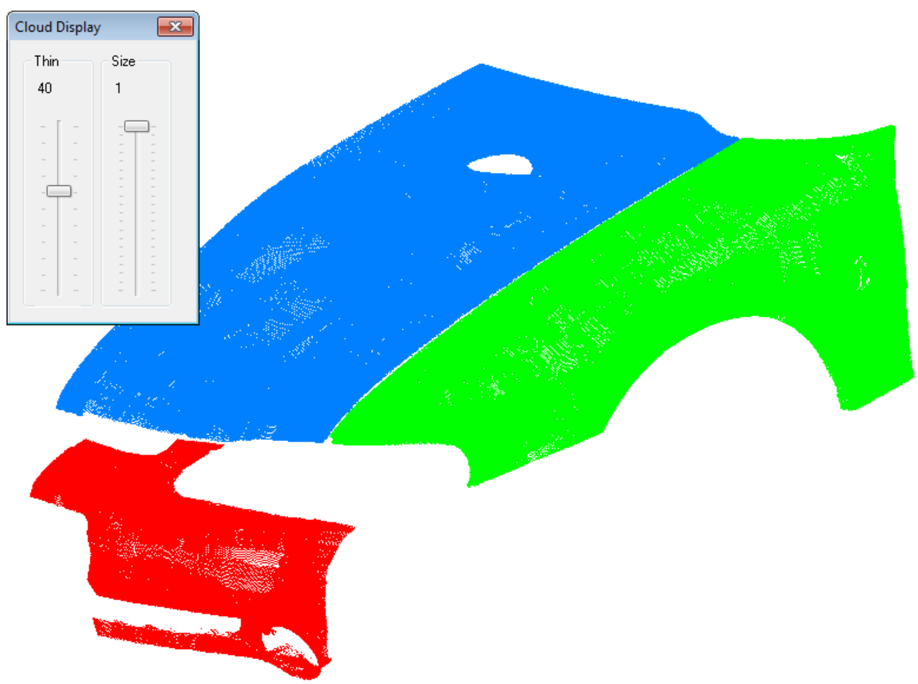


Figure 6-32. Viewing a cloud at 1:40 density, with each cloud point drawn as a 1-pixel dot.

To Display the Cloud Display Control:

- From the menu, select **View>Cloud Display Control**, or

- Press **Ctrl+T** (default keyboard shortcut).

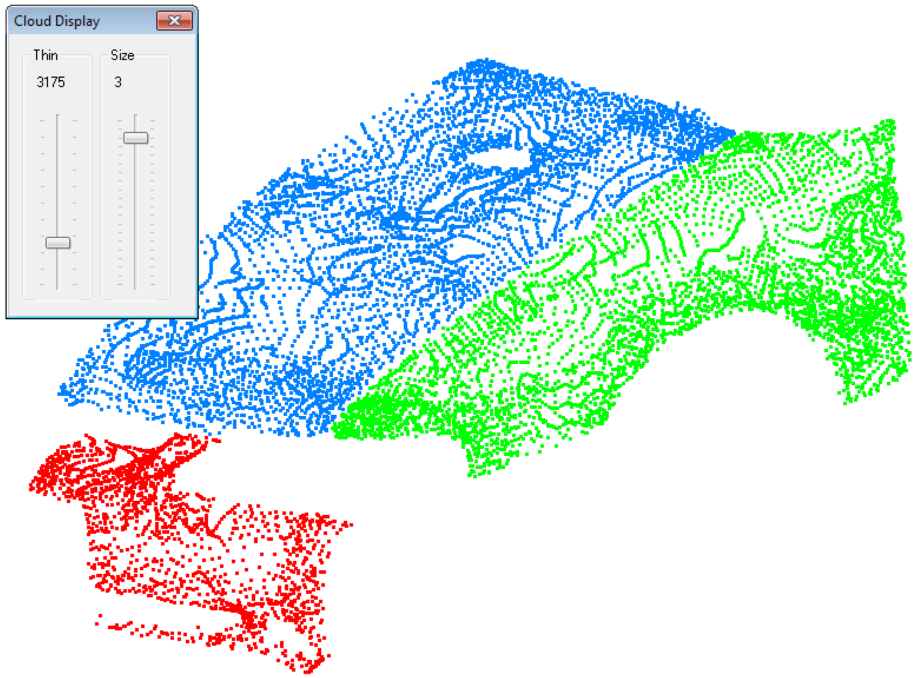


Figure 6-33. Viewing a cloud at 1:3175 density, with each cloud point drawn as a 3x3 pixel square.

Tip: When working with large numbers of cloud points, selecting graphically can be a very slow operation. Select a cloud using F2 selection wherever possible.

- **Thin.** The thinning factor for display. 1 indicates that all cloud points are displayed. 100 indicates that only every hundredth cloud point is displayed.
- **Size.** The size of the rendered cloud point, in pixels. 2 indicates the cloud is to be drawn as a 2x2 pixel square.

When cloud thinning is enabled and you graphically select cloud points, all cloud points will be selected—not just those that are rendered.

Highlighting

As a job file grows to hold more objects and measurements, it may become somewhat difficult to distinguish objects in the graphical view. While hiding items not of concern is probably the most common solution to this, often times it is faster to simply select an object by name in the tree and highlight it. Highlighting an item does two things: it changes the item in the graphical view to the current highlight color, and it also colors the item in the tree a bold blue. Doing this enables you to pinpoint the selected object while keeping all objects of interest visible. In addition to individually highlighting objects, you can also highlight points, collections, instruments, and folders. (Highlighting a collection or folder highlights everything inside that collection or folder).

There are also cases in which simply hiding items does not provide sufficient information. For instance, you can right-click a relationship and ask it to highlight its associated points or objects in order to identify which entities are associated with that relationship. This is information that cannot be revealed by simply hiding objects.

To Highlight a Point, Object, Instrument, Collection, or Folder:

- Right-click the item in the tree or graphical view and select **Highlight**.

To Highlight the Points or Objects Associated With a Relationship:

- Right-click the relationship in the tree and select **Highlight Entities**.

To Clear Highlights:

- Right-click the item(s) in the tree or graphical view and uncheck the **Highlight** setting, or
- Select **View>Clear all Highlights** to clear all highlighted items.

View Refresh

Rarely, you might occasionally find the need to manually refresh the view. If you are uncertain that the view updated after performing an action, you can refresh the view to ensure that you're looking at the latest rendering of the graphical view.

To Refresh the Graphical View:

- Select **View>Refresh** from the menu, or
- Press **F5**, the default keyboard shortcut.

Copying the Graphical View to the Clipboard

You can quickly and conveniently copy just the current graphical view to the clipboard for pasting in other applications (such as into an Excel or Word document).

To Copy the Graphical View to the Clipboard:

- From the menus, select **Edit>Copy graphics to clipboard**.

Creating a Movie Capture of the Screen

SA comes with a tool for capturing video of the screen (or certain windows) to a video file. This may be useful for sharing your work with others or communicating an important feature to a customer.

To Create a Movie Capture of the Screen:

1. From the menus, select **View>Create AVI Movie>New File**.
2. When prompted, specify a filename for the resulting video.

3. The ***Image Capture Options*** dialog appears (Figure 6-34).

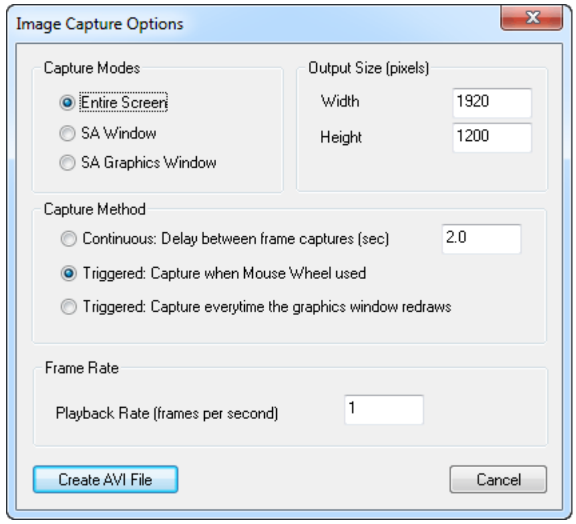


Figure 6-34. Options for an video capture.

4. Select the desired settings, then click Create AVI File.

Capture Modes

- **Entire Screen.** The entire screen will be captured.
- **SA Window.** Only the boundary of the SA window will be captured.
- **SA Graphics Window.** Only the graphical view will be captured.

Output Size (pixels)

Controls the resolution of the resulting video file.

Capture Method

- **Continuous.** Captures frames with a specified delay between frames.
- **Triggered: Capture when mouse wheel used.** Captures a frame only when the mouse wheel is used.
- **Triggered: Capture everytime the graphics window redraws.** Captures a new frame whenever the graphical view is redrawn.

Frame Rate

- **Playback Rate (frames per second).** Indicates the playback rate of the video—the number of video frames displayed per second.

5. If applicable, choose the appropriate video compression settings.

6. Start recording by selecting **View>Create AVI Movie>Start Recording** or press `Alt+1`.

To Pause or Resume Video Recording:

- To pause, select **View>Create AVI Movie>Stop Recording** or press **Alt+2**.
- To resume, select **View>Create AVI Movie>Start Recording** or press **Alt+1**.

To Finish Recording:

1. First stop recording using the **View>Create AVI Movie>Stop Recording** command or press **Alt+2**.
2. Select **View>Create AVI Movie>Close File**.

Controlling Target Icon Size

You can control the size of constructed and measured points (targets) as displayed in the graphical view. Targets can be drawn at a fixed size—which does not change regardless of zoom level—or a varied size, in which the points get larger as you zoom in. The former avoids clutter when multiple points are spaced closely together, whereas the latter gives a visual indication of scale on the current view.

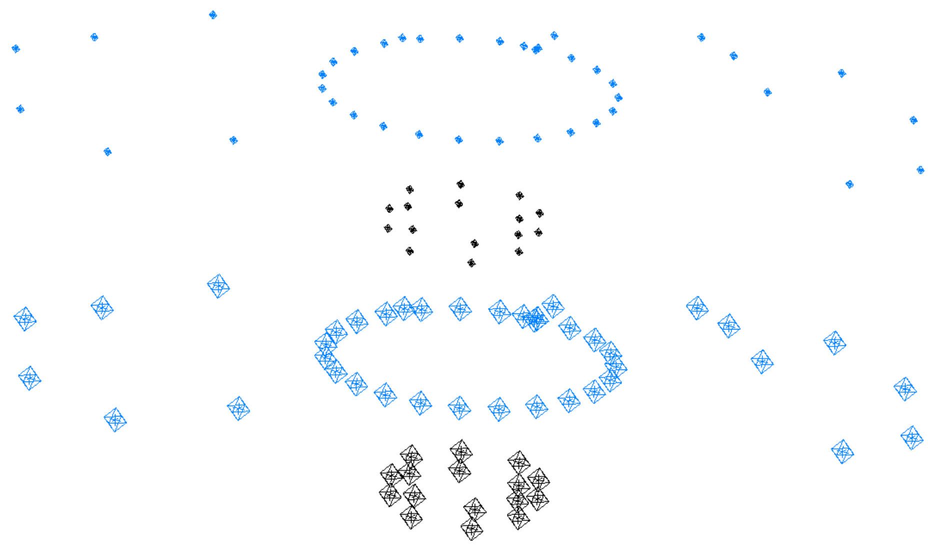


Figure 6-35. Targets displayed at 5 pixel size (top) and 15 pixel size (bottom).

To Use a Fixed Target Size:

1. Click the User Options icon in the toolbar.
2. In the *Targets* area of the *Display* tab, ensure that *Fix size in view* us checked, and enter the desired pixel size for the targets.

To Use a Variable Target Size:

1. Click the User Options icon in the toolbar.
2. In the *Targets* area of the *Display* tab, ensure that *Fix size in view* is unchecked. Set the *Icon radius* to the desired size, in job units.

This Page Intentionally Left Blank.

Working In SA

CHAPTER 7

This chapter will help you to learn the ins and outs of working in SA on a day-to-day basis, which will speed up your workflow and allow you to complete your work as quickly as possible.

Selection

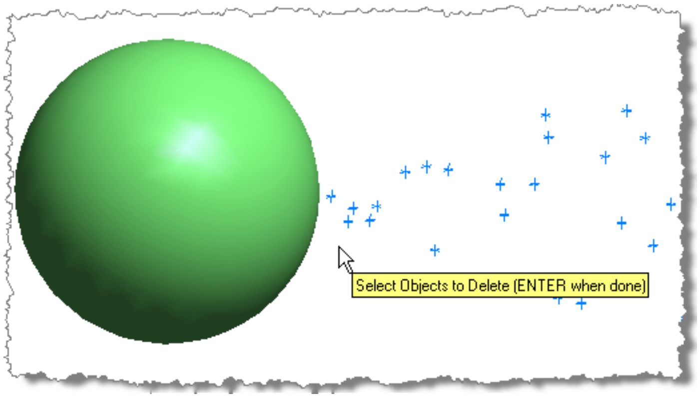


Figure 7-1. A selection prompt displayed in the graphical view.

SA generally follows the *Action-Object* selection paradigm when it comes to issuing menu commands. This implies that you first choose a command or operation, and then select the items to which the operation should be applied. Depending on the operation, multiple selection steps may be required. When an object is selected, it will change to the selected highlight color and its listing in the tree will become **bold blue**.

You can select entities visually in the graphical view. When an operation requires a user to make a selection, a selection prompt will appear as in Figure 7-1.

The entities that you will be prompted to select will depend on the selected command (one of the benefits of the Action-Object paradigm). SA will filter entity types for you, so that if you are prompted to select points, you will only be allowed to select points—not instruments or objects.

To Abort a Command and Exit the Selection Mode:

- Press the **Esc** key. The selection prompt will disappear and the command will be cancelled.

In many commands, the prompt permits you to select more than one item—usually as many items as desired. You can continue to select items using the selection methods below until satisfied, at which time you must indicate to the prompt that you are finished selecting items.

To Complete Selection:

- Press the Enter key.

There are several ways to make selections in SA:

- **Graphically.** Pick items directly in the graphical view.
- **Treebar Selection.** Pick items using the tree.
- **F2 Selection.** Pick items from a filtered list.
- **Wildcard Selection.** Pick items based on naming criteria using wildcards.
- **By Color.** Pick items based on their color.
- **By Type.** Pick items based on their type.

Each of these selection methods is discussed in more detail in the following sections.

Graphical Selection

One way to graphically select items is to simply click them in the graphical view.

To Graphically Select Items One at a Time:

- With a prompt displayed, single-click the item in the graphical view. The item's selection state will be toggled—once selected, clicking an item again will remove it from the selection.

You can also graphically select multiple items at one time. To do this, SA provides graphical selection modes.

Graphical Selection Modes

There are two graphical selection modes in SA: rectangle and polygon.

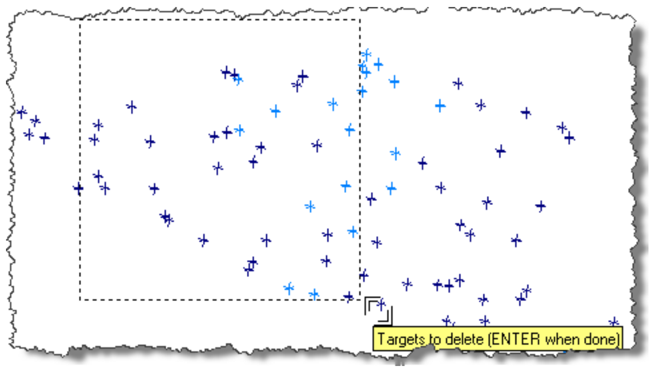


Figure 7-2. Selecting a set of points using a rectangular selection marquee.

Tip: If you've started to define a selection in the wrong place, drag the mouse back over the starting corner of the rectangle. When the cursor changes back to an arrow, release the mouse button. You will remain in the selection mode but nothing will be selected.

Rectangle mode allows you to draw a rectangle around the entities of interest, as illustrated in Figure 7-2. Everything inside the rectangle is either selected (or deselected), whereas the selection states of items outside of the rectangle remain unchanged.

To Select Multiple Items Using Rectangular Selection:

Note: If you release the Shift key prior to releasing the mouse button, you will define an autoscale region, not an entity selection.

Note: To remove selected items from the selection, hold down both the **Shift** and **Alt** keys.

1. Ensure you are in rectangle graphical selection mode (see "Graphical Selection Options" on page 190).
2. While pressing the **Shift** key, click and hold the mouse button to define one corner of the rectangle.
3. Drag to the opposite corner of the rectangle, then release the mouse button.

Polygon selection mode allows you to draw a polygon of any shape to define the selection, as depicted in Figure 7-3.

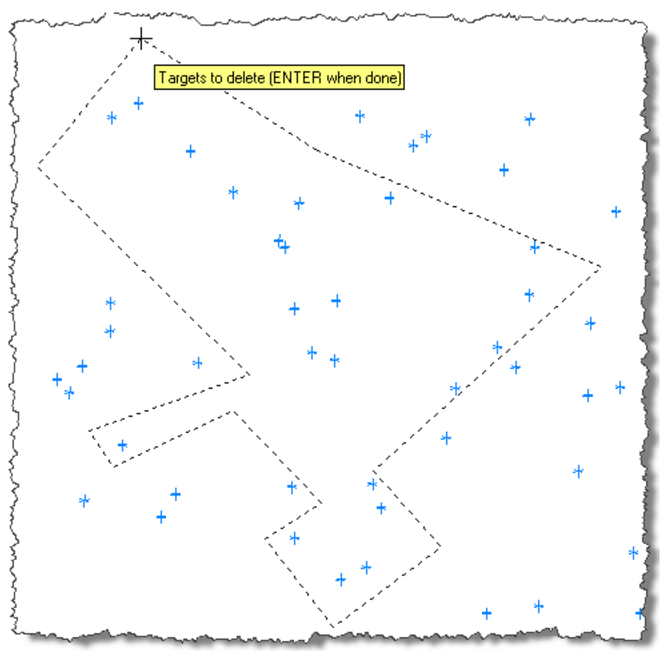


Figure 7-3. Points are being selected through the use of a polygon selection marquee.

Items inside the polygon are either selected (or deselected), whereas items outside of the polygon are not affected.

To Select Multiple Entities Using Polygon Selection:

> **Note:** To remove entities from the selection, hold down **Shift**, then hold down **Alt** and make your selection. When finished defining the polygon, release the **Shift** key, but keep the **Alt** key depressed.

1. Ensure you are in polygon graphical selection mode (see "Graphical Selection Options" on page 190).
2. While pressing the **Shift** key, single-click to define each vertex of the polygon.
3. For the last vertex, locate the mouse cursor where you'd like the vertex to be and release the **Shift** key.

To Change the Selection Mode During a Selection:

- Press the **F3** key.

Graphical Selection Options

> **Note:** The graphical selection mode options will persist until SA is closed.

The ***Graphical Selection Mode*** dialog (Figure 7-4) provides a number of options to allow you to control graphical selection. You can pick the selection mode as well as define whether entities inside, outside, or crossing the selection boundary should be selected.

To Open the Graphical Selection Mode Dialog:

- Choose **View>Graphical Selection Mode**, or
- Select the Graphical Selection Mode icon from the Main Toolbar.

To Change Graphical Selection Modes:

- Choose the *Rectangle* or *Polygon* radio button.

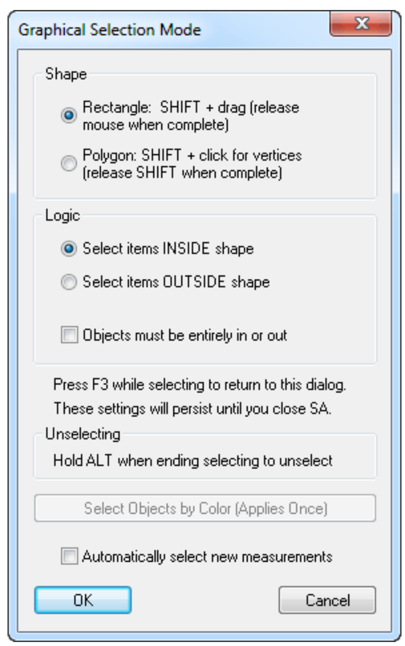


Figure 7-4. The Graphical Selection Mode dialog allows you to define the behavior of the selection marquee.

To Select Whether Entities Inside or Outside the Selection Region Should Be Selected:

- Choose the *Select items INSIDE shape* or *Select items OUTSIDE shape* radio button.

To Define Whether Entities Crossing a Selection Region are Selected:

- Check or uncheck the *Objects must be entirely in or out* check-box.

Treebar Selection

Any item can also be selected in the tree. Selecting items in the tree provides an advantage over the graphical view in cases where there are many entities overlapping, when the item is not visible, does not have a representation in space, or when the geometry of the view makes it difficult to determine which items are actually selected.

To Select Items in the Tree:

- With a prompt displayed, double-click the item in the tree. Double-click the entity again to toggle its selection state.

F2 Selection

The F2 selection window (Figure 7-5) provides significant advantages for selection in cases where multiple items need to be selected. F2 selection filters out entity types that don't apply to the current selection and lists the remaining items in a hierarchical view similar to the treebar.

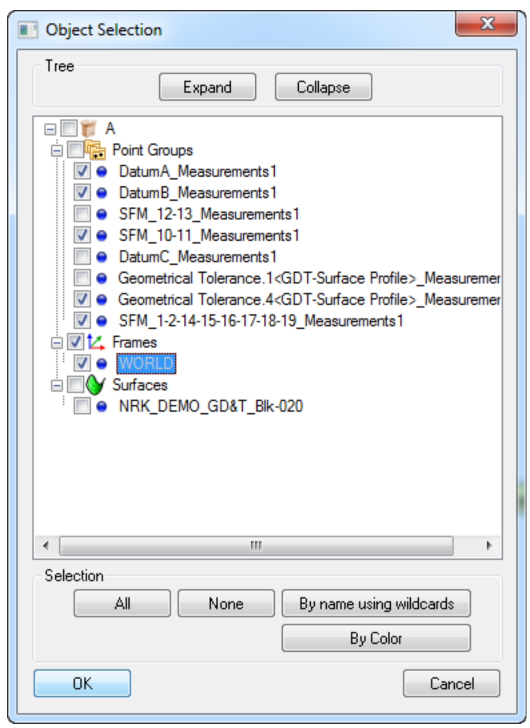


Figure 7-5. The F2 Selection dialog.

There are three variants of the F2 selection window: there is a window specific to point selection mode, object selection mode, and instrument selection mode. Each is slightly different and has options particular to the specific mode.

To Use the F2 Selection Window:

1. During selection (while the prompt is displayed), do one of the following:

- Press **F2**,
- Click the F2 - Select from a list icon in the toolbar, or
- From the menu, select **View>Point Selection Tree-View**.

2. Select the desired items, then click the OK button.
3. Continue selecting items using other methods (if desired), then confirm your selection by pressing the **Enter** key.

To Expand or Collapse the Entire Tree of Items (Point/Object Selection Modes):

- Click the Expand or Collapse button.

To Select All Items in a Folder or Collection:

- Check the box next to the folder or collection name.

To Select an Entire Category or an Entire Point Group (Point/Object Selection Mode):

- Check the box next to the category/point group (for example, Spheres, Point Groups, or Frames).

To Select All But a Few Items in a Category or Point Group (Object/Point Selection Modes):

- Check the box next to the category type, then uncheck individual items within that category.

To Select Contiguous Rows of Items (Object/Point Selection Modes):

1. Check the box next to the first item in the list.
2. Hold down **Shift** and check the box next to the last item in the list.

To Select All Items in the List:

- Click the All button.

To Deselect All Items in the List:

- Click the None button.

Wildcard Selection

Wildcard selection is a powerful way to select items by name from the F2 window. To use it, click the By name using wildcards button, which opens the dialog in Figure 7-6.

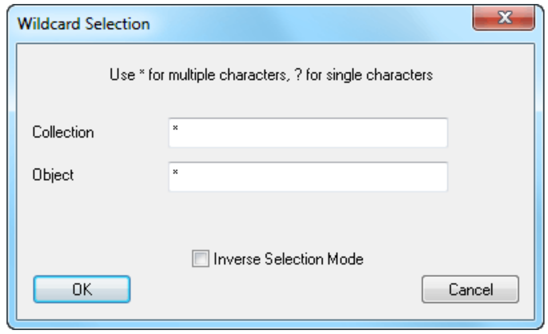


Figure 7-6. The Wildcard Selection dialog.

Warning: Wildcard selection automatically adds an asterisk (*) at the beginning and end of your search text. This may result in selection of more items than desired.

Note: The wildcard selection criteria, like all names in SpatialAnalyzer, are not case sensitive.

This dialog may appear slightly different depending on whether the current command allows you to select points or objects. In this dialog, asterisks (*****) represent a set of contiguous characters, and question marks (**?**) represent a single character.

This dialog allows you to enter selection criteria based on collection, object, and point names. In each criteria field, enter a search string. Any entities that match all of the entered criteria will be selected.

Examples (all based on point selection):

To Select:	Enter the following into the fields:		
	Collection	Group	Target
All points in all collections	*	*	*
All points in all collections containing "Survey"	SURVEY	*	*
All points in the Survey collection from point groups whose names contain "Instrument", followed by any string, followed by "Measured"	SURVEY	INSTRUMENT*MEASURED	*

To select everything except the items selected by the search criteria, check the Inverse Selection Mode option. After completing a wildcard selection, the items are selected (signified by the checked checkbox next to their entry in the list) but the F2 dialog remains open so that you can modify your selection or add to it. You can even add the result of another wildcard selection to your currently selected items.

To Select Based on Wildcards:

1. At a selection prompt, open the F2 Selection Window.
2. Click the By Name Using Wildcards button.
3. Type in your criteria, then click OK.
4. Modify the selection as needed, or perform another wildcard selection.
5. Click OK to select the items.

Selection By Object Type

The F2 selection window enables you to select items by their type (see Figure 7-7). Although this is not particularly useful when selecting items in a single collection (because those items are already organized by type and can be easily selected by clicking one checkbox), if you have multiple collections of items that you want to select by type, this can save you some time.

To Select Objects By Type:

1. In an object selection mode, call up the F2 selection dialog.
2. Click on the By Object Type button.
3. In the ***Select Object Types*** dialog, select the object types you'd like to select, then click the OK button.

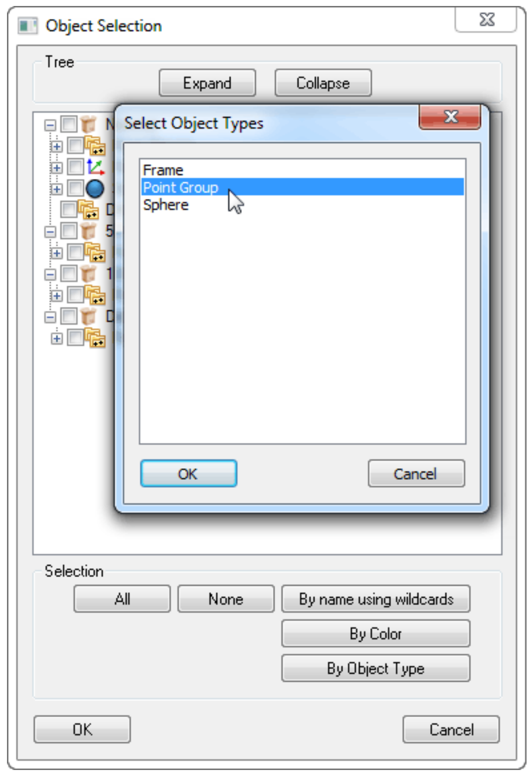


Figure 7-7. Selecting all point groups by their object type.

Selection By Color

It may be desirable to use object color to characterize different entities in the job file. For instance, you may decide to color all nominal objects yellow, and all as-built or measured objects blue. If you follow this colorization scheme, then you can easily select all nominal or measured objects using selection by color.

There are two ways to select objects by color. You can select graphically by color, or you can F2-select by color. Both color selection modes apply to object selection only, because only objects can be assigned a color. When selecting graphically by color, you choose a single item whose color defines the color to select. SA will then automatically also select all objects that match that color. F2 selection by color allows you to directly pick a color (not an object) and SA selects all objects of that color.

To Graphically Select Objects By Color

1. While in object selection mode, press the **F3** key or click on the Graphical Selection Mode icon.
2. In the ***Graphical Selection Mode*** dialog, click the Select Objects by Color (Applies Once) button.
3. Click on an object whose color defines the color you'd like to select.

To F2 Select Objects By Color

1. With a selection prompt displayed, press the **F2** key or click the list selection icon.
2. In the resulting ***Object Selection*** dialog, click the By Color button.
3. In the ***Select Color*** dialog, check one of the colors in the list. (Only the colors in use in the file will be displayed).
4. Click OK. You will be notified how many objects matched the color criteria, and they will be added to your selection.

Context Menus

Context menus play an important part in the SpatialAnalyzer workflow. They allow you to select commands directly related to an item by selecting that item instead of looking for a command in a menu. In some cases, commands are available in context menus and nowhere else. To access a context menu, simply right-click an item or object. If it has a context menu, it will be displayed adjacent to the mouse cursor.

Context menus are frequently used in the graphical view. Right-clicking an entity in the graphical view reveals its context menu, allowing quick access to commands related to that entity (see Figure 7-8). There are several commands that appear in the graphical view context menus that are not found anywhere else.

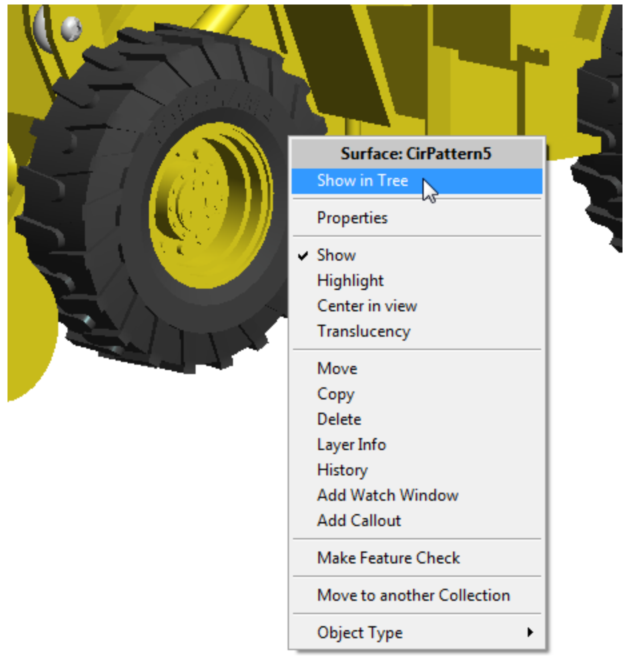


Figure 7-8. The context menu for this tire contains a command to reveal the surface in the tree.

Right-clicking an entity in the SA tree will also present a context menu and is one of the most common actions performed in the tree (Figure 7-9).

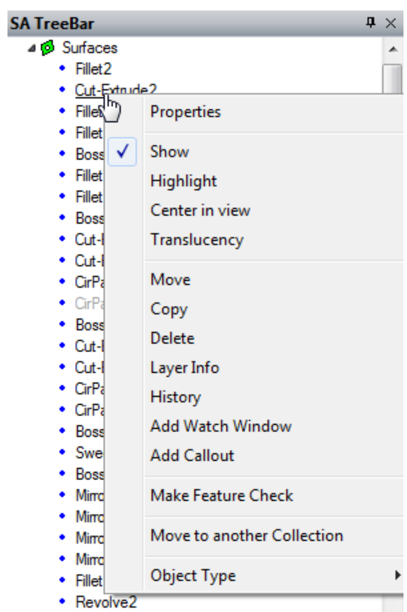


Figure 7-9. The context menu in the tree.

Organizing Your Job

In a typical measurement job, you'll be working with many different types of objects in a file: instruments, coordinate frames, measured points, geometry, CAD data, reports, and more. Without a way to organize this data, it would be easy to quickly get lost amongst the variety of entities in the file. Fortunately, SA provides a flexible way to organize this data so that it can be accessed conveniently and without confusion. The primary player in all of this organization is the tree, in which the items in your job are displayed in a hierarchical fashion.

Folders and *Collections* are two primary constructs that enable this organizational capability.

Folders

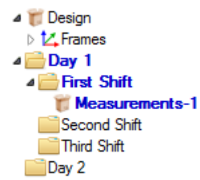


Figure 7-10. Multiple folders and collections in a hierarchy in the tree.

Folders allow you to sort and arrange your job information in the tree in a way that makes the most sense based on the way you work (Figure 7-10). You can create nested folders and move collections from one folder to another. You can expand and collapse a folder hierarchy, hide folders, rename and move folders, and change their order in the tree. Folders can only contain other folders or collections. You cannot put an instrument, point group, coordinate frame, or any other entity directly inside of a folder.

To Create a New Folder:

- Right-click an empty area of the tree or on any folder and choose **New Folder**.

To Create a New Sub-Folder:

- Right-click the folder you'd like to be the parent of the new sub-folder, and choose **New Sub-Folder**.

To Delete a Folder:

- Right-click a folder in the tree and choose **Delete**. All items within the selected folder will be deleted.

To Delete Multiple Folders by Name:

1. From the menu, select **Edit>Delete>Delete Folders By Wildcard...**
2. Type a wildcard search criteria (* and ? are allowable wildcard characters) into the *Search Criteria* field.
3. Click the OK button and confirm the deletion action.

To Rename a Folder:

- Right-click the folder and choose **Rename**.

To Expand or Collapse an Entire Folder Hierarchy:

- Right-click the folder and choose **Expand All** or **Collapse All**.

To Change the Order of Folders in the Treebar:

- Right-click a folder and choose **Change Order In List>Move Up (Move**

Down).

Collections

> **Note:** If you measure points (through an instrument interface) into a collection that does not already exist, the collection will be created automatically for you.

A *collection* refers to a group of objects, measured data, reports, and other entities that can be managed as a group from the tree, and as depicted with an open box icon . This "collection" of objects serves to group related data in a logical manner, making measurement and analysis more intuitive.

A job may have any number of collections. However, only one of these collections may be the *active* collection at any one time. When a collection is active, it is considered to be the *default* collection. Any newly-created entities (e.g., Points, Instruments, Planes, Lines, Reports, Relationships, etc) are created in the active collection unless a different collection is explicitly specified. The active collection is depicted in the tree by a *bold blue font*. Notice in Figure 7-10 that any folders enclosing the active collection are also highlighted with a *bold blue font* so that you can find the active collection if the folder hierarchy is collapsed.

To Create a New Collection:

- Right-click an empty area of the tree, any collection, or any folder and choose **New Collection**, or
- Measure a point or object into a collection that does not already exist (see Measurement).

To Rename a Collection:

- Right-click the collection in the tree and choose **Rename**.

To Delete a Collection:

- Right-click a collection in the tree and choose **Delete**.

To Delete Multiple Collections:

- Right-click a collection in the tree and choose **Delete Multiple Collections**, or
- Right-click an empty area of the tree and choose **Select Collections to Delete**.

To Delete Multiple Collections By Name:

1. From the menu, select **Edit>Delete>Delete Collections By Wildcard...**
2. Type a wildcard search criteria (* and ? are allowable wildcard characters) into the *Search Criteria* field.
3. Click the OK button and confirm the deletion action.

To Copy a Collection:

- Right-click a collection and choose **Copy**.

To Change the Order of Collections in the Treebar:

- Right-click the collection and choose **Change Order In List>Move Up (Move Down)**.

To Make a Collection Active:

- Right-click the collection and choose **Make active default collection**.

To Move a Collection to a Folder:

> **Note:** To move a collection out of all folders, choose the Root folder as the destination.

> **Note:** A common mistake for new users is to try to double-click a category in the tree to select it. You must expand the category to reveal the actual items inside it.

1. Right-click the collection and choose **Move Collection to Folder**.
2. Choose the destination folder from the resulting dialog.

When objects are created inside of a collection (either explicitly or as the result of an operation), SA automatically organizes those objects by their type inside what is referred to as a *category*. A category is simply a container that holds all like items in a given collection. For instance, all point groups in a given collection are placed into the **`Point Groups`** category, and all instruments in a collection are placed into the **`Instruments`** category. This behavior makes it easy to keep your file's elements organized by type.

Relocating Items to Another Collection

A common use for collections is to organize nominal and measured data separately. Another use is for organizing multiple states of measured data (for instance, measuring an assembly at various stages in the production process). You may occasionally come across the need to move items from one collection to another.

To Relocate an Item to Another Collection:

1. Right-click the item in the tree or graphical view and select **Move to Another Collection** from the context menu.
2. In the ***Select Collection*** dialog, double-click the destination collection (or select it and click OK). The item will be moved into the destination collection.

To Relocate Multiple Objects to Another Collection:

1. From the menu, select **Edit>Relocate Objects to another Collection**.
2. Select the objects you'd like to relocate.
3. In the ***Select Collection*** dialog, double-click the destination collection (or select it and click OK). The objects will be moved into the destination collection.

To Relocate Multiple Instruments to Another Collection:

1. From the menu, select **Edit>Relocate Instruments to another Collection**.
2. Select the instruments you'd like to relocate.

3. In the ***Select Collection*** dialog, double-click the destination collection (or select it and click OK). The instruments will be moved into the destination collection.

To Relocate Multiple MPs to Another Collection:

1. From the menu, select **Edit>Relocate MPs to another Collection**.
2. Select the scripts you'd like to relocate.
3. In the ***Select Collection*** dialog, double-click the destination collection (or select it and click OK). The MPs will be moved into the destination collection.

Rearranging Points

At some point while working in a file, you will likely want to rearrange your points by moving them from one point group to another. There are several ways to do this.

To Relocate Points into a Separate Group:

1. From the menu, select **Construct>Points>From Existing Points>Move**.
2. Select the points to relocate.
3. In the dialog, enter a name for a new group into which to place the points. This group must not already exist.
4. The points will be removed from the current group and placed into the new group. The point names will be prepended with the source group name.

To Relocate a Single Point into a Different Group:

Renaming a point and giving the point a different collection or group will move it into that collection/group. If the collection or group does not already exist, it will be created for you.

1. Right-click the point to relocate and choose **Rename** from the context menu.
2. In the ***Rename Point*** dialog, enter a new collection or group name, then click OK.

The Group Manager

You can rearrange points in an intuitive way using the ***Group Manager***. The ***Group Manager*** is a Windows Explorer-like dialog that allows you to drag and drop points from one group to another, and also provides the capability for renaming, group creation/deletion, search and replace, sorting, and more.

To Call Up The Group Manager:

- From the menu, select **Edit>Group Manager...**

The Group Manager is shown in Figure 7-11.

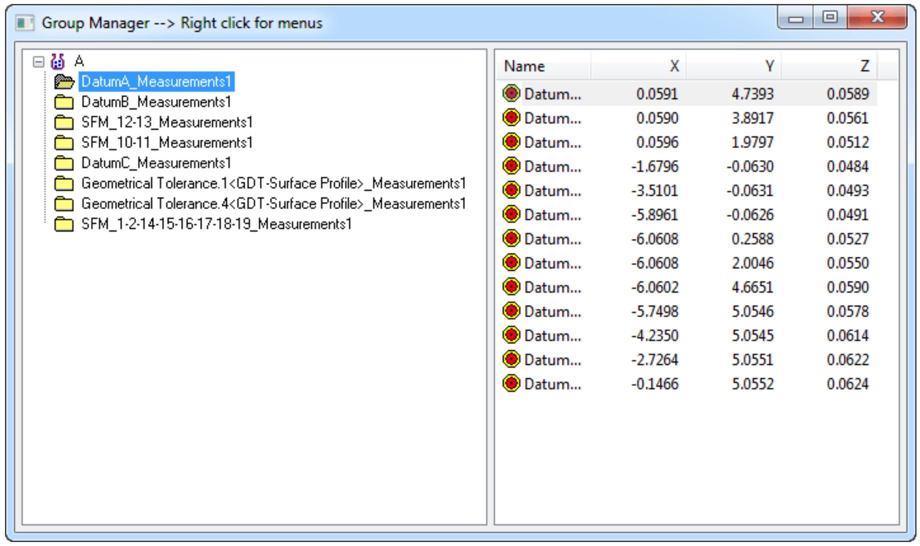


Figure 7-11. The Group Manager.

On the left side of the group manager is a list of all groups in your job, organized by collection. At right is a list of the points in the selected group, as well as their coordinates.

To Create a New Group:

1. Right-click anywhere on the left side of the Group Manager and select **New** from the context menu.
2. Type a new group name, then click OK.

To Delete a Group:

- Right-click a group on the left side of the ***Group Manager*** and select **Delete** from the context menu.

To Rename a Group:

1. Right-click a group on the left side of the ***Group Manager*** and select **Rename** from the context menu.
2. Enter a new name for the group, then click OK.

To Relocate Points to a Different Group:

1. Select the points to relocate from the right side of the window. You can only select points from one group at a time, although multi-selection is allowed.
2. Do one of the following:

- Drag the selected points to the destination group, or
- Right-click the selection and select **Cut** from the context menu, then right-click the destination group and select **Paste** from the context menu.

To Copy Points To Another Group:

1. Select the points to copy on the right side of the window. You can only select points from one group at a time, although

multi-selection is allowed.

2. Do one of the following:

- Right-click the selection and choose **Copy** from the context menu, then right-click the destination group and select **Paste** from the context menu, or
- Right-click the selection and choose **Copy selections into a New Group** from the context menu. Type a new group name, then click OK.

To Create New Points:

1. Select the group to contain the new points on the left side of the ***Group Manager***.
2. Right-click the right side of the window and select **New** from the context menu.
3. Enter the Cartesian coordinates for the point, then click OK.
4. Enter the name for the new point, then click OK.

To Delete Points:

- Select one or more points, then right-click the selection and choose **Delete** from the context menu.

To Sort Points by Name or Coordinate Value:

- In the ***Group Manager***, click a column header to sort by that value. Click again to sort in the reverse direction.

To Find Points:

1. Right-click a point or selection (select points to search amongst first, if desired) and select **Find...** from the context menu.
2. Type a search string, and choose either the *Selection* option to search the selection, or the *Group* option to search the entire group.
3. Click the OK button. The list of points will be filtered to show only the matching points. (Click the group name to reset the list).

To Perform Search and Replace in Point Names:

1. Select points (if desired), then right click the selection and choose **Replace...** from the context menu.
2. In the ***Replace*** dialog, enter text to search for, and the text to replace it with. Choose *Selection*, *Group*, or *All* to indicate whether you want to search among the selection, entire group, or all points.
3. Click the OK button. All matching text substrings are replaced.

To Access Point Properties:

- Double-click a point in the ***Group Manager***, or
- Right-click a point and select **Properties** from the context menu.

To Access Group Properties:

- Right-click a group and select **Properties** from the context menu.

Naming & Renaming Items

If you create objects in SA without properly naming them—that is, by using the default names provided by SA—you will soon find that your job file becomes completely unmanageable. In all but the simplest of measurement files, there are enough entities created that you can quickly lose track of what each entity represents. Therefore, best practice is to clearly name all items in your file when you create them. Doing so makes the file easier to work with, keeps it organized, and also allows you (or another person) to look at the file perhaps years later and still understand the data inside it.

Choosing a name is entirely up to you, although there are a few rules that should be followed.

- Names are not case-sensitive. A group named `TEST` is not distinguishable from a group named `test`.
- Spaces are allowed, but depending on the letters surrounding the space, it is often difficult to determine if a name contains a space at all (or perhaps multiple spaces). If you're worried about potential confusion, avoid the use of spaces.
- Multiple points in the same point group cannot have the same name. They must be unique.
- Multiple objects may have the same name in a given collection, however this is not recommended if you expect to use scripting (Measurement Plans), as this leads to potential ambiguity within your script as to which object you are referring.
- Avoid long consecutive sequences of numeric digits at the end of a point or group name. Keep them to less than 20 digits. (Otherwise, incrementing and decrementing of point and group names will not function correctly).

To Rename a Group of Points:

Do one of the following:

1. Do one of the following:

- From the menu, select **Edit>Rename>Group of Points**, or
- Right-click a group in the tree and select **Rename** from the context menu.

2. At the prompt, enter a new name for the group, then click OK.

To Rename a Point:

1. Right-click a point and select **Rename** from the context menu, or select **Edit>Rename>Single Point**.
2. At the prompt, enter a new Collection/Group/Target name (as desired), then click OK.

To Rename Objects:

- Right-click the object and select **Rename** from the context menu. (If this menu item is not available, choose the **Properties** menu item and the name will be found in that dialog).

Renaming Using Unique Names

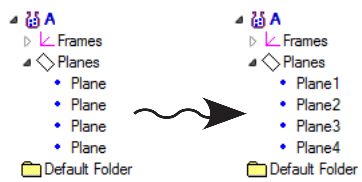


Figure 7-12. Automatically removing duplicate names.

Occasionally, you will be presented with a file in which several objects have the same name. This is sometimes the case when importing an external CAD file, for instance. You can easily automatically rename a series of objects, ensuring that they all have unique names (Figure 7-12). SA will rename the objects by appending the type of object with a unique number and using that for the new name.

To Rename Objects Using Unique Names:

1. From the menu, select **Edit>Rename>Objects by Assigning Unique Names**.
2. At the prompt, select the objects to rename.

SA will rename each of the objects in turn, ensuring that the names are unique.

Renaming Using A Name Pattern

You can rename points and objects using a *name pattern*. Suppose you wish to rename a set of objects and call them **Obj1**, **Obj2**, **Obj3**, etc. This would use a name pattern that looks like this: **Obj%d**. When renaming using a name pattern, you can specify any prefix or suffix for the name, then add the wildcard **%d** to the search field. The **%d** characters will be replaced by an actual number when the command is performed. By default, the first number is 1, but you can specify your own starting value.

To Rename Objects Using a Name Pattern:

1. From the menu, select **Edit>Rename>Objects Using a Name Pattern**.
2. Select the objects you'd like to rename.
3. In the ***Pattern for Rename*** dialog (Figure 7-13), specify the

pattern and a starting value, then click OK.

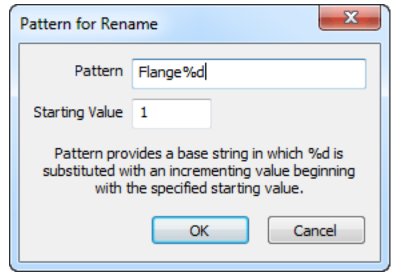


Figure 7-13. Renaming a group of objects using a name pattern.

To Rename Points Using a Name Pattern:

1. From the menu, select **Edit>Rename>Points Using a Name Pattern**.
2. Select the points to rename.
3. In the ***Pattern for Rename*** dialog (Figure 7-13), specify the pattern and a starting value, then click OK.

Renaming By Proximity

Renaming by auto-corresponding to points uses a group of points as a name reference. A different group of points is then renamed based on the reference group's names.

You can rename points by proximity to a set of reference points. This is extremely useful for quickly renaming points in preparation for a Point to Point Best-Fit operation. When renaming by proximity, SA will search for a point in a reference group that is closest to a point in the group to rename. As long as the reference point is within the specified threshold, the point to rename will be assigned the same name as the reference point (see Figure 7-14). Note that the reference group and group to rename need not have the same number of points. Points that don't have a match will be ignored.

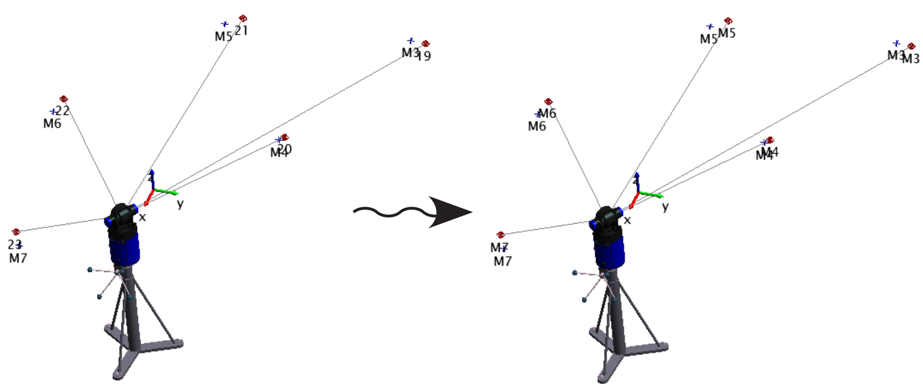


Figure 7-14. Before renaming points by proximity (left) and after (right).

The renaming operation is controlled by a *Proximity Distance Match Tolerance*. This is the furthest that a reference point can be located relative to the point to rename to be considered a match.

Probably the most common use for this is when remeasuring a set of tie-in points after an instrument has *drifted* (moved) beyond the desired threshold. After remeasuring the tie-in points, since the instrument has only slightly changed position, they will be relatively close to the nominal tie-ins. In this case, you need not be concerned with the proper naming of the points. Simply measure them, and SA will take care of automatically renaming them for you.

To Automatically Rename Points by Proximity:

1. From the menu, select **Edit>Rename>Points by Auto-Corresponding 2 Groups>Proximity**.
2. At the prompt, select the reference group—you may need to select it from the tree. This is the group containing the proper point names that you'd like to assign to the other group.
3. At the second prompt, select the group to be renamed—you may need to select it from the tree.
4. In the ***Input*** dialog, specify the *Proximity Distance Match Tolerance*.
5. If any matches are found within the specified threshold, you will be presented with the ***Residuals*** dialog. This dialog lists the original point names, and their proposed new names. Click the Save to File button to save this list to a text file, or the Accept button to accept the proposed new names.

Note: If (due to the threshold) matches are found for some points but not others, points that did not have matches will have `Orig.` prepended to the point name. This is to ensure uniqueness—another point that did have a match may have been renamed to the unmatched point's name.

Note: If the threshold is very large, SA will only use the closest reference point to each measured point.

Renaming By Inter-Point Distance

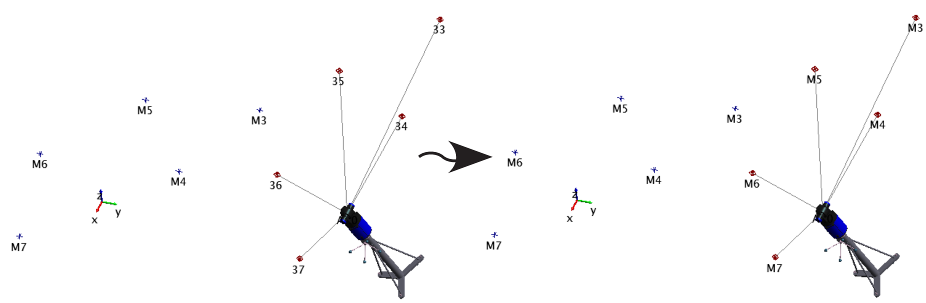


Figure 7-15. A set of points before renaming by inter-point distance (left) and after (right).

You will often want two sets of points to have matching corresponding point names. Suppose you have a set of nominal reference points in your SA file that you'd like to use for tie-ins. Assuming your instrument hasn't yet been located to the nominal points (as it typically will not be), the measured points will not be close to the reference points. In order for a Point to Point Best-Fit operation to successfully execute, corresponding point names must match up. If you forget to name the points correctly as you measure them, the fit will not succeed. While you could manually rename each individual point, this command performs that process automatically (Figure 7-15), and is a big time-saver in this situation. The command is governed by an *Inter-point distance match tolerance*. This tolerance is the maximum allowable *difference* between the spacing of any proposed pair of renamed points and the corresponding spacing of that same pair of points in the reference group. If the difference exceeds this value, the set is not considered a match.

Note that the reference group and group to rename need not have the same number of points. Any points that don't match will be ignored.

To Rename Points By Inter-Point Distance:

> **Note:** If (due to the threshold) matches are found for some points but not others, points that did not have matches will have `Orig.` prepended to the point name. This is to ensure uniqueness—another point that did have a match may have been renamed to the unmatched point's name.

> **Note:** If the threshold is very large, SA will only use the closest reference point to each measured point.

1. From the menu, select **Edit>Rename>Points by Auto-Corresponding 2 Groups>Inter-Point Distance**.
2. At the prompt, select the reference group containing the desired point names.
3. Then, select the group containing the points to rename.
4. In the ***Input*** dialog, enter the desired *Inter-point distance match tolerance*.
5. If any matches are found within the specified threshold, you will be presented with the ***Residuals*** dialog. This dialog lists the original point names, and their proposed new names. Click the Save to File button to save this list to a text file, or the Accept button to accept the proposed new names.

Deleting Items

To keep your job file clean, you should delete any items that you no longer need. If you are certain that a measurement was erroneous, for instance, it is best to delete it on the spot and remeasure. This minimizes clutter in the job file and helps make the file more understandable. There are numerous ways to delete items in SA.

To Delete Objects:

1. From the menu, select **Edit>Delete>Delete Objects**, or press the `Delete` key.
2. Select the objects to delete.

To Delete an Entity:

1. Right-click the item in the tree or graphical view.
2. From the context menu, select **Delete**.

To Delete Points:

1. From the menu, select **Edit>Delete Points**, or press `Alt+D`.
2. Select the points to delete.

To Delete Cloud Points:

1. From the menu, select **Edit>Delete Cloud Points>Selected Points**.
2. Select the cloud points to delete.

Sometimes you may wish to delete cloud points based on their radial distance from a set of points. For instance, once you've scanned a set of registration spheres and extracted the sphere centers, you may wish to delete the scan data of the spheres themselves (so they do not interfere with other analysis).

To Delete Cloud Points by Their Radial Distance From Reference Points:

1. From the menu, select **Edit>Delete Cloud Points>By Radial Distance from Points**.
2. Select the point cloud(s) of interest.
3. When prompted, pick the points to use as the reference points.
4. Specify a radial distance.
5. At the prompt, specify whether the cloud points inside or outside of the radius should be deleted.

To Delete Vectors:

1. From the menu, select **Edit>Delete Vectors**.
2. Select the vectors to delete.

SA keeps track of the source of objects—whether they were created natively in SA or imported from CAD files. As a result, you can delete objects that were imported from a specific CAD file.

To Delete Objects From a Specific CAD Source:

1. From the menu, select **Edit>Delete>Delete CAD Objects By Source**.
2. In the ***Delete Objects by CAD Source*** dialog, select the source CAD file, then click OK.

To Delete Pictures:

1. From the menu, select **Edit>Delete>Delete Pictures**.
2. Select the pictures to delete.

To Delete Relationships:

1. From the menu, select **Edit>Delete>Delete Pictures**.
2. Select the pictures to delete.

The Layer Manager

Often, when working with complex files, you may find yourself showing and hiding sets of related objects in the tree. For instance, if you have a CAD model and measurements of several sub-systems on an assembly, you may wish to view only a given sub-system in the graphical view.

For this purpose, you can create *layers*. You can assign objects to certain layers, then show or hide layers. This serves as a very quick way to show and hide multiple objects at once.

Every file contains a **`Default`** layer that cannot be deleted.

To Show the Layer Manager:

- From the menu, select **View>Layer Manager**. The ***Layer Manager*** dialog appears (Figure 7-16).

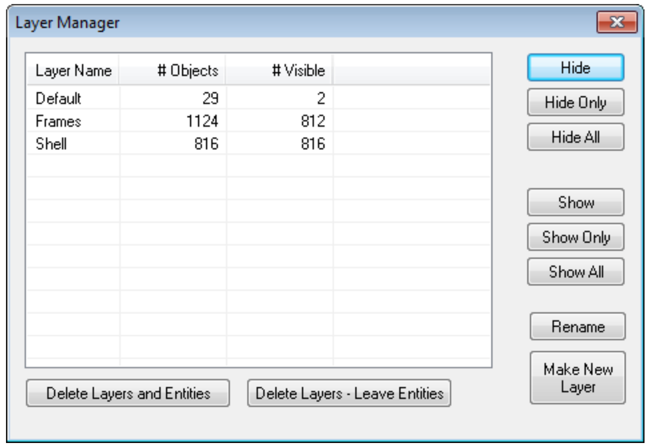


Figure 7-16. The Layer Manager.

To Define a New Layer:

1. In the SA file, show only those objects you'd like to appear on the layer.
2. In the ***Layer Manager***, click the Make New Layer button.
3. Type a new name for the layer, then click OK.
4. The ***Layer Manager*** will show the new layer and indicate the number of objects on the layer, and the number of objects currently visible.

To Show or Hide Layers:

1. In the ***Layer Manager***, select one or more layers to hide or show.
2. Click one of the following buttons:

- **Hide.** Hides the selected layer. Objects already hidden remain hidden.
- **Hide Only.** Hides *only* the selected layer. Objects on other layers are shown.
- **Hide All.** Hides *all* layers. The only remaining visible objects will be those that don't belong on a layer.
- **Show.** Shows the selected layer in addition to the objects already visible.
- **Show Only.** Shows *only* the contents of the layer, hiding all other objects.
- **Show All.** Shows *all* layers. The only remaining hidden objects will be those that don't belong on a layer.

To Rename a Layer:

1. In the ***Layer Manager***, select the layer to rename, then click the Rename button.
2. Type a new name for the layer, then click OK.

To Delete a Layer:

1. In the ***Layer Manager***, select one or more layers to delete.
2. Click one of the following buttons:

- **Delete Layers and Entities.** Deletes the layers and any objects on those layers.
- **Delete Layers - Leave Entities.** Deletes the layers but any objects on those layers remain.

To Add or Remove an Object From a Layer:

1. Right-click on the object in the tree or graphical view and select **Layer Info** from the context menu.
2. The ***Layer List*** dialog appears, showing the layers that currently contain the object.

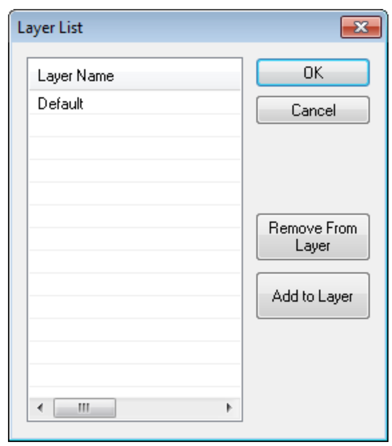


Figure 7-17. The ***Layer List*** dialog. This object is only on the default layer.

3. Do one of the following:

- To remove the object from a layer, select the layer and click Remove From Layer.
- To add the object to one or more layers, click the Add to Layer button. In the ***Add to Layers*** dialog, select the layers you'd like to add the object to, then click OK.

Configuring SpatialAnalyzer

There are a wide variety of settings in SpatialAnalyzer that provide significant control over the application's configuration. These settings can be stored in one of several different ways, depending on the scope of the setting:

- **Factory Defaults.** Factory default settings are stored as part of SA itself. These are the settings that apply when no other settings override them.
- **Job Files.** These are settings that are stored as part of the job file because it makes the most sense for them to be a property of a specific job file. Opening up a job file will restore its job file settings, regardless of the computer on which file is opened.
- **User Interface Profile.** These are settings that are stored as part of a customized user interface profile.
- **Machine Settings.** These settings are stored in the Windows Registry and apply at the computer level. Machine settings apply on a given computer, regardless of the current job file. (Although a user interface profile is stored in a profile file, the *active* profile is stored as a machine setting).
- **Persistence Files.** Persistence files contain configuration information and can easily be transferred from one computer to another.

Job File Settings

A number of the more general job file settings are described below—however be aware that this list is not all-inclusive.

Units

Anywhere a number with units is displayed in SA, it is presented in the active unit system, unless indicated otherwise. A job file can be set to one of five different length units: meters, centimeters, millimeters, feet, or inches. Temperature units can be set to either Fahrenheit or Celsius. The active job's length units are displayed in the status bar (Figure 7-18). Angular units are always presented in decimal degrees, but can be specified to present data in ±180° format or 0-360° format.

Figure 7-18. The active length units are displayed in the SA Status Bar.

To Change the Job File Units

1. Click the User Options icon in the Main toolbar.
2. Click to activate the *Units* tab.
3. Select the desired length or temperature units.

To Change the Representation of Angles

1. Click the User Options icon in the Main toolbar.
2. Click to activate the *Units* tab.

3. Select the appropriate setting in the *Angle Representation* box.

To Set the Default Job File Units (No Template File In Use)

1. Click the User Options icon in the Main toolbar.
2. Click to activate the *Machine Configuration* tab.
3. Click the No Template Default Options button.
4. In the ***New File Defaults*** dialog, select the preferred units.

Decimal Precision

Internally, calculations in SA are performed (and numbers are stored) with a high degree of decimal precision. However, when numeric results are displayed in the user interface, they are typically rounded to a preferred decimal precision in order to simplify the display. You can control the number of decimal places used when displaying different categories of values. Specifically, you can independently control decimal precision for length, angle, scale, unit vector, and weight values.

You can change the decimal precision for a given file at any time without incurring any additional internal numerical error.

To Change the Decimal Precision

1. Click the User Options icon in the Main toolbar.
2. In the *Decimal Digits for Display* box of the *Display* tab, change the *Length, Angle, Scale, Unit Vector,* and *Weight* decimal precision as desired.

To Set the Default Job File Decimal Precision (No Template File In Use)

1. Click the User Options icon in the Main toolbar.
2. Click to activate the *Machine Configuration* tab.
3. Click the No Template Default Options button.
4. In the ***New File Defaults*** dialog, enter the preferred display decimal precision.

A Closer Look

Having a different decimal precision for different types of numbers is useful, since different types of numbers are expressed in different units. For example, three digits of decimal precision for an angle (eg. 0.001°) is only accurate to 3.6 arcseconds--not enough for some applications. However, three digits of decimal precision in length when working in millimeters is too much precision for most applications (1 micron).

Reporting Tolerances

Reporting tolerances specify the threshold at which pointing errors or uncertainties are flagged in reports.

Angle-Distance Weights

Angle-distance optimization weighting values provide you with control over the weight of angular measurements compared to distance measurements for bundle adjustments and point calculation. You can weight distance and angular measurement components equally, or you can completely disregard distance or angular measurement components.

To Edit the Angle-Distance Weights

1. Click the User Options icon in the Main toolbar.
2. In the *Optimization Parameters* box of the *Units* tab, click the Edit Angle-Distance Weights button.
3. In the resulting ***Optimization Weighting Values*** dialog, drag the slider to the left to apply more weight to angular components, or to the right to apply more weight to distance components. You can optionally type relative weights for each into the two text fields.

Optimization Parameters

You have control over a number of parameters involved in certain calculations. These are found in the *Optimization Parameters* section of the *Units* tab in the User Options.

Convergence Tolerance

Optimizations are involved in many different aspects of SpatialAnalyzer's computations. While these optimizations usually have a single "best" answer in theory, in reality often they can only be found through iterative analysis. As a result, finding the "correct" answer would theoretically take an infinite amount of time.

Consider a best fit optimization: once the computer finds what it perceives as the solution that best fits a series of measurements to a CAD surface, it will stop. However, if you were to increase the internal decimal precision and "jiggle" the points around with even more fine movements and rotations, it could find a better solution than the solution initially proposed.

Continuously increasing the decimal precision and making finer adjustments becomes unwieldy after a short time, and it would most certainly try your patience. While a best fit showing $1x10^{-12}$ inches of RMS error is better than a best fit showing $1x10^{-11}$ inches of error, we don't particularly care, because the difference is much less than the accuracy of the measurement in the first place. The additional computation required to reach the better solution is just not worth our time.

As a result, SA follows a set of rules that tell it when an optimization is "good enough". When SA notices that the difference between two iterations in a solution is smaller than these thresholds, the solution terminates, because the result is considered to be "good enough". The default values provided for the convergence tolerances have been found to work well for a wide array of measurement devices and geometric optimizations.

To Edit the Convergence Tolerance Settings

1. Click the User Options icon in the Main toolbar.
2. In the *Optimization Parameters* box of the *Units* tab, specify new length and angular convergence tolerance values.

- **Length (object fits).** When the RMS error of a fit (such as a geometry fit, surface fit, or relationship fit) does not change by more than this value, the optimization process terminates. Using a smaller value here will theoretically give a better solution, but will increase the computation time required for the fit.
- **Angular (bundle adj).** When the RMS pointing error in a bundle adjustment does not change by more than this value from one iteration to the next, the bundle adjustment process terminates. Using a smaller value here will theoretically give a better solution, but will increase the computation time required for the fit.

Perturbations/Damping

For many of the numeric optimizations within SA, the partials matrix is computed using numerical differentiation. A value is changed by a delta (referred to as a perturbation), and the result is computed. The deltas used to offset the values in an optimization are set here. At each step, one portion of the partials matrix is populated. If these values are too small, you can experience sensitivity problems. The perturbation may be so small that the change in the objective is negligible and therefore the optimization proceeds without accurate direction. If the values are too large, it is possible to miss nuances in the solution space by moving over the ideal solutions.

To Adjust the Perturbation/Damping Values

1. Click the User Options icon in the Main toolbar.
2. In the *Optimization Parameters* box of the *Units* tab, specify new length and angular perturbation values, or specify a new damping factor.

- **Length Perturbation.** The length perturbation is the amount of perturbation along the X, Y, and Z axes. In the case of a best-fit transformation, for example, the X value is adjusted by the length perturbation. The objective function is recomputed, and this process is repeated with adjustments to the Y and Z values.
- **Angular Perturbation.** After perturbing X, Y, and Z, the Rx val-

ue is perturbed by the angular perturbation, then Ry, then Rz. At each step, one portion of the partials matrix is populated. If these values are too small, you can experience sensitivity problems. The perturbation may be so small that the change in the objective is negligible and therefore the optimization proceeds without accurate direction. If the values are too large, it is possible to miss nuances in the solution space by moving over the ideal solutions.

- **Damping.** Once the partials matrix is computed for a given step in an optimization, the matrix is inverted, and multiplied by the residual errors. This results in a vector describing how to move the input variables to minimize the objective is the system were truly linear. Since the systems we deal with in coordinate metrology are usually non-linear, the delta vector will most likely not move to the optimal solution. For this reason, we scale the vector by the damping factor to retard its effect. After making the move, the partial determination process repeats until an acceptable solution is reached.

Machine Settings

Although most parameters associated with work in SA are stored as part of the job file, many are instead stored on the computer. These settings are those that are more appropriate to be consistent on a given computer and do not necessarily have anything to do with a given SA file. For instance, the setting defining the file system location for SA backup files is a setting more appropriate to the computer instead of the SA file—regardless of which SA file is opened, it should always be saved to a certain backup folder.

SpatialAnalyzer Directories

SA uses a folder structure for storing certain files, such as measurement backups, templates, and instrument interface settings. You can control where this folder structure is located. By default, the folder structure is placed into the root of the `C:\` drive and consists of the following:

- **C:\Analyzer Data\\.** The base of the SA folder structure.
- **C:\Analyzer Data\Reports\\.** The location for generated reports.
- **C:\Analyzer Data\Templates\\.** The location for the default template file, as well as any other template files or custom user interface configuration files.
- **C:\Analyzer Data\Scripts\\.** This folder can be used to store Measurement Plan Scripts, although scripts can be located anywhere on the file system.

- **C:\Analyzer Data\Persistence\.** This folder contains instrument interface settings files, configuration files, parameter files, and other configuration information.
- **C:\Analyzer Data\Backup\.** All automatic backup files are saved to this folder. Measurement backups are saved to a subdirectory of this directory named `Measurements`.

To Change the Location of SA's Entire Folder Structure

1. Click the User Options icon in the Main toolbar.
2. In the *Machine Configuration* tab, click the Select Root Directory to contain "Analyzer Data" and its subdirectories button.
3. Choose a new folder.

The individual folder locations and names can also be changed directly from this dialog.

To Change the Location of an Individual SA Directory

1. Click the User Options icon in the Main toolbar.
2. In the *Machine Configuration* tab's *SpatialAnalyzer Directories* box, select the desired directory.
3. Click the Pick new Directory for Selected Entry button.
4. Select the desired folder from the folder browser.

Parallel Processing

Note: When a multi-threaded operation is underway, the two gears on the right side of the SA status bar will light up.

Many operations in SA are multi-threaded, such as USMN calculations, surface queries and projection, and point cloud filtering. However, in order to reap the substantial performance benefit of multi-threading, SA's parallel processing must be enabled (and your computer must support multi-threading).

To Enable Parallel Processing

1. Click the User Options icon in the Main toolbar.
2. In the *Machine Configuration* tab, check the *Use multiple processors if available* option.

Fault Handling

Should you ever experience a crash or other serious runtime error while using SA, you can configure SA to automatically collect information on this error and send it on to New River Kinematics for analysis. (This is similar to the Windows error reporting functionality). You can also choose to send a *large diagnostic file*, which is a more complete record and picture of SA's state at the time of the error.

To Enable Fault Handling

1. Click the User Options icon in the Main toolbar.

2. In the *Fault Handling* section of the *Machine Configuration* tab, enable the *Collect Diagnostic info from faults* option.

3. To also send large diagnostic files, select the *Enable large diagnostic file* option.

Security

You can place security restrictions on SA in order to restrict the access of different classes of users to SA's functionality. For example, you can prevent a user or class of users from saving a job file, measuring, deleting objects, moving objects, or using Measurement Plans.

To enable security restrictions, you first must create at least two *security classes*: One that provides administrator privileges, and another to provide restricted access. A security class defines the rights and privileges a group of people belonging to that class will have. The different rights that can be restricted are listed here:

- **Edit Security Options.** Members can edit security settings (normally this privilege is only granted to an administrator class).
- **Save Jobs.** Members can save files.
- **Print Jobs.** Members can print reports.
- **Measure.** Members can take measurements.
- **Delete Objects.** Members can delete objects.
- **Delete Points.** Members can delete individual points.
- **Delete Collections.** Members can delete entire collections.
- **Import Files.** Members can import files into the current job file.
- **Move Objects.** Members can move (transform) objects.
- **Copy Objects.** Members can make copies of objects.
- **Edit Points.** Members can change point and target properties.
- **Use MP.** Members can run Measurement Plan scripts.
- **Instrument Properties.** Members can modify instrument properties.
- **Bundle Adjust.** Members can perform bundle adjustments.

To Create a Security Class

1. Click the User Options icon in the Main toolbar.

2. In the *Security Classes* section of the *Security* tab, click the New button.

3. Give the class a name, and deselect the privileges you'd like to

restrict. Then click OK.

To Remove a Security Class

- In the *Security Classes* section of the *Security* tab, select the class you'd like to delete and click the Remove button.

To Edit a Security Class

1. In the *Security Classes* section of the *Security* tab, select the class you'd like to edit and click the Edit button.
2. Change the name of the class and/or the associated privileges, then click OK.

Once two or more classes have been created, you need to create users. These users need not refer to unique people (although they can)—that is, multiple people can use the same user name.

To Create Users

1. In the *Users* section of the *Security* tab, click the New button.
2. Enter a name for the user, then type and confirm a password for the user.

To Remove Users

- In the *Users* section of the *Security* tab, select the user you'd like to delete, then click the Remove button.

To Edit Users

1. In the *Users* section of the *Security* tab, select the user you'd like to edit, then click the Edit button.
2. Modify the username and/or the password for the user, then click OK.

The last step, once classes and users have been created, is to assign users to a class. A single user can belong to one or more classes. When a user belongs to more than one class, the user receives the combined privileges of all classes to which he belongs.

To Assign Classes To a User

1. In the *Users* section of the *Security* tab, select the user you'd like to assign to a security class.
2. Next to the *Belongs to classes* box, click the Add button.
3. Select a class to add to the user, then click OK.

To Remove a Class From a User

1. In the *Users* section of the *Security* tab, select the user to modify.
2. In the *Belongs to classes* box, select the class to remove from the user.

3. Click the Remove button.

Once your users, security classes, and privileges have been set up, you must enable security restrictions in SA, or else no change in behavior will occur.

To Enable Security

1. Click the User Options icon in the Main toolbar.
2. In the *Security* tab, ensure that all classes and users are set up correctly. Also verify that at least one user has full access to the *Edit Security Options* privilege, and that the user's username and password is known.
3. Uncheck the *Always Auto-login with full access* checkbox.
4. Restart SA. A username and password will be required to start the application.

With the *Always Auto-login with full access* option disabled, you can also use the Login Now button to log in as a different user without restarting SA. This is useful for conveniently testing the privileges of different security classes.

To Disable Security

1. Login to SA using a username that has access to the *Edit Security Options* privilege (typically the "administrator").
2. Click the User Options icon in the Main toolbar.
3. In the *Security* tab, enable the *Always Auto-login with full access* option.

System Diagnostics

SA provides a number of system tools and utilities to monitor configuration and diagnose potential issues with your system or file setup. These tools are listed below.

Memory Monitor

Large files (particularly those containing dense scan data or complex CAD files) can easily consume the available RAM on your system. To monitor your available/used RAM, SA provides a convenient memory monitor utility, which displays SA's memory usage, total system memory usage, total RAM, and operating system information.

To View the Memory Monitor

- From the menu, select **Help>Diagnostics>Memory Monitor**.

Check Job Data Consistency

You can verify the integrity and consistency of any SA file. This tool will examine an SA file and verify that there are no corrupt internal references.

To Verify the Integrity of an SA File

1. Open the SA file.
2. Select **Help>Diagnostics>Advanced>Check Job Data Consistency**.
3. A dialog will be displayed indicating whether any errors were found, and a log of the findings will be saved to the `Analyzer Data\Reports\_SAJobFileConsistencyReport.txt` file.

Regenerate References

If any issues are found with the ***Check Job Data Consistency*** tool, you can attempt to repair them by regenerating references. This will walk through the measurements and ensure that each measured point has the proper references to its targets.

To Re-Generate Target References

Note: Regenerating references is not normally necessary. It should only be performed if a job file consistency check indicates errors.

1. Open the SA file.
2. From the menu, select **Help>Diagnostics>Advanced>Re-Generate Measurement to Target References**.

Repeating Commands

Occasionally, the need comes up to repeat a command, perhaps several times. While you could search through the menus to find the desired command and select it again, such an approach is not efficient. Instead, you can use a keyboard shortcut to repeat the last executed command.

To Repeat the Last Executed Command:

- Press **`Ctrl+Tab`**, or select **Scripts>Repeat Command**.

In other cases, you may need to repeat a series of non-consecutive commands several times. In such a situation, the **Repeat Command** menu option will not help you, since it only executes the most recent command. Instead, you can ask SA to display a command history of the most recently executed commands (Figure 7-19), with the most recent at the top.

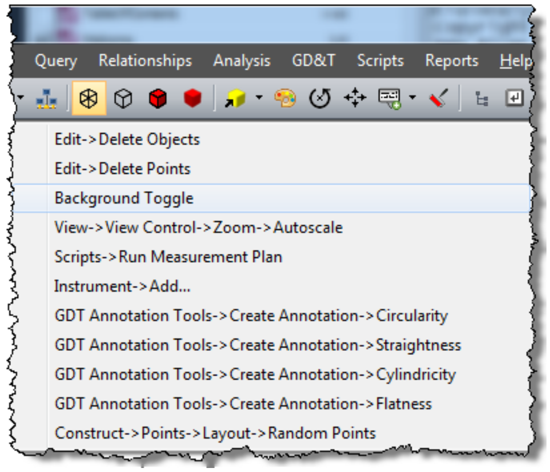


Figure 7-19. The command history.

To Select a Recent Command From the Command History:

- Press **Ctrl+Shift+Tab** or select **Scripts>Command History**, then select the desired command from the resulting list.

Working With Items

Many items in an SA job have some sort of representation in space. In a few cases in which the items are containers (collections, folders, point groups, and categories), the position and orientation of an item is not clearly defined, but still exists. For example, a collection can be moved and rotated (that is—a relative movement can be applied to it), but you would have a difficult time determining where that collection is specifically located in space. The list below indicates the defined positions and orientations of the various items in SA that have a representation in space:

- **Folders.** The specific position and orientation is undefined.
- **Collections.** The specific position and orientation is undefined.
- **Coordinate Frames.** The position is defined by its origin, and its orientation is defined by its axes. The primary direction of a coordinate frame is its Z axis.
- **Point Groups.** The specific position and orientation is undefined.
- **Lines.** The position is defined by its "begin" point. The line itself describes its Z direction.
- **Instruments.** The position is determined by the base, optical center, or other defined origin of the instrument. The standing axis of an instrument is its Z axis, and typically the X axis is defined by the "front" of the instrument (although this is not always the case).

- **B-Splines.** The specific position and orientation is undefined.
- **Planes.** The position is defined by its centroid. The Z axis is defined by its normal, and its X axis is typically defined by the first defined point in the plane.
- **Circles.** The position is defined by its center. The Z axis is defined by its normal, but the X and Y axes are undefined.
- **Ellipses.** The position is defined by its center and the Z axis is defined by its normal. The X axis is defined by its major axis, and the Y axis is defined by its minor axis.
- **Cylinders.** The position is defined by the center of its "begin" end. The Z axis is defined by its axis. The X and Y axes are undefined.
- **Spheres.** The position is defined by its center, and the X axis passes through its poles. The Y and Z axes are undefined.
- **Cones.** The position is defined by its apex, and the Z axis is defined by its axis. The X and Y axes are undefined.
- **Paraboloids.** The position is defined by its focal point, and the Z axis by its axis. The X and Y axes are undefined.
- **Polygonized Surfaces, Surfaces, Vector Groups, Point Clouds, and Perimeters.** The specific positions and orientations are undefined.
- **Graphical Entities.** Each has a defined position and orientation, but graphical entities are only for graphical purposes and are not useful in analysis.

If you're wondering why this is important, it's because many different operations in SA that require you to define a position or orientation allow you select most of the items above to describe that position or orientation. For instance, if in the process of constructing a new coordinate frame you're prompted to specify the X direction for the frame, selecting a circle is a valid selection for that prompt—SA will use the normal direction of the circle to define the X direction.

All of the objects listed above have a number of common characteristics, discussed below.

Moving Objects

While there are many commands to automatically apply transformations to objects based on special alignments and fits, all of the items mentioned in "Working With Items" above can be moved manually through any of several methods. These methods are described below.

Entering a Transform

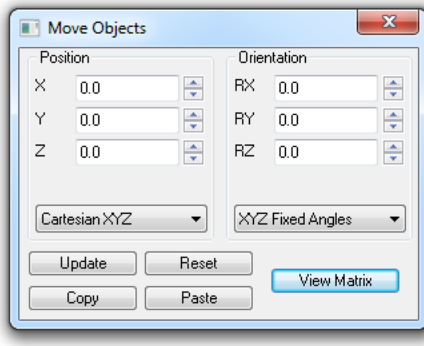


Figure 7-20. The Move Objects dialog.

The ***Move Objects*** dialog (Figure 7-20) provides the ability to apply a relative movement to an object by typing in values, expressed in the active coordinate frame. In this dialog, you can explicitly type in the positional and rotational deltas—these values are initially zero because they are relative movements from a starting position. Positional deltas can be entered in Cartesian (xyz), Cylindrical (rθz), or Spherical (rθϕ) values. Rotational deltas can be entered in *Fixed notation*, *Euler notation*, *Equivalent Angle Axis notation*, or as *Euler Parameters*.

To Move One or More Objects by Typing In Values:

1. Right-click an object in the tree or graphical view and select **Move**. Or, to move multiple objects simultaneously, select **Edit>Move Objects>Enter Transformation** and select the objects you wish to move.
2. In the ***Move Objects*** dialog (Figure 7-20), select the proper format in which to enter the values from the dropdowns in the *Position* and *Orientation* boxes. Usually, *Cartesian XYZ* and *Fixed XYZ* is the intended format.
3. Type in the appropriate values, then press **`Enter`** or click the Update button.

To Reset an Object Back To Its Original Position:

- Click the Reset button, or
- Manually type zeroes into all numeric fields, then press **`Enter`**.

Transforms can be copied from one dialog and pasted in another. This allows you to manually transform an object in the ***Move Objects*** dialog, then apply that same transform to another object.

To Copy and Paste Transforms Between Instruments:

1. In the ***Move Objects*** dialog, ensure the transform you want to copy is entered.
2. Click the Copy button.
3. Call up the ***Move Objects*** dialog for the destination object (see "To Move One or More Objects by Typing In Values:" above).

4. Click Paste.

If you are performing an analysis of your own, you can copy and paste raw 4×4 transform matrices between objects, and also export transforms to a comma-delimited .CSV file.

To Copy and Paste Transform Matrices Between Objects:

1. In the ***Move Objects*** dialog, click the View Matrix button.
2. In the ***Transformation Matrix*** dialog, click the Copy button to copy the current matrix to the clipboard, or click Export .csv to send it to a comma-delimited file.
3. Call up the ***Transformation Matrix*** dialog for the destination object.
4. Check the *Allow Editing* checkbox, which enables manual editing of an object's 4×4 transform matrix.
5. Click the Paste button, or choose Import .csv to import the matrix from a comma-delimited file.

To Edit An Object's Transformation Matrix Directly:

1. In the ***Move Objects*** dialog, click the View Matrix button.
2. Enable the *Allow Editing* checkbox.
3. Type the desired values into the 4×4 matrix.

Graphical Dragging

You may wish to move objects by dragging them graphically in the view. This is a great way to visually position objects in an approximate way without being concerned with exact transformation values.

To Drag Objects Graphically:

Tip: To rotate the object about different axes, use saved viewpoints to switch to different views without cancelling out of the command.

1. From the menus, select **Edit>Move Objects>Drag Graphically**.
2. At the prompt, select the objects you wish to move, then press **Enter**.
3. Use the left mouse button to move the object, and the right mouse button to rotate it about an axis perpendicular to the view orientation. When finished, press **Enter**.

Frame to Frame Transformations

Frame to frame transformations make note of the transformation required to go from one coordinate frame (the *source frame*) to another (the *destination frame*), and then applies that transformation to selected objects (Figure 7-21).

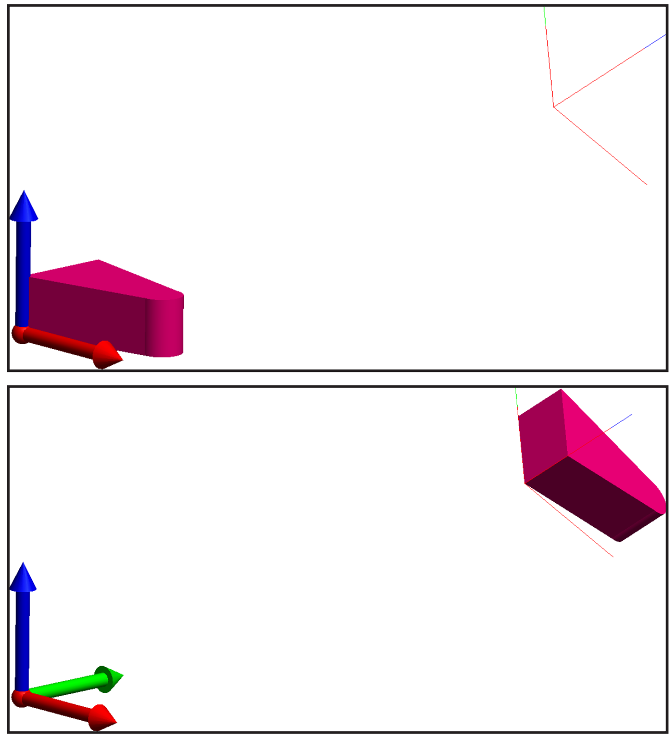


Figure 7-21. An object prior to (top) and after (bottom) a frame to frame transformation.

An interesting characteristic about coordinate frames is that they store complete information on position and orientation. As a result, they can be used to keep a record of an object's position. As an extension to that idea, two different coordinate frames store a positional and rotational delta (a transformation). Frame to frame transformations take advantage of this fact. If you plan on applying a transformation to an object, you can store the object's initial position by creating a coordinate frame at its location (the **Construct>Frames>On an Object** command is very handy for this). If, after moving the object, you change your mind and would like to put it back to its original position, you can create a frame on the object's current position, then use a frame to frame transformation to move the object back to it's original location.

To Apply a Frame to Frame Transformation:

1. From the menu, select **Edit>Move Objects>Frame to Frame Transformation**.
2. At the prompt, select the source frame. This is the coordinate frame that the transformation will be calculated *from*.
3. Select the destination frame. This is the coordinate frame that the transformation will be calculated *to*.
4. Now that the positional and rotational deltas have been calculated, select the objects to which you'd like to apply the transform. You may choose the source or destination frame, if desired.

5. Confirm or reject the transformation at the prompt.

Point to Point Delta Translations

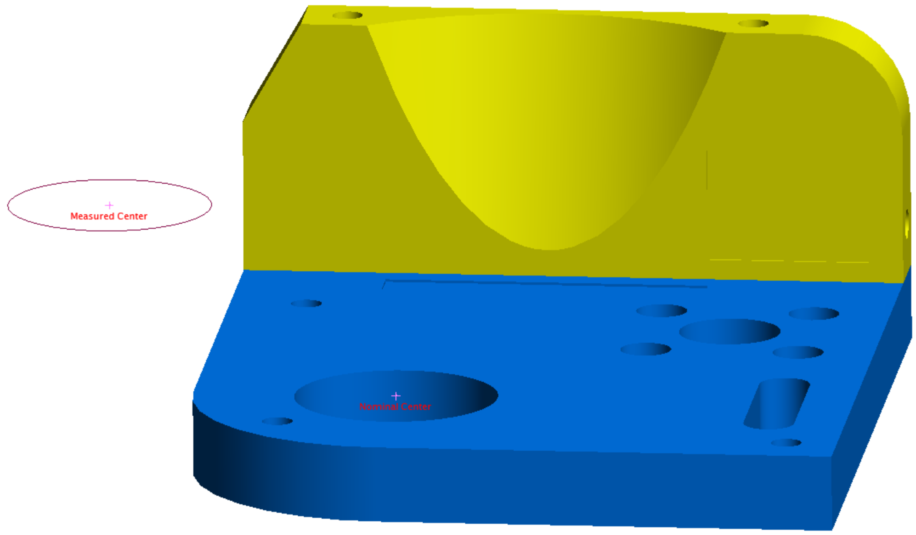


Figure 7-22. A circle is being moved to a nominal position through the use of a point to point translation. In this example, **`Measured Center`** is the source point, and **`Nominal Center`** is the destination point.

Often, you will want to apply a pure translation (movement in XYZ but no rotation) to one or more objects. Point to point delta transformations use a source point and a destination point to calculate a positional delta to apply to one or more objects (Figure 7-22). A point to point delta transformation is the 3 degree of freedom analog to a frame to frame transformation (which involves movement in 6 degrees of freedom). Just as frames can record an object's position and orientation in space, points can record an object's position in space. As such, point to point translations are reversible and can be used to "undo" a translation applied to an object—simply reverse the source and destination points.

To Move Objects Using a Point to Point Translation:

1. Select **Edit>Move Objects>Translate by Point to Point Delta**.
2. At the prompt, select the first (source) point.
3. Select the second (destination) point.
4. Choose the objects to translate.

Translating Using Instrument Updates

Updates from a 3 degree-of-freedom device such as a laser tracker can be used to translate one or more objects. Note that this is a pure translation--no rotations are applied.

When the command is initiated, the initial target position is recorded. As the target moves from this initial position, the translation is applied to the selected objects.

To Translate Objects Using Instrument Updates:

1. Select **Edit>Move Objects>Translate Using Instrument Updates**.

2. At the prompt, select the instruments (if more than one exists in the file) and the objects you'd like to move. These objects will move together as a rigid body.

3. The ***Translate Objects - by Instrument*** dialog will appear (see Figure 7-23).

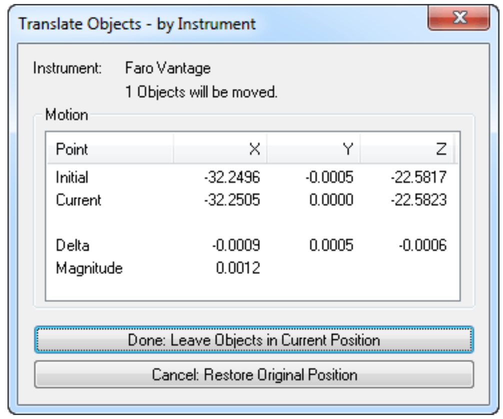


Figure 7-23. Translating objects using an instrument's 3 degree-of-freedom updates.

4. When measurement updates are received from the instrument, the objects will be translated.

5. When finished, you can either click Cancel: Restore Original Position, which will put the objects back in their original location, or click OK: Leave Objects in Current Position, which will accept the transformation and apply it to the objects.

Transforming in 6D Using Instrument Updates

Updates from a 6 degree-of-freedom device (such as a T-Probe, portable arm, T-Mac, or STS) can be used to transform one or more objects. The positional and rotational delta experienced by the device is applied to the objects involved in the transformation.

When the command is initiated, the position and orientation of the device is recorded. As the device moves and rotates from this initial position, the transformation is applied to the selected objects.

To Transform Objects in 6 Degrees of Freedom Using an Instrument's Updates:

1. Select **Edit>Move Objects>Transform in 6D Using Instrument Updates**.

2. At the prompt, select the instruments (if more than one exists in the file) and the objects you'd like to transform. These objects will move and rotate together as a rigid body.

3. The ***Transform (6D) Objects - by Instrument*** dialog will appear (Figure 7-24).

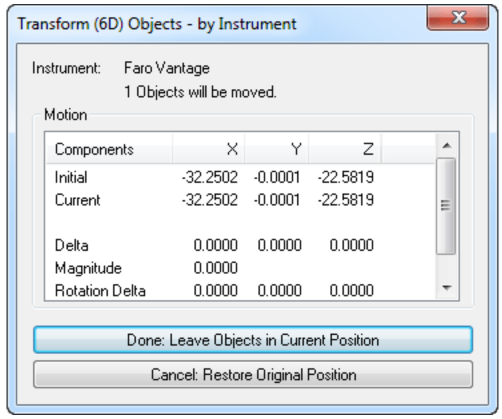


Figure 7-24. Transforming objects using an instrument's 6 degree-of-freedom updates.

4. When 6-DOF measurement updates are received from the instrument, the object position and orientation will be updated automatically.

5. When finished, you can either click Cancel: Leave Objects in Current Position to accept the transformation, or Cancel: Restore Original Position to move the objects back to their original position.

Transforming in 6D Using Multiple Instrument Updates

SA has the capability of synchronizing three degree-of-freedom measurements from 3 or more instruments and combining them to calculate a six degree-of-freedom transformation. This is commonly referred to as *Trans-Track*. For Trans-Track to function correctly, three or more instruments must first be tied to a common coordinate system. This can be achieved using any accepted technique, such as USMN, bundling, or best-fit. Once the instruments are tied together, each instrument must measure a single point, which becomes the reference point for that instrument.

The Trans-Track operation is then started, and the instruments are simultaneously polled for updates. A real-time transformation is then calculated, and that transformation is applied to any objects of choice.

To Transform in 6D Using Multiple Instrument Updates:

> **Tip:** Spread reference points apart as far as possible to minimize angular errors in Trans-Track.

> **Tip:** The accuracy of the Trans-Track operation is limited to the accuracy of the network, so try to tie instruments together as accurately as possible.

1. Ensure that you have three or more instruments tied together to the same reference system (usually a set of common points), and that their interfaces are all running in SA.

2. Affix three targets to the moving object, with one instrument watching each target--these will be your reference points.

3. Measure a single reference point from each of the instruments.

4. From the menus, select **Edit>Move Objects>Transform in 6D Using Multiple Instrument Updates**.

Tip: Be careful to place targets such that they will be visible throughout the range of motion of the moving object.

5. When prompted, select the instruments that will be part of the operation. You must be explicit about the order in which you pick the instruments.
6. Now select the earlier-measured reference points that correspond with the instruments. The points must be selected in the same order as the instruments. (That is, select the first instrument's reference point first, then the second instrument's point, etc.)
7. When prompted, select the object or objects to which you'd like the transformation applied.
8. The ***Transform Using Multiple Instrument Updates*** dialog will appear (see Figure 7-25).
9. Click the Query Instruments button, and SA will begin polling the instruments and calculating the transformation. The dialog will list the fit error between the live set of points and the reference set. The time lapse among the measurements is also displayed.
10. When finished, press Done: Leave Objects in Current Position to accept the transformation, or Cancel: Restore Original Position to undo any movement.

Tip: Measurements cannot be perfectly synced, therefore slower movement will result in more accurate real-time transformations due to a smaller time delay among the instrument measurements. Allowing the targets to become stationary for a moment will remove sync-based error from the transformation.

The Trans-Track dialog also has a few settings that can be modified (see Figure 7-26):

- **Point Best-Fit Error Tolerance.** If a point's best fit error to the corresponding reference point exceeds this threshold, the point will be highlighted as a warning.
- **Time Lapse Tolerance (sec.).** If the time difference between the first point and another point exceeds this threshold, the point will be highlighted as a warning, indicating that the measurement sync was not acceptable. This can be ignored whenever the targets are stationary.
- **Delay after update (sec.).** The time delay between one set of measurements and the next.

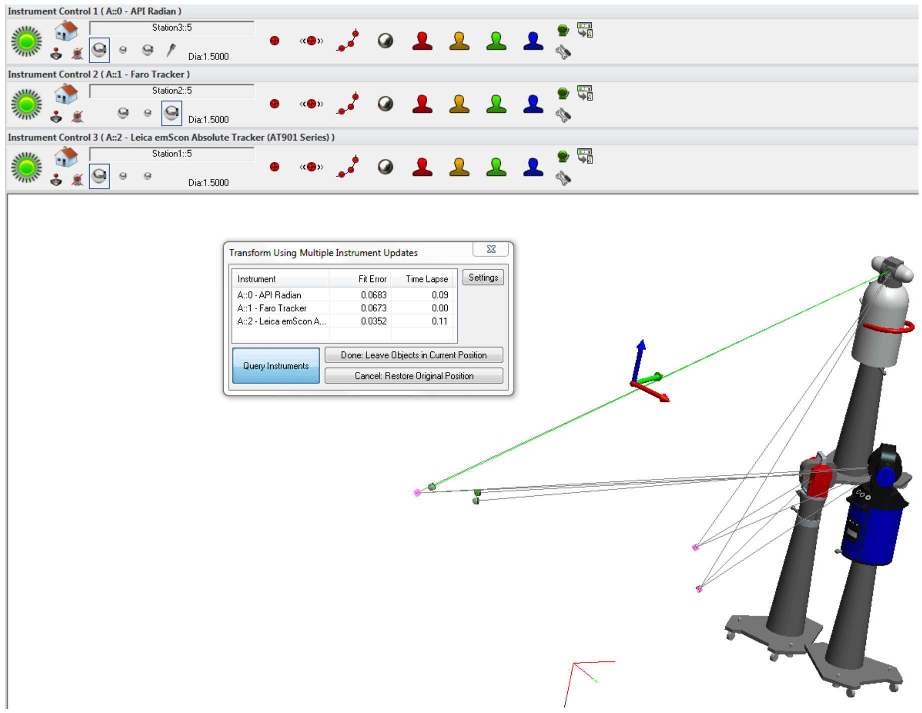


Figure 7-25. Performing a Trans-Track operation with three laser trackers.

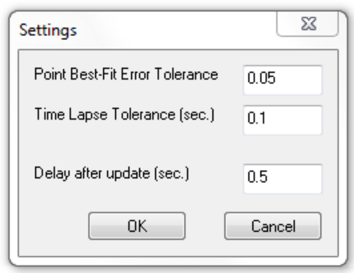


Figure 7-26. Trans-Track settings.

Transforming in 6D Using a Batch of Points from an Instrument

Some instrument systems, such as photogrammetry systems, are capable of tracking several points simultaneously. SA is capable of watching this batch of points and updating the transformation of one or more objects as the updates arrive. Since this command only involves one instrument, alignment among multiple instruments is not required as it is with **Transform in 6D Using Multiple Instrument Updates**.

To Transform Objects in 6D Using a Batch of Points From An Instrument:

1. From the menus, select **Edit>Move Objects>Transform in 6D Using a Batch of Points from an Instrument**.
2. Select the object(s) that you would like to transform.
3. The ***Trans-Track a Point Set*** dialog appears (Figure 7-27).
4. Before moving anything, measure a set of points. This will be used as the reference for the initial position.

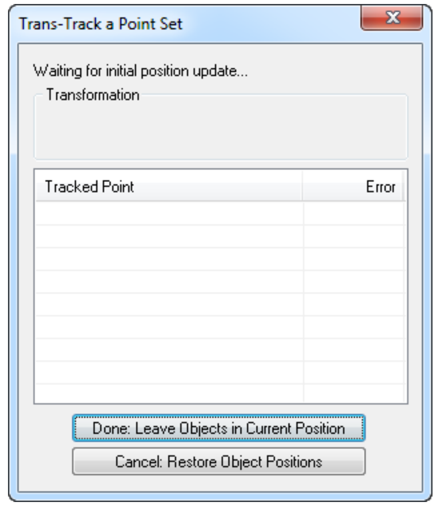


Figure 7-27. The ***Trans-Track a Point Set*** dialog appears.

5. As each set of measurements are taken, the position of the selected objects will be updated, and the fit error will be reported in the dialog.
6. When finished, click Done: Leave Objects in Current Position to accept the transformation, or Cancel: Restore Object Positions to cancel the command and put all objects back into their original positions.

Guiding Objects in 6D Using Point Measurements

If you would like to apply 6 degree-of-freedom transformations to one or more objects, but only have a 3 degree-of-freedom device available, then you can measure a set of reference points to serve as the controls for a 6 degree-of-freedom transformation.

The process involves measuring a set of reference points affixed to the moving object, and creating a corresponding set of "goal" points which represent these reference points transformed to the desired final location. You then repeatedly measure the reference points on the moving object, and SA will update the position of the moving objects and also display a watch window indicating the required motion to transform from the current position to the final position (Figure 7-28).

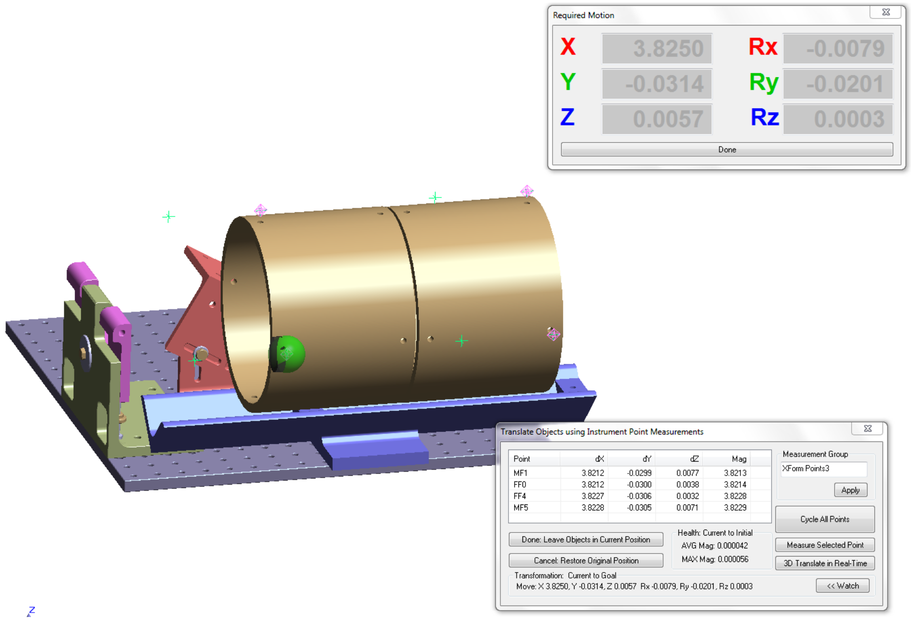


Figure 7-28. Guiding objects using point measurements.

The controls for guiding in 6D are shown in Figure 7-29.

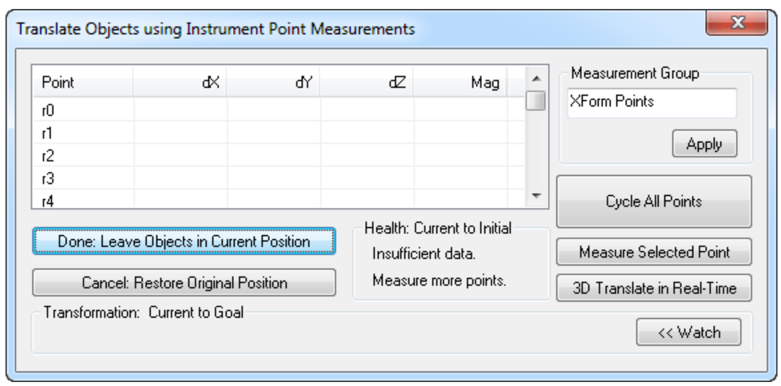


Figure 7-29. The ***Translate Objects using Instrument Point Measurements*** dialog.

- **Measurement Group.** Defines the name of the point group containing the measured points. If you change this, you must click the Apply button.
- **Cycle All Points.** When clicked, this button cycles through all reference points, points at their last measured position, searches, locks, and measures each point (applies to steerable, targeted instruments only).
- **Measure Selected Point.** Measures a point and associates it with the selected point in the table.
- **3D Translate in Real-Time.** Select a point in the table, then click this button. SA will track the target as this point and *translate* the moving objects. This is useful for getting real-time feedback on movement, without considering rotational information. Once you dial in the position, you can switch back to full 6 degree-of-freedom measurement of your reference

points by toggling this mode back off.

- **Watch.** Click this button to open a watch window showing the transformation, in the active frame, required to get from the current position to the goal position.

To Guide Objects in 6D Using Point Measurements:

Tip: Just as with a best fit or trans-track operation, spread your reference points out to improve your baseline and reduce angular errors in the solution.

1. Locate into a coordinate system as necessary.
2. Measure a set of reference points on the moving object. Three are required at a bare minimum, more are better.
3. Create a set of corresponding goal points in the destination position. This is usually accomplished by copying the reference group and applying an appropriate transformation to the copy.
4. From the menu, select **Edit>Move Objects>Guide Objects in 6D using Point Measurements**.
5. At the prompt, select the destination (goal) group.
6. Select the reference group on the moving object.
7. Select any additional objects you'd like to transform.
8. When the dialog appears, select the appropriate point in the table and click Measure Selected Point to measure it. Or, ensure targets exist at each reference location and click Cycle All Points to measure them all automatically.
9. Using the reported transform, apply the desired movement to the moving objects, then repeat the measurement.
10. When satisfied with the final position, click Done: Leave Objects in Current Position to accept the transformation.

Note: When using "Cycle All Points", movements must be small enough relative to the previously measured position to ensure that the instrument can still find the targets.

Copying Objects

Objects can be copied. The copy is always placed into the active collection. If the collection containing the copied object is the active collection, the object will be renamed as necessary to maintain uniqueness. There are two main methods for copying an object: Copying and then applying a transform (which can be unity to leave the object unmoved), or copying and then applying a point to point translation. Keep in mind that since point groups are objects, copying a point group will duplicate that group of points. However, since the copied points were not actual measured points (after all, they are copies) they will be depicted as constructed points.

It is probably most common for copies to be made without applying any transformation so that the copy is in the same position as the original.

Copy, Then Transform

This method first creates a copy of one or more objects, then permits you to optionally type in a transformation to move the copies.

To Copy then Transform:

1. Right-click an object in the tree or graphical view and select **Copy**; or, to copy multiple objects, select **Edit>Copy Objects>Copy Then Transform** or press `Ctrl+C`.
2. If copying multiple objects, select the objects.
3. The ***Copy Objects*** dialog will appear, which is identical to the ***Move Objects*** dialog. Enter the desired transformation to apply, or leave the values zeroed out to leave the copy in place.

Copy, Then Translate Point to Point

If you'd just like to translate the copies only, you can create a copy and apply a point to point translation to them. This is the copy-based version of moving objects using a point to point translation.

To Copy Using Translation Only:

1. In the menus, select **Edit>Copy Objects>Translate by Point to Point Delta**.
2. At the prompt, select the first (source) point.
3. Select the second (destination) point.
4. Specify the objects you'd like to copy, then press `Enter`.

Mirroring Objects

Objects can be mirrored (reflected) about any of a frame's three planes (XY, XZ, or YZ). This is most often used with symmetric parts. For example, if certain nominal features of an aircraft model are only available on the left side, it will need to be mirrored to the right side when it comes time to work on the right side of the aircraft.

You can either create a mirrored copy of selected objects (creates a duplicate, then reflects the duplicate), or you can mirror the objects directly without creating any copies. Mirrored copies are placed into the active collection and are renamed as necessary to ensure uniqueness. An object moved via a mirroring operation will not be renamed at all. Always use care when mirroring—depending on the object being reflected and the plane being mirrored about, the object's normal may flip directions.

To Mirror One or More Objects:

1. Select **Edit>Mirror Objects>Move** (to mirror the source objects) or **>Copy** (to mirror copies of the source objects).

2. Select the frame about which to mirror.
3. Select the objects to mirror, then press **Enter**.
4. In the ***Select Mirror Plane*** dialog, choose the plane about which to mirror, then click the Mirror button.

Scaling Objects

Warning: The most common situation in which an object is scaled is when you have point groups or other data that is not associated with an instrument and you want to apply a temperature compensation to it. Other than this situation, it is not common and is usually more appropriate to apply a scale factor to an instrument.

Note: The list of materials in the CTE Editor is stored in the SA file. If you'd like a modification to appear in all files, save your SA file as a default template.

Any object may be scaled by an arbitrary amount. This yields an object that has grown or shrunk by the corresponding scale factor relative to the active coordinate frame. Since the scaling is applied relative to the active coordinate frame, scaling generally also results in object translation (except for a point on the origin of the active frame).

Two different methods can be used for scaling:

- **Calculate Scale Based on CTE.** With this option, you select a material and an initial and final temperature, and SA calculates the scale factor automatically using the linear thermal expansion equation. The material entered should be the material of the objects being scaled. For example, if you are scaling measurements of a steel plate, steel should be used for the material.
- **Enter Scale Manually.** You can enter your own scale factor. As an example, a scale factor of 2.0 will double the size of the object.

To Scale One or More Objects:

1. From the menus, choose **Edit>Scale Objects**.

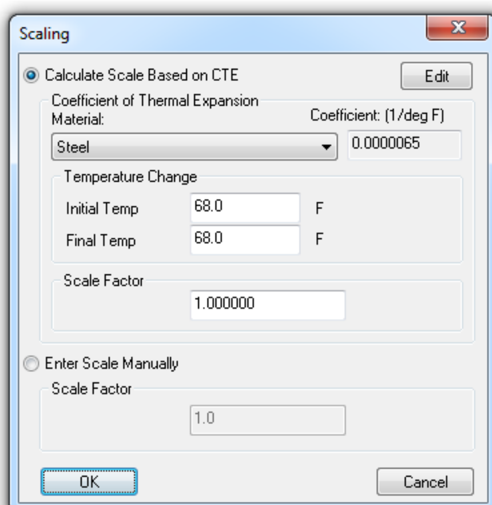


Figure 7-30. The Scaling dialog.

2. At the prompt, select the objects to scale, then press **Enter**. The ***Scaling*** dialog will be displayed (see Figure 7-30).
3. Choose the scaling method (either *Calculate Scale Based on CTE* or *Enter Scale Manually*).

4. If scaling manually, enter the desired scale factor. If scaling using CTE, choose the appropriate material and initial/final temperature. Then click OK.

If the default list of materials is not sufficient for your needs, you may wish to enter your own material with a custom coefficient of thermal expansion.

To Enter a Custom Material and CTE:

1. In the ***Scaling*** dialog, click the Edit button.
2. In the ***CTE Editor***, use the Add, Edit, and Remove buttons to modify the list of materials, then click Done.

Finding Items In The Tree

While working in the graphical view (particularly if the job file is complex and has many entities), you may need to find an item in the tree. Searching for the item through the tree is cumbersome in complex files, so SA provides the ability to find any item in the tree.

To Find an Item In the Tree:

- In the graphical view, right-click the item and select **Show In Tree** from the context menu. SA will expand, autoscroll, and select the item in the tree for you.

Network Browser

SA is a very network-capable application. You may be viewing an SA instance running on another computer on your network, or you may be connecting an instrument interface to another computer on the network. The SA Network Browser allows you to view the network and perform a few network-related diagnostic operations.

To Access the SA Network Browser:

- Click the SA Network  icon in the Main Toolbar.

The ***SA Network Browser*** dialog (Figure 7-31) displays at the root level all computers running SA that are visibile on the network, along with their IP address. In Figure 7-31, there are four computers running SA on the network.

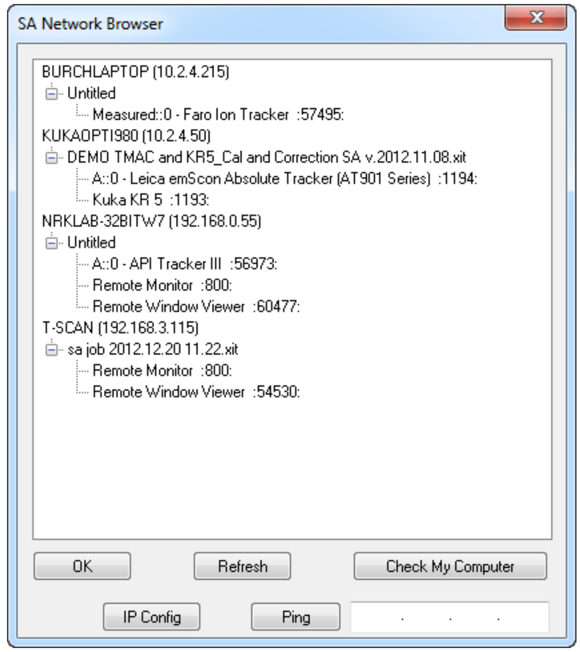


Figure 7-31. This ***SA Network Browser*** lists a number of instances of SA running on four different computers.

Below the computer name will be listed all open SA files on each computer, and below that all unconnected instruments or other connections that can be made. In Figure 7-31, the computer named **T-SCAN** has two possible connections: A connection to the Remote Monitor and the Remote Window Viewer.

To Refresh the Network Browser List:

- Click the Refresh button.

To Run the Check My Computer Tool:

- Click the Check My Computer button.

To View the IP Settings of Your Computer:

- Click the IP Config button.

To Ping an IP Address:

- Type an IP address into the text field, then click the Ping button.

Remote Monitoring

SA provides the capability to pull an open SA file from any computer reachable on the network and work with it. You can also remotely view the SA application, graphical view, or HUD windows from another computer (if applicable).

For privacy and security reasons, both of these remote viewing capabilities require the computer being monitored to have its *Remote Viewable* option set.

To Make a Computer Viewable Via Remote Monitoring:

1. Visit the User Options by clicking the User Options icon on the toolbar.
2. Navigate to the *Machine Configuration* tab and check the *Remote Viewable* option.

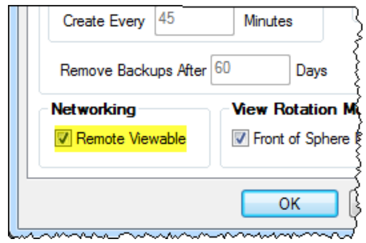


Figure 7-32. The *Remote Viewable* option.

To Work With the Current File From Another Network Computer:

> **Note:** Computers with the *Remote Viewable* option turned off will not be visible in this list.

1. From the menu, select **File>Remote Monitor another SpatialAnalyzer**.
2. When the ***Remote Monitor Options*** dialog appears, double-click any SA file's **`Remote Monitor`** option. Or, select the *Use manual entry of IP (or computer) and port* option and manually enter the IP address or computer name and port number to connect to, then click OK.

> **Note:** When remote monitoring, SA essentially saves the file from the remote computer and loads it on the local computer. No link is maintained between the two files after that point.

The file will be pulled from the other computer and opened. A timestamp will appear in the corner of the graphical view to indicate the time when the file was pulled (Figure 7-33).

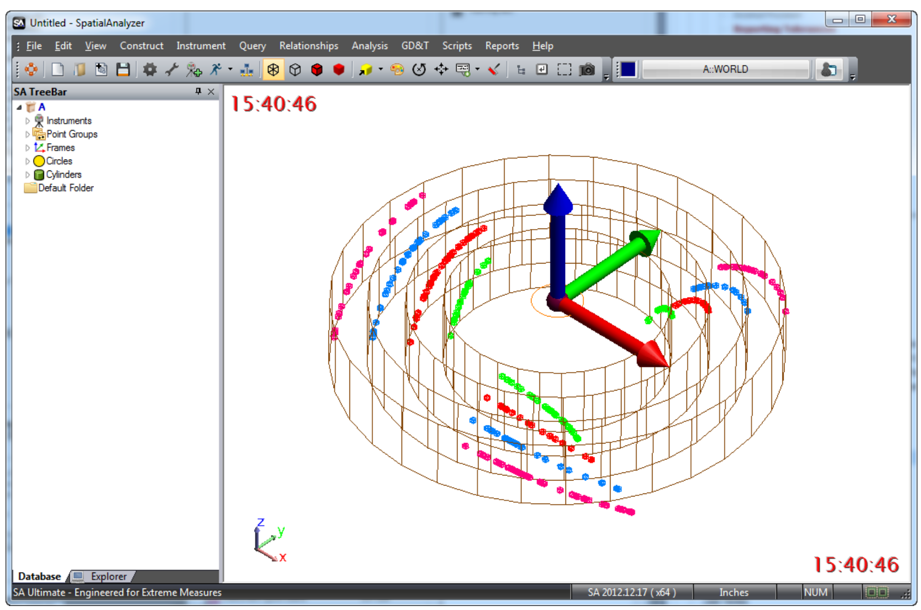


Figure 7-33. Remotely monitoring another instance of SA on the network.

To View an Active Window From a Remote Instance of SA:

1. From the menu, select **File>Remote Window Viewer**.
2. In the resulting ***Remote Window Monitor Options*** dialog, double-click the **`Remote Window Viewer`** entry that you'd like to

connect to, or select the *Use manual entry of IP (or computer) and port* option and enter an IP address/computer name and port number, then click OK.

3. When the ***Select Remote Window*** dialog appears, select the desired option, then click OK. A separate window will appear, which will periodically get updated. You are now watching the other computer's SA instance, live.

Working With The Clipboard

The Windows clipboard is the fastest way for transferring data within or among applications. It is far better than manually typing data into a different program, and it faster and "cleaner" then exporting data to a file and re-importing in the other application.

You can copy the graphical view to the clipboard as an image for pasting into any program that supports standard image formats.

To Copy the Graphical View to the Clipboard:

- From the menu, select **Edit>Copy graphics to clipboard**.

If you've copied point data from another source—perhaps an Excel file or Notepad text file, you can paste it directly into SA.

To Paste Points From the Clipboard:

1. In the source application (Notepad, Excel, etc.), select the data and copy it to the clipboard.
2. From the SA menu, select **Edit>Paste Points from Clipboard**.
3. Select the appropriate ASCII import options (see the chapter on File Import for detailed information on these options), then click Import.

Setting the Working Frame

While working in SpatialAnalyzer, all data is represented relative to the active (working) coordinate frame, unless you have specified otherwise. When you examine a point's coordinates, for instance, you are seeing those coordinates relative to the working frame. When moving an object, you are entering positional and rotational deltas with respect to the working frame. When performing a best-fit and restricting rotation about the X or Y axis, the X and Y axes are defined by the working frame. As a result, staying mindful of the active frame while working in SA is very important.

You can create as many coordinate frames in your job file as you want or need. You can also freely switch from one active frame to another at any time. This allows you to view deltas, specify rotational values, or look at coordinates with respect to any frame of reference.

To Change the Working Frame:

Do one of the following:

- Click the Working Frame button A::WORLD, then select the new desired working frame from the list, or
- From the menu, select **Edit>Pick Working Frame**, and select the new frame, or
- Press **Alt+W** (the default keyboard shortcut), and select the new frame, or
- Right-click any coordinate frame you'd like to activate and select **Make Working Frame**.

About the World Frame

Warning: Moving the **World** frame can be dangerous if you are not careful. Modifying this frame requires care and is generally not necessary.

The **WORLD** frame exists in every SA file and cannot be removed. This frame serves as the reference or base coordinate system for everything in your file. While you can move the world frame, there is usually no reason to do so. Deleting a world frame which has been moved from its original location will result in automatic creation of a new world frame in the original location (which explains why moving a world frame could cause confusion if you are not careful).

Object Color

Objects can be given any color, or *painted*, which you can use for any purpose. You can use the color of an object for organizational purposes, to assign special meaning to an object, or simply to make it look good. It's up to you to decide.

SA uses the concept of a *working color*. Every time a new object is created, it is given a color. The color that is assigned is the current—or working—color. Think of the working color as an open can of paint—each object you create is painted, and the working color is used for that painting job.

By default, the working color is set to automatically increment. That is, when an object is created and assigned a color, the next color in a list of 13 random colors is automatically made the working color. The result of this is that repeated object creation results in objects of different colors so that they are more clearly distinguishable in the graphical view.

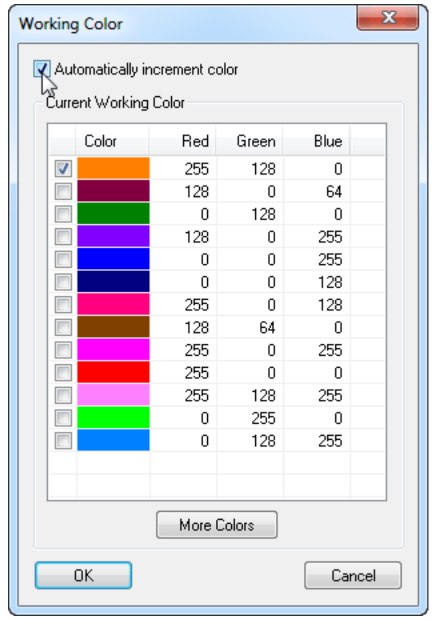


Figure 7-34. Toggling the *Automatically increment color* option in the ***Working Color*** dialog.

Successive creation of multiple objects in different colors may be a behavior that you like—or, depending on what you're doing, it may not. For instance, you may wish to create a number of nominal objects all of the same color, using the assigned color as a cue that the objects are in their nominal configuration. In this situation, the automatically incrementing working color is not desirable. You can disable working color auto-increment so that the current color remains the working color until you explicitly change it.

To Set the Working Color:

1. Do one of the following:

- In the Color/WCF toolbar, click the color swatch button , which dispays the working color, or
- From the menu, select **Edit>Change Working Color**.

2. Select a new working color, then click OK. The next object created will be assigned the working color.

> **Note:** Selecting **Edit>Change Working Color** does not allow selection from a list of existing working colors. On the other hand, selecting the toolbar button does.

To Turn Working Color Auto-Increment On or Off:

1. In the Color/WCF toolbar, click the color swatch button .
2. In the ***Working Color*** dialog (Figure 7-34), toggle the *Automatically increment color* option.

To Change the Color of One or More Objects:

1. Choose the desired working color as described above in "To Set the Working Color:" on page 241.
2. From the toolbar, click the Color Objects button , or select **Edit>Set Objects to Working Color**.
3. Select the objects to paint.

You can also change the color of an object directly through its properties.

To Change the Color of an Object Through Its Properties:

1. Right-click the object in the tree or graphical view and select **Properties** from the context menu.

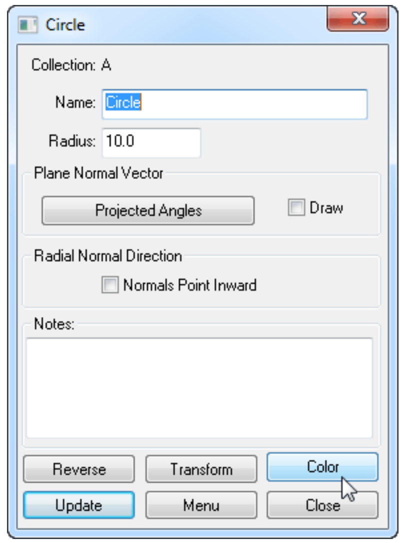


Figure 7-35. Changing the color of a circle through its properties dialog.

2. In the dialog (Figure 7-35), click the Color button. Select a new color, then click OK.

Accessing Item Properties

Nearly every item in your job has properties associated with it. Some properties are general—such as the name of the item and any notes associated with it—and other properties are more specific and depend on the object type. These more specific properties are discussed in detail later.

To Access the Properties of an Item:

Do one of the following:

- Right-click the item in the tree or graphical view and select **Properties** from the context menu, or
- From the toolbar, click the Object Properties button, then select the object to examine (applies to objects only), or
- From the menu, select **Edit>Object Properties**, then select the object to examine (applies to objects only), or
- Press **Alt+Q** (default keyboard shortcut), then select the object to examine (applies to objects only).

The Calculator

Since you may find the need to perform a quick calculation fairly frequently in SA, a shortcut is provided for calling up the Windows calculator utility.

To Start the Windows Calculator Utility:

- From the menu, select **Edit>Calculator**.

User Notes

Occasionally you may be interested in recording some field notes. You can add user notes to an SA job for later reference or for adding to a report. A user note will be added to the tree under the **Events** category, and can optionally be added to the job's log file as well.

To Add a User Note:

1. From the menu, select **Edit>Add User Note**, or press **Ctrl+N** (default keyboard shortcut).
2. In the ***Enter User Note*** field, type a summary and details for the note.
3. If you'd like the note to also be logged in the job's log file, select the *Log File* option.
4. Click OK.

Adding Notes to Items

You can also add notes and associate them with specific items. For instance, if you're measuring a cylindrical tank to check for deformation when it's filled, you might want to add a note that a best-fit cylinder was created when the tank was filled, and add the quantity of fluid in the tank for later analysis.

To Add a Note to an Item:

Note: Some items, such as polygonized surfaces, surfaces, and relationships, do not have associated notes.

1. Right-click the item in the tree or graphical view and select **Properties** from the context menu.
2. In the *Notes* field, enter the desired note, then close the dialog.

Item History

Many different items in your job have a *history* associated with them. An item's history shows significant events related to the item's creation or manipulation. For instance, all item histories indicate when the item was added to the job,. A circle's history may indicate the details of the geometry fit used to create that circle.

To View an Item's Historical Log:

- Right-click the item in the tree or graphical view and select **History** from the context menu.

To Expand or Collapse all Entries in an Item's History:

- In the item's history dialog, click the + or - button.

To Change the Chronological Order in Which an Item's History is Displaye:

- In the item's history dialog, choose the *Ascending* or *Descending* option.

To Copy an Item's History to the Clipboard:

- In the item's history dialog, select the entries to copy (or select none to copy them all), then click the Copy to Clipboard button.

To Export an Item's History to a Text File:

- In the item's history dialog, click the Export Entire Log as Text File button. Choose a filename, then click Save.

To Search an Item's Historical Log:

- In the item's history dialog, click the Search button. In the resulting ***Search*** dialog, type a string to search for, then click Find Next.

To Trim an Item's Historical Log:

- In the item's history dialog, click the Trim Log button. Confirm the action, then specify the number of log entries to keep and click the OK button.

Print Setup

Default printing options can be setup through the ***Print Setup*** dialog (Figure 7-36).

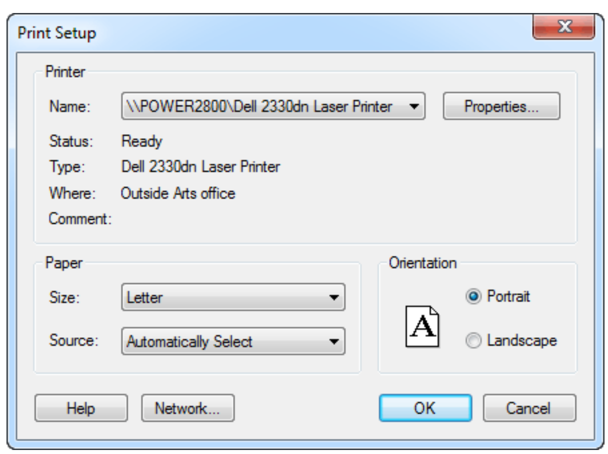


Figure 7-36. The ***Print Setup*** dialog.

This standard dialog's options depend on the selected printer but define how output from printed reports is handled by the printer. For more information, refer to your printer's documentation.

To Modify Print Settings:

1. From the menu, select **File>Print Setup...**
2. Choose the desired settings, then click OK.

Command-Line Arguments

SpatialAnalyzer has a series of command-line arguments that allow you to automate some tasks when launching the program:

Argument	Description
`/?`	Displays a help window showing all of the available command-line arguments.
`{FILENAME.XIT}`	Automatically opens the specified SA job file upon launch. Replace {filename.xit} with the path to the desired file.
`/MP {FILENAME.MP}`	Runs the specified Measurement Plan file upon launch. Replace {filename.mp} with the path to the desired measurement plan file.
`/MPE {COLLECTION}::{MP NAME}`	Runs the specified embedded Measurement Plan upon launch. Replace {COLLECTION} with the collection name, and {MP Name} with the name of the embedded MP in the SA tree.
`/QUICK {INSTRUMENT NAME}`	Launches SA, starts a new file, adds the entered instrument name, and starts its interface. Replace {Instrument Name} with the name of the instrument (as specified in the Add Instrument to SA dialog).
`-MINIMIZE`	Starts SA in a minimized window state.

Combinations of these arguments may be executed at the same time. Quotes are required around the elements (filenames, instrument names, and object names) if they contain spaces.

Launching SA with Command-Line Arguments

Command-line arguments were traditionally used when running a program from the DOS (command prompt) window. Now that graphical operating systems like Windows have become standard, command-line arguments are usually entered by use of a shortcut icon. Regardless, they can still be executed from the Windows command prompt.

To Use Command-Line Arguments from the Windows Command Prompt:

1. Windows XP: From the Windows Start menu, choose *Run*. In the dialog, type cmd and press **Enter**. Windows Vista/7: From the Windows Start menu, click in the text box labeled *Search programs and files*. In the box, type **cmd** and press **Enter**.
2. In the resulting window, navigate to the SpatialAnalyzer in-

stall directory by typing **cd {PATH}**, where **{PATH}** is the path to the SA install directory. For example, **cd C:\Program Files\New River Kinematics\SpatialAnalyzer 2011.01.24**. Press **Enter**.

3. Type **"Spatial Analyzer" {ARGUMENTS}**, where **{ARGUMENTS}** includes one or more of the arguments listed above. Press **Enter**.
4. SA should launch, using the parameters that you specified in the arguments.

To Use Command-Line Arguments via a Shortcut Icon:

1. Create a shortcut for the SpatialAnalyzer program. (See Microsoft Windows help if you are unsure how to do this).
2. Right-click on the icon and choose **Properties**.
3. In the ***Properties*** dialog, locate the *Target* field.
4. Add a space at the end of the text in this field, then add the arguments as specified above. When finished, click the OK button.
5. Double-click the icon to start SA with the specified parameters.

Example Command-Line Arguments

To:	Command
Start SA With A File	"Spatial Analyzer" "C:\MyFolder\My Template File.xit"
	"Spatial Analyzer" FileInThisFolder.xit
View the Command-Line Argument Help	"Spatial Analyzer" /?
Start an External MP in a Specified File	"Spatial Analyzer" /MP C:\MyScripts\MyMP.mp MyFile.xit
Start an Embedded MP in a Specified File	"Spatial Analyzer" /MPE Nominal::MyEmbeddedMP MyFile.xit
Add/Startup a Leica Tracker in a New File	"Spatial Analyzer" /QUICK "Leica emScon Absolute Tracker (AT901 Series)"
Add/Startup an API AXXIS Arm in a Specified File	"Spatial Analyzer" /QUICK "AXXIS 6-200 Arm (3.2m 6 dof)" myFile.xit

Multiple Instances and Exiting SA

SA does not have a "tabbed" interface. If you would like to have multiple copies of SA open on a single computer, simply open the application again.

To Exit SA:

- From the menu, select **File>Exit**, click the "X" in the top-right corner of the application window, or double-click the application icon in the top-left corner of the window.

This Page Intentionally Left Blank.

Working With Files

CHAPTER 8

While working in SA you will likely be working with many types of files—SA job (.XIT) files, CAD files containing nominal designs, and text files containing point coordinates, for example. This chapter explains how to work with external files in SpatialAnalyzer.

SA Files

Creating New Files

Creating a "new" file in SA does not necessarily mean that you're working with a blank slate, as you would be in some applications. Even a completely new file has at the very least a collection, a coordinate frame, and myriads of settings for anything from colors and units to decimal precision.

When a new file is created without the existence of a default template, SA opens a factory default file—that is, a file that has default settings hardcoded by NRK.

To Create A New File:

- From the menu, select **File>New**, or click the icon on the toolbar. If you currently have an SA file open, you will be asked if you would like to save it.

If you have an existing file open, you can restore it to the factory default settings (colors, decimal precision settings, etc).

To Revert a File's Settings to the Factory Default:

- With the file open, select **File>Restore Default Settings**. The settings will return to the defaults. You will of course need to save the file to store the changes.

Opening Files

The Open File command permits opening native SpatialAnalyzer files. These are commonly referred to as *job files* and come in one of two formats:

- **xit.** This is a standard 32-bit .xit file. It can be opened by both 32- and 64-bit versions of SA, but cannot have extremely large data sets. It is limited to storing data that will fit in a 32-bit software environment.
- **xit64.** This is a 64-bit SA file. It can only be opened in the 64-bit version of SA, but is able to store very large data sets (such as extremely dense scan sets).

SpatialAnalyzer job files are backward compatible—you can open an `.xit` or `.xit64` file saved in an older version of SA in a newer version. However, the reverse is not true—attempts to open a job file saved in a newer version of SA into an older version will fail.

To Open an Existing File:

Note: If you double-click a file in Windows Explorer, it will be opened in the most recent version of SA that has been opened.

- From the menu, select **File>Open**, or
- Click the icon on the toolbar, or
- Press the default keyboard shortcut of **`Ctrl+O`**, or
- Double-click an SA file in Windows Explorer.

To Open a Recent File:

- From the menu, select **File>[Filename]** toward the bottom of the file menu.

Saving Files

You can save a SpatialAnalyzer job file in one of two formats, as described in “Opening Files” on page 250. Any current job file can be saved in either format, however it’s important to keep a few things in mind:

- If you want to be able to open the file in either the 32-bit or 64-bit version of SA, you must save the file in the 32-bit (**`.xit`**) format.
- If the file contains a very large dataset, and you want to preserve the integrity of the data, it must be saved in the 64-bit (**`.xit64`**) format and cannot be opened in the 32-bit version of SA.
- A very large file can be saved in the 32-bit (**`.xit`**) format if you’d like, however there will likely be loss of data. You will be

warned if this occurs.

File Checksums

To protect against data integrity issues, SA can perform a checksum of your SA file immediately after a save operation. This essentially verifies that the file on disk matches the currently opened file, assuring that the save was successful.

To Verify File Checksums After Saving:

1. In the User Options dialog, navigate to the *Machine Configuration* tab.
2. Enable the *Verify File Checksums After Saving* option.

To Save an SA file:

1. From the menu, select **File>Save**, click the icon on the toolbar, or use the default keyboard shortcut (**`Ctrl+S`**).
2. In the ***Save As*** dialog, select the desired format from the *Save as type:* dropdown, then click Save.

The first time you save a file, you will be prompted to choose a name and format. Subsequent saves will simply save the file and not prompt you for a file name or format.

Save As

If you choose to save the current file under a different filename or format, you can select the **Save As** command at any time. This permits you to specify a new filename or format, even if you've already saved the current file.

To Save the File With a New Name or Format:

1. From the menu, select **File>Save As**.
2. In the ***Save As*** dialog, specify a new filename and format, then click Save.

Backing Up Files

SA can be configured to capture periodic, automatic backups of your working file which requires no user intervention. However, you can also automatically save a backup copy of your current file. This backup will be saved to the current backup folder and the filename will be appended with a timestamp.

To Force A Backup File To Be Saved:

- From the menu, select **File>Backup Now**. After a short delay, the file will be saved to the backup folder.

Automatic Backups

Losing measurement data or your job progress can be a critical and expensive problem. To mitigate this risk, you can set your machine to make automatic SA file backups while SA is running. There are two types of backups that can be made:

- **Job File Backups.** These backup files save the same information as a standard job file. (In fact, job file backups are simply regular SA files that are saved automatically for you).
- **Measurement Backups.** These backups only store instruments and measurements.

So why are there two types? The main reason is to save space. You might be working in a file with a very large complex nominal CAD file. There is simply no reason why all of this information should be saved in a backup, because the CAD surfaces are typically not modified. As a result, they mostly just waste hard disk space. Measurement backups, on the other hand, only store instrument and measurement data which is much more critical and significantly smaller in file size. As a result, frequent measurement backups can be made without wasting hard drive space or causing performance delays, whereas frequent job file backups would cause a much bigger hit to performance and available drive space. Measurement backups are also regular SA files, and can easily be re-merged into the SA file containing the nominal CAD data using the **File>Import** command.

Sometimes, you might open a file, make a few changes, and accidentally overwrite your original file, which could be disastrous without a backup. To help prevent this type of mistake, you can optionally ask SA to create a file backup automatically upon opening the file, so you have a pristine, untouched backup in place.

You can also ask SA to automatically create Measurement Plan backup files while making changes in the MP Editor. If the specified time has elapsed and changes have occurred to your MP, SA will store a backup of your script to the Backups folder.

To Enable Automatic Backups:

1. Click the User Options icon in the Main toolbar.
2. In the *Automatic Backups* section of the *Machine Configuration* tab, click the *Enabled* checkbox for the appropriate backup type.
3. Select a time interval at which enabled backups should occur.
4. If desired, enable the *Automatically Create on File Open* option to store a backup upon opening a file.

or

- To quickly toggle automatic backups without changing other backup settings, toggle the **Edit>Automatic File Backup** setting in the menus (this does not apply to MP Backups).

To Restore a Job File Backup

- Select **Edit>Undo to Restore Point** from the menus. Select a desired restore point from the list, or
- Select **File>Open** and select the desired file in your backup folder.

To Restore a Measurement Backup

- Select **File>Open** and select the desired file in your backup folder.

To Restore a Measurement Plan Backup

- In the MP Editor, select **File>Open** and select the desired .MP file in the backup folder.

Restore Points

Restore points can be thought of as "quicksaves". At any time, you can create a quick backup of your current files by creating a restore point. You can then revert back to the restore point to continue from that point. They are not a complete alternative to an undo system (which is not available in SA) as they require you to consciously create a restore point. However, they are very useful as quicksaves of your working job file.

To Create a Restore Point:

Do one of the following:

- From the menu, select **Edit>Create Restore Point**, or
- Press `Alt+Space` (the default keyboard shortcut).

To Undo to a Restore Point:

Do one of the following:

- From the menu, select **Edit>Undo to Restore Point**, or
- Press `Ctrl+Z` (the default keyboard shortcut).

Password-Protecting SA Files

Warning: The password protection scheme is not intended to be a secure method of data storage. Password protection can be circumvented by a hacker relatively easily. If you need high security, you should use a tool designed for secure data storage.

For safety purposes, you may choose to password-protect your SA file to prevent accidental or malicious modification. This involves assigning a password to the current job file, then saving it. When the file is opened in the future, it requires a password to access the data.

a Job File:

1. From the menu, select **File>Password Protection>Set Password**.

2. In the ***Set Password*** dialog, enter your password into both fields.
3. When prompted, confirm that you wish to apply a password to the file.
4. Save the file. It will now require a password to open.

To Remove a Password From a Job File:

1. From the menu, select **File>Password Protection>Remove Password**.
2. At the prompt, confirm that you would like to remove the password.
3. Save the file. A password will no longer be required to open it.

File Import

SA supports a wide variety of common file formats, making data sharing easy. You can work with various CAD formats, text-based point formats, custom formats and embedded formats. The act of importing a file will always merge it into the existing job file.

When a file is imported, the data is transformed by aligning the working frame or active coordinate system of the imported file to the working frame of the current file.

Auto Import

You can let SA choose the proper import method for a file based on its file extension. You do this using the **Auto Import (by file extension)** command. Be aware that this depends on the file extension to match the format of the file. If there is a mismatch, the import will fail.

To Auto-Import a File by Type:

1. Do one of the following:

- From the menu, select **File>Auto Import (by file extension)**, or
- In the toolbar, click the Auto Import icon, or
- Drag the file from Windows Explorer into the graphical view, or
- In the tree, switch to the *Explorer* tab, find the file to import, then double-click it.

2. The appropriate file import command will be triggered.

Importing SpatialAnalyzer Job Files

It is often desirable to merge another SA job (.xit file) into the current SA file. This is particularly useful when portions of measurement have been accomplished on various computers and files. You can choose which collections should be imported.

To Import Another SA File Into the Current Job Using the Menus:

Note: If any imported collections have the same name as existing collections, they will be automatically renamed.

1. Choose **File>Import>SpatialAnalyzer Job File (.xit)**. Choose the file to import.
2. From the ***SA File Import Filter*** dialog, check the collections you'd like to import.
3. Click Import Selected Collections.

Direct CAD Access

The **Direct CAD Access** import option allows a user to import native CAD files directly into SpatialAnalyzer. The following is a list of supported formats:

Manufacturer	Format	Version	Polygonized Surfaces	Surfaces	Annotations
Standard	PRC (.PRC)	All Versions	✓	✓	✓
	IGES (.IGS, .IGES)	5.1, 5.2, 5.3	✓	✓	N/A
	Industry Foundation Classes (.IFC, .IFCZIP)	IFC2x Editions 2, 3, & 4	✓	N/A	N/A
	STEP (.STP, .STEP)	AP 203 E1/E2, AP 214, AP 242	✓	✓	✓
	Stereo Lithography (.STL)	All Versions	✓	N/A	N/A
	VDA-FS (.VDA)	v1.0 & v2.0	✓	✓	N/A
	VRML (.WRL, .VRML)	v1.0, v2.0	✓	N/A	N/A
Adobe	Adobe 3D PDF (.PRC)	All Versions	✓	✓	✓
McNeel North America	Rhinoceros (.3DM)	v4, v5	✓	✓	N/A
Dassault Systemes	CATIA V4 (.MODEL, .SESSION, .DLV, .EXP)	Up to 4.2.5	✓	✓	✓
	CATIA V5 (.CATDRAWING, .CATPRODUCT, .CATPART, .CATSHAPE, .CGR)	R4 to R21, V5-6R2013 (R22)	✓	✓	✓
	CATIA V6 (.3DXML)	2011 to 2013	✓		✓
	SolidWorks (.SLDASM, .SLDPRT)	Up to 2014	✓	✓	
	ACIS (.SAT, .SAB)	Up to v23	✓	✓	N/A

Manufacturer	Format	Version	Polygonized Surfaces	Surfaces	Annotations
SIEMENS	NX (.PRT)	Unigraphics v11.0 to NX 9.0	✓	✓	✓
	JT (.JT)	Up to 9.5	✓	✓	✓
	Parasolid (.X_T, .X_B, .XMT, .XMT_TXT)	Up to v26	✓	✓	N/A
	Solid Edge (.ASM, .PAR, .PWD, .PSM)	V19-20, ST-ST6	✓	✓	
	I-DEAS (.MF1, .ARC, .UNV, .PKG)	Up to 13.x (NX 5) & NX 6	✓	✓	✓
PTC	Pro/ENGINEER (.ASM, .NEU, .PRT, .XAS, .XPR)	Up to Wildfire 5	✓	✓	✓
	Creo Elements/Pro/Parametric (.ASM, .NEU, .PRT, .XAS, .XPR)	v5.0 (Pro) v2.0 (Parametric)	✓	✓	✓
Autodesk	Inventor (.IPT, .IAM)	Up to 2014		✓	N/A

To Natively Import a CAD File:

1. From the menu, select **File>Import>Direct CAD Access**.
2. Select the file you'd like to import.
3. In the ***Direct CAD Access*** dialog (if you've chosen to show this dialog), choose the desired import options.
4. If the *Selective Import* option is checked in the ***Direct CAD Access*** dialog, the ***Choose CAD Features*** dialog will be shown (Figure 8-1). Choose the desired features to import, then click OK. For more on this dialog, see "Selective CAD Import" on page 257.

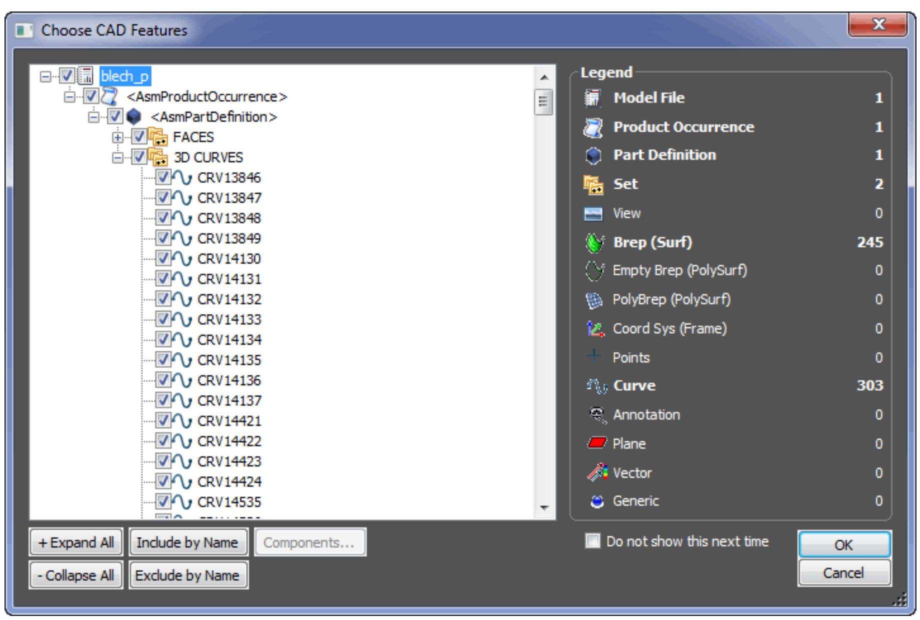


Figure 8-1. The selective CAD import dialog.

Selective CAD Import

The ***Choose CAD Features*** dialog (Figure 8-1) is only displayed when the *Selective Import (Choose which Components to Import)* option in the ***Direct CAD Access Settings*** dialog is enabled (see "Direct CAD Access Settings" on page 257). It lists all components of the CAD model or assembly in a tree-like view on the left, and a legend at right indicates the type and quantity of all of the entities to be imported. In this dialog you can choose to deselect entities from the source CAD file that you do not wish to import. This provides several benefits:

- It saves you the effort of having to clean up the CAD file of unwanted items after import.
- It makes the import process shorter.
- It allows you to circumvent portions of a CAD file that may be causing import errors.

Direct CAD Access Settings

The Direct CAD Access Settings control a number of options related to import via Direct CAD Access.

To Open the Direct CAD Access Settings:

- From the menu, select **File>Import>Direct CAD Access Settings**, or
- From the User Options *Machine Configuration* tab, click the Direct CAD Access Settings button.

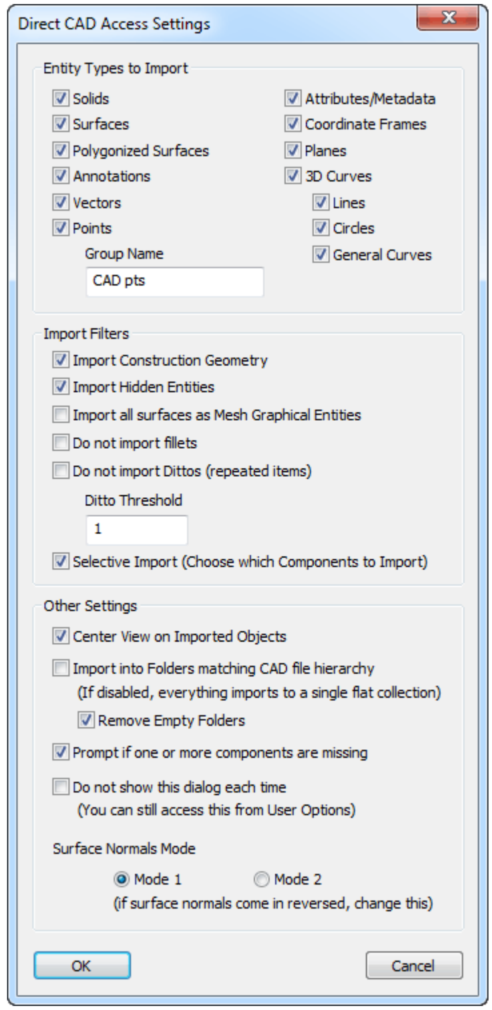


Figure 8-2. The ***Direct CAD Access Settings*** dialog.

The options for this dialog are explained below.

- **Entity Types to Import.** These checkboxes give you control over which types of entities to import. Any unchecked entities will not be imported.
- **Group Name.** All points from the CAD file will be imported into a group with this name.
- **Import Construction Geometry.** For CAD formats that support it, unchecking this box will exclude construction geometry from the import.
- **Import Hidden Entities.** For CAD formats that support it, unchecking this box will exclude hidden entities from the import.
- **Import all surfaces as Mesh Graphical Entities.** When checked, surfaces from the CAD file are imported as Mesh Graphical Entities instead of true surfaces. Mesh Graphical Entities are useful for visualizing surfaces and have a significantly lower requirement for computing resources than surfaces, but you cannot analyze against them.

- **Do not import fillets.** Usually, CAD surfaces contain minor features that may not be important to a metrology inspection. Fillets or modeled weld beads are one example of this. To keep your imported model as efficient as possible, check this option to exclude fillets from the import (if the file type supports marking the features as such).
- **Do not import Dittos (repeated items).** Some file formats support "dittos", which are efficient memory copies of surfaces that are repeated over and over in a CAD file. For instance, hardware such as bolts, nuts, and screws are often modeled as dittos to save memory in the CAD application. These are features that are rarely inspected by metrology—check this option to exclude them from the import.
- **Ditto Threshold.** Some formats support multiple levels of dittos (dittos of dittos, for instance). If necessary, you can modify this number to indicate the threshold below which dittos are not imported. Higher numbers import less dittos.
- **Selective Import (Choose which Components to Import).** When enabled, the ***Choose CAD Features*** dialog is displayed to allow you to more finely control which entities are imported from the CAD file (see "Selective CAD Import" on page 257).
- **Center View on Imported Objects.** When enabled, the graphical view will be centered on the imported objects after import.

> **Tip:** Importing while matching CAD hierarchy can result in a somewhat cumbersome, cluttered SA job file. Use this option sparingly, and only when needed.

- **Import Into Folders Matching CAD File Hierarchy.** When enabled, if the source CAD format supports a hierarchy, that hierarchy will be preserved in the SA tree using a hierarchy of folders and collections.
- **Remove Empty Folders.** Importing a CAD file into folders maching the CAD file hierarchy (see previous option) can sometimes result in empty folders in the tree. Enabling this option removes these empty folders after the import operation.
- **Prompt if One or More Components Are Missing.** Several CAD formats support the idea of a product or assembly file, which is a master file that has several supporting component CAD files. When importing these master files, this option will warn you if any subcomponent CAD files that are part of the master assembly are not found.
- **Do Not Show This Dialog Each Time.** When enabled, the ***Direct CAD Access Settings*** dialog will not display on import, but will use the previous settings. If you want to access the settings again, follow the steps in "To Create A New File:" on page 249.
- **Surface Normals Mode.** This setting has two possible options:

Mode 1 or Mode 2. Some CAD formats or CAD packages do not consistently define or store the direction of surface normals. Incorrectly oriented CAD surface normals will cause incorrect analysis in SA. While surface normals can always be corrected manually, if you find that one mode is importing surfaces in the wrong direction, trying Mode 2 might solve the problem.

Importing Standard CAD Formats

SA can import several standard CAD formats using a "classic" importer. These formats include:

- **STEP.** Import via **File>Import>Standard CAD Formats>STEP**.
- **IGES.** Import via **File>Import>Standard CAD Formats>IGES**.
- **VDA.** Import via **File>Import>Standard CAD Formats>VDA**.
- **SAT.** Import via **File>Import>Standard CAD Formats>SAT**.

In addition to providing additional import formats, you can also use this classic import method if you encounter problems attempting to import using Direct CAD Access. During the CAD import you can select from a variety of import options (Figure 8-3):

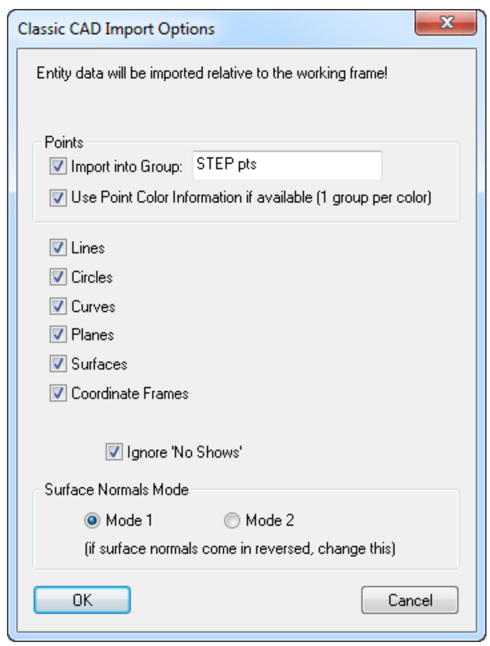


Figure 8-3. The ***Classic CAD Import Options*** dialog.

- **Import into Group.** Points from the CAD file will be imported into a group with the specified name.
- **Use Point Color Information if available (1 group per color).** Upon importing points from the CAD file, this option will set the group color to match the current point color from the source file, if that color exists. Each point will be organized into a group by its color.

- **Lines/Circles/Curves/Planes/Surfaces/Coordinate Frames.** Choose which types of entities to import.
- **Ignore 'No Shows'.** When selected, items hidden or on a "no show" layer in the file (if supported) will not be imported.
- **Surface Normals Mode.** Equivalent to the same option in the ***Direct CAD Access Settings*** dialog (see "Direct CAD Access Settings" on page 257).

To Import a Standard CAD Format Using the Classic Importer:

1. Choose **File>Import>Classic CAD Importers>(Format)**.
2. Select the file to import from the dialog, and click OK.
3. Choose the desired CAD import options, then click OK.
4. A summary of the file information will be displayed. To save it to file, click Save to File. Otherwise, click Accept.
5. If the current file's units do not match the source units, you will be prompted whether you'd like to convert the units. Click Yes or No.

Note: If units between files differ, the units are converted (if you choose). The CAD data is not physically scaled at all.

Importing Predefined ASCII Formatted Files

ASCII text files are common in coordinate metrology. They provide a standard, human-readable format that many software packages can easily import. SpatialAnalyzer supports many formats of ASCII data. Figure 8-1 shows an example of a file in "Point Name, X, Y, Z" format.

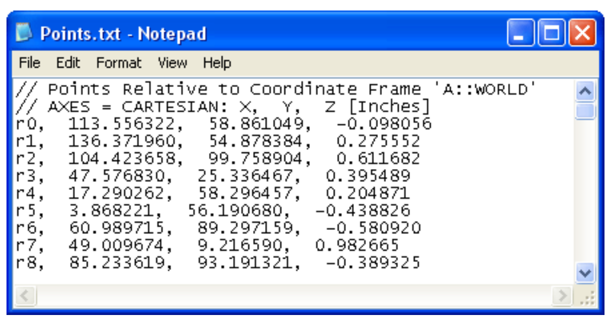


Figure 8-4. A text file containing point data in `Point-name, X, Y, Z` format.

When a file is selected, you will be presented with several options to define how SA will import the file (Figure 8-5):

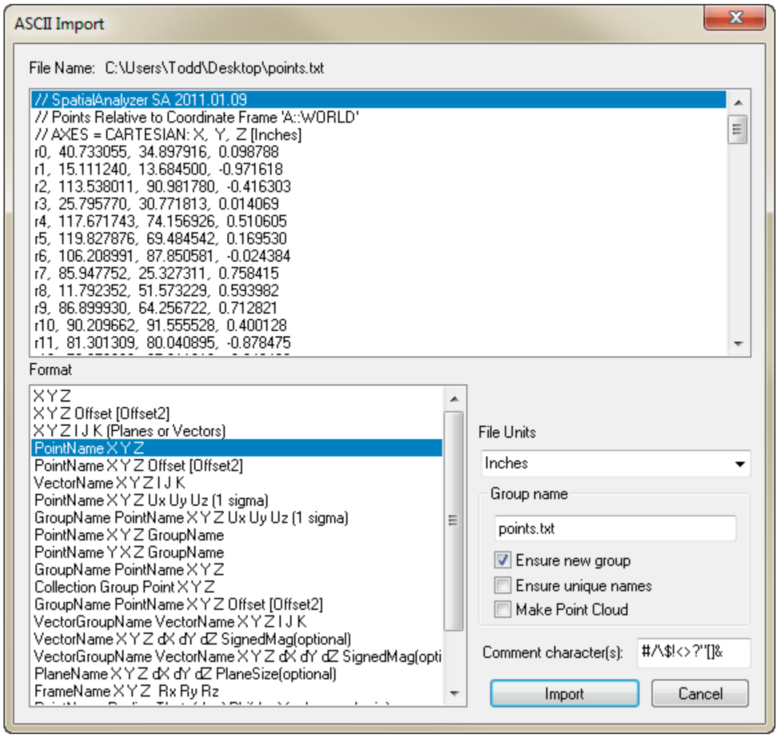


Figure 8-5. The ASCII Import dialog.

- **Format.** Choose the format of the ASCII file. SA will automatically delimit using commas, tabs, or spaces (depending on how the ASCII file is set up).
- **File Units.** Define the units used in the ASCII file.
- **Group Name.** Specify the name to import the points into (only applicable if not specified already by the format).
- **Ensure new group.** Select this option to ensure that points are placed into a new point group if the specified group already exists.
- **Ensure unique names.** Verifies that imported point names are unique. If there is a naming conflict, the offending point will have an asterisk appended.
- **Make Point Cloud.** Select this option to import points as a point cloud instead of a point group.
- **Comment character(s).** Any lines that start with the specified characters are treated as comment lines and are ignored.

To Import an ASCII-Formatted File:

1. Select **File>Import>ASCII: Predefined Formats**.
2. Choose the file to import.
3. In the ***ASCII Import*** dialog, specify the desired options, then click Import.
4. The file will be imported, and you will receive a summary of

lines imported and ignored.

Importing ASCII Formatted Points With Offsets

There are two additional ASCII formats for importing points with offsets, depending on the format of your source data.

To Import Points With Offsets:

- From the menu, select **File>Import>ASCII: Points with Offsets><Format>**.

<Name><X><Y><Z><Offset>

This option is used when the source file is in the format **`<Name><X><Y><Z><Offset>`**.

2 Files: <Name><X><Y><Z> + <Name><Offset>

This option is used when there are two source files: One in the format **`<Name><X><Y><Z>`** and the other in the format **`<Name><Offset>`**.

Importing XML Files

XML files containing nominal points and coordinates can be imported into an SA file. At a later time, measured point data can be merged back into the XML file, which will associate the actual coordinates with the nominal coordinates. The nominal points will be imported into a group matching the name of the XML file.

XML Format: Nominal Points

The format of the XML file should have the standard XML format, such as the introductory tag:

```
<?xml version=”1.0” encoding=”UTF-8” ?>
```

Following this, an enclosing group may contain one or more **`<POINT>`** elements, containing a **`NAME`** attribute and a child **`<NOMINAL>`** element which itself contains **`X`**, **`Y`**, and **`Z`** attributes. For example:

```
<?xml version=”1.0” encoding=”UTF-8” ?>
  <left>
    <POINT NAME=”Center1”>
      <NOMINAL X=”1.000” Y=”4.000” Z=”7.000” />
    </POINT>
    <POINT NAME=”Center2”>
      <NOMINAL X=”3.000” Y=”6.000” Z=”9.000” />
    </POINT>
  </left>
```

XML Format: Nominal and Measured Points

Once measured data has been obtained, the measured point names are matched up with the nominal names in the XML file, and an **`<AC-`**

TUAL> element is added as a child of each matching **<POINT>** element. This **<ACTUAL>** element contains **X**, **Y**, and **Z** attributes holding the measured coordinates. The above XML file might look like the following after merging measured data:

```
<?xml version="1.0" encoding="UTF-8" ?>
  <left>
    <POINT NAME="Center1">
      <NOMINAL X="1.000" Y="4.000" Z="7.000" />
      <ACTUAL X="1.002" Y="3.996" Z="7.003" />
    </POINT>
    <POINT NAME="Center2">
      <NOMINAL X="3.000" Y="6.000" Z="9.000" />
      <ACTUAL X="3.007" Y="6.005" Z="8.998" />
    </POINT>
  </left>
```

To Import an XML File:

1. From the menu, select **File>Import>XML**, then select the desired XML file.
2. In the XML import dialog, the structure of the points to import will be listed (Figure 8-6). Click the Import Points button.

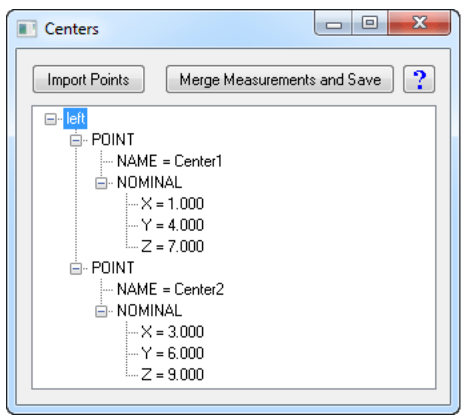


Figure 8-6. The XML import dialog for the above example file.

To Merge Measurements to an XML File:

1. From the menu, select **File>Import>XML**, then select the desired XML file.
2. In the XML import dialog (Figure 8-6), click the Merge Measurements and Save button.
3. From the ***Object Selection*** dialog, select the point group you'd like to merge.

Custom Formats

Many custom file formats can be imported, including the following:

- **xyz, ijk Files (IJK).** Files in IJK format (containing both position and probe orientation data).

- **Digital Network Level Measurements.** Measurements from a digital level. This is an ASCII file expected to be of the format `<group>`, `<target>`, `<elevation>`.
- **VSTARS .xyz Files.** .xyz files from VSTARS.
- **VSTARS Cameras.** Camera files from VSTARS in the outstar.txt format.
- **DMIS Files.**
- **Polyworks Files.** .POL, .PIF, .PF, and .DPI formats are all supported.
- **IMETRIC Files.**
- **Leica GSI8/GSI16 Files.** Files from Leica total stations in GSI8 or GSI16 format.
- **Steinbichler AC Files.** This imports data from a Steinbichler scanner (such as the T-Scan) originating in .AC format.
- **Wavefront Objects Files.** This imports data in Wavefront Technologies' Objects file (.obj).
- **Leica ADF format.** Files in Leica's .adf format.
- **1-D Data (DataMyte).**

To Import a Custom File Format:

- From the menu, select **File>Import>Custom Formats>...**

Embedding Files

You can import any file from the file system and embed it in the SA job file. The embedded file is placed into the active collection. This is convenient for transporting relevant files with the job file. Of course, when doing so the SA job file grows accordingly to store the file. At any time, this file can be exported back to the file system.

To Embed a File:

1. Activate the collection into which you'd like to embed the file.
2. From the menu, select **File>Import>File as Embedded File**.
3. At the prompt, select the file to embed. The file will appear under the *Embedded Files* category of the tree.

Importing Images

You can import any common image format into the SA job. In the tree, embedded images appear under the *Pictures* category in the active collection. Importing images is particularly useful because these images can be used in reports. For example, you might have a few JPEG images of a measured part taken with a digital camera that you'd like to include in your report. Importing these images into the SA file gives you the ability to use them in a report.

To Import an Image Into an SA File:

1. Ensure the collection into which you'd like to import the image is active.
2. From the menu, select **File>Import>Image**.
3. At the prompt, select the image file to import. In the tree, the image will appear in the *Pictures* category of the active collection.

File Export

There are many ways to export your data from SA. Points, Point Clouds, Vector groups and many CAD formats are supported under the **File>Export** menu option.

Exporting ASCII Formatted Points

Points are easily exported from SA to an external ASCII text file. Using this method, the points are always exported in the working coordinate frame. There are many export options to give you fine control over the export format and data (Figure 8-7). These options are summarized below:

- **Target Name.** The name of each exported target can be one of four options: *Collection Group Target*, *Group Target*, *Target*, or *None*. These options indicate which parts of the point/target name are included. For example, exporting using *Collection Group Target* exports in the format `CollectionName GroupName TargetName x y z`.
- **Coordinate System.** Point coordinates can be exported in Cartesian, Cylindric, or Spherical coordinates.
- **Tolerance.** If selected, point tolerances will be exported for each point (if available).
- **Coordinate uncertainties.** If selected, coordinate uncertainties for each point will be exported (if available).
- **Time Stamps.** If selected, the point's time stamp will be exported.
- **Target Offsets.** If selected, the planar and radial offset of each point will be exported.

- **Target Comments.** If selected, any comments stored with the point will be exported.
- **Measurement Details.** If selected, measurement details (such as RMS error, measurement mode, and sampling method) will be exported, if available.
- **Separator.** Data can be delimited by spaces or commas in the exported format.
- **Decimal Precision.** Numerical values can be exported using fixed decimal precision (you specify the number of digits) or using full precision scientific notation.
- **Include SA Version and Frame Comments.** If selected, the SA version and working coordinate frame name will be listed at the beginning of the file, preceded by **//** to denote that the line is a comment.
- **Include Axis Comments.** If selected, information about the exported coordinate system will be listed at the beginning of the file, preceded by **//** to denote that the line is a comment.

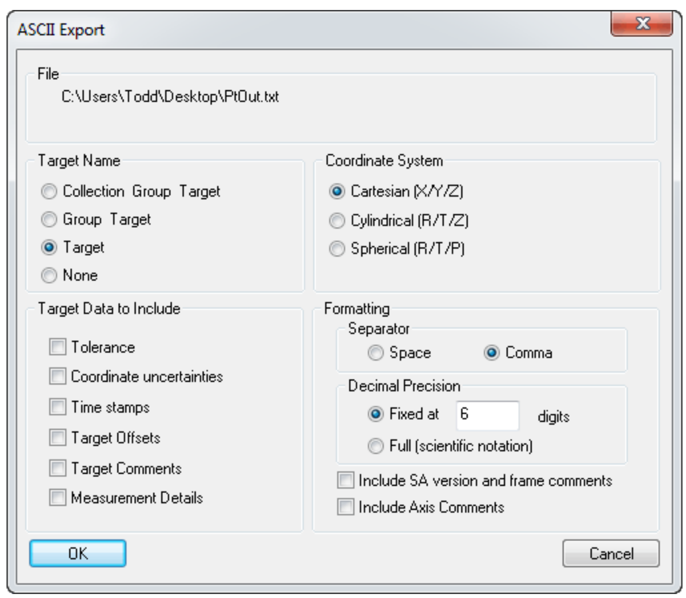


Figure 8-7. The options for exporting ASCII points.

To Export Selected Points:

1. From the menu, select **File>Export>Points (ASCII File)**.
2. When prompted, select the desired points to export, then press **Enter**.
3. In the ***Save As*** dialog, specify the name for the exported file, then click Save.
4. When the ***ASCII Export*** dialog appears, choose the desired options, then click OK. For convenience, the resulting exported file will be opened in Notepad.

To Export an Entire Point Group:

1. In the tree, right-click the desired point group to export, then select **Export to ASCII File** from the context menu.
2. In the *Save As* dialog, specify the name for the exported file, then click Save.
3. When the ***ASCII Export*** dialog appears, choose the desired options, then click OK. For convenience, the resulting exported file will be opened in Notepad.

Exporting Point Clouds

You can export point clouds to an external ASCII file. You can specify whether each point is assigned a label in the exported file.

To Export One or More Point Clouds to an ASCII File:

1. From the menu, select **File>Export>Point Clouds**.
2. Select one or more clouds to export.
3. In the *Save As* dialog, specify the filename to export, then click Save.
4. When prompted, indicate whether you would like each cloud point to have a label in the exported file.

To Export a Single Point Cloud to an ASCII File:

1. Right-click any point cloud in the tree and select **Export to ASCII File**.
2. In the *Save As* dialog, specify the filename to export, then click Save.
3. When prompted, indicate whether you would like each cloud point to have a label in the exported file.

Exporting Vector Groups

Vector groups can be exported to an ASCII file. The file is exported in the following format:

- `Name, X, Y, Z, dX, dY, dZ, Signed Magnitude`

To Export a Vector Group to an ASCII File:

1. From the menu, select **File>Export>Vector Group**.
2. At the prompt, select the vector group to export.
3. In the save dialog, specify the name for the ASCII file to export. For convenience, the exported file will be opened for you in Notepad.

Exporting Object Frames

The coordinate frames for any objects that have representable frames (such as coordinate frames, circles, and spheres) can be exported in one of two formats:

- `Name, X, Y, Z, Rx, Ry, Rz`
- `Name, r0c0, r0c1, r0c2, r0c3, r1c0...`

The first option exports the name of the objects and their positions and orientations in space. The second exports the name of the objects and the 16 elements of their transformation matrix.

To Export Object Frames:

1. From the menu, select **File>Export>Frames (and objects' frames)**.
2. Select all of the objects defining the frames you'd like to export.
3. In the ***Frame Export Settings*** dialog, select the desired format for the exported file, then click OK.
4. In the ***Save As*** dialog, specify the name for the exported file. Notice that this file is automatically exported as a .CSV file. For convenience, the file is automatically opened after successful export.

Exporting to CAD Formats

SA jobs can be exported to several different formats, with support for points, geometry and surfaces. Only the entity types supported by the selected file format will be exported. The supported formats are:

- **STEP.** You can export the entire SA file or specific objects.
- **IGES.** You can export the entire SA file or specific objects.
- **VDA.** You can export the entire SA file or specific objects.
- **SAT.**
- **DXF.**
- **Metric Vision Instrument Compensation Data Format.**
- **Imageware Cloud File.**
- **Surface Wireframe Curve File(s) (.crv).** Writes a wireframe representation of the selected surfaces to a text file containing comma-separated values.
- **Polyworks POL File.**
- **STL.** Files can be exported in either ASCII or binary format.

To Export an SA File to a CAD Format:

1. From the menu, select **File>Export>[Format]**. If supported, specify whether the entire model or specific objects should be exported.
2. If applicable, select the objects you'd like to export.
3. When prompted, specify the name of the file to export.

Exporting Window Placements

When automating SA using Measurement Plans or the SDK, it is sometimes desirable to preserve the placement and size of SA windows and dialogs. Consider an example in which a number of client computers running at a specified screen resolution will be using a series of automation routines. In order to ensure that these measurement computers have all of the windows and dialogs in the exact same place as the "development computer", you can export window placements on the development computer, then have the automation routine import these placements on the client computers using the **Import SA Windows Placement** MP command. This will ensure that the client computers position and size the windows exactly the same as the development computers.

To Export Window Placements:

1. From the menu, select **File>Export>Window Placements**.
2. Specify a name for the exported file. The file will be exported as a `.POS` file.

Templates

When you create a new file or start up SpatialAnalyzer, it opens in a "factory default" configuration. However, you might wish to specify your own read-only configurations for one or more files that are used as starting points for your work. Or, you may wish to create a configuration that is used as the default starting point for your job when starting up SA or creating a new default file. For instance, you likely have a preferred set of default units, decimal precision settings, colors, and other settings you wish to use. Setting these manually each time you start a new job would be tedious.

To satisfy this need, SA employs *templates*. A template is not very different from a regular job file. In fact, it *is* a job file. The only difference is that it is marked as read-only and is typically stored in a standard folder on disk (`C:\Analyzer Data\Templates`), although this is not a requirement.

To Create a New Template File:

> **Note:** Template files are typically stored in `C:\Analyzer Data\Templates`, although this is not mandatory.

1. Configure a job file as desired.
2. From the menu, select **File>Save As Read-Only Template**.
3. In the dialog, specify the name and location for the template file. It will be set as read-only.

To Open an Existing Template File:

1. From the menu, select **File>Open Template File**.
2. From the dialog, choose the file to open.

The Default Template

Instead of having templates as starting points for a specific job, you may also have a preferred default starting configuration for all files. SpatialAnalyzer provides the capability to define a *default template*. A default template is a template that is automatically loaded in the following situations:

- SpatialAnalyzer is started from the Windows start menu or a desktop shortcut.
- **File>New** is selected from the menu, or the New File icon is selected in the toolbar.

Default templates help you avoid repeating the same job file preparation steps by allowing you to define a starting point for all of your job files. They also provide a convenient method for standardizing file setup and settings across an organization. If a default template does not exist, SA will use factory default settings.

To Create a Default Template

1. Configure an SA file with the settings for your template. Remember to set all settings that are saved with an SA file (such as the active units and decimal precision settings) as desired.
2. From the menu, choose **File>Save As Read-Only Template**.
3. In the save dialog, ensure you are in the `Analyzer Data\Templates` folder (SA will take you there by default), then save the template as `default.xit`. When you start SA or a new file, this file will automatically be restored.

To Remove a Default Template

1. Navigate to SA's `Analyzer Data\Templates` folder in Windows Explorer.
2. Delete the `default.xit` file. Without this file, the factory default settings will be used.

Screen & Graphics Capture

At any time, you can capture either the entire screen or the graphical view for printing or output to a standard image file (BMP, JPG, PNG, GIF, or TIFF).

To Print or Capture the Screen or Graphical View:

1. From the menu, select **File>Capture Screen[Capture Graphics]>To Printer[To File (BMP/JPG/PNG/GIF/TIFF)]** as appropriate.
2. If capturing to printer, choose the printing options and confirm. If capturing to file, select the filename and type, then click Save.
3. The **Capture Graphics > To File (BMP/JPG/PNG/GIF/TIFF)** command allows you to specify a render scale. A value of **1.0** uses the actual pixel count of the graphical view for the resulting image. A value of **3.0** will create an image three times the resolution of the graphical view.

CHAPTER 9

Points

Points are the most fundamental aspect of any measurement. They are the basis from which virtually all analysis is performed in portable metrology—therefore having a good understanding of how they work in SpatialAnalyzer is crucial.

About Points

A point is defined as a specific location in three-dimensional space, and may be represented in Cartesian (X/Y/Z), Spherical (r/θ/Φ), or Cylindrical (r/θ/Z) coordinates (depending on the application). With a few exceptions coordinates are reported to the user in the working coordinate frame.

Points can be created by many methods. For example, they can be created from a point construction command or can be the result of measurement with a portable metrology device. They may be imported from a CAD file, extracted from sphere scans, or the result of a fit to existing data.

Point Groups

In SA, all points must exist inside of a *point group*. A point group is a container for a set of points that are related in some way. The point group itself is considered an object, whereas its contained points are not objects. All points inside of a point group are treated as a rigid body. That is, when you move or rotate a point group, all of the enclosed point groups move or rotate together, as if rigidly connected.

The point group is the organizational element that is used to organize your point data by feature. All measurements of a specific hole should typically be placed into their own point group. All measurements of reference (tie-in) points should typically be placed into their own group. Likewise, measurements of a complex surface should typically be placed into their own group as well. As mentioned earlier in this manual, organizing your point data in this manner will make analysis and reporting easier later on.

Since point groups are objects and points themselves are not, only point groups have a color. All points inside that group inherit the color of their point group.

There are two types of points that you will see in SA: *Constructed Points* and *Measured Points* (also referred to as *Targets*). Both of these types of points may appear in the same point group.

Constructed Points

A *constructed point* is a point that is not associated with any instrument observations or measurements. There are many ways to create constructed points. They may have been created manually by entering coordinates, by intersecting a line with a plane, or may be the result of importing data from an ASCII file. A constructed point may also result from deleting a target's individual measurement observations.

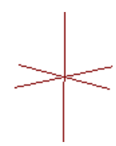


Figure 9-1. A constructed point as seen in the graphical view.

A constructed point is represented in the graphical view and the SA tree by a 3-D "plus" symbol, as pictured in Figure 9-1 and Figure 9-2.

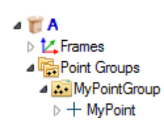


Figure 9-2. A constructed point in the tree named **`MyPoint`** sits inside the **`My-PointGroup`** group.

Expanding the triangle next to any point will display its coordinates in the working frame.

Measured Points

A *measured point* (also referred to as a *target*) is a point which has been determined from one or more direct instrument observations. For traceability purposes, targets they maintain a record about who and what measured them, when and how they were measured, weather data, sampling statistics, raw measurement data from the instrument, and any other pertinent metadata. Because they are tied to instrument measurements, measured points cannot be created through typical point construction or analysis methods--they must be measured. (There is, however, a way to create targets through measurement simulation).

Measured points are represented in the graphical view by a 3-D plus symbol with a diamond around it (Figure 9-3).

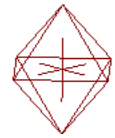


Figure 9-3. A measured point as it appears in the graphical view.

In the tree, targets are represented by a crosshair icon (Figure 9-4).

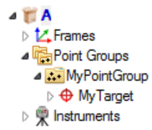


Figure 9-4. A measured target named **`MyTarget`** as it appears in the tree.

Expanding the triangle next to any point will display its coordinates in the working frame.

Point Characteristics

There are a number of characteristics common to both constructed and measured points. The ***Point (Target) Information*** dialog, shown in Figure 9-5, contains the properties of an individual point. In this particular figure, a constructed point's properties are shown.

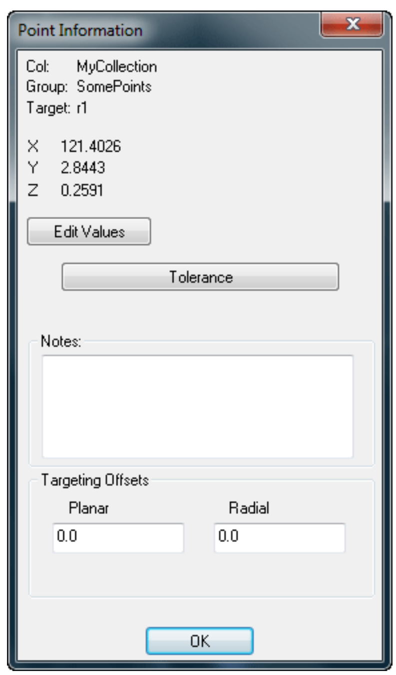


Figure 9-5. The ***Point Information*** dialog for a constructed point.

To Display the Properties for a Point:

- Double-click a point in the graphical view or tree, or
- Right-click a point in the graphical view or tree and select **Properties**.

At the top of the ***Point Information*** dialog, the collection, group, and target name are listed. Below that are the coordinates of the point in the working coordinate frame.

To Edit the Coordinates of a Point:

1. Click the Edit Values button. The ***Edit Point Coordinates*** dialog will appear as shown in Figure 9-6.

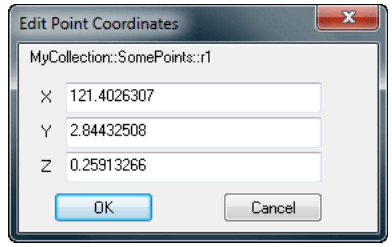


Figure 9-6. The ***Edit Point Coordinates*** dialog.

2. Type in the new values and click OK.

If you attempt to edit the coordinates of a measured point (something you typically should not do), you will receive a warning and the measured point will turn into a constructed point. The original measurement data is still preserved with the point, but the measurements become deactivated and they need to be reactivated (and the target recomputed) to restore the measured point.

Point Tolerance

Each point can be assigned a tolerance (Figure 9-7). Tolerances that have been assigned to a point will be used for watch windows and other comparisons where applicable. A point tolerance must be associated with a specific coordinate frame (because it may be asymmetrical). Tolerances can be applied on any combination of axes, and in any combination of sides (high or low).

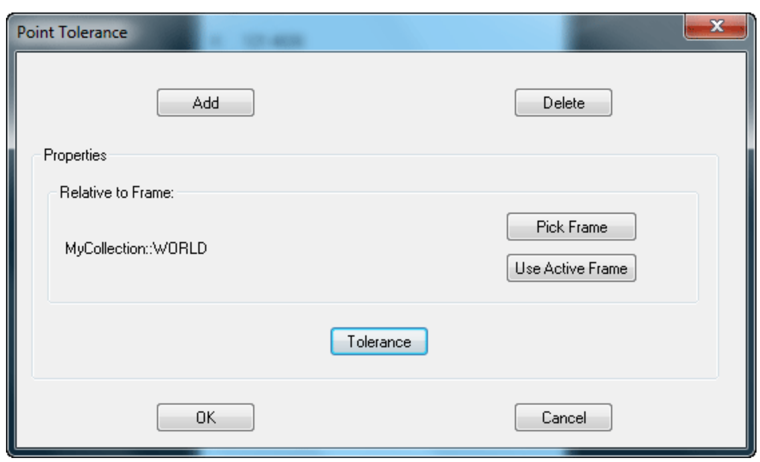


Figure 9-7. The ***Point Tolerance*** dialog allows you to specify the tolerance for a point.

To Add a Tolerance to a Point:

1. In the ***Point (Target) Information*** dialog, click the Tolerance button, or right-click the point in the tree or view and choose

Tolerance.

2. In the resulting ***Point Tolerance*** dialog, click the Add button, as shown in Figure 9-7.

3. To pick the applicable coordinate frame from a list, click the Pick Frame button and choose the frame from the resulting dialog. To specify that the tolerance will apply to the current working frame, click the Use Active Frame button.

4. To assign the numeric tolerance value, click the Tolerance button, which will open the ***Vector Tolerance*** dialog (Figure 9-8).

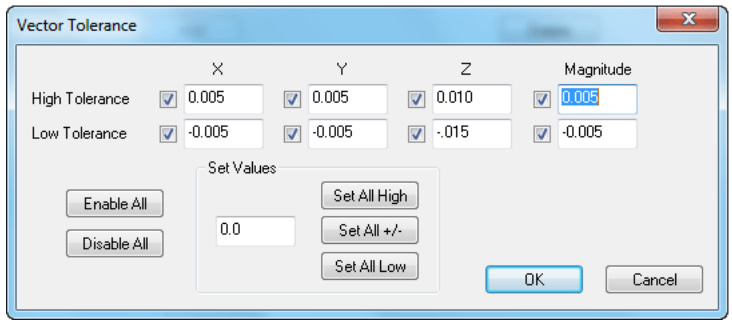


Figure 9-8. The Vector Tolerance dialog.

5. Check the tolerances that you want to apply, and enter the numeric values. The Enable All and Disable All buttons are shortcuts to check and uncheck all tolerances. If a value is entered into the *Set Values* field, then clicking the Set All High, Set All +/-, or Set All Low buttons apply the value to all of the high tolerances, all of the high and low tolerances, or all of the low tolerances, respectively.

6. Click OK to apply the tolerance.

Notes

The *Notes* field in the ***Point (Target) Information*** dialog can be used to enter descriptive notes about an individual point.

Targeting Offsets

Each point can be assigned a planar and radial target offset. Enter the values in these fields to change them for a single point. These offsets will be automatically compensated for when comparing the point to surfaces and other geometry. By default, constructed points have offset values of zero.

Planar Offset

The planar offset is the most commonly-used offset. It is used in almost all situations. To provide examples, the planar offset is used when compensating in the planar direction for a pin nest (Figure 9-9), when comparing to a cylinder or best-fitting a cylinder to measured points, or when comparing a set of measured points to a surface.

Radial Offset

The radial offset is only used in one situation: when using tooling intended to measure circular and elliptical holes that has a radial offset associated with it—that is, when using a pin nest. In Figure 9-9, the radius of the pin nest's shank determines how far outward or inward the circle radius or ellipse axis must be offset.

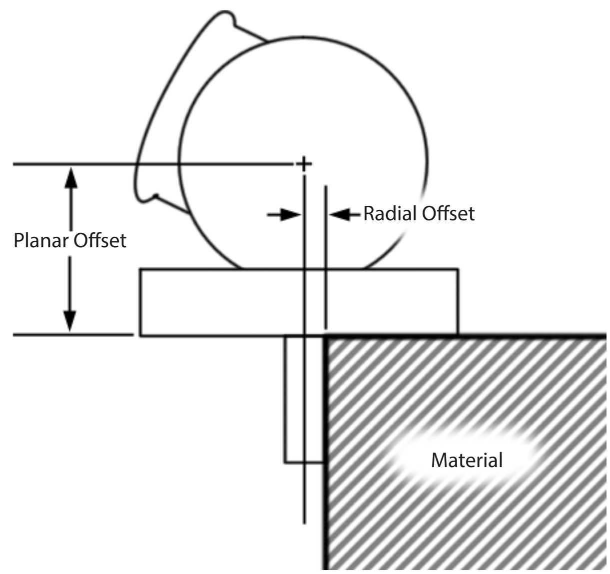


Figure 9-9. Measuring a surface hole with a laser tracker's SMR and Pin Nest.

Spherical probes (such as SMRs and balls) have the same planar and radial offset, although technically the radial offset is not used.

The following are some sample measurement instrument/tooling combinations, and the proper planar and radial offsets associated with each:

Measurement Tooling	Planar Offset	Radial Offset
PCMM arm with 6 mm probe	3 mm (0.1181")	3 mm (0.1181")
PCMM arm with 3 mm probe	1.5 mm (0.0591")	1.5 mm (0.0591")
PCMM arm with Point Probe	0	0
Tracker with 1.5" (38.1 mm) SMR	0.75" (19.05 mm)	0.75" (19.05 mm)
Tracker with 1.5" SMR and Flat Nest[1]	1.0" (25.4 mm)	N/A (0)
Tracker with 1.5" SMR and Pin Nest	1.0" (25.4 mm)	0.125" (3.175 mm)
Tracker with 1.5" SMR and Edge Nest	1.0" (25.4 mm)	0
Laser Radar (non-contact measurement)	0	0

1 When a flat nest is used, a feature such as a hole cannot be measured because there is no tooling feature to properly locate the SMR radially. As a result, there is no valid radial offset for this tooling combination.

Accessing A Point's Group Menu From the Graphical View

Sometimes, you want to select a point and access the properties of its enclosing point group. For instance, to change the color of a point you would need to access its parent point group. The graphical view's context menu allows you to do just that.

To Access a Point's Enclosing Group Menu:

- In the graphical view, right-click a point and select **Group Menu** from the context menu. The enclosing group's context menu will be displayed.

Measurements

Measured points carry a number of additional properties that are important to understand.

How SA Computes a Measured Point's Location

Determining the location of a measured point at first seems trivial. However, it gets significantly more complex when you realize that some instruments (such as theodolites) are angle-only measurement devices, and do not measure a target's position in space. With theodolites, at least two measurements of a target need to be recorded from different points in space to triangulate the location of the point.

Another complication is that one or more instruments may measure a single target several times. We will discuss these situations, but let's first look at how SA computes a measured point's location in a simple case, when an instrument provides a point position in space.

When measurements are recorded, SA always records the raw measurement data that came from the instrument. For theodolites, this means a horizontal angle (azimuth) and vertical angle (elevation). For other line-of-sight devices, it means azimuth, elevation, and distance, and for non-spherical measurement devices (such as portable arms) the coordinates are provided from the instrument in Cartesian space.

The instrument always provides the location of the measurement relative to its own internal reference system—typically at the optical center or at the base of an instrument. Since each instrument has a three-dimensional position and orientation in the job file, SA takes these raw measurements and calculates their coordinates in job space based on the position of the instrument. When an instrument moves, the position of all of its measurements are recalculated.

At any time, you can move a point group containing measured points. When you do so, you will break the connection between the measured points and their instrument, and the measured points will instantly turn into constructed points to show that you are no longer looking at actual measurements. You haven't lost the original measurement data, however. You can recalculate these measured points and turn them back into measurements.

To Recalculate Measured Points:

- From the menu, select **Analysis>Re-Compute Targets From Shots**. This will snap all measurements back into place relative to the instrument and convert them back into measured points.

About Target Observations

An observation is a single reported measurement from an instrument. A target, by definition, must have one or more instrument observations. While the coordinates for a constructed point are easy to understand and unambiguous, the coordinates for a target take more explanation. It is fairly common, particularly with theodolite measurements, for a given target to have multiple observations that together define the coordinate of the point. Consider, for instance, the case where three theodolites have measured a common point (see Figure 9-10 on page 281). Since each measurement has finite error (due to encoder error, operator differences, alignment, and other factors), the observations will not coincide in space. Put another way, each instrument will "see" a slightly different location when measuring the same point in space. (The error in Figure 9-10 is greatly exaggerated for clarity).

It would be easy to simply take an average of each observation to determine the computed target location, but that is not an optimal technique, because it assumes that each observation has equal error. Instead, SA uses a more accurate technique of computing the coordinate for the target based on an optimization. In Figure 9-10, the computed target location is closer to the leftmost instrument's observation, due primarily to the fact that it is closest to the measured point and therefore should have the least measurement error.

A view of this situation from above (see Figure 9-11 on page 281) makes it even clearer: The closest instrument is most accurate, and gets the most influence in calculating the target location. The net result of this optimization technique is that the computed target location's coordinates are more accurate than they would be if the average of the observations were used instead.

In the case of a single instrument measuring a point multiple times, the end result is that the target coordinate is almost exactly an average of the individual observations, as the distance to the target is approximately constant for all observations.

Figure 9-10. A measured target as computed by observations from three instruments (exaggerated for clarity).

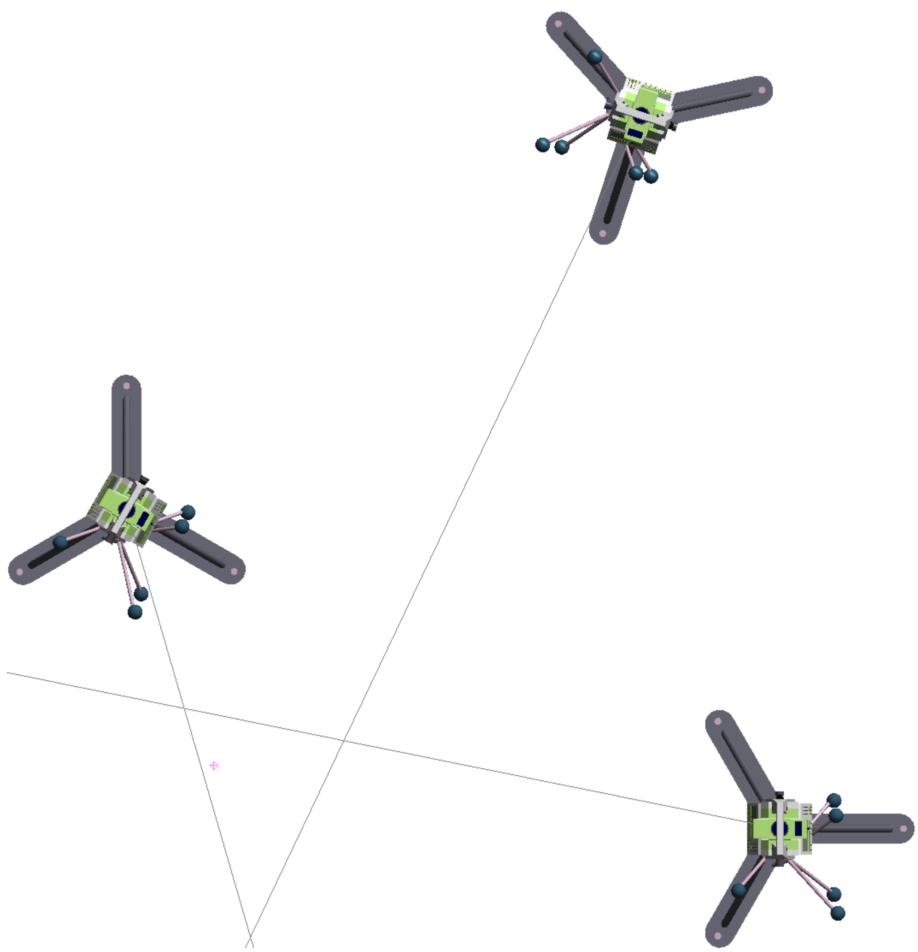


Figure 9-11. The instrument closest to the target has the greatest accuracy and therefore gets the largest influence in target computation.

About Target Offsets

Regardless of the type of instrument used, when a measurement is performed in SA, the point that is recorded is the center of the probe (or the surface intersection point for non-contact measurements). For example:

- Laser tracker measurements record the center of the SMR.
- PCMM arm measurements record the center of the probe tip.
- Laser radar measurements record the returned point of contact with the object's surface.

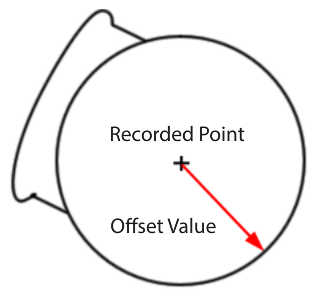


Figure 9-12. The center of the probe is the recorded point, and the offset is its radius.

In order to properly compare measured points to geometry or construct geometry from measured points, SA must always know the planar and radial offset for the measurement tooling that is currently in use. In normal use, an instrument interface has an offset assigned based on the current probe and tooling being used, and this offset is automatically applied to points as they are measured. For example, suppose a pin nest and SMR are being used to measure the inside diameter of a hole with a laser tracker. If SA is set up to construct a circle as a result of the measurement, then once the circle is fit to the measured probe centers, it will be offset normal to the circle by the planar offset amount, and its radius will be increased by the radial offset value. These offsets will align the constructed circle with the true location of the edge of the hole, regardless of the measurement tooling or probe used for the measurements.

In addition to the target's offsets, for traceability purposes each individual measurement observation has a radial and planar offset as well.

Characteristics Unique to Measured Points

There are a few characteristics that are unique to measured points that you will not see on constructed points (Figure 9-13).

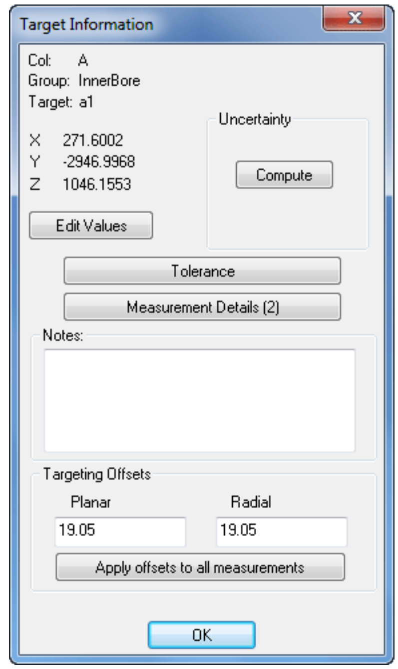


Figure 9-13. The ***Target Information*** dialog looks like the ***Point Information*** dialog for constructed points, but has a few subtle (but important) differences.

Uncertainty

There is an *Uncertainty* box with a Compute button inside. This is used for computing the uncertainty of the measured point and is discussed in more detail in the chapter on Measurement Uncertainty.

Measurement Details

To view the detailed measurement information that comprises a target, click on the Measurement Details button. (Note that the button will display the number of active observations that are being used to calculate the target position).

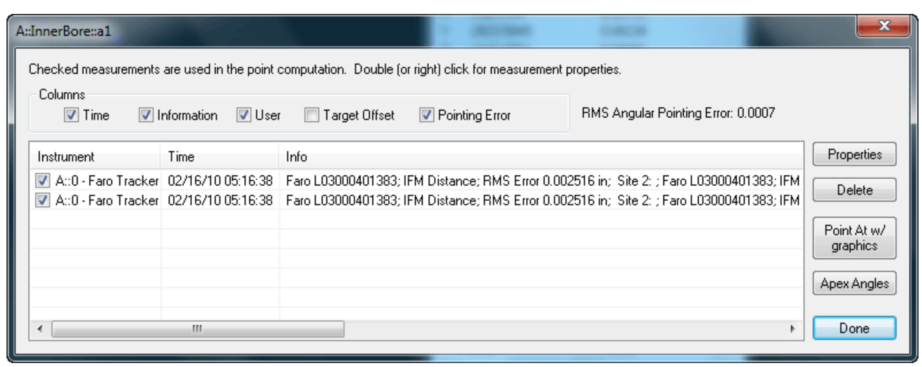


Figure 9-14. Two observations comprise this measured target, and both are active.

Warning: Deactivated observations are not used when calculating the target coordinate.

In this example (Figure 9-14), there are two active instrument observations that are being used to calculate the target coordinates. The checkboxes at the top of this dialog control which items are visible in the table.

- **Time.** Toggles whether or not the timestamp (date and time) for the measurement is displayed.
- **Information.** Toggles whether measurement details (such as

instrument serial number, mode, weather information, etc.) are displayed.

- **User.** Toggles whether the username is displayed.
- **Target Offset.** Toggles whether the stored offsets for each observation are displayed.
- **Pointing Error.** Displays the instrument's angular and linear *pointing errors*. (Pointing error is the angular error between the observation and the computed target location).

Next to these checkboxes, the dialog displays the RMS value of the combined angular pointing errors.

To Activate or Deactivate an Observation:

- Check or uncheck the observation in the *Instrument* column.

To Activate or Deactivate All Observations on a Target:

- Right-click the point in the tree or graphical view and select **Activate All Measurements** or **Disable All Measurements** as applicable.

To Activate All Measurements to All Targets:

- From the menu, select **Analysis>Activate All Measurements**.

To Permanently Delete an Observation:

- Select the row in the table and click the Delete button.

An *Apex Angle* is the angle between two measurement rays, or lines of sight, from measurement devices (see Figure 9-15). Ideal measurements with the least amount of error and uncertainty are obtained with apex angles of 90 degrees. As the apex angles increase or decrease from this value, the measurements have more uncertainty and are less reliable. SA can calculate and display the worst apex angle from a series of observations comprising a target.

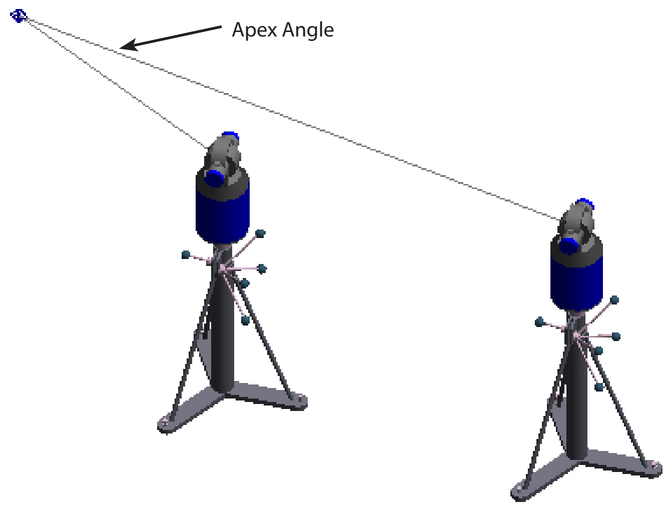


Figure 9-15. The apex angle between two lines of sight to a common target.

To Determine the Worst Apex Angle Comprising a Target:

- Click the Apex Angles button. SA will determine the two observations that form the worst apex angle, list the angle, and also list the two observations that form this angle.

To Point an Instrument's Graphical Model at its Observation:

- Select the observation of interest and click the Point At w/ Graphics button. The instrument's head will point at the measurement in the graphical view.

Observation Details

You can view the details for an individual instrument observation.

To View the Details for a Single Observation:

- Double-click or right-click the observation, or
- Select the desired row and click the Properties button.

The details for an observation are displayed in the ***Polar Measurement*** dialog (see Figure 9-16 on page 286).

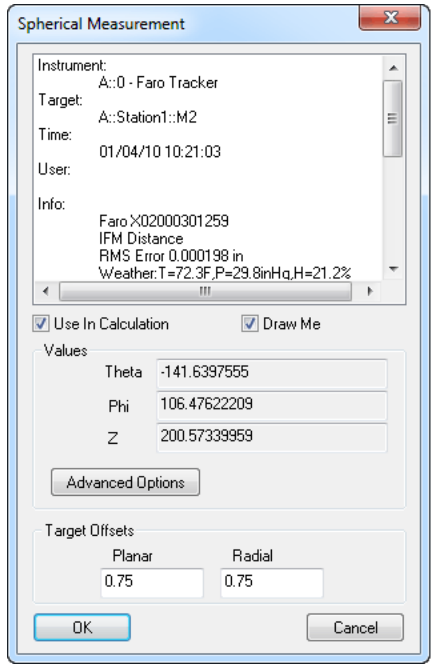


Figure 9-16. A single observation's details. This shows a laser tracker's observation information.

The top section displays all of the metadata associated with the observation.

- **Use In Calculation.** Toggling this option is equivalent to activating/deactivating an observation in the ***Measurement Details*** dialog.
- **Draw Me.** Turn this option off to prevent SA from drawing the instrument shot (ray) for the observation.
- **Theta/Phi/Z (X/Y/Z).** Read-only values showing the raw data from the instrument, expressed in the instrument's coordinate frame.
- **Advanced Options (theodolites only).** Mark observations as normal, collimation shots, or mirror cube shots (see below).

Each observation can store a planar and radial offset. These offsets are normally recorded when the measurement is taken. However, you can change them by filling in the *Planar* and *Radial* offset values to assign custom offsets to a specific observation.

The Advanced Options button allows you to specify whether the observation is a *Normal Shot*, a *Collimation Shot* (you must specify the instrument you're collimating to), or a *Mirror Cube Shot* (you must specify which mirror face was measured). These settings are used to help constrain bundle adjustments (see Figure 9-17 on page 287).

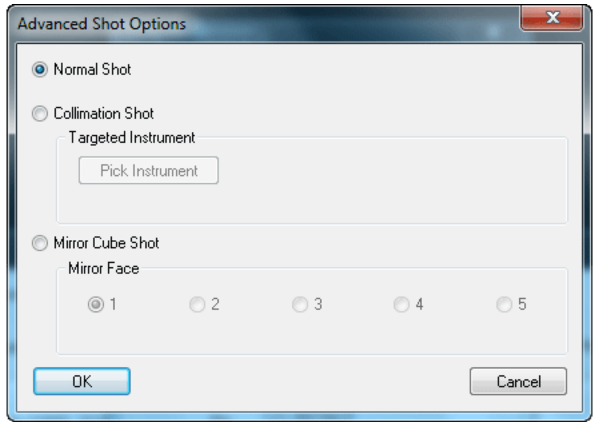


Figure 9-17. Advanced shot options for a spherical measurement.

Targeting Offsets

Individual observations can have their own stored planar and radial offsets. Constructed points do not have to deal with this, as they do not have any observations.

A button labeled Apply offsets to all measurements (see Figure 9-13 on page 283) is visible below the planar and radial offset fields for measured points (you will not see it for constructed points). Clicking this button will apply the specified planar and radial offsets to all individual observations for the target.

Setting Properties for Multiple Points

Some situations arise in which you'd like to change the properties of multiple points at once. For instance, if you accidentally measure a large number of points with the wrong target offsets set in the instrument interface, you will need to correct these offsets in order for SA to properly apply probe compensation.

You can apply new tolerances or offsets to multiple points using the ***Set Properties for Multiple Points*** dialog (Figure 9-18).

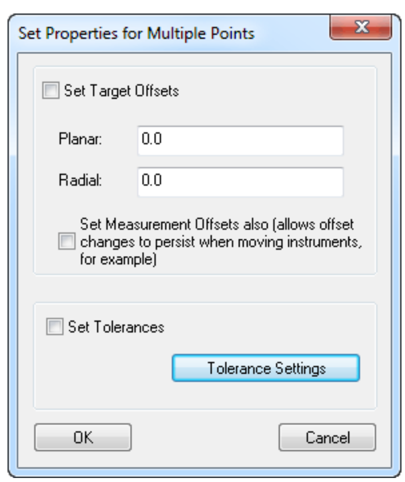


Figure 9-18. The ***Set Properties for Multiple Points*** dialog.

To Change Offsets or Tolerances For Multiple Points:

- From the menu, choose **Edit>Set Properties for multiple points** and select the points to modify. This will call up the ***Set Properties for Multiple Points*** dialog.

To Change Offsets or Tolerances For An Entire Group:

- Right-click a point group and select **Set Point Properties** from the context menu. This will call up the ***Set Properties for Multiple Points*** dialog.

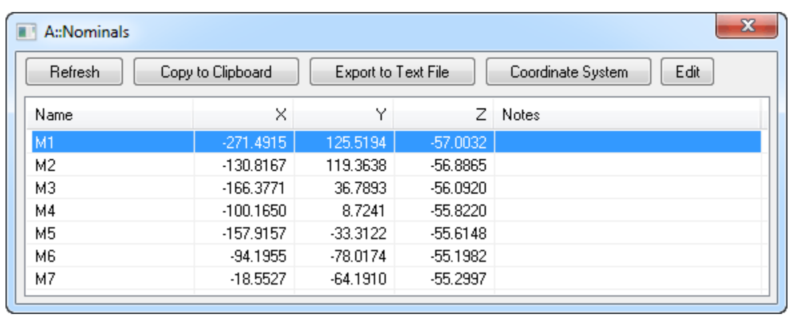
This dialog has two regions. The top region is for changing the stored target offsets, and the bottom region is for changing the point tolerances.

- **Set Target Offsets.** Check this option to change the planar and radial offsets of the measured points to the values entered into the *Planar* and *Radial* fields.
- **Set Measurement Offsets Also.** This applies the entered offsets to the individual observations within a target as well. Very rarely would you not want to select this option.
- **Set Tolerances.** When checked, the tolerances specified via the Tolerance Settings button will be applied to the points.

Note: The tolerance settings dialog is the same as described in "Point Tolerance" on page 276.

Working With Point Lists

It is often useful to view a list of points and point coordinates in tabular format, so that you can easily compare against nominals or quickly modify a set of points at once (similar to working in Excel). SA's point lists provide this functionality (Figure 9-19).



Name	X	Y	Z	Notes
M1	-271.4915	125.5194	-57.0032	
M2	-130.8167	119.3638	-56.8865	
M3	-166.3771	36.7893	-56.0920	
M4	-100.1650	8.7241	-55.8220	
M5	-157.9157	-33.3122	-55.6148	
M6	-94.1955	-78.0174	-55.1982	
M7	-18.5527	-64.1910	-55.2997	

Figure 9-19. A point list.

To View a Point List:

Do one of the following:

- From the menu, select **View>Point Lists for Groups**, then select the group(s) to view, or
- Right-click a group and select **View Point List** from the context menu (to view a single group).

The list has several buttons at the top:

- **Refresh.** Refreshes the list to ensure it's showing the latest information.
- **Copy to Clipboard.** Copies the selected rows (or all rows if nothing is selected) to the clipboard.
- **Export to Text File.** Exports the selected rows (or all rows if nothing is selected) to a text file.
- **Coordinate System.** Click this button to switch among Cartesian, Cylindrical, and Spherical coordinates.
- **Edit.** Opens up a new window allowing you to modify the point information (see below).

Instead of just viewing the information, a point list can be used for modifying point names or coordinates. This is particularly useful because you can change multiple values at once. For instance, you can quickly change the x-coordinate of multiple points in one action.

To Edit a Point List:

Do one of the following:

- With the point list already displayed (see "To View a Point List:" on page 288), click the Edit button, or
- Right-click a point group and select **Edit Point List** from the context menu to edit that group of points.

The ***Edit Point List*** dialog is displayed in Figure 9-20.

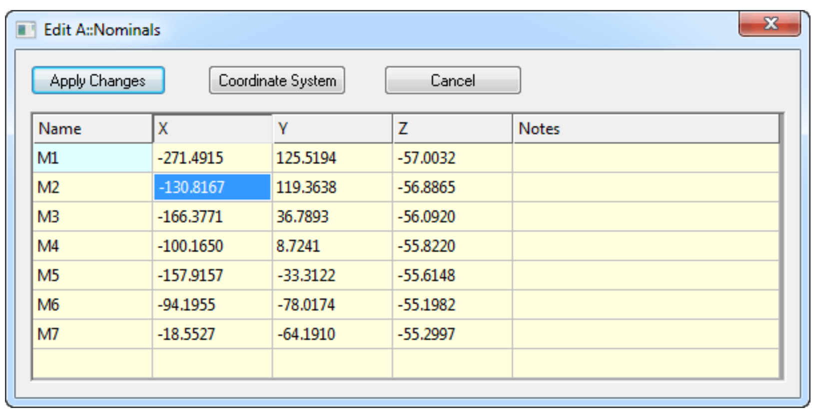


Figure 9-20. The ***Edit Point List*** dialog.

To Change a Point's Name, Coordinate, or Notes:

- Double-click the field you'd like to modify, enter a new value, then press **`Enter`**.

To Modify Multiple Values At Once:

1. Select multiple fields using **`Shift+Click`** (standard Windows range selection) or **`Ctrl+Click`** (standard Windows non-contiguous selection).
2. Double-click one of the selected fields, type a new value, and press **`Enter`**. All fields will be updated with the new value.

The dialog provides a few additional buttons at the top.

- **Apply Changes.** Modifications are not actually applied to the points until you click this button.
- **Coordinate System.** Choose to switch the display between Cartesian, Cylindrical, or Spherical coordinates.
- **Cancel.** If you cancel, no changes will be applied the points.

Constructing Points

There are many different ways to create points in SA. You can explicitly create a point at a specified coordinate or create points through an array of geometric analysis functions.

Entering Point Coordinates

The most basic way to create a point is to explicitly enter a point by specifying its collection, group, target, and coordinate system. This will create a single constructed point in your job.

To Enter a Point:

1. From the menu, select **Construct>Points>Enter**, or press **Ctrl+P** (default keyboard shortcut). The ***Add Points to Model*** dialog will appear (Figure 9-21).

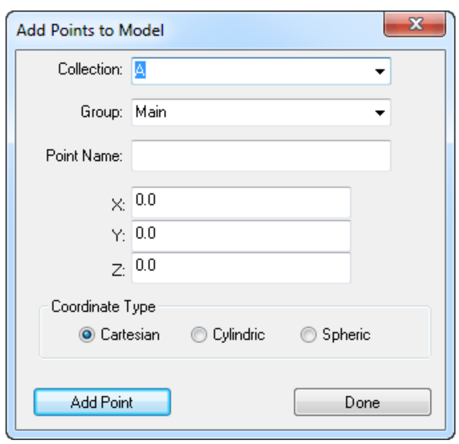


Figure 9-21. Constructing a point by entering its coordinates.

2. Enter a collection, group, and point name (or select one from the dropdown of existing collections or groups).
3. Specify the desired coordinate type (*Cartesian*, *Cylindric*, or *Spheric*).
4. Enter the desired coordinate values, then click the Add Point button to create the point and continue adding more, or click Done to create the point and close the dialog.

Constructing a Point at the Center of a Line

Creating a point at the center of a line constructs a point halfway between its endpoints.

To Create a Point at the Center of a Line:

1. From the menu, select **Construct>Points>Center>Line**.
2. Select the desired line.
3. The ***Construct New Point*** dialog will appear (Figure 9-22).

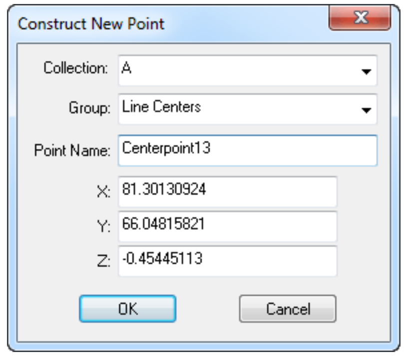


Figure 9-22. The ***Construct New Point*** dialog.

4. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown).
5. Click the OK button.

Constructing Points at the Center of Spheres and Circles

You can construct points at the center of multiple circles or multiple spheres with a single command.

To Construct Points at the Center of Circles or Spheres:

1. From the menu, select **Construct>Points>Center>Circles** or **>Spheres**. The ***Point Name*** dialog will appear (Figure 9-23).

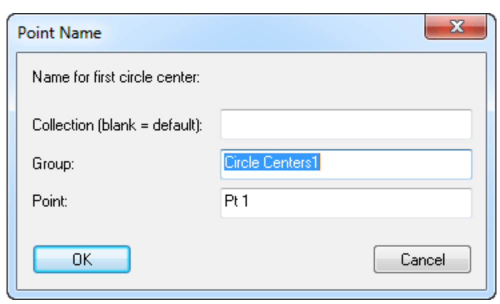


Figure 9-23. The ***Point Name*** dialog.

2. Enter a collection, group, and point name for the first point to construct. (Leave the *Collection* field blank to put the points into the active collection). Click OK.
3. Select one or more circles or spheres. Each center point will be created, and successive point names will be incremented.

4. When finished creating centers, press **Esc** or **Enter**.

Constructing Points at the Center of a Cylinder

When creating points at the center of cylinders, SA will actually create three points: one point at each end on the cylinder's axis (called **EndA** and **EndB**), and a third point halfway between the endpoints (called **Center**).

To Create Points at the Center of Cylinders:

1. From the menu, select **Construct>Points>Center>Cylinder**.
2. Select the cylinder. The points will be placed into the active collection, into a group with a name matching that of the cylinder.

Projecting to the Closest Point on Objects

Projecting points to the closest point on an object is equivalent to creating points on the object where that object's surface normals intersect with the original points.

With this command, you can select multiple objects and SA will find the closest point among all selected candidate objects. You can also select multiple points, and SA will find the closest point for each of the candidate points.

You can also offset points away from the surface. This is particularly useful when you have nominal points on a surface that you'd like to back away from the surface by the probe radius (so that those points can serve as proximity triggers for measurement).

To Project Points to the Closest Points on an Object:

1. From the menu, select **Construct>Points>Project Points to>Objects>Closest Point**.
2. Select one or more objects to project to.
3. Select one or more points to project. The ***Point Naming*** dialog will appear (Figure 9-24).

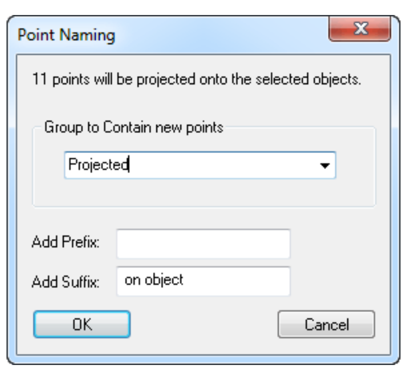


Figure 9-24. The ***Point Naming*** dialog.

4. Enter or select the desired group to contain the projected points. In the *Add Prefix* or *Add Suffix* field, enter a prefix or suffix that you'd like to attach to the point names, then click OK.
5. The ***Probe Offset*** dialog will appear. Enter a distance to offset the projected points back away from the object's surface. A positive value shifts along the positive direction of the surface normal.

Projecting Points Parallel to a Working Coordinate Frame Axis

You can project points along a working coordinate frame axis until it intersects with one or more surfaces (Figure 9-25). (If there is no intersection, no projected point will be created).

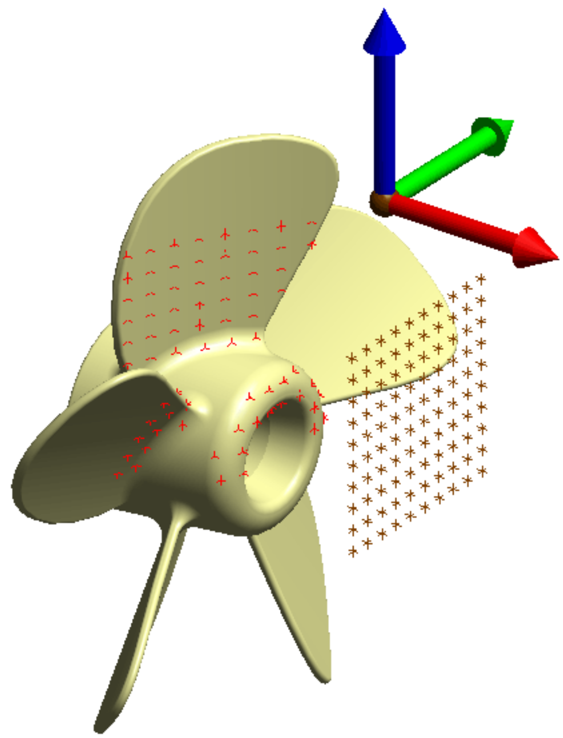


Figure 9-25. Projecting a grid of points along the working X axis to a surface.

To Project Points Along the Working Coordinate Frame Axis:

1. From the menu, select **Construct>Points>Project Points to>Objects>Relative to Coordinate Frame Axis>Parallel to WCF Axis**.
2. When prompted, pick the surfaces to project to.
3. Select the points you'd like to project.
4. When the ***Select Axis*** dialog appears, choose the axis to project along.
5. The ***Point Naming*** dialog will appear (Figure 9-24). Enter or select the desired group to contain the projected points. In the *Add Prefix* or *Add Suffix* field, enter a prefix or suffix that you'd like to attach to the point names, then click OK.

Projecting Points Radially from a Working Coordinate Frame Axis

You can also project points radially from a coordinate frame axis. Here, one of the axes of the working coordinate frame serves as the axis of a cylindrical coordinate system, and the points are projected out radially from that axis until they intersect with one or more surfaces (Figure 9-26).

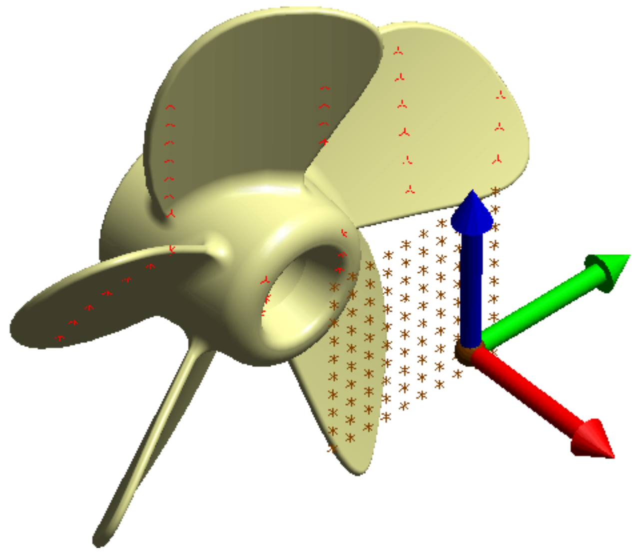


Figure 9-26. Projecting radially from a coordinate frame axis.

To Project Radially from a Working Coordinate Frame Axis:

1. From the menu, select **Construct>Points>Project Points to>Objects>Relative to Coordinate Axes>Radial from WCF Axis**.
2. When prompted, pick the surfaces to project to.
3. Select the points you'd like to project.
4. When the ***Select Axis*** dialog appears, choose the axis from which to project radially.
5. The ***Point Naming*** dialog will appear (Figure 9-24). Enter or select the desired group to contain the projected points. In the *Add Prefix* or *Add Suffix* field, enter a prefix or suffix that you'd like to attach to the point names, then click OK.

Projecting Points Spherically from a Working Coordinate Frame Origin

Points can be projecting spherically outward from the origin of the working coordinate frame (Figure 9-27).

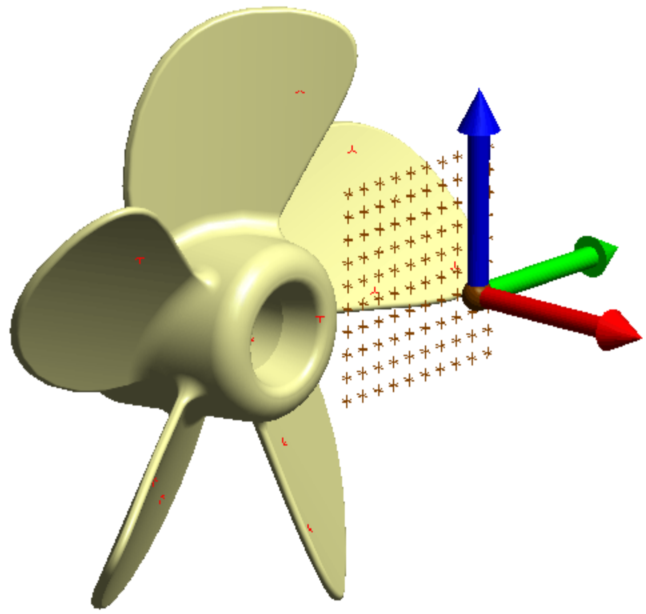


Figure 9-27. Projecting points spherically from the working coordinate frame origin.

To Project Points Spherically From the Working Coordinate Frame Origin:

1. From the menu, select **Construct>Points>Project Points to>Objects>Relative to Coordinate Frame Axis>Spherical From WCF Origin**.
2. When prompted, pick the surfaces to project to.
3. Select the points you'd like to project.
4. The ***Point Naming*** dialog will appear (Figure 9-24). Enter or select the desired group to contain the projected points. In the *Add Prefix* or *Add Suffix* field, enter a prefix or suffix that you'd like to attach to the point names, then click OK.

Projecting Points to a Curve

You can project points directly to any arbitrary curve or curves (Figure 9-28). The projected points can lie directly on the curves or can be backed off the curve by a specified amount. Each point will be projected to the closest curve among all candidates.

To Project Points to a Curve:

1. From the menu, select **Construct>Points>Project Points to>Curves**.
2. At the prompt, select the curves to project to.
3. Select the points to project.
4. The ***Point Naming*** dialog will appear (Figure 9-24). Enter or select the desired group to contain the projected points. In the *Add Prefix* or *Add Suffix* field, enter a prefix or suffix that you'd like to attach to the point names, then click OK.
5. At the ***Probe Offset*** dialog, enter a distance to offset the pro-

jected points back away from the curve. This value must be positive, and always shifts away from the curve.

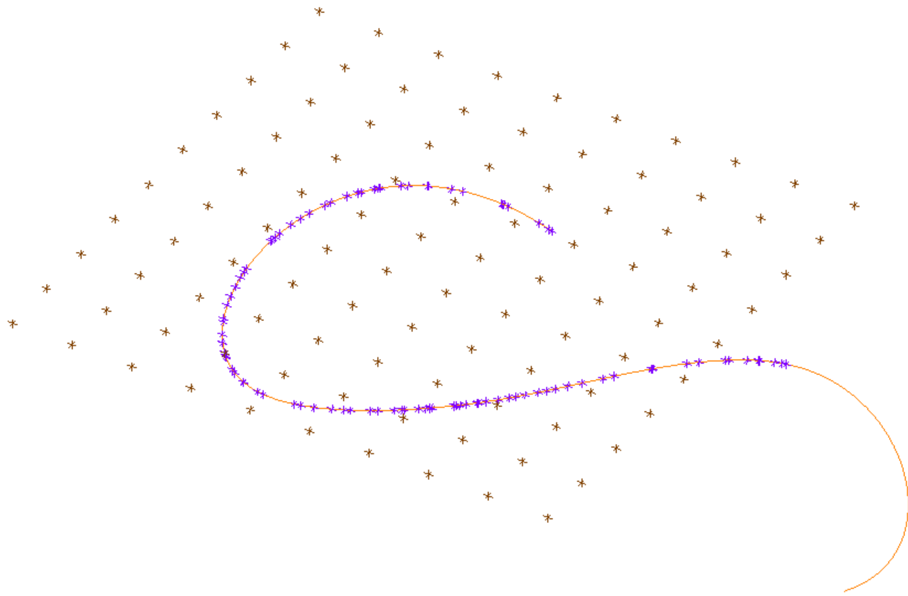


Figure 9-28. Projecting points to a curve.

Creating Points at a 3-Plane Intersection

A point can be created at the intersection of 3 planes. This of course requires that no two planes may be parallel.

To Create a Point at the Intersection of 3 Planes:

1. From the menu, select **Construct>Points>Intersection>3 Planes**.
2. Select the three planes successively.
3. The ***Construct New Point*** dialog (see Figure 9-22 on page 291) will appear. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown).
4. Click OK to create the point.

Creating Points at the Intersection of a Plane and Circle

Intersecting a plane and a circle will actually create two points as seen in Figure 9-29. It should be noted that the plane and circle cannot be parallel or coplanar.

To Create Points at the Intersection of a Plane and Circle:

1. From the menu, select **Construct>Points>Intersection>Plane and Circle**.
2. Select the plane, then the circle. Two new points will be created in the active collection's `Intersections` group named `Plane-Circle1` and `Plane-Circle2`.

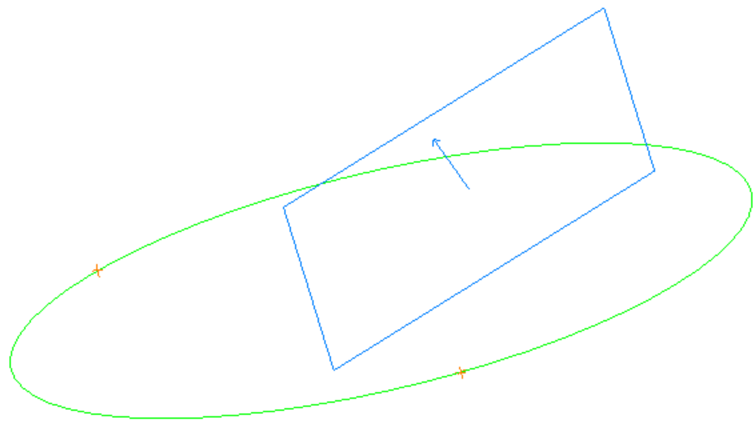


Figure 9-29. Intersecting a plane with a circle.

Constructing Points at the Intersection of a Line and Plane

You can intersect a line with a plane to create a point at the intersection (Figure 9-30).

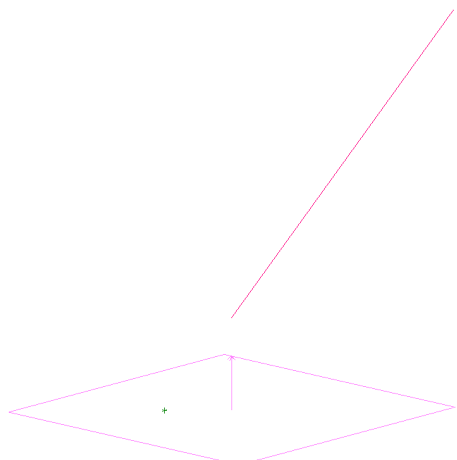


Figure 9-30. A line and plane intersection creating a point, even though their depictions do not intersect.

There are a few things to keep in mind:

- The line and plane need not physically intersect, since planes and lines are treated as infinite in size.
- If the line lies in the plane, or the line is parallel to and not in the plane, no points will be created.

To Create a Point at the Intersection of a Line and Plane:

1. From the menu, select **Construct>Points>Intersection>Line and Plane**.
2. Select the line, then the plane.
3. The ***Construct New Point*** dialog (Figure 9-22) appears.
4. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown).
5. Click the OK button.

Constructing a Point at the Intersection of 2 Lines

You can create a point at the intersection of two lines (Figure 9-31). Or, if the lines do not intersect, a point will be created at their mutual perpendicular midpoint. (The mutual perpendicular midpoint is halfway between the closest points between the two lines). If the lines are parallel, no point will be created.

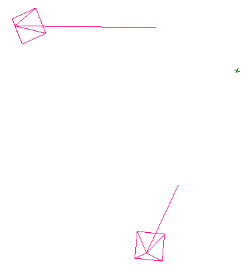


Figure 9-31. A point constructed at the mutual perpendicular midpoint between two lines. In this case, these two lines do not intersect.

To Create a Point at the Intersection of Two Lines:

1. From the menu, select **Construct>Points>Intersection>2 Lines (mutual perpendicular midpoint)**.
2. Select the first line, then the second.
3. The ***Construct New Point*** dialog (Figure 9-22) appears.
4. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown).
5. Click the OK button.

Constructing Points at the Intersection of a Line and Circle

Points can be created at the intersection of a line and a circle (Figure 9-32). The line is first projected to the plane on the circle and then the intersection points are created.

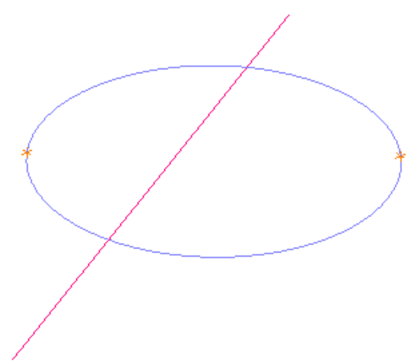


Figure 9-32. A line and circle intersection creating two points.

To Create Points at the Intersection of a Line and Circle:

1. From the menu, select **Construct>Points>Intersection>Line and Circle**.
2. Select the line, then the circle. Two new points will be created

in the active collection's **`Intersections`** group.

Constructing Points at the Intersection of a Line and Cylinder

You can create a point at each intersection of a line and cylinder. The cylinder and line need not physically intersect, since cylinders and lines are treated as infinite in length (Figure 9-33). However, the line must pass within the cylinder's radius.

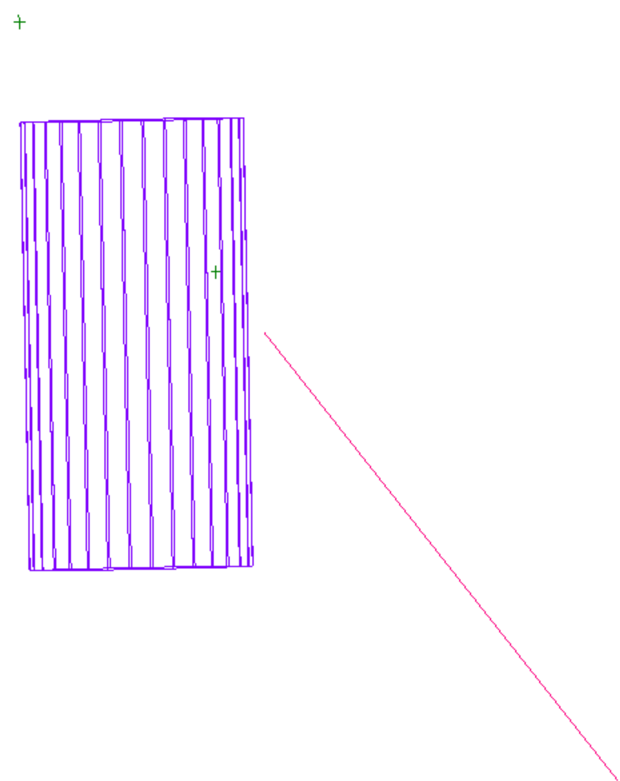


Figure 9-33. A point at each intersection of the line and cylinder are created, even though their depictions do not intersect. Note the length of the cylinder does not need to be increased, since cylinders are infinite in size.

To Create a Point at the Intersection of a Line and Cylinder:

1. From the menu, select **Construct>Points>Intersection>Line and Cylinder**.
2. Select the line, then the cylinder. A naming dialog appears (Figure 9-35).
3. Specify the desired collection, group, and point names (or select an existing collection and group from the dropdown).
4. Click OK to create the point.

Constructing Points at the Intersection of a Line and Sphere

You can create a point at each intersection between a line and a sphere.

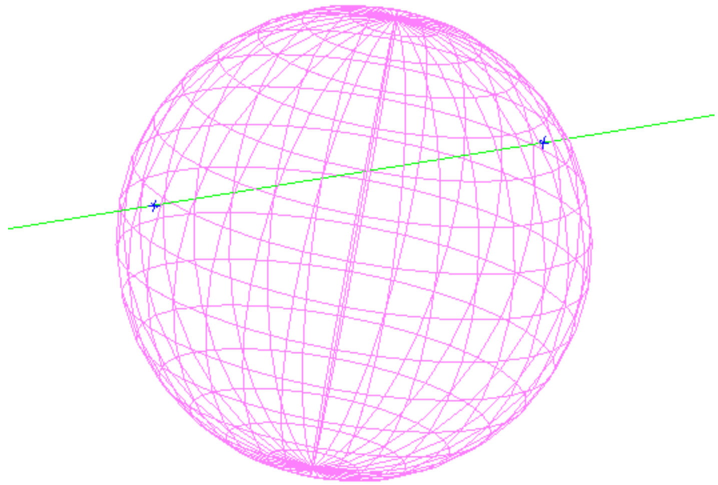


Figure 9-34. A point is constructed at each intersection of the line and sphere.

To Create a Point at the Intersection of a Line and Sphere:

1. From the menu, select **Construct>Points>Intersection>Line and Sphere**.
2. Select the line, then the sphere.
3. A naming dialog appears (Figure 9-35).
4. Specify the desired collection, group, and point names (or select an existing collection and group from the dropdown).
5. Click OK to create the point.

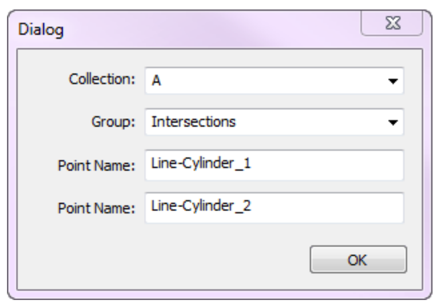


Figure 9-35. The naming dialog for line intersections with cylinders and spheres.

Constructing Points at the Intersection of 2 B-Splines

Points can be constructed from the intersections of two B-Splines. A point will be created at the midpoint of the closest location between the two B-splines if they do not intersect (Figure 9-36).

To Create a Point at the Intersection of 2 B-Splines:

1. From the menu, select **Construct>Points>Intersection>2 B-Splines**.
2. Select the first B-spline, then the second.
3. The ***Construct New Point*** dialog (Figure 9-22) will appear. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown.

4. Click OK to create the point.

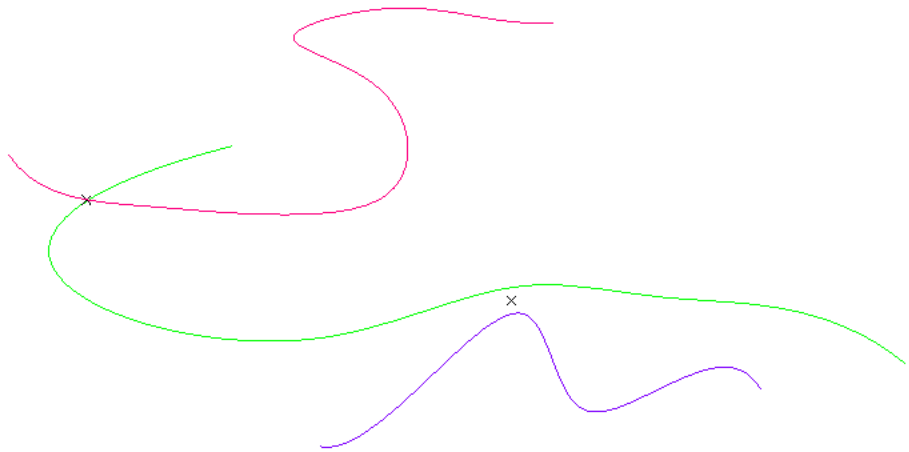


Figure 9-36. The pink and green B-splines create a point where they physically intersect, whereas the green and purple B-splines create a midpoint between their closest locations since they do not intersect.

Constructing Points at the Intersection of a B-Spline and Surfaces

Note: If the B-Spline intersects multiple times or intersects multiple selected surfaces, only one point is created. Which point is created is not generally clear.

You can create a point at the intersection of a B-Spline and a surface (Figure 9-37). The command follows the B-Spline from its tail toward its head and creates a point when the spline first pierces the surface.

You can select multiple surfaces with this command, but still only one intersection point will be created.

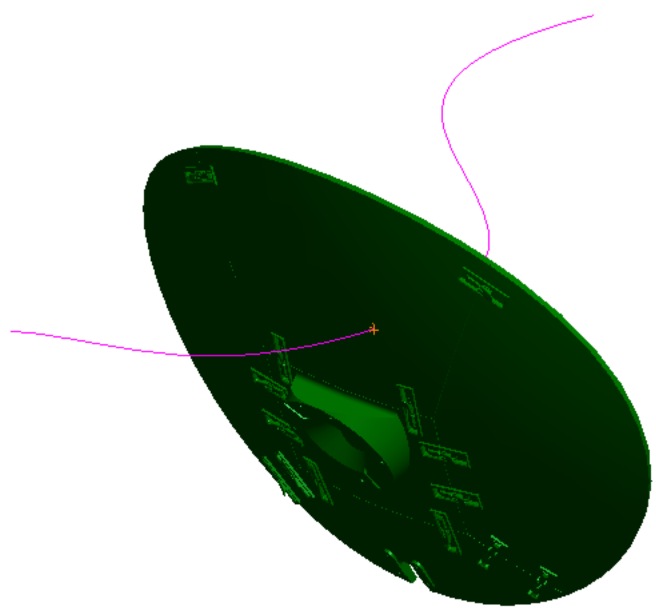


Figure 9-37. Intersecting a B-Spline with a surface to create a point.

To Create a Point at the Intersection of a B-Spline and Surfaces:

1. From the menu, select **Construct>Points>Intersection>B-Spline and Surfaces**.
2. Select the B-Spline.
3. Select one or more surfaces to test.

4. When prompted, enter an approximation tolerance. Generally, the default value is fine.
5. The ***Construct New Point*** dialog (Figure 9-22) will appear. Leave the coordinates as-is (they are calculated for you), and specify the desired collection, group, and point name (or select an existing collection or group from the dropdown.
6. Click OK to create the point.

Constructing Points at the Intersection of Principle Object Axes and Surfaces

A point can be constructed at the intersection of an object's axis and a surface (Figure 9-38). The definition of the axis depends on the type of object, but the axis must intersect at a point on the surface and not through a hole in the surface.

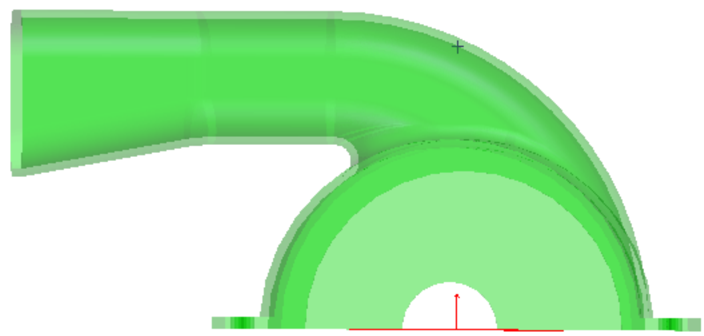


Figure 9-38. A point created at the intersection of a plane's axis and a surface.

To Construct a Point at the Intersection of an Object Axis and Surface:

1. From the menu, select **Construct>Points>Intersection>Principle Object Axes and Surfaces**.
2. Choose the axis object, followed by the surface.
3. Enter the destination point group name. You also have the option of entering a suffix for the points (which will have the same name as your object).
4. Click OK.

Shifting Points In a Direction

You can use a selected object to define a direction in which to shift a point or selection of points by a specified distance (Figure 9-39). If a frame is selected as the object, you will be prompted to choose along which axis to shift the points.

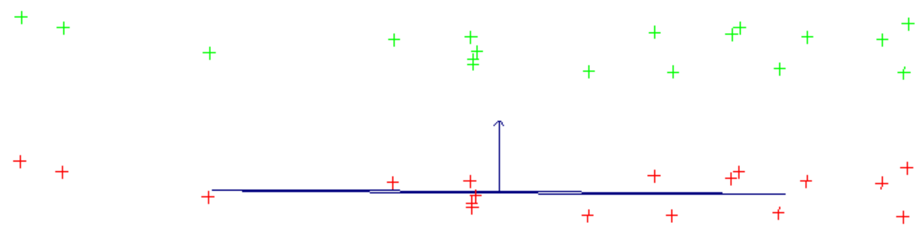


Figure 9-39. Green points were copied from the original points (red) and shifted along the circle's surface normal.

To Shift Points in a Direction:

1. From the menu, select **Construct>Points>Shift Points>In a Direction**.
2. Select the object defining the shift direction, then select the points you would like to shift.
3. The ***Shift Points in Direction*** dialog will appear (Figure 9-40). Enter the shift distance and the point group destination.

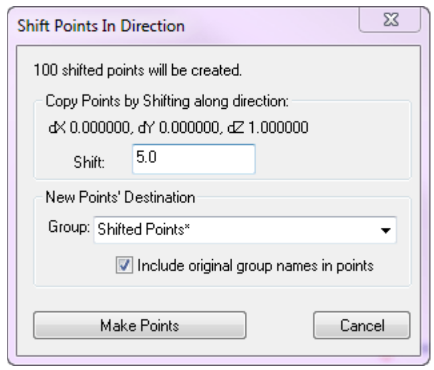


Figure 9-40. Shifting points along a direction.

4. Click Make Points to create the shifted points.

Shifting Points In Cylindric Coordinates

Points may be shifted cylindrically. Any object that contains directional information may be selected to define the axis of the cylinder about which the points are shifted (Figure 9-41).

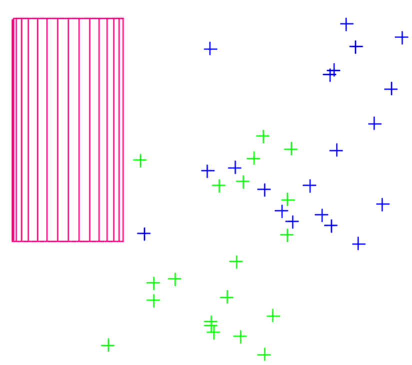


Figure 9-41. The cylinder is the object defining the cylindrical coordinate system and the blue points were copied from the original points (green) and shifted 5.0" along the planar and radial directions.

To Shift Points in Cylindric Coordinates:

1. From the menu, select **Construct>Points>Shift Points>In Cylindric Coordinates**.
2. Select the object defining the cylindric coordinate system,

followed by the points to be shifted.

3. The ***Shift Points in Cylindric Coordinates*** dialog will appear (Figure 9-42). Enter the radial and planar shifts and the point group destination.

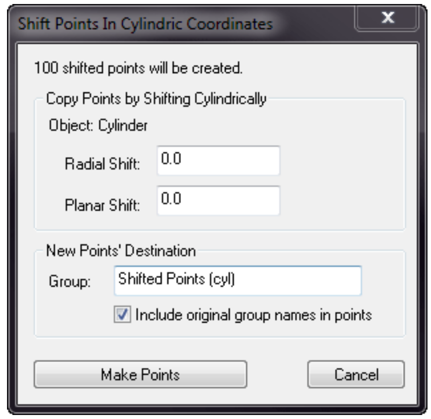


Figure 9-42. Specifying the radial and planar shift.

4. Click Make Points to create the shifted points.

Laying Out Points on a Grid By Count

A grid of points can be laid out with respect to the XY plane of the working coordinate frame. Using this command, you specify the range of the grid and the number of points desired along each axis, and SA determines the spacing for you automatically.

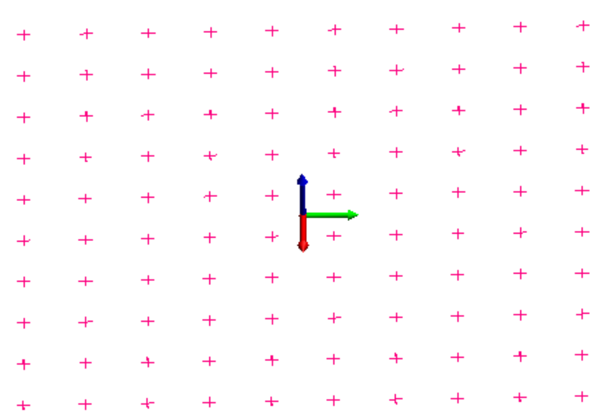


Figure 9-43. 10x10x1 Grid layout of 100 points.

To Lay Out Points on a Grid by Count:

1. Activate the frame you'd like to use to define the grid orientation.

2. From the menu, select **Construct>Points>Layout>Grid by Count**.

3. The ***Make a Grid of Points*** dialog will appear (Figure 9-44). In the *Min* and *Max* fields, specify the limits along the coordinate directions for the three-dimensional grid. In the *Count* fields, specify the number of rows of points along each direction. You can also enter a destination collection, group, and target

prefix for the resulting points.

4. Click Create Points to construct the grid.

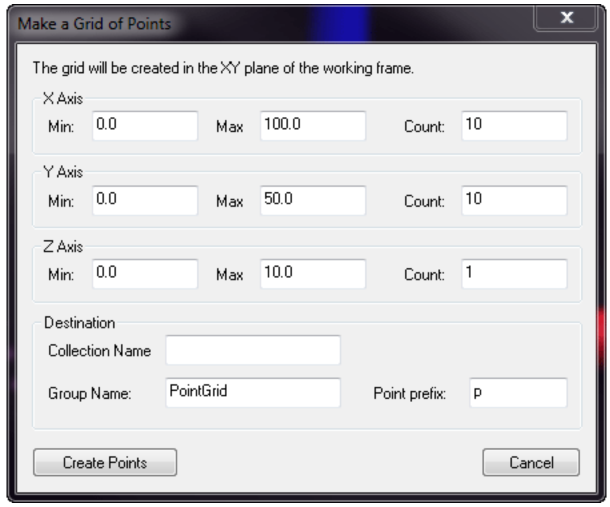


Figure 9-44. Entering the X, Y, and Z range of the grid, and the number of rows of points along each axis direction.

Laying Out Points on a Grid By Distance

A grid of points can be laid out with respect to the XY plane of the working frame. Using this command, you specify the range of the grid and the distance between points, and SA determines the number of points for you automatically.

To Lay Out Points on a Grid by Distance:

1. Activate the grid you'd like to use to define the grid orientation.
2. From the menu, select **Construct>Points>Layout>Grid by Distance**. The ***Make a Grid of Points*** dialog will be displayed (Figure 9-45).

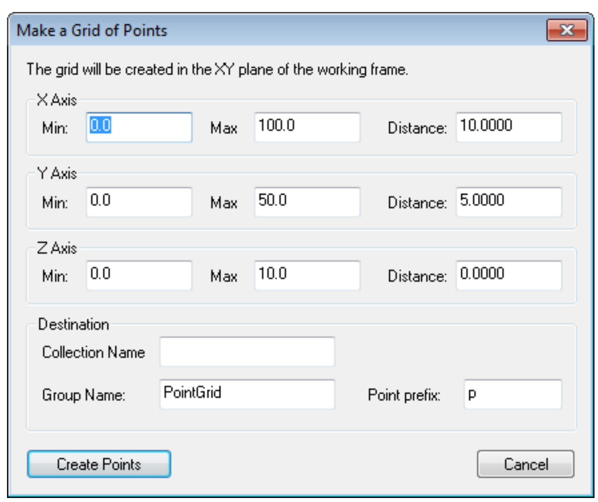


Figure 9-45. Defining parameters to create a grid by spacing.

3. Specify the range of the grid in all three component directions, as well as the desired spacing between points along each direction. You can also enter a destination collection,

group, and target prefix for the resulting points.

4. Click Create Points to construct the grid.

Laying Out Points On a Circle

It is often useful—particularly when laying out nominal bolt hole patterns—to lay out points on a circle. A circular point layout can be constructed in the XY plane of the working frame (Figure 9-46).

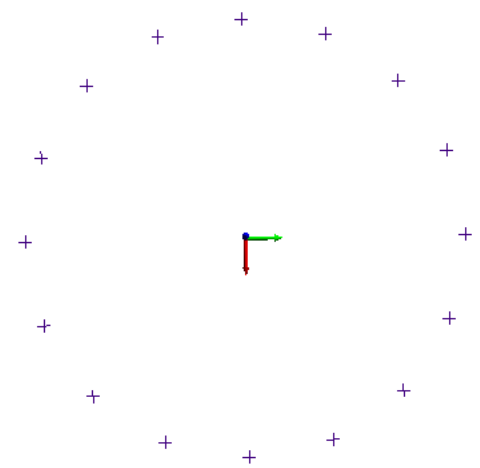


Figure 9-46. Laying out 16 evenly-spaced points on a circle.

The ***Layout Points on Circle*** dialog is shown in Figure 9-47. A number of options are available when laying out points, which are discussed below:

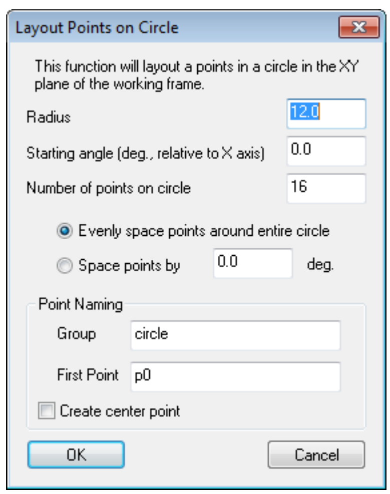


Figure 9-47. The ***Layout Points on Circle*** dialog.

- **Radius.** The radius of the circle defining the points.
- **Starting angle.** The angle in the working frame's XY plane, from the X axis, at which to lay out the first point.
- **Number of points on circle.** The number of points to create on the circle.

There are two methods for laying out points on the circle:

- **Evenly space points around entire circle.** The points will be constructed evenly around the entire circle.

- **Space points by.** You can specify the angle between points. If (based on the specified spacing and the number of points) more than 360° of the circle is specified, the points will continue to be laid out around the circle.

You can also control the naming of the resulting points using the *Group* and *First Point* fields, and whether a center point is created with the *Create center point* option.

To Construct a Layout of Points on a Circle:

1. From the menu, select **Construct>Points>Layout>Circle**.
2. The ***Layout Points on Circle*** dialog will appear (Figure 9-47).
3. Fill in the desired parameters, then click OK to create the points.

Generating Random Points

Occasionally, you may come across the need to generate random points. (This tends to be more common in training and testing than actual production environments).

When generating random points (Figure 9-48), you can specify the number of points and the range of the points along each of the three axis directions. The points will be randomly created inside this defined box.

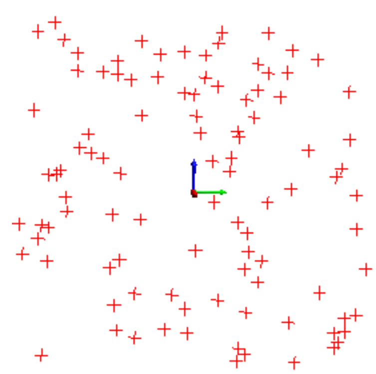


Figure 9-48. 100 randomly-generated points.

To Construct a Random Layout of Points:

1. From the menu, select **Construct>Points>Layout>Random Points** or use the default keyboard shortcut **`Ctrl+Alt+Z`**.
2. The Random Point Generator dialog will appear (Figure 9-49). Specify the number and range of point coordinates as well as the desired group name.

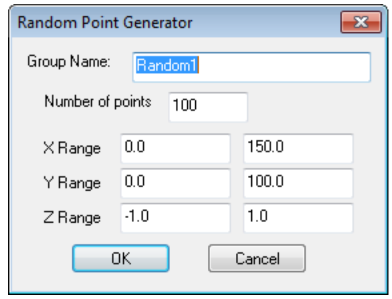


Figure 9-49. The ***Random Point Generator*** dialog.

3. Click OK to create the points.

Laying Out Points On A Surface By Clicking

You can click anywhere on a surface to create a point on that surface (Figure 9-50).

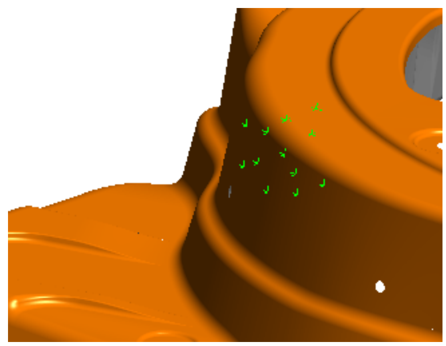


Figure 9-50. Laying out points on a surface.

To Lay Out Points on a Surface:

Note: The first click on a surface will incur a small delay as the software performs some calculations. Remaining clicks on the surface will be much faster.

1. From the menu, select **Construct>Points>Layout>On Surface By Clicking**.
2. Place the cursor anywhere on the surface and click to create a point at that location. If the cursor is not over a surface, a point will not be created.

Laying Out Points On An Object Vertex By Clicking

Points can be constructed on an object's vertices simply by clicking near the vertex (Figure 9-51).

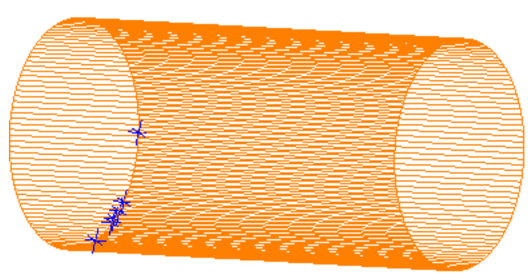


Figure 9-51. Several points have been laid off on this cylinder's vertices.

To Lay Out Points on Object Vertices by Clicking:

Note: The points will always be created on the surface, regardless of render settings.

1. From the menu, select **Construct>Points>Layout>On Object Vertex by Clicking**.

Tip: Increasing surface rendering detail will create more vertices for a given surface.

2. At the prompt, enter a desired group and starting point name.
3. Click the object near the desired vertices.
4. Press **Enter** when finished.

Laying Out Points On An Object's Vertices

You can construct points on all vertices of one or more objects at once (Figure 9-52).

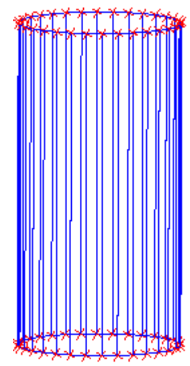


Figure 9-52. Points laid out on a cylinder's vertices.

To Lay Out Points on an Object's Vertices:

1. From the menu, select **Construct>Points>Layout>On Object's Vertices**.
2. Select the objects on which you'd like to create vertex points.
3. Enter a name for the destination group, then click OK.

Laying Out Points Evenly On Curves

You can construct any number of evenly-spaced points on one or more B-Spline curves (Figure 9-53).

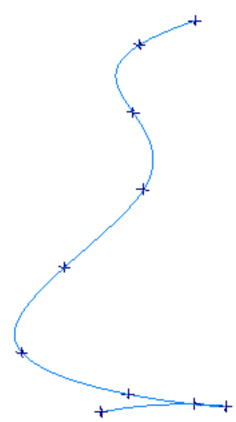


Figure 9-53. 10 points spaced evenly on a B-Spline.

To Lay Out Evenly-Spaced Points on a Curve:

1. From the menu, select **Construct>Points>Layout>On Curves>n Spaced Points**.
2. Select one or more B-splines.

3. Type a name for the destination group, then click OK.
4. At the prompt, enter the number of points to create on each curve, then click OK.

Laying Out Points On Curves With Prescribed Spacing

SA can construct points on one or more curves spaced by a prescribed distance (Figure 9-54). This is not a straight-line distance, but a distance along the curve (as if you'd laid a measuring tape of the prescribed length along the curve). When creating the points, SA places the first point exactly on the starting end of the curve.

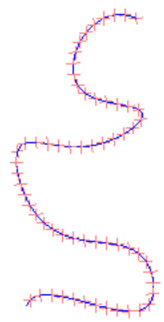


Figure 9-54. Points on a B-spline spaced exactly 2" from each other along the curve.

To Create Points Spaced on a Curve by a Prescribed Distance:

1. From the menu, select **Construct>Points>Layout>On Curves>Spaced at Distance**.
2. Select one or more B-splines.
3. Type a name for the destination group, then click OK.
4. Enter the specified distance between points, then click OK.

Laying Out Points On Curves by Clicking

This command allows you to click anywhere on one or more curves and create a point at that position on the curve (Figure 9-55).

Figure 9-55. Several points created by clicking on a B-Spline.

To Create Points on a Curve by Clicking:

1. From the menu, select **Construct>Points>Layout>On Curves>By Clicking**.
2. Enter a destination group name for the constructed points, then click OK.
3. Type a starting point name, then click OK.

4. Select one or more curves to consider.
5. Click at the desired locations. SA will create the point on the closest point of the closest curve.

Laying Out Points On Lines

As with a curve, you can lay points out on a line by specifying the number of points (in which they're evenly spread along the line) or by specifying the spacing (Figure 9-56).

Figure 9-56. A line with 10 evenly-spaced points.

To Lay Out Points on a Line:

1. From the menu, select **Construct>Points>Layout>On Lines**.
2. Select one or more lines.
3. The ***Layout Points on Lines*** dialog will appear (Figure 9-57).

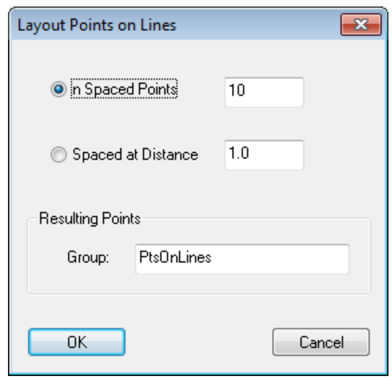


Figure 9-57. The ***Layout Points on Lines*** dialog.

- **n Spaced Points.** SA will create the specified number of points spaced evenly along each of the selected lines.
- **Spaced at Distance.** SA will create points on each line starting from the tail of the line, using the prescribed spacing.
- **Group.** The name for the resulting point group.

4. Choose the desired options, then click OK.

Laying Out Points At The Center of Surface Holes

It is often useful to create points at the center of holes on a surface. You can click on a single surface containing surface holes, and SA will create points at the center of the holes based on prescribed minimum and maximum diameter thresholds (Figure 9-58).

If a surface is dissected and you therefore select just a single surface face with holes, the points will be created along that face. If a surface is merged and you select a closed shell, points will be created on each end of the hole (if it is a thru hole).

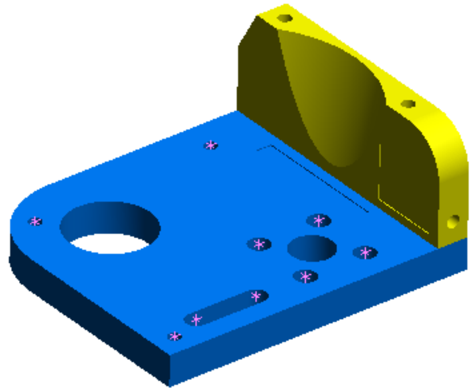


Figure 9-58. These points have been constructed at the center of surface holes.

SA considers any surface opening that has a circular arc for an edge as a surface hole.

To Construct Points at the Center of Surface Holes:

1. From the menu, select **Construct>Points>Layout>At Center of Surface Holes**.
2. Select one or more surfaces with holes.
3. The ***Points from Surface Holes*** dialog will appear (Figure 9-59).

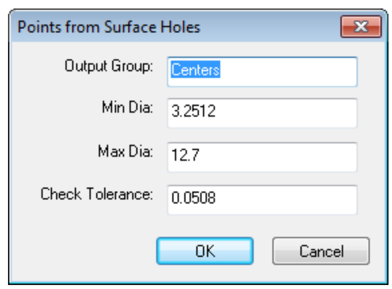


Figure 9-59. The ***Points from Surface Holes*** dialog.

- **Output Group.** The name for the destination group.
- **Min Dia.** The minimum diameter hole for point construction. Holes smaller than this diameter will not have center points constructed.
- **Max Dia.** The maximum diameter hole for point construction. Holes larger than this diameter will not have center points constructed.
- **Check Tolerance.** This describes the accuracy of the diameter check. Smaller values may take incrementally longer, but will more accurately check the diameter of the hole. Typically, you need not adjust this value unless you encounter unexpected behavior.

4. Select the desired options, then click OK.

Laying Out Points In The View By Clicking

Primarily intended for training and testing purposes, SA enables you to construct points by clicking anywhere in the graphical view. These points will be created on a plane perpendicular to the viewing direction (Figure 9-60).

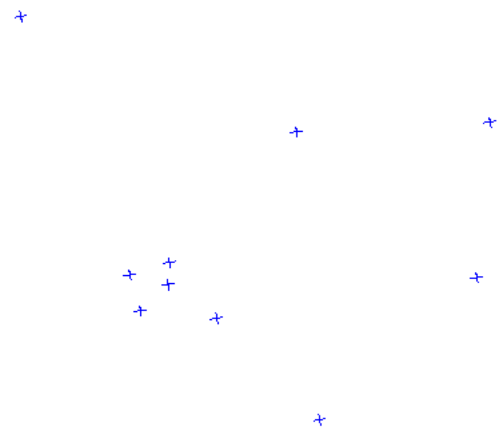


Figure 9-60. Random points laid out in the view.

To Construct Points in the Graphical View by Randomly Clicking:

1. From the menu, select **Construct>Points>Layout>In View By Clicking**.
2. Enter a name for the destination point group.
3. Specify a name for the starting point (subsequent point names will be incremented.
4. Click anywhere in the graphical view to create each point. Press **Enter** when finished.

Copying Existing Points

Note: Any copied measurements become constructed points.

Any set or subset of existing points can be easily copied. The copies are placed into a new point group.

To Copy Points from Existing Points:

1. From the menu, select **Construct>Points>From Existing Points>Copy**.
2. Select the points that you want to copy.
3. When prompted, type a destination group name, then click OK.

Moving Existing Points Into A New Group

Note: This command does *not* physically move points, as the name might imply.

You can quickly and easily relocate a set of points into a new group (Figure 9-61). The points may start in the same group or separate groups, but they will always end up in the same group. The points are relocated (not copied), which means that measurements remain as measured points after the command is executed. After the command, the point names will carry a prefix indicating their source group.

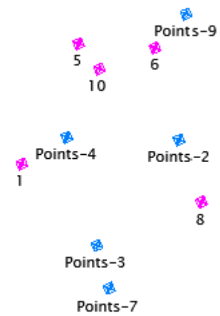


Figure 9-61. Relocating points into a new group.

To Move Existing Points to a New Point Group:

1. From the menu, select **Construct>Points>From Existing Points>Move**.
2. Select the points you want to move.
3. At the prompt, enter a name for the destination group, then click OK.

Creating a Subset of Points With Greatest Spacing

It is not uncommon to be confronted with point data that is more dense than is necessary for a certain operation (Figure 9-62).

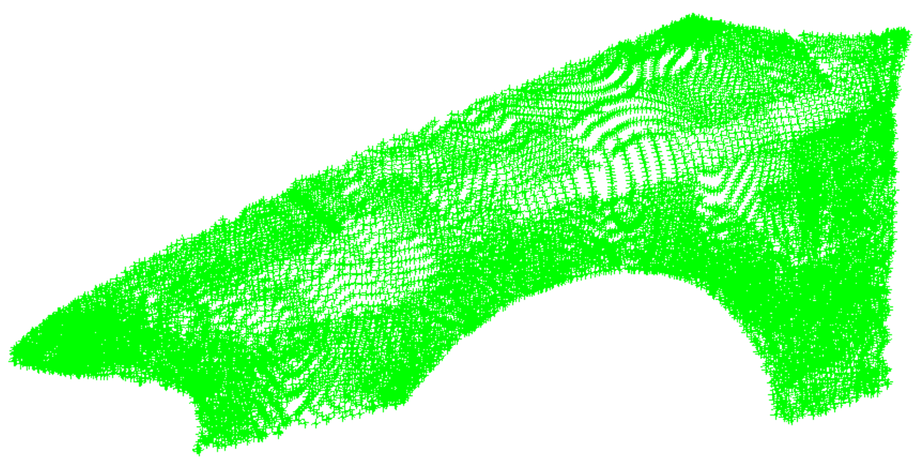


Figure 9-62. An overly-dense point group.

SA can sample an existing point group and create a new group of points at a lower spatial density than the source group (Figure 9-63). You can specify the desired number of points, and SA will choose a subset of points spread out to match that number.

The original points are untouched. Any subsampled points are a copy of the originals.

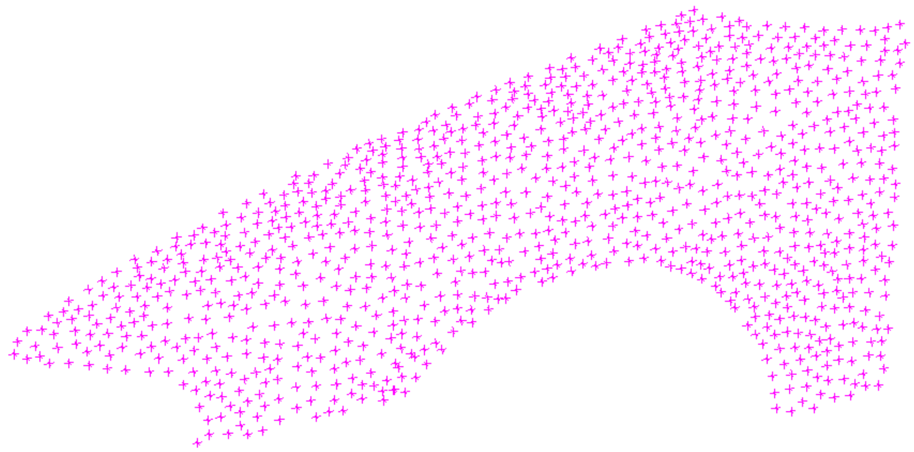


Figure 9-63. A subset of the original point group.

To Create a Subset of Points with Greatest Spacing:

1. From the menu, select **Construct>Points>From Existing Points>Subset with Greatest Spacing**.
2. Specify the number of points to create.
3. Enter a name for the destination point group.

Converting Cloud Points to Points

Note: Creating more than about 20,000 constructed points will result in increasingly sluggish interaction with the software. When dealing with very large datasets, use cloud points.

You can convert cloud points gathered from a scan into normal constructed points. The newly-constructed points retain their existing offsets, but will appear as constructed points instead of measured points—they will not have any additional measurement metadata (which is why cloud points are so much more efficient in the first place).

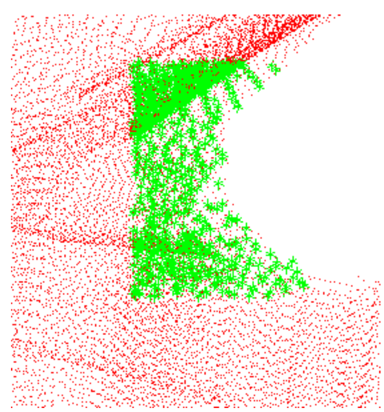


Figure 9-64. Points constructed from cloud points.

To Convert Cloud Points Into Constructed Points:

1. From the menu, select **Construct>Points>From Cloud Points>Pick Cloud Points**.
2. Select the points to convert.
3. The ***Make Points from Clouds*** dialog will appear (Figure 9-65).

Note: If the cloud is being thinned, only the thinned subset will be selected.

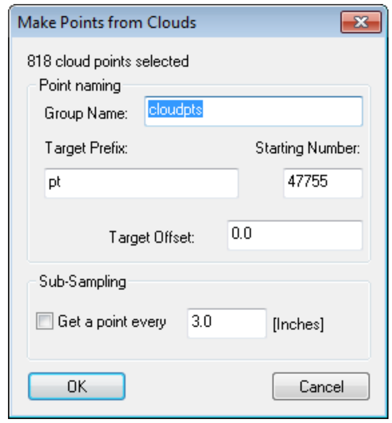


Figure 9-65. The ***Make Points from Clouds*** dialog.

4. Specify the group name, an optional target prefix and starting number, and an offset for the resulting points. If desired, select the *Get a point every xxx* option to subsample the cloud.
5. Click OK. If subsampling, confirm the dialog.

Converting Clouds to Points

Similar to converting cloud points to points, you can convert an entire cloud to points. The procedure is almost identical to creating points from cloud points.

To Create Points from Clouds:

1. From the menu, select **Construct>Points>From Cloud Points>Pick Clouds**.
2. Select the points to convert.
3. The ***Make Points from Clouds*** dialog will appear (Figure 9-65).
4. Specify the group name, an optional target prefix and starting number, and an offset for the resulting points. If desired, select the *Get a point every xxx* option to subsample the cloud.
5. Click OK. If subsampling, confirm the dialog.

Filtering Clouds to Points

Given a set of clouds and a set of points, SA can filter the clouds to the points. You provide a desired proximity and maximum number of points, and choose the existing points to serve as filter points. SA will create a new point group containing a subsampled set of up to the prescribed number of points (within the proximity) for each filter point (Figure 9-66).

This command is useful, for instance, when you have a set of nominal points on a design surface that indicates where you'd like to measure, and you have a surface scan of the part. The command will filter the clouds to those points.

To Filter Clouds to Reference Points:

1. From the menu, select **Construct>Points>From Cloud Points>Filter Clouds to Points**.
2. Select the points to use as filter points.

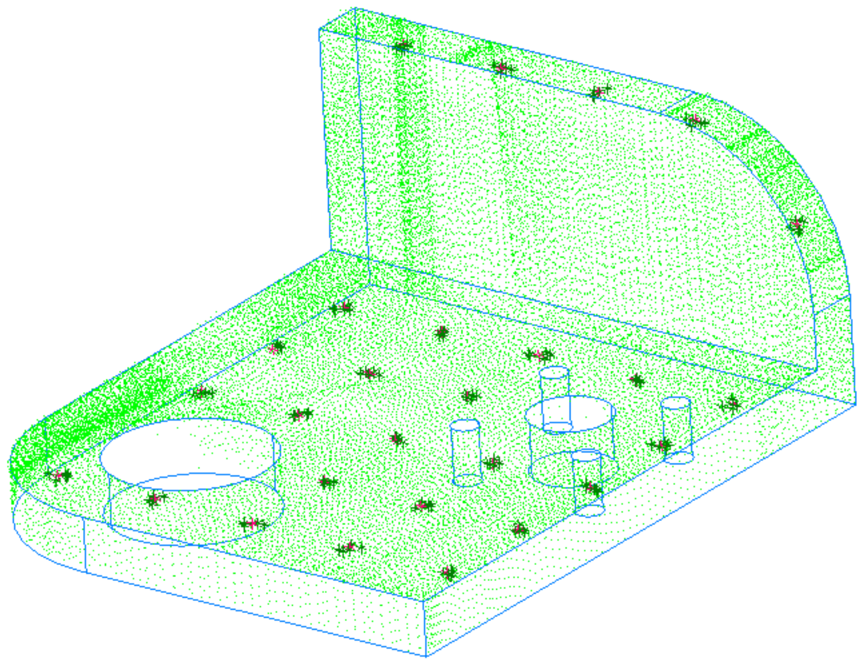


Figure 9-66. Filtering a cloud to a set of reference points. Here, up to 5 points were created for each reference point.

3. Pick one or more clouds to filter.
4. At the prompt, specify the resulting group name.
5. Enter a filter proximity—the maximum distance that a resulting cloud point can be from a filter point. Entering **0** will create a point only at the closest cloud point.
6. Specify the maximum number of points to create for each filter point. Entering **0** will pick all points within the prescribed proximity.

Note: You may wish to average the resulting points after this operation.

Filtering Clouds to a Plane

You can filter one or more clouds to a plane. After specifying a proximity to the plane, SA will create a point group or cloud containing all cloud points within the prescribed proximity to the plane (Figure 9-67).

This is particularly useful for creating clean cross-sections from surface scan data.

To Filter Clouds to a Plane:

1. From the menu, select **Construct>Points>From Cloud Points>Filter Clouds to Plane**.
2. Select the desired filter plane.

3. Select one or more clouds to filter.

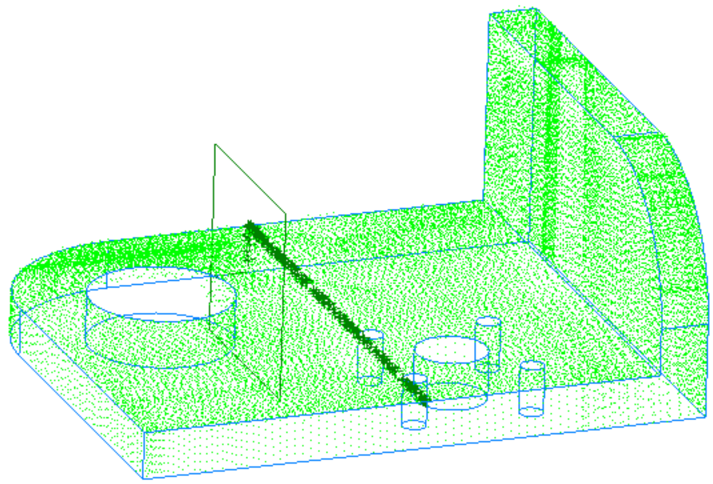


Figure 9-67. Filtering a cloud to a plane.

4. At the prompt, enter the desired destination group name (or cloud name, if you choose to create a cloud).
5. Specify a filter proximity, which indicates how close a cloud point must be to the plane to be used. If you enter `0`, all source cloud points will be used.
6. The ***Filter Clouds to Plane*** dialog will appear (Figure 9-68):

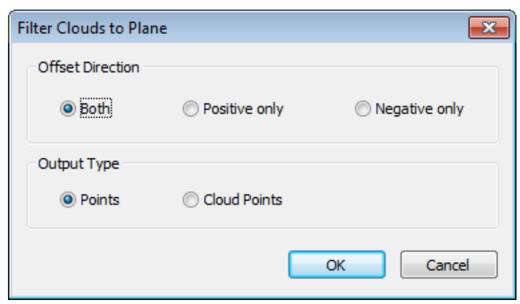


Figure 9-68. Options for filtering clouds to a plane.

- **Offset Direction.** Determines the location of the proximity region. *Both* accepts points up to the prescribed filter proximity on both sides of the plane. *Positive only* only accepts points on the positive side of the plane, and *Negative only* only accepts points on the negative side of the plane.
- **Output Type.** Determines the format of the resulting points—should they be *Points* or *Cloud Points*?

7. Select the desired options, then click OK.

Filtering Clouds to Vectors by Resolving Points

Given a set of clouds and a set of vectors, SA can filter the clouds to the vectors. You provide a desired threshold around each vector, a proximity to the vector's base, and the minimum number of required points, and SA will—for each vector—extract the points that meet the specified thresholds and average them into a single point (Figure

9-69).

This command is useful, for instance, when you have a set of nominal vectors on a design surface that indicates where you'd like to measure, and you have a surface scan of the part. The command will filter the clouds to those vectors.

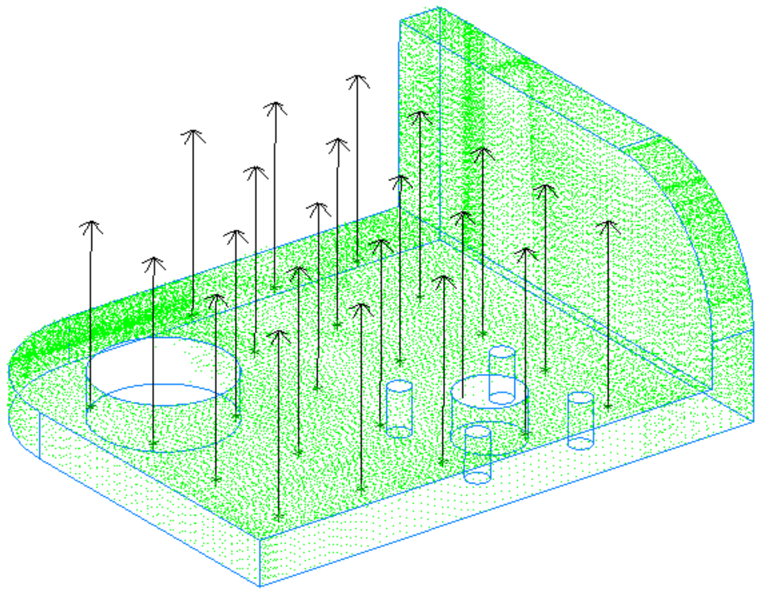


Figure 9-69. This cloud has been filtered to nominal vectors.

To Filter Clouds to Vectors:

1. From the menu, select **Construct>Points>From Cloud Points>Filter Clouds to Vectors-Resolve Points**.
2. Pick the clouds you'd like to filter.
3. Select the vector groups you'd like to filter to.
4. The ***Filter Clouds to Vectors*** dialog will be displayed (Figure 9-70):

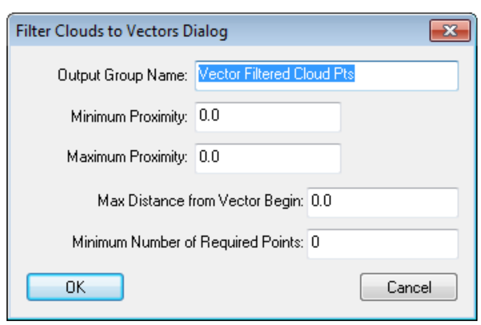


Figure 9-70. The ***Filter Clouds to Vectors*** dialog.

- **Output Group Name.** The name for the resulting point group.
- **Minimum Proximity.** The minimum allowable distance from a point to the vector to be considered.
- **Maximum Proximity.** The maximum allowable distance from a point to the vector to be considered.
- **Max Distance from Vector Begin.** The maximum distance

Note: *Max Distance from Vector Begin* prevents ambiguity when there are two or more coaxial vectors.

that a cloud point can be from the base of the vector to be considered.

- **Minimum Number of Required Points.** The minimum number of points required for an averaged point to be created. If this minimum is not met, no averaged point is created.

5. Enter the desired options, then click OK.

Extracting Center Points From Scanned Spheres

Given a raster scanned point cloud, SA can extract the centers of registration spheres from the scan data. You provide the command with the expected sphere diameter, acceptable RMS tolerance, and minimum number of points required, and the command will detect spherical data matching the proposed diameter from the scan data, fit spheres to them, and create their center points (Figure 9-71).

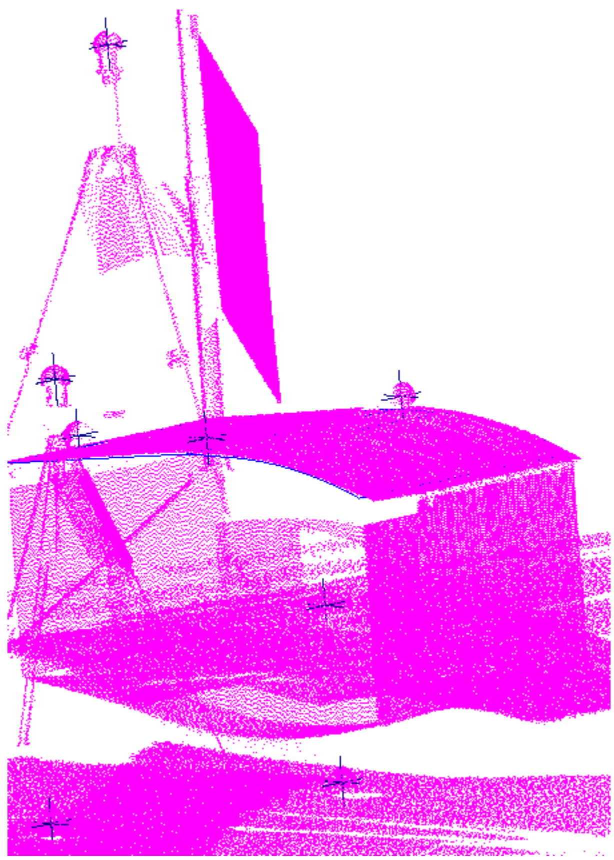


Figure 9-71. Registration sphere centers have been automatically extracted from this scan.

To Auto-Detect Sphere Centers from Raster Scan Data:

1. From the menu, select **Construct>Points>From Cloud Points>Auto-Detect Sphere Centers**.
2. The ***Sphere Extraction Settings*** dialog appears (Figure 9-72):

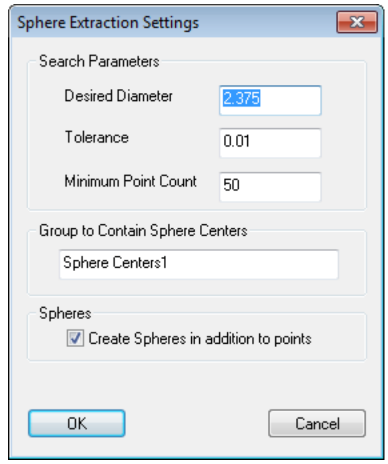


Figure 9-72. Specifying sphere extraction parameters.

- **Desired Diameter.** The nominal diameter of the registration spheres.
- **Tolerance.** The maximum allowable deviation for a given point from the desired diameter in order to be considered.
- **Minimum Point Count.** The minimum number of points required to satisfy the tolerance to consider a sphere to be "found".
- **Group to Contain Sphere Centers.** The name for the resulting group to contain the sphere centers.
- **Create Spheres in addition to points.** When enabled, SA will create the spheres (at the desired diameter) in addition to the center points.

3. Select the desired options, then click OK.
4. After calculation, SA will display the ***Sphere Auto-Detection Results*** dialog (Figure 9-73). This contains the RMS error, maximum error, and point count for each detected sphere.

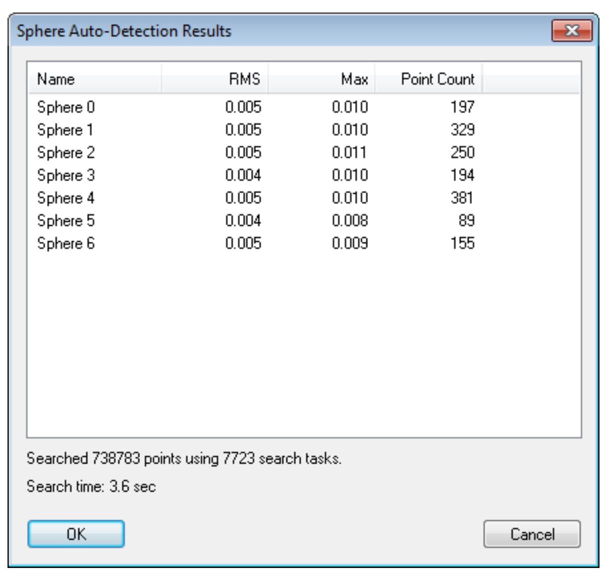


Figure 9-73. Sphere Auto-detection results.

5. Click OK to accept the results, or Cancel to cancel.

Auto-Corresponding Two Groups By Proximity

Given a group of reference points and a group of roughly-aligned corresponding points, this command will copy the corresponding group and automatically rename the copied points to match the names of the reference points.

To Auto-Correspond Two Groups by Proximity:

1. From the menu, select **Construct>Points>Auto-Correspond 2 Groups>Proximity**.
2. When prompted, select the reference group of known point names.
3. Select the group to copy and rename. The ***Autocorrespond Points*** dialog will appear (Figure 9-74):

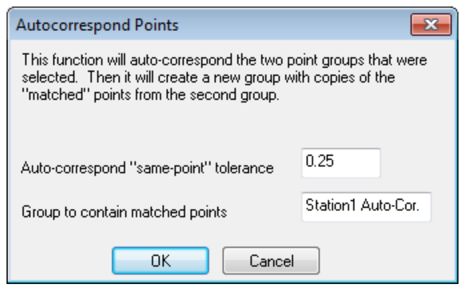


Figure 9-74. Setting parameters for an auto-correspond operation.

- **Auto-correspond "same point" tolerance.** Each point must be within this proximity to a reference point, or it will not be renamed.
- **Group to contain matched points.** The name for the copied group with the corrected point names.

4. SA will propose new names for your review. Click Accept to create a copy of the source point group and assign the appropriate point names.

Auto-Corresponding 2 Groups by Inter-Point Distance

Given a set of reference points and a group of source points, this command copies the source group and renames the points based on the reference point names. This command does not require the source and reference groups to be roughly aligned.

To Auto-Correspond Two Groups by Inter-Point Distance:

1. From the menu, select **Construct>Points>Auto-Correspond 2 Groups>Inter-Point Distance**.
2. Select the reference group.
3. Select the group to be copied. The ***Autocorrespond Points***

dialog will appear (Figure 9-74).

- **Auto-correspond "same point" tolerance.** Once "aligned", each point must be within this proximity to its reference point to be considered a match.
- **Group to contain matched points.** The name for the copied group with the corrected point names.

4. SA will propose new names for your review. Click Accept to create a copy of the source point group and assign the appropriate point names.

Averaging Points

You can create a single averaged point from a set of points.

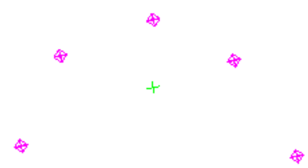


Figure 9-75. The average of 5 selected points.

To Average a Set of Points:

1. From the menu, select **Construct>Points>Fit to Points**.
2. Select the points to average, then press **Enter**.
3. The ***Construct New Point*** dialog appears (Figure 9-22).
4. Specify the name for the resulting point—leave the calculated coordinates alone—then click OK.

Creating Points at the End of a Line

You can construct a point at either or both ends of a line by clicking near the desired end of the line.

To Construct a Point at the End of a Line:

1. From the menu, select **Construct>Points>End of a Line**.
2. The ***Point Name*** dialog will appear (Figure 9-76).

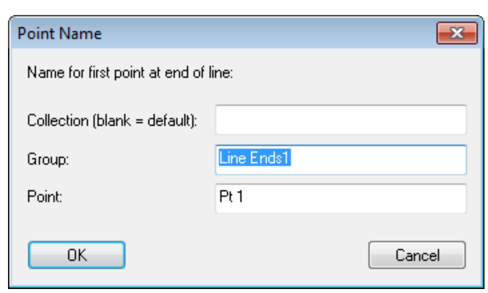


Figure 9-76. The ***Point Name*** dialog.

3. Specify the name for the first created point, then click OK.
4. Click near the end of a line to create the point. When finished

creating points, press **Enter**.

Constructing a Hidden Point

Note: This command assumes you've already defined your hidden point bar in SA. For details, see "Measurement".

If you have measurements on a hidden point bar, you can construct the hidden point. An **A** and **B** point are used to define the vector of the hidden point bar, and the hidden point is constructed along this vector based on the prescribed distance from the **A** point to the hidden point.

To Construct a Hidden Point:

1. From the menu, select **Construct>Points>Hidden Point Bar**.
2. When prompted, select the **A** point.
3. Select the **B** point.
4. The ***Pick Hidden Point Bar*** dialog will appear. Select the appropriate hidden point bar, then click OK.
5. The ***Construct New Point*** dialog will appear (Figure 9-22). Specify the point name, leaving the coordinates alone—then click OK.

Creating Points at a Plane Cross Section:

This command utilizes the database order of a given set of points to create a new set of points at a cross section of a plane.

To Construct Points at a Plane Cross Section:

1. From the menu, select **Construct>Points>Plane Section Crossing.**
2. When prompted, select the ***Group of Points.***
3. Select the ***Section Plane.***

Constructing Points Evenly Spaced on a Surface:

Given a surface, this command allows you to lay out a grid of evenly spaced points along both the U and V directions (Figure 9-77).

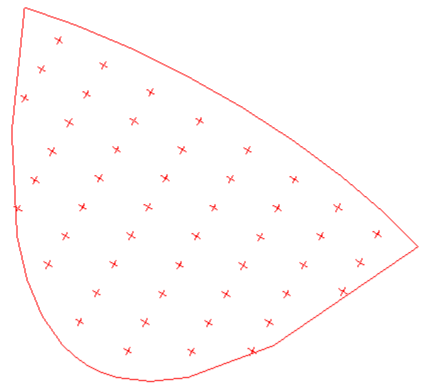


Figure 9-77. Points laid out in ten divisons along U and V.

To Construct a Grid of Points Evenly Spaced on a Surface:

1. From the menu, select **Construct>Points>Mesh Surfaces in UV.**
2. When prompted, select the appropriate surface(s), then press `Enter`.
3. The ***Make Points From Surface*** dialog will appear (Figure 9-78).

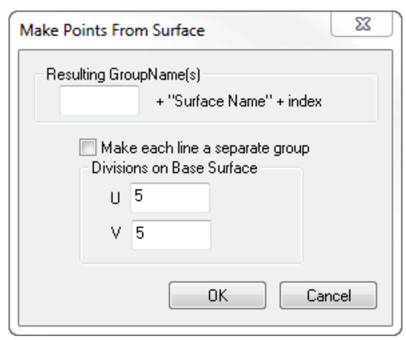


Figure 9-78. The ***Make Points From Surface*** dialog.

4. Specify a prefix for the resulting point group.
5. If you would like each row to be in a separate group, check the *Make each line a separate group* option.
6. Specify the number of divisions along the U and V directions, then click OK.

Note: The resulting point group name will be the prefix specified + "Surface Name" + index.

Constructing Points From Vectors

This command will create a point at the origin of each selected vector (Figure 9-79).

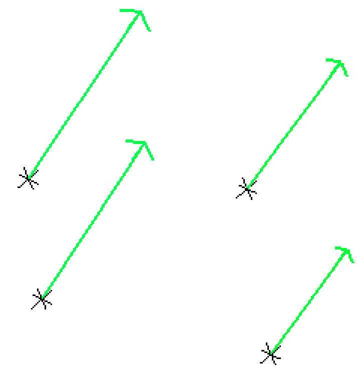


Figure 9-79. A point is constructed at the origin of each vector.

To Construct Points from Vectors:

1. From the menu, select **Construct>Points>From Vectors**.
2. When prompted, select the appropriate vector(s), then press `Enter`.
3. The ***Input*** dialog will appear (Figure 9-80).

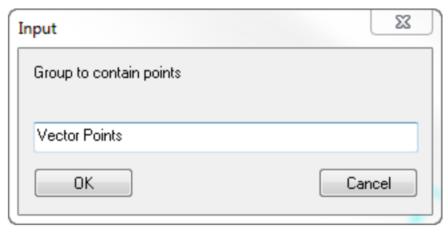


Figure 9-80. The ***Input*** dialog.

4. Specify a name for the point group containing the resulting points, then click OK.

Constructing a Single Point through Patch Normal Shift

This command allows you to project a single point to a defined set of patch points. A theoretical plane is created from the patch points and and the end result is a constructed point at a specified offset along its probing direction (Figure 9-81).

Figure 9-81. A point constructed from a selected point and shifted 0.75" along the probing direction of the theoretical patch point plane.

To Construct a Point by Shifting along Normal:

1. From the menu, select **Construct>Points>Patch Normal Shift>Single Point** or press **Ctrl + M**.
2. When prompted, select the point, then select the patch points you wish to use. Press **Enter**.
3. The ***Add Shifted Point*** dialog will appear (Figure 9-82).

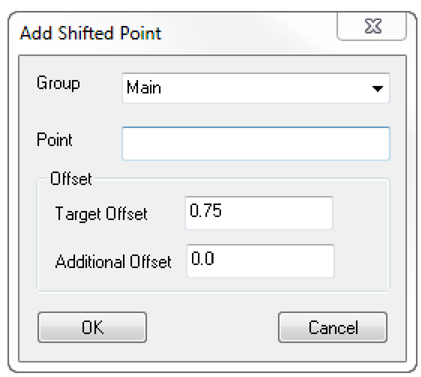


Figure 9-82. The ***Add Shifted Point*** dialog.

Note: The target offset of the selected initial point will populate in the dialog by default.

4. Specify the point group and point name of the resulting point, as well as the target and additional offsets (if applicable).
5. Click OK.

CHAPTER 10

Objects

Anything that has both a position and orientation in space—with the exception of instruments—is considered an object in SA. In this chapter, we'll discuss how to work with objects, and what you can do with them.

Point Groups

When points are created (constructed or measured), they are placed into point groups, often referred to just as groups. Point groups act as "containers" to hold points, and are best used to organize measured data in a logical manner.

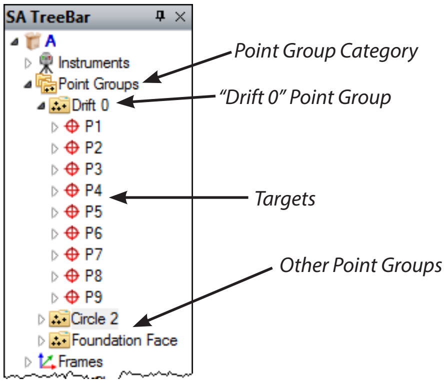


Figure 10-1. Point groups as they appear in the tree.

Point groups are listed under the *Point Groups* category in the treebar (see Figure 10-1). Each individual point group can be expanded to reveal the contained points by clicking the disclosure triangle next to the point group name. Click the triangle again to collapse the list of points.

A point group is an object, whereas an individual point is not. As such, the point group can be moved, scaled, rotated, mirrored, etc. When a point group is manipulated in this manner, all of the points inside the group are treated as a rigid body. For example, if a point group is scaled by a factor of 2, the space between the points inside the group will be doubled. In Figure 10-2, a point group has been copied, then translated up in the Z direction by 10 units.

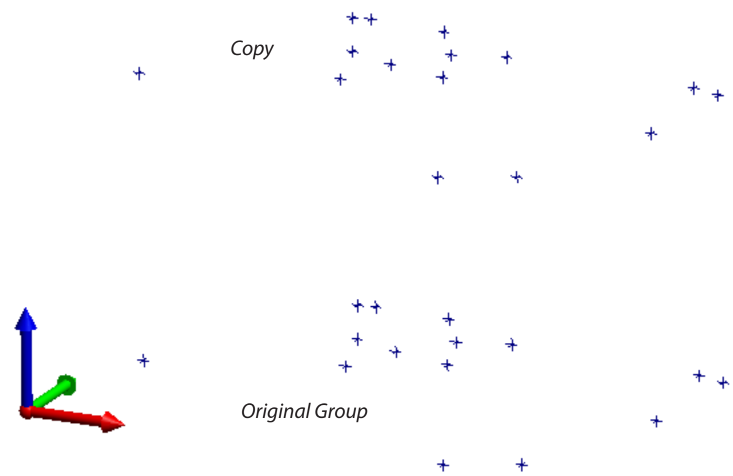


Figure 10-2. A point group has been copied and translated in the positive Z direction.

If a point group contains targets, and the point group is moved, then the targets will temporarily convert to constructed points. You can revert these constructed points back to targets (and their original locations relative to the instrument) by using the **Analysis>Re-Compute Targets From Shots** command.

Points will also inherit some of the properties of their containing group. For example, an object color can be assigned to the point group, and all points will inherit the color of their containing group. If a point group is hidden, then all of the points inside the group are hidden as well.

Frames

SpatialAnalyzer allows you to create a virtually unlimited number of coordinate frames (referred to as frames in SA), each representing a different aspect of your job that is important to you.

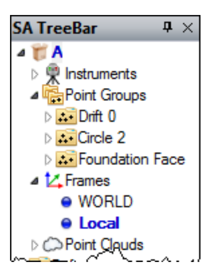


Figure 10-3. This collection has two coordinate frames, **World** and **Local**. The **Local** frame is active (working).

While there can be many coordinate frames in a given SA job, only one can be designated as the *working* frame. When a coordinate frame is designated as the working frame, it will be drawn in a **bold blue font** in the tree (see Figure 10-3 on page 330), and the ***Color/WCF Toolbar*** will display the name of the frame (right).

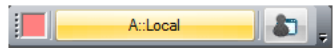
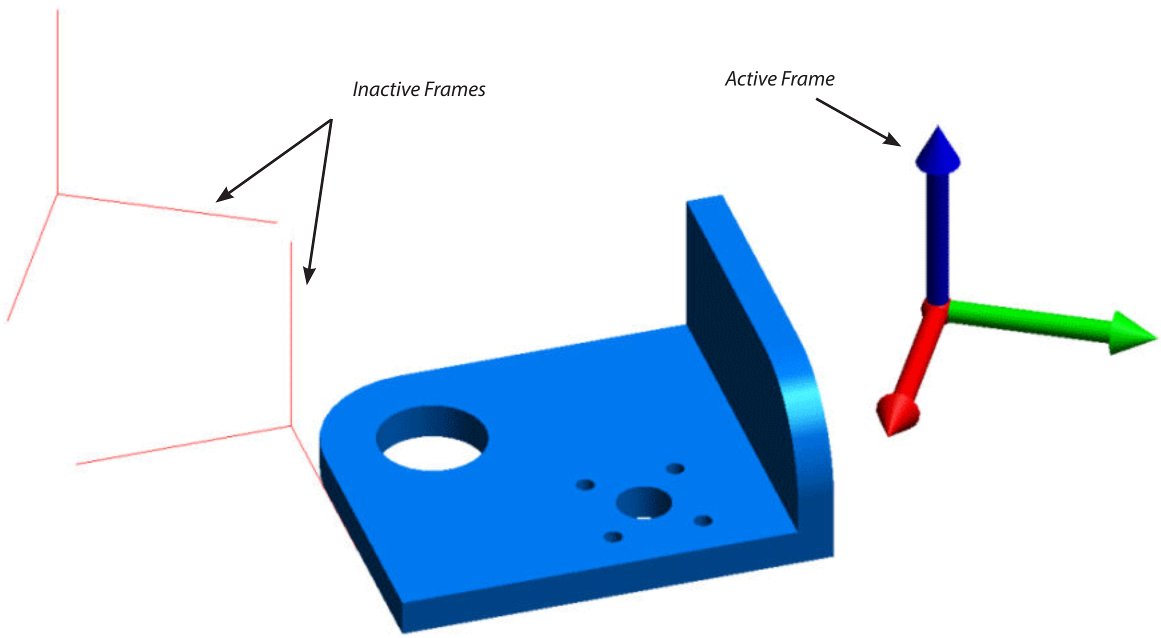


Figure 10-4. The Color/WCF Toolbar.

All coordinates, query results, and other values are always displayed relative to the active coordinate frame (Figure 10-4), unless explicitly specified otherwise. For example, when you view the coordinates of a point, you are seeing the coordinates relative to the active coordinate frame.



To Activate a Coordinate Frame:

Do one of the following:

- Right-click a frame in the graphical view or SA tree and select Make Working Frame.
- Click the WCF button on the ***Color/WCF toolbar*** and choose a new working frame from the ***Object Selection dialog***.
- Right-click a frame in the tree and select **Properties**. Then click the Make Working button.
- From the menus, choose **Edit>Pick Working Frame** and select the frame to activate.
- Press **`Alt+W`** (default hotkey) and choose the desired frame from the ***Object Selection*** dialog.

Frame Properties

Frames, like virtually all entities in SA, have properties associated with them.

To View a Coordinate Frame's Properties:

Do one of the following:

- Right-click the frame in the SA tree or graphical view and choose **Properties**.
- Double-click a frame in the SA tree.
- Click the ***Object Properties*** icon in the ***Main Toolbar***, then double-click a frame in the graphical view.

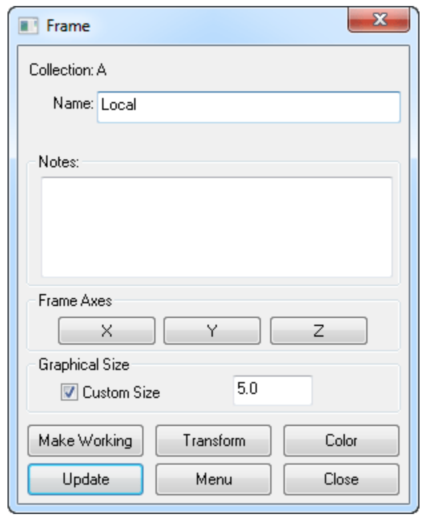


Figure 10-5. The Frame properties dialog.

- **Collection.** Indicates the collection in which the frame resides.
- **Name.** The name for the coordinate frame.
- **Notes.** The notes field allows you to add text-based notes to a coordinate frame, which you can reference at a later time.
- **Frame Axes.** By clicking on the X, Y, or Z buttons, you can view the projected angle of the selected coordinate axis, presented relative to the working coordinate frame (Figure 10-6).

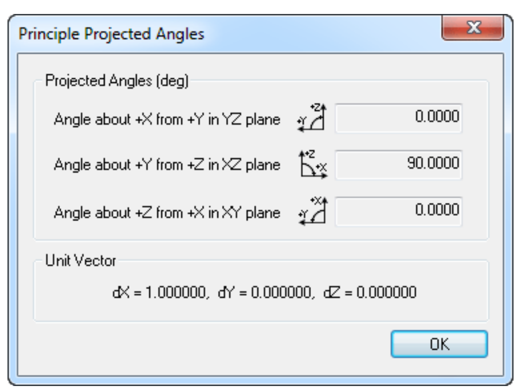


Figure 10-6. A frame's principle projected angles.

- **Custom Size.** Enable this option to assign a custom size to the coordinate frame, and enter the size in the resulting text field.

- **Make Working.** Sets the coordinate frame active so that it becomes the working coordinate frame.
- **Transform.** Opens the transformation dialog to adjust the position and orientation of the frame.
- **Color.** Opens the color dialog so that you can assign a new color to the coordinate frame.
- **Update.** Updates SA with any changes in the object properties without actually closing the ***Frame*** dialog. Note that closing the dialog also applies changes to SA.
- **Menu.** Exposes object-specific menu options. This is equivalent to right-clicking the object in the tree.
- **Close.** Closes the properties dialog.

Note: The assigned color of the frame is only shown when it is not the working frame.

Frame Construction

SpatialAnalyzer offers many methods for constructing Frames. To fully understand frame construction it is important to understand the components needed to construct a frame. A frame has four basic components: an origin point, primary axis, secondary axis and tertiary axis. A frame can usually be constructed by defining an origin point and two of the axes.

During frame construction many commands will prompt the user to define the primary, secondary and tertiary axis. An axis can be described by a point or a vector (object direction). When using a point to describe an axis, a vector is formed from the origin point and the selected point. If an object is selected its normal will be used to define the axis. The following examples show several combinations of points and vectors.

Enter

Creates a frame at the origin and orientation of the active frame. The transform dialog will appear allowing the user to enter the position and orientation values.

Plane, Line, Point (3-2-1)

A 3-2-1 frame consists of a locator point for the origin, line/vector for one axis and plane for the second axis (see Figure 10-7 on page 334). The Plane, Line, Point Transformation dialog allows control over the axis for the plane and line/vector. The user may also specify an alternate coordinate for the locator point.

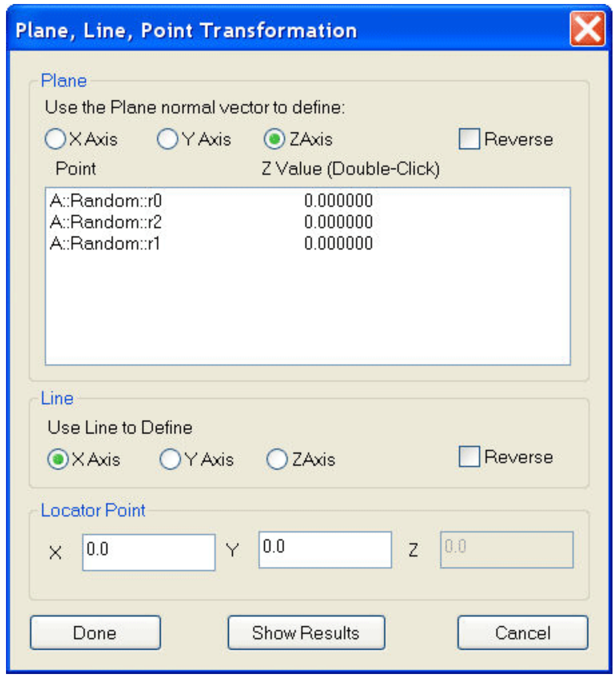


Figure 10-7. The Plane, Line, Point Transformation dialog.

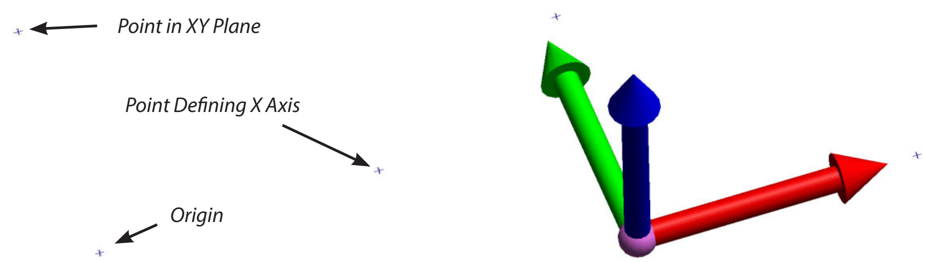


Figure 10-8. Points used to define a 3-2-1 frame (left) and the resulting constructed frame (right).

Plane, Line, Line (Sheet Metal Corner)

Uses a plane and two lines to define a frame. A user can select the axis for each object selected.

Using Points and Vectors

Example using only three points. **Construct>Frame>3 points>Origin, X-axis>Point on XY plane**

3 Points

Uses three point to describe the origin, primary axis and secondary axis.

Z-Axis

Constructs a frame with the Z-axis normal to a plane, circle or a line and uses the selected points for origin and X-axis.

On Instrument

This will construct a frame on the instrument base with the Z-axis being the standing axis. A frame can also be made on the "scope or end effector". This will generate a frame using the probing direction as the vector for the Z-axis.

Using Working Frame Orientation

Uses the current active frame orientation to define a new frame. A copy can be made by defining a new origin point or maintain the Z-axis and clock the remaining axis.

3 Planes

Uses three planes to create a coordinate system. User selects X, Y and Z planes. The user interface allows control over priority and direction.

Mirror Cube

Often, users wish to construct a coordinate frame based on the faces of a mirror cube (see Figure 10-9). The Mirror Cube frame assignment method provides a convenient method for repeatedly and accurately locating a coordinate frame with two instruments (typically theodolites). SpatialAnalyzer contains a powerful mirror cube frame assignment tool.

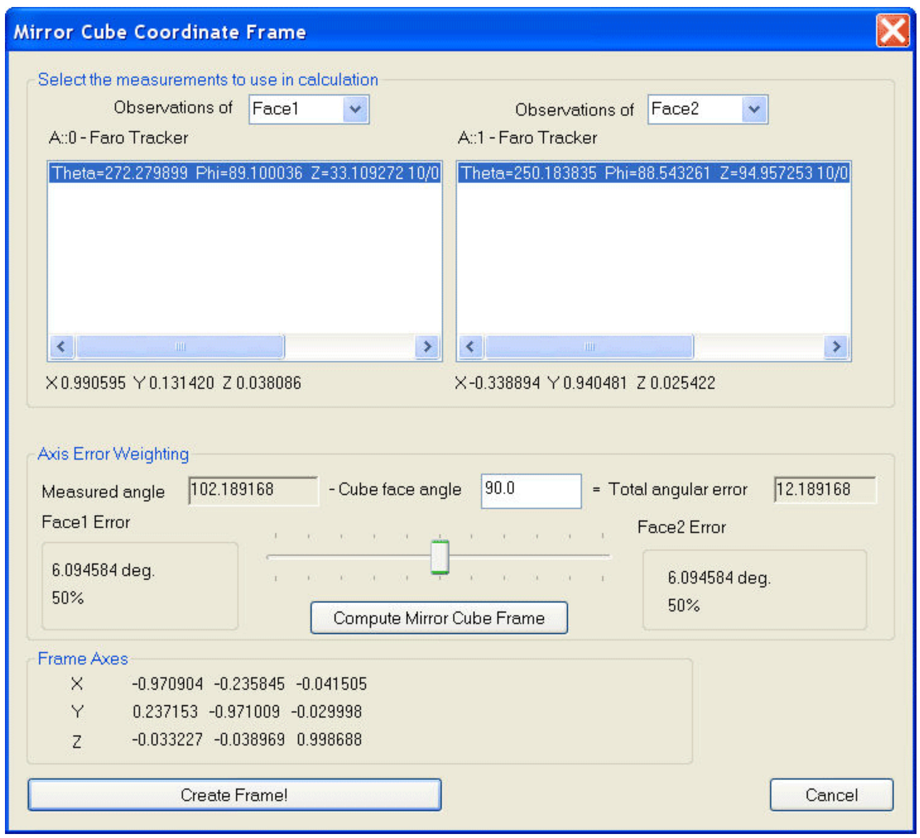


Figure 10-9. The Mirror Cube Coordinate Frame dialog.

To accomplish a mirror cube frame assignment, point two theodolites at the mirror cube and collimate the instruments with the mirror faces. Record a shot from each theodolite at the cube using the same target name, just as if you were shooting the mirror as a target. If you desire reversed shots, reverse the scope and take another shot, or take multiple shots in each face. In SpatialAnalyzer, select Construct, Frame, and Mirror Cube. You will be prompted to select the target for the mirror. Pick the common target name you used for the measurements. The mirror cube frame assignment dialog will appear. At this point, you may press the Create Frame button to make the coordinate frame, or you may change the options.

Possible Options:

- You may select which instrument represents the X-axis of the coordinate frame.
- You may select or deselect specific shots from each instrument. When there are multiple shots, their vector directions are averaged to calculate the vector displayed below the measurement box.
- The check box for coordinate axes determines whether the axes of the frame are outward along the theodolite vectors or inward toward the theodolites.
- The Axis weighting box allows you to control the distribution of the errors between the two axes.
- Set the Cube face angle (normally 90 degrees). Press Compute after doing this to update!
- The difference between the cube "ideal" angle and the theodolite vector angle will be displayed.
- Use the slider bar to control how far the frame axes are away from the theodolite axes. Sliding the bar to the left places more of the error on the "right-side" instrument.
- Choose carefully when distributing the error. In some cases, it may be better to have no error on one axis since it is difficult to tell if the "error" is coming from the theodolite measurements or the mirror cube itself.

On Object

Constructs a frame with the Z-axis normal to the selected object.

Fit to Coordinate Components

In this method of frame assignment, we assume that we have several points which share a known X coordinate, several points which share a Y coordinate, and several points which share a known Z coordinate. This is often used in a factory setting where a designer has established a reference block with one face representing a certain XY, one plane representing a certain XZ plane, and one face representing a certain YZ plane. Initially the user will be prompted for the points of known X value, then Y and Z in turn. After selection of the points is completed, the user will see the dialog shown below. Enter the known values and hit OK. The frame assignment will be made.

Average of Other Frames

Creates a frame based on the average of selected frames. Usually used to create an average between gravity frames.

Copy and Make Left-Handed

Makes a copy of the selected frame and prompts for the axis to reverse.

Spheres

Fit to Points

Constructs a best-fit sphere for the selected points. See the Geometry Fit section for more information.

Enter

Constructs a sphere at a position and orientation relative to the working frame. Use the ***Sphere Properties*** dialog to edit parameters.

Sphere Section

Creates a spherical section. Use the ***Sphere Section*** dialog to edit parameters.

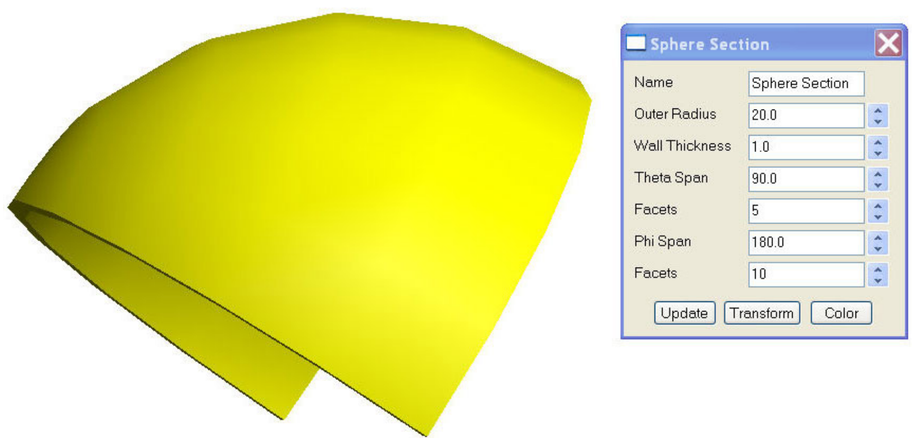


Figure 10-10. Creating a spherical section.

B-Splines

Properties

The B-Spline properties will report the Curve Parameters and allow directional arrows to be drawn.

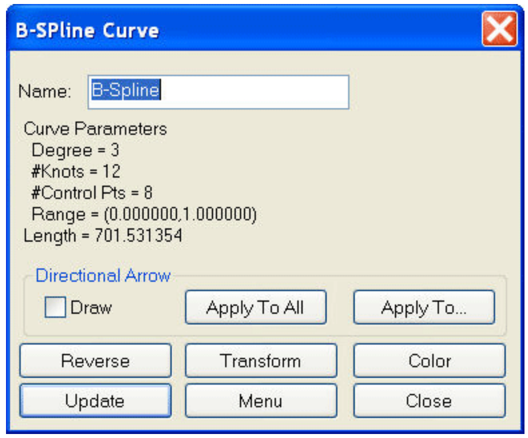


Figure 10-11. The B-Spline curve properties dialog.

Construction

From Points

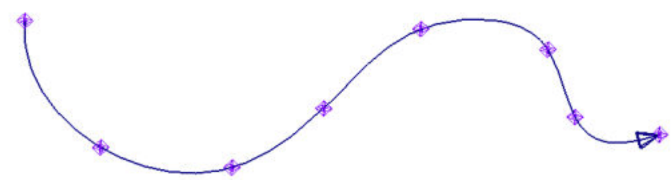


Figure 10-12. A B-Spline fit to a series of measured points.

Creates a B-Spline curve from selected points using either an interpolated fit or cubic fit (see Figure 10-12 and Figure 10-13).

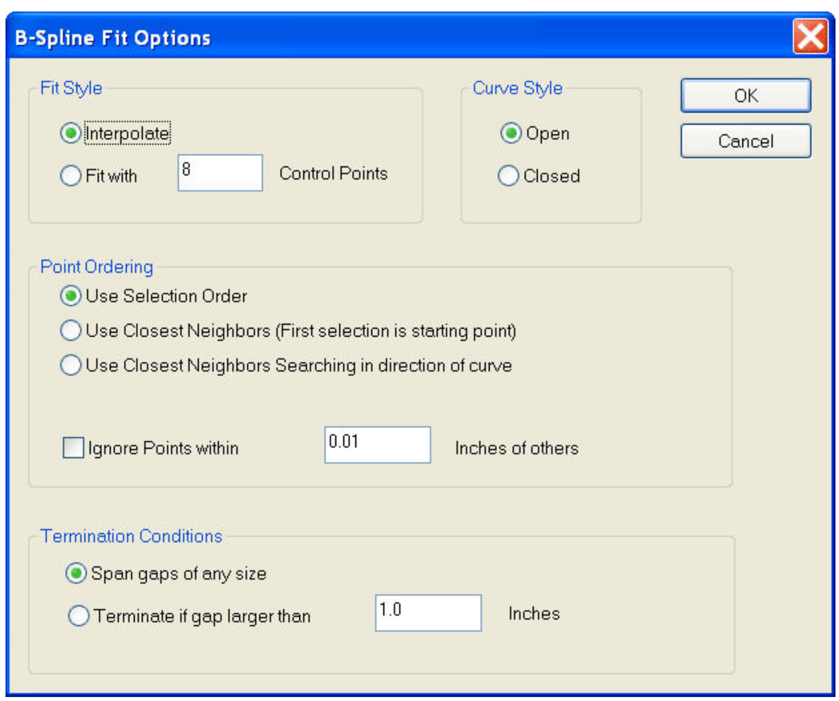


Figure 10-13. B-Spline Fit options.

- **Fit Style.** A B-Spline can be fit one of two ways, **Interpolation** or **Fit with control points**. An interpolated fit will pass through all of the selected points and interpolate the curve's shape between each point. Fitting with control points allows for a best fit.

- **Curve Style.** If closed the B-spline will end at the beginning point of the B-spline.
- **Point Ordering.** The direction and path of the B-spline is determined by the order of the points. The user can select the points in order or use the closest neighbor approach. Points can also be ignored if their neighbor is within a prescribed distance.
- **Termination Conditions.** The B-Spline creation will terminate if a gap of a prescribed distance is exceeded.

From Surface

Creates B-Splines from surface boundaries (Figure 10-14).

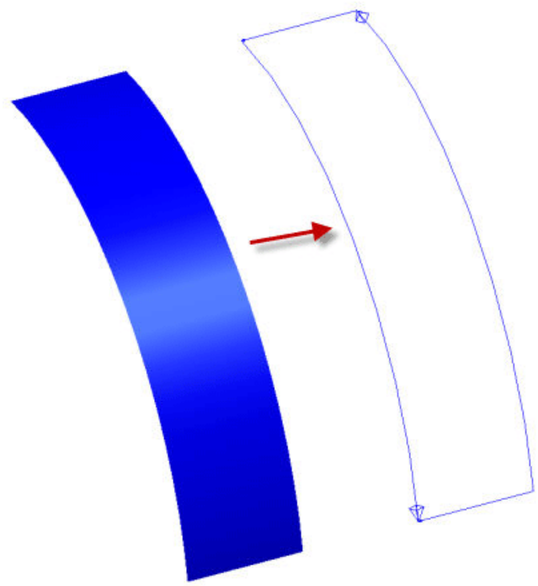


Figure 10-14. Creating B-Splines from the boundaries of a surface.

Intersect Surface

Creates a B-Spline from the intersection of two surfaces (Figure 10-15).

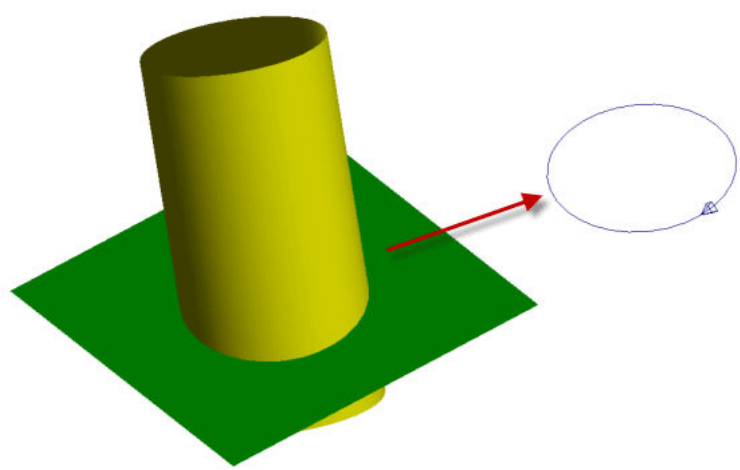


Figure 10-15. This B-Spline has been created by intersecting cylindrical and planar surfaces.

Intersect Plane and Surface

Creates a B-Spline at the intersection of a plane and surface. Very similar to the **Intersect Surface** command.

From a Collection of B-Splines

Creates one single B-Spline from the selected individual B-Splines.

Circles

Properties

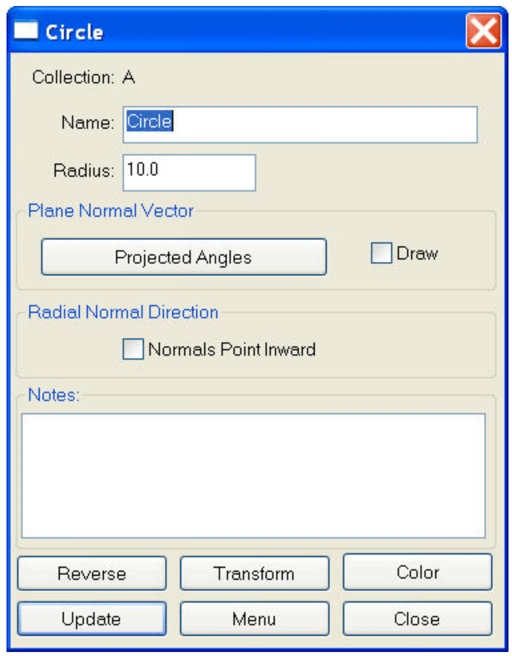


Figure 10-16. The properties dialog for a circle.

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name
- **Radius.** Radius of circle
- **Projected angles.** Displays the Principle Projected Angles dialog. These angles represent the projection of the object normal vector onto the coordinates planes of the active frame.
- **Draw.** Displays the normal vector in the graphics.
- **Radial Normal Direction.** Reverses the normal radial vector direction of the circle.
- **Notes.** Contains information entered by user or information regarding construction method, fit, etc.
- **Reverse.** Reverses the normal direction of the object.
- **Transform.** Opens the transformation dialog.

- **Color.** Opens color dialog and allows the object color to be changed.
- **Update.** Updates SA with any changes in the object properties.
- **Menu.** Exposes object specific menu options. Emulates right-clicking the object in the tree.
- **Close.** Closes the object properties dialog.

Construction

Fit to Points

Constructs a best-fit circle for the selected points. See the Geometry Best-Fit section for more information.

Enter

Constructs a circle where the normal is in the Z-axis of the working frame. Use the ***Circle Properties*** dialog to edit parameters.

Maximum Material Condition (from Scan Patch)

Creates a best-fit circle with a diameter that bounds all of the selected points. All of the points will either be touching the circle or inside of the circle.

From CAD Surfaces

Uses selected CAD surfaces to look for circular segments and creates circles based on the search criteria.

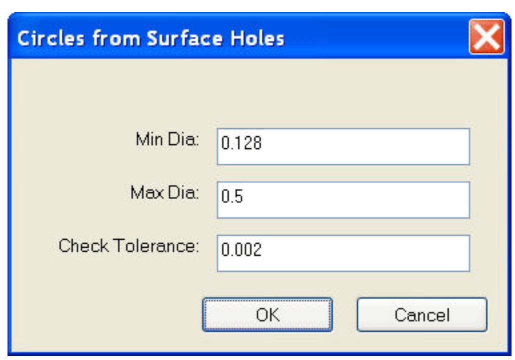


Figure 10-17. Creating circles from surface holes.

Normal to Object, Centered at Point

Creates a circle using the selected object’s normal and is centered at the selected point.

Planes

Construction

Fit to Points

Constructs a best-fit plane for the selected points. See the Geometry Best-Fit section for more information.

Enter

Constructs a plane where the normal is in the Z-axis of the working frame. Use the ***Plane Properties*** dialog to edit parameters.

Normal to Object, Through Point

Plane created normal to a selected object and passes through a selected point.

Normal to Line, Through End

A plane is created that is normal to the line and passes through a selected end of line.

Mirror from Two Measured Points

The mirror plane function is used to determine the location of a mirror by measuring a point both directly and through the mirror.

1. Pick the instrument.
2. Pick the point measured directly (not through mirror).
3. Pick the virtual point that is really the same point, but measured through the mirror.

SA creates a plane representing the mirror location (see Figure 10-18 on page 343). You can make a frame on this plane if needed.

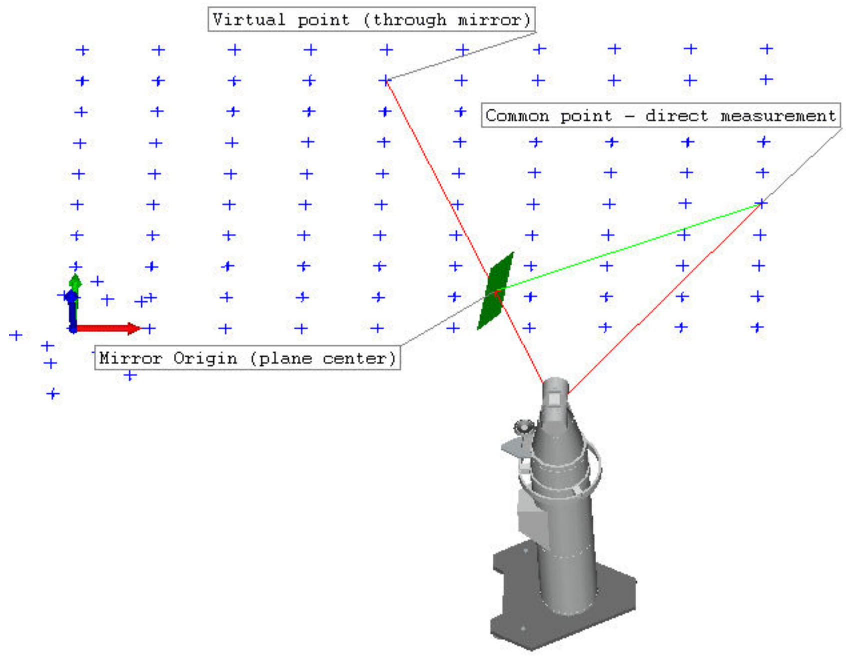


Figure 10-18. Creating a plane from two measured points using a mirror.

If the points used in the construction have uncertainty fields (**Analysis>Coordinate Uncertainty>Create Point Uncertainty Fields**), SA will automatically compute the uncertainty of the mirror plane and add the results to the notes field for the constructed plane.

Establish Bounding Planes for Points

Creates two planes parallel to the selected plane and are spaced so that they bound the selected points.

From Surface Faces

Creates planes from the selected surface faces.

Lines

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **End-Point Method.** Enter two point coordinates to determine length, position and orientation of the line.
- **Ray Method.** Control the length or the transformation of the line.
- **Projected angles.** Displays the Principle Projected Angles dialog. These angles represent the projection of the object normal vector onto the coordinates planes of the active frame.

- **Draw.** Displays the normal vector in the graphics.
- **Notes.** Contains information entered by user or information regarding construction method, fit, etc.
- **Reverse.** Reverses the normal direction of the object.
- **Shift Ends.** Enter values to shift beginning and end point along the vector of the line.
- **Color.** Opens color dialog and allows the object color to be changed.
- **Update.** Updates SA with any changes in the object properties.
- **Menu.** Exposes object specific menu options. Emulates right-clicking the object in the tree.
- **Close.** Closes the object properties dialog.

Construction

Fit to Points

Constructs a best-fit line for the selected points. See the Geometry Best-Fit section for more information.

Enter

This will allow you to enter the end-points of the line or designate a direction and a length. These values are entered using the Line Property dialog.

2 Points

Constructs a line between two selected points. The lines direction is from the first to second point.

Projection, Line to Plane / Circle

After selecting either a plane or a circle as the destination, you will be prompted to select the line you wish to project. A new line will be created on the projection surface.

Between a Point and a Line

This function creates a line between a user-selected point and a line.

2 Plane Intersection

A line is created at the intersection of two planes. If the planes are parallel, the function reports that the line can not be created.

Center of Cylinder

Constructs a line from the cylinder end points and inherits the cylinder's direction.

Along Frame Axes

Constructs a line down each axis of the working frame.

From Instrument Shot

Creates a line from a measured point and its associated instrument. The point selected must be a measured point.

Normal to Object(s)

Creates a line normal to an object. All objects (plane, line, cylinder, etc) have a normal direction. The line created will represent that normal direction.

Normal to Objects, through Point

Creates a line parallel to the selected object's normal and intersects to selected point.

Parallel to Line, through Point

Creates a line parallel to the selected line and pass through the selected point.

Cross Product

The cross product is a function that takes two lines (vectors) and produces a line that is perpendicular to them.

From a Vector Group

Creates lines from each vector in a vector group.

Cylinders

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **Transformation Method.** Controls the length or the transformation of the cylinder.
- **End-Point Method.** Enter two point coordinates to determine length, position and orientation of cylinder.
- **Diameter.** Diameter of the cylinder.
- **Surface Normals.** Reverses the surface normal direction.
- **Projected angles.** Displays the Principle Projected Angles dialog. These angle represent the projection of the object normal vector onto the coordinates planes of the active frame.
- **Draw.** Displays the normal vector in the graphics.

- **Notes.** Contains information entered by user or information regarding construction method, fit, etc...
- **Reverse.** Reverses the normal direction of the object.
- **Color.** Opens color dialog and allows the object color to be changed.
- **Update.** Updates SA with any changes in the object properties.
- **Menu.** Exposes object-specific menu options. Emulates right-clicking the object in the SATreebar.
- **Close.** Closes the object properties dialog.

Construction

Fit to Points

Constructs a best-fit cylinder for the selected points. See the Geometry Best-Fit section for more information.

Enter

Constructs a cylinder where the normal is in the Z-axis of the working frame. Use the ***Cylinder Properties*** dialog to edit parameters.

Cylinder Slice

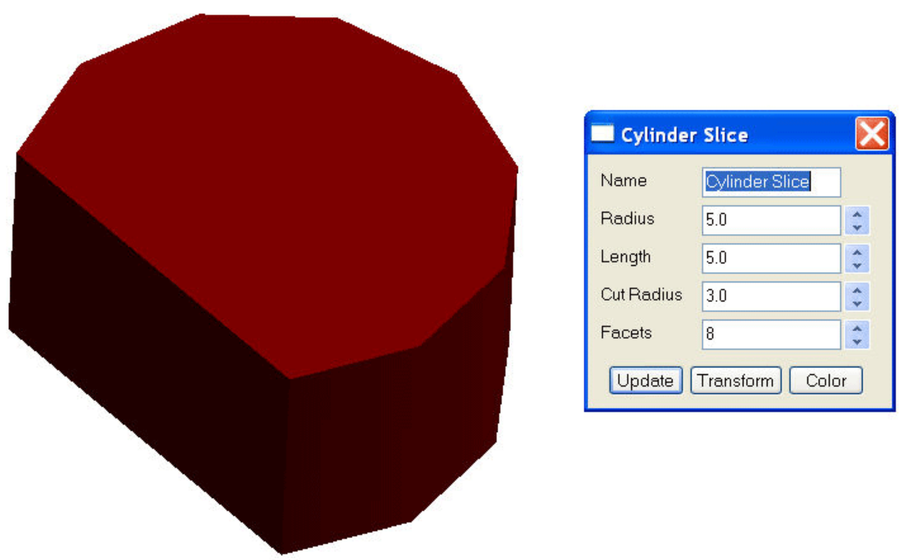


Figure 10-19. Creating a cylinder slice.

Pie Wedge

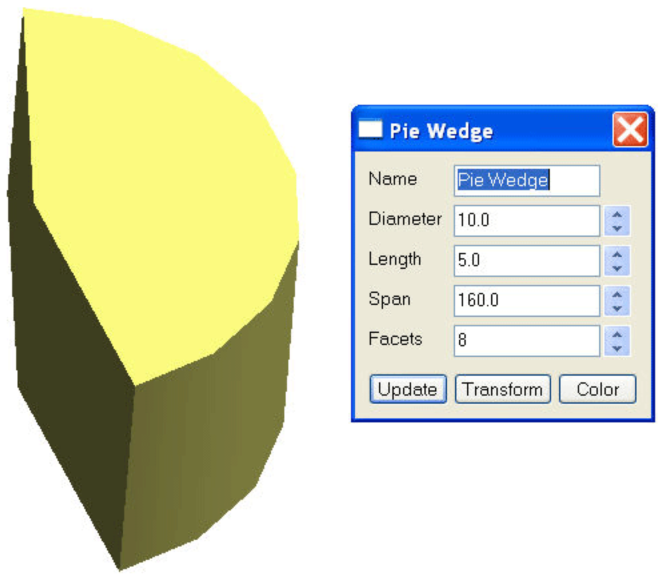


Figure 10-20. Creating a pie wedge.

Pipe

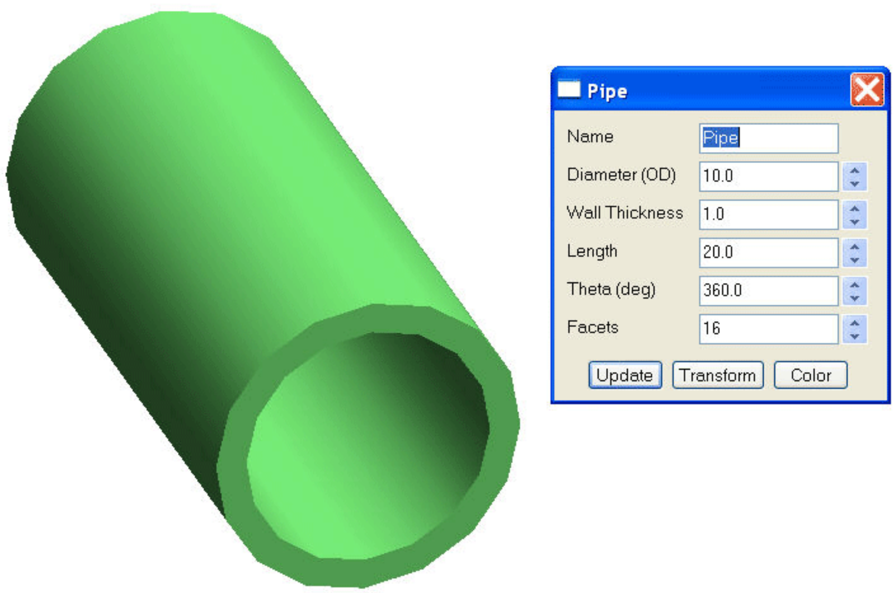


Figure 10-21. Creating a pipe.

Tapered Pipe

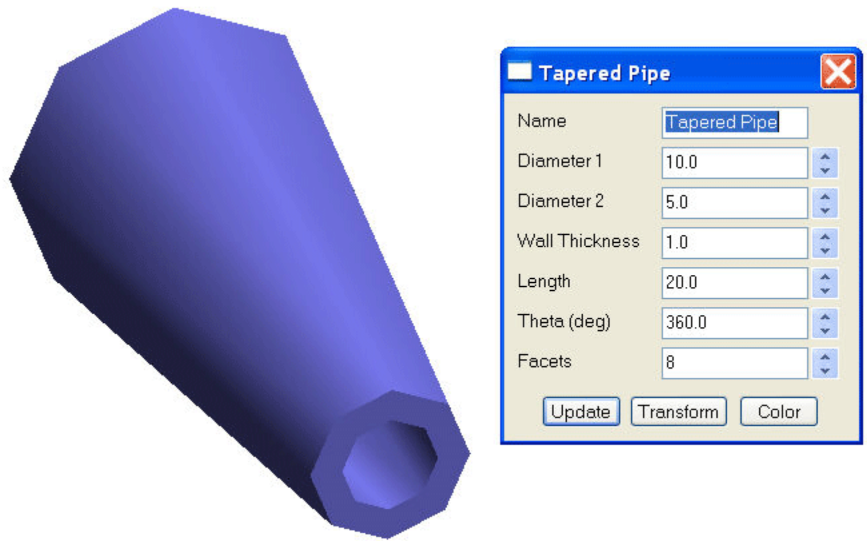


Figure 10-22. Creating a tapered pipe.

Ellipses

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **Transform.** Opens the transformation dialog.
- **Major Axis Radius/Minor Axis Radius.** The major (long) and minor (short) axis lengths (Figure 10-23).

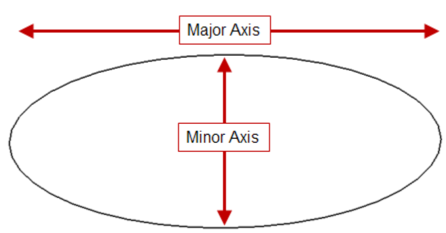


Figure 10-23. The major/minor axes of an ellipse.

- **Notes.** Contains information entered by user or information regarding construction method, fit, etc...
- **Update.** Updates SA with any changes in the object properties.
- **Color.** Opens color dialog and allows the object color to be changed.

Construction

Fit to Points

Constructs a best-fit ellipse for the selected points. See the Geometry

Best-Fit section for more information.

Enter

Constructs an ellipse about the working frame. Use the ***Ellipse properties*** dialog to edit parameters.

Paraboloids

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **Graphical Settings.** Control the length, span and facets of the graphical representation of the cone.
- **Geometry Parameters.** Control length and alternative parameters.

Construction

Fit to Points

Constructs a best-fit paraboloid for the selected points. See the Geometry Best-Fit section for more information.

Enter

Constructs a Paraboloid about the working frame, use the ***Plane properties*** dialog to edit parameters.

Cones

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **Graphical Settings.** Control the length, span and facets of the graphical representation of the cone.
- **Included Angle.** The entire angle that contains the taper.
- **Surface Normals.** Reverses the surface normal direction.
- **Notes.** Contains information entered by user or information regarding construction method, fit, etc.
- **Transform.** Opens the transformation dialog.

- **Color.** Opens color dialog and allows the object color to be changed.
- **Update.** Updates SA with any changes in the object properties.
- **Menu.** Exposes object specific menu options. Emulates right-clicking the object in the tree.
- **Close.** Closes the object properties dialog.

Construction

Fit to Points

Constructs a best-fit cone for the selected points. See the Geometry Best-Fit section for more information.

Enter

Constructs a cone about the working frame, use the ***Cone properties*** dialog to edit parameters.

Surfaces

Offset Surfaces

Currently under construction...however, this will allow a user to select a surface to offset and enter an offset.

From a Collection of Surfaces

Merges selected surface into one surface entity.

By Dissecting Surfaces

Dissects the surface into individual surfaces. The user has the option to dissect any surface or pick surface faces to dissect.

Select Faces by Projecting Points

Creates separate surfaces by checking point projections. This can be useful for speeding analysis when trying to reduce surface faces to analyze.

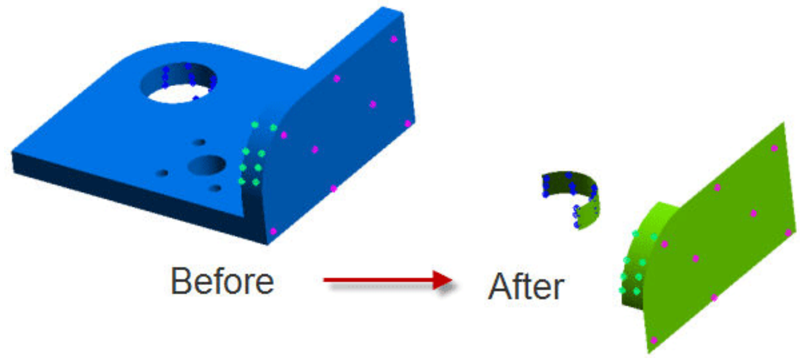


Figure 10-24. Creating surfaces by projecting points.

From Objects

Creates surfaces from supported SA Geometry. Examples are plane, cylinders, etc...

From Plane, Cylinder, Sphere, and Cone

Performs same functionality as above, these are just geometry specific. The above command is more universal.

Tubular Surface from a B-Spline

Creates a tubular surface that uses the B-Spline path. Users has the ability to enter Tubular Diameter.

From Point Groups

Functionality only available in SA Ultimate. Please see the Advanced Surface Fitting section.

From B-Splines

Functionality only available in SA Ultimate. Please see the Advanced Surface Fitting section.

From Plane and Closed B-Spline

Functionality only available in SA Ultimate. Please see the Advanced Surface Fitting section.

Fit from Bounding Curves and Internal Points

Functionality only available in SA Ultimate. Please see the Advanced Surface Fitting section.

Fit from Nominal Face and Actual Data

Functionality only available in SA Ultimate. Please see the Advanced Surface Fitting section.

Polygonized Meshes

From Surfaces

Converts selected surface to a Polygonized Mesh.

From Cloud Points and from Selected Cloud Points

Generates a Polygonized Mesh from selected point clouds or points. By default the mesh orientation uses the current Point of View, however, the working frame can be used as well (Figure 10-25). Enter the grid resolution based on point cloud density. Rule of thumb, the data density should be four times greater than the grid resolution.

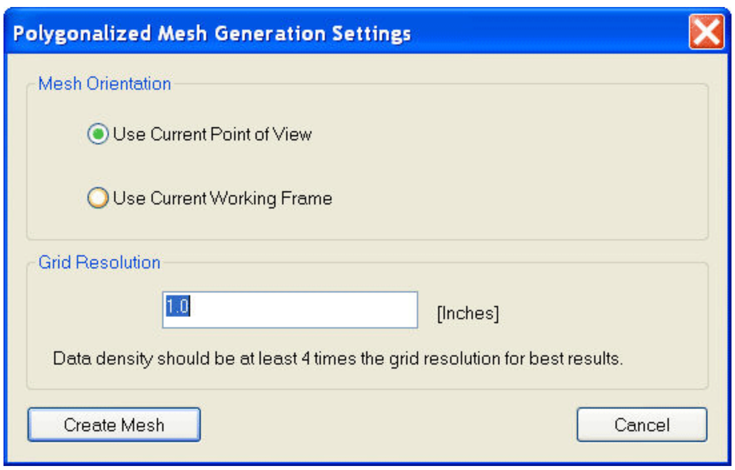


Figure 10-25. Polygonized mesh generation settings.

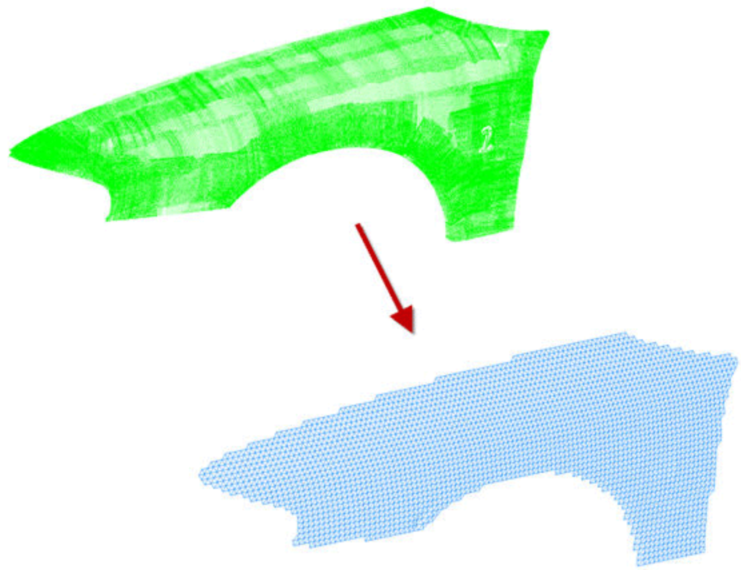


Figure 10-26. Generating a polygonized mesh from a point cloud.

Colorized Mesh from Point Proximity

Creates a Colorized Polygonized Mesh depicting the error between the selected points and the selected surface (Figure 10-27). The colorization options can be accessed through the Polygonized Surface properties.

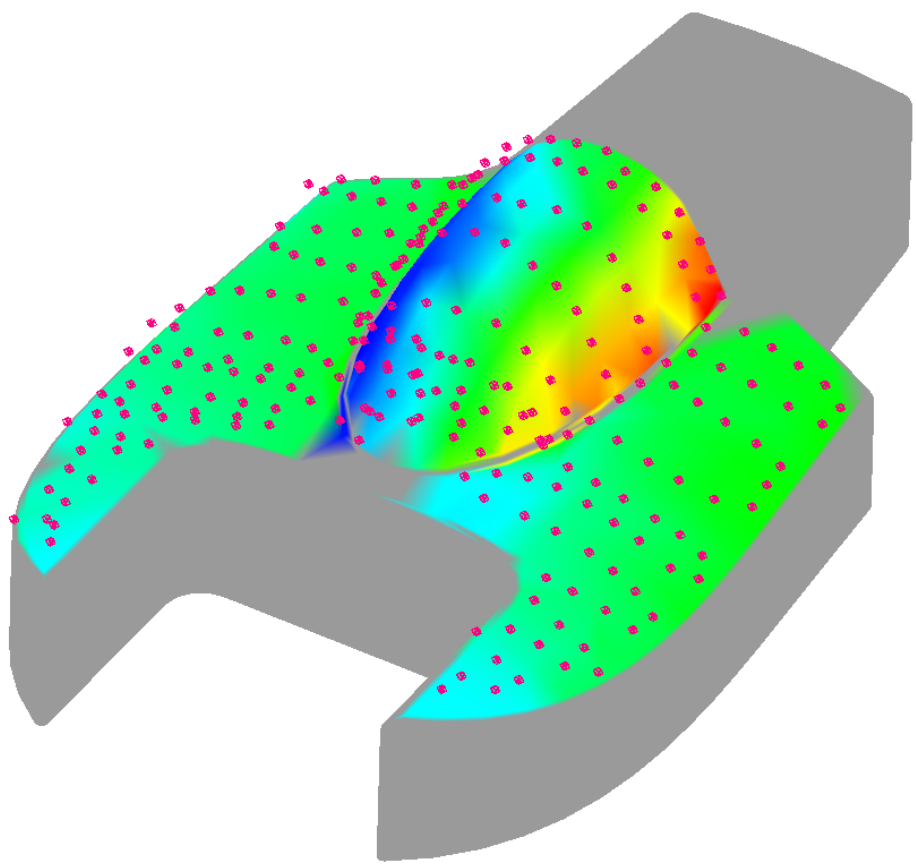


Figure 10-27. A colorized graphical mesh created using proximity of measured points to a nominal surface.

Perimeters

Properties

- **Collection.** Specifies collection name where object resides.
- **Name.** Object name.
- **Closed Perimeter.** When unchecked the perimeter is opened.
- **Notes.** Contains information entered by user or information regarding construction method, fit, etc.
- **Color.** Opens color dialog and allows the object color to be changed.
- **Apply.** Updates SA with any changes in the object properties.
- **Cancel.** Closes dialog without saving changes.

Construction

This command will create an open or closed perimeter object from selected points. Its purpose is to provided a boundary for scanning instruments. The below example shows perimeters to capture data from a targetless scanner.

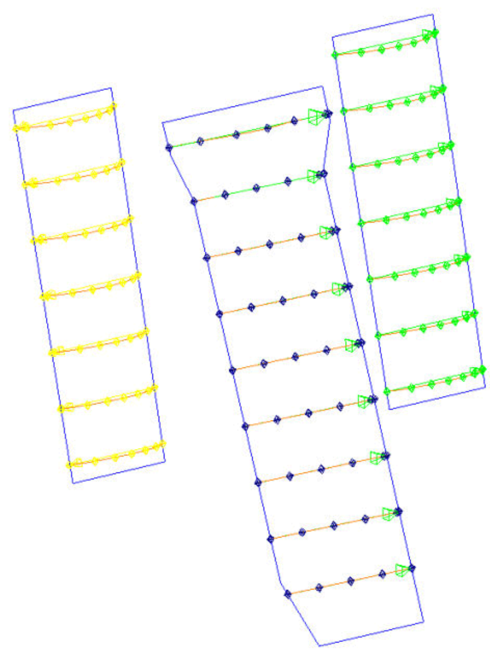


Figure 10-28. Perimeters defining the boundary of measurements from a total station.

Point Clouds

Point Clouds are unique because unlike Point Groups they do not give access to individual point coordinates. All cloud points are treated as a rigid body. Point Clouds are often used for large amounts of data, such as data measured with a laser scanner. Point clouds require less memory to store than a point group. By default the display options for a point cloud are automatically set to maintain an optimal graphical performance. As a Point Cloud is rotated or translated in the graphic view, the display resolution of the cloud will reduce automatically and will be restored once the movement is stopped.

From Existing Clouds

Creates a new point cloud based on selected point clouds.

From Existing Cloud Points

Creates a new point cloud based on the selection of individual cloud points.

Random Data

Used to generate random point cloud data.

From Existing Point Group

Converts a point group into a point cloud.

Auto Filter to Faces

Creates new point cloud based on proximity to surfaces.

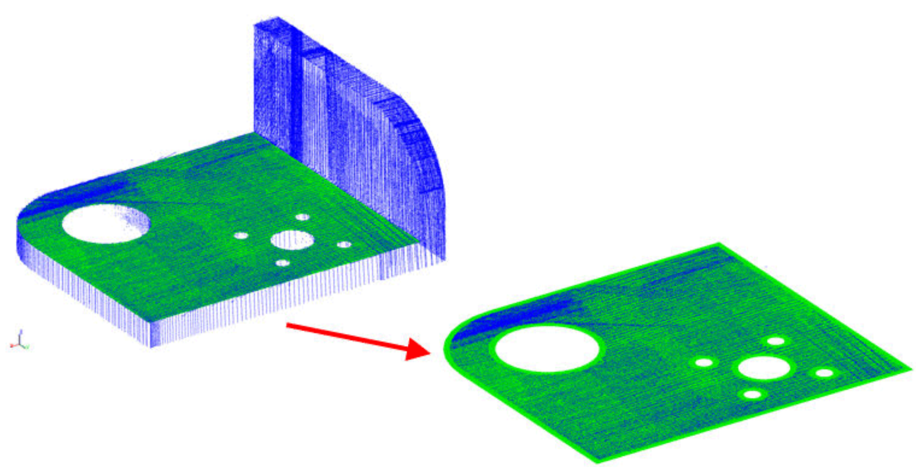


Figure 10-29. A cloud created based on proximity to a surface face.

Filter settings allow for offset boundaries to be created. Thinning options are also provided to reduce data density.

Instruments

CHAPTER 11

Instruments are the devices that gather accurate measurement data and provide a foundation for this data's position in a digital world. This chapter discusses how to work with instruments in general.

SA supports over 100 different measurement devices, from PCMM arms and laser trackers to scanners, laser radars, digital levels, and theodolites. An instrument in SA contains the basic properties of the instrument and defines its location in the workspace. Instruments appear in the treebar under the Instruments category, and each instrument is assigned a number (starting from zero), referred to as the *Instrument Index Number*. This number appears next to the name of the instrument in the treebar (Figure 11-1).

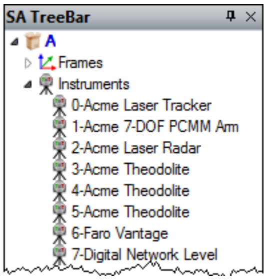


Figure 11-1. Instruments in the tree.

It is very important to keep in mind that *instruments are not objects*. In SA, objects are treated differently than instruments. Commands and functions that explicitly ask for objects will not accept an instrument. They are completely independent entities.

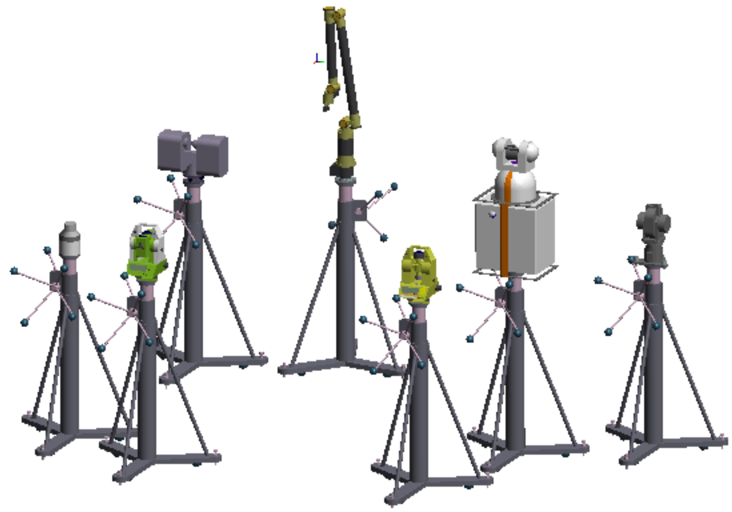


Figure 11-2. A number of instruments in the graphical view.

For spherical measurement devices (total stations, laser trackers, and the laser radar, for example), the position of the instrument defines the origin of the instrument's measurement volume. For a PCMM arm, the instrument position defines the base of the kinematic chain of linkages leading toward the probe.

When a point is measured by an instrument, the instrument reports back a point in its own internal coordinate system. This point is recorded in the measured target's observation details. For spherical measurement devices, the coordinate is represented in a spherical coordinate system. For a Cartesian measurement device (such as an arm), the coordinates are represented in XYZ coordinates.

Instrument Properties

To Access an Instrument's Properties:

- Right-click the instrument in the tree or graphical view and choose **Properties** from the context menu,
- Double-click the instrument in the tree, or
- Select **Instrument>Properties**, and double-click the instrument of interest.

The properties for an instrument (Figure 11-3) will vary depending on the class of instrument (laser tracker vs. arm vs. scanner, etc.).

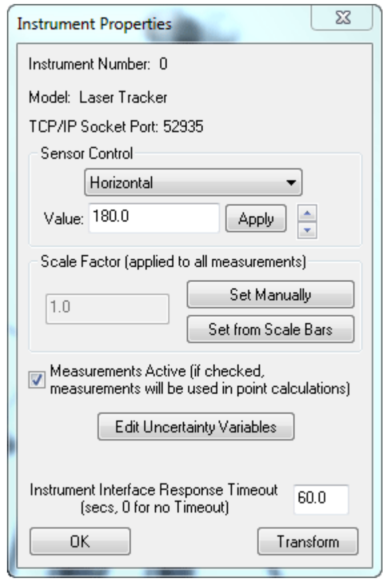


Figure 11-3. The Instrument Properties dialog.

- **Instrument Number.** The index number for the instrument (referred to as the *Instrument Index Number*).
- **Model.** The name assigned to the instrument. (By default, this will be the manufacturer's name and instrument type).
- **TCP/IP Socket Port.** The TCP/IP port that the instrument has been assigned. This is a random port number that is assigned by SA.

Sensor Control

The sensor control section allows you to adjust the graphic model of the instrument. For spherical devices, you can adjust the vertical, horizontal, and distance degrees of freedom. For an arm, you can adjust the rotation value for each link. Other instrument types (such as laser scanners) may not have any adjustment at all. After selecting the degree of freedom and associated value, click the Apply button to see the changes. Or, click the up/down arrows to increment the rotation value.

Scale Factor

Warning: Scaling an instrument's measurements requires caution. Once an instrument's measurements have been scaled, querying one measured point to another will not represent the true measured real-world distance between the two points.

It may be desirable to apply a scale to an instrument's measurements to account for thermal expansion and contraction of a measured part. For example, suppose you have a nominal model that has been designed for 68°F (20°C) conditions. If you are laying off nominal features from that model on a hot day--say 95°F (35°C)--then you will want to scale the instrument's measurements down from 95°F to 68°F so that the instrument and the nominal model are adjusted to the same temperature reference. One way to achieve this is to scale the instrument's measurements.

When an instrument scale factor is applied, all measurements from the instrument will be scaled appropriately. For example, an instrument observation that is normally at a radius of 100" from the instrument will be 101" from the instrument at a scale factor of 1.01. Instruments that are currently using a scale factor other than 1.0 will appear with a special yellow icon in the tree and will be listed as a scaled instrument, as depicted in Figure 11-4.

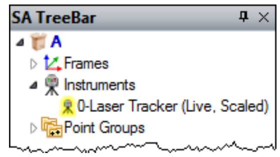


Figure 11-4. This laser tracker has a scale applied.

The scale for an instrument can be set in several different ways. It can be set manually (by directly entering a scale factor), set by measuring scale bars, or set in a command (such as **Instrument>Locate (Transform to Part)>Measure Nominal Points**), which allows you to vary the instrument's scale as part of the best-fit procedure.

To Set an Instrument's Scale Factor Manually:

1. In the ***Instrument Properties*** dialog, click the Set Manually button.

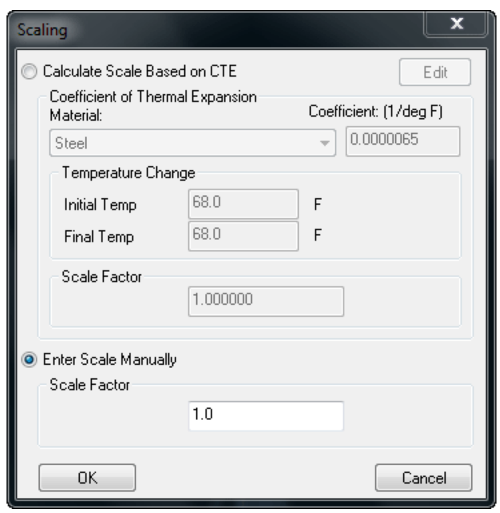


Figure 11-5. The scaling dialog.

2. In the ***Scaling*** dialog (Figure 11-5), select the *Enter Scale Manually* radio button and enter a scale factor, then click OK.

To Calculate a Scale Based on a Coefficient of Thermal Expansion:

Consider an example of measuring a steel part sitting in the sun, which has expanded in the 95°F midday heat. If we want to compare measurements against the nominal model for this part, which is at 68°F, then it is important to scale the instrument's measurements from 95°F (initial temperature) to 68°F (final temperature). This would be a situation where entering a scale for the instrument based on a CTE could be appropriate. The resulting scale factor will shrink the instrument's measurements so that the measurements can be compared to the 68°F CAD model, effectively removing the differences caused by thermal expansion.

1. In the ***Instrument Properties*** dialog, click the Set Manually button.
2. Select the *Calculate Scale Based on CTE* radio button.
3. In the dropdown, choose the desired material for the part that will be measured. If the material of interest is not available, click the Edit button, which allows you to add, delete, and edit materials and their associated coefficients of thermal expansion.
4. Enter an initial and final temperature. A higher final temperature will result in a scale increase, and a lower final temperature will result in a scale decrease.

To Calculate a Scale Based on Measured Scale Bars:

Scaling based on measured scale bars should only be performed when the material of the scale bar matches the material of the part to be measured, and when the nominal scale bar temperature matches the nominal (modeled) CAD temperature of the part. Consider an example of measuring a steel part sitting in the sun, which has expanded in the 95°F midday heat. If a steel scale bar with a nominal length of 36" at 68°F is then measured in the same conditions as the part, it will also have expanded, and the measured scale bar points will be more than 36" apart. Scaling based on this measured scale bar will scale the instrument's measurements down to match the nominal (68°F) length of 36". Subsequent measurements from the instrument will now be properly compensated to the nominal 68°F temperature.

1. In the ***Instrument Properties*** dialog, click the Set from Scale Bars button.
2. If the instrument has measured points that belong to a defined scale bar, the scale factor will be instantly calculated.
3. Review the results of the scale calculation, and click Yes to apply the scale factor to the instrument.

Note: If the scale bar points have not been measured by the instrument in question, then no scale will be applied.

- **Measurements Active.** Unchecking this box will deactivate all of this instrument's measurements from target calculations. This might be useful, for example, if you've measured the same point with several instruments, and have found an instrument to be out of calibration to the point where its measurements are corrupting the overall coordinates of the targets. Re-check this box to activate the measurements again. (Note that this will require you to recompute targets from shots using **Analysis>Re-Compute Targets From Shots**).

Uncertainty Variables

All instruments have error in their measurements, and as a result every measurement from an instrument has some uncertainty. Each class of instrument (laser tracker, PCMM arm, total station, etc.) has different set of uncertainty characteristics. For example, laser trackers have uncertainties for the horizontal and vertical encoders, as well as an uncertainty for the distance measurement. PCMM arms, on the other hand, have an angular uncertainty for each link of the arm. Additionally, each instrument within a specific instrument class has different values for the uncertainties. The uncertainty values for a newly calibrated laser tracker in pristine condition from a manufacturer will be different from those of an old, dirty, poorly calibrated tracker from the same manufacturer.

The uncertainty variables assigned to an instrument determine what the resulting uncertainties of the individual measurements from that instrument will be. Points measured from an instrument with small uncertainty values will have less uncertainty than those measured from an instrument with large uncertainty values.

The default uncertainty values that SA applies to a given instrument are identical for a given class of instrument. Due to variations in instrument calibration and performance, manufacturer, and measurement parameters, it is impossible to use a specific set of numbers for each instrument--the numbers can vary significantly. Instead, a set of conservative default values have been chosen (see Figure 11-6 on page 363). These values, while usable, will not be truly representative of the instrument's true uncertainty variables, so there is a procedure to determine an instrument's values. An instrument's true uncertainty variables are determined through a process using USMN. Once these values have been determined for a given instrument and measurement situation, they can be plugged into the instrument to get truly accurate uncertainties for a given instrument's measurements.

Note that all instrument uncertainty values are entered as 1 sigma values, regardless of the confidence interval set for the job in the User Options. Clicking the Edit Uncertainty Variables button will present you with a dialog in which you can enter specific values that apply to the selected instrument. The values in Figure 11-6 apply to spherical measurement devices such as laser trackers, total stations, and laser radars.

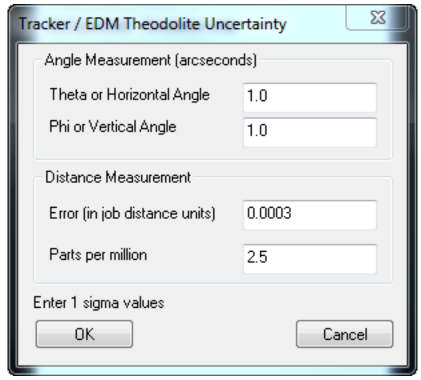


Figure 11-6. Generic uncertainty variables for a spherical measurement device (laser trackers shown here).

- **Theta or Horizontal Angle.** The angular uncertainty for a spherical measurement device's horizontal encoder, in arcseconds.
- **Phi or Vertical Angle.** The angular uncertainty for a spherical measurement device's vertical encoder, in arcseconds.
- **Error (in job distance units).** The fixed component of uncertainty for a spherical measurement device's distance measurement. The instrument above has been assigned a static uncertainty of 0.0003" in distance measurement.
- **Parts per million.** Additional uncertainty (on top of the fixed component described above) for a spherical measurement device's distance measurement that increases linearly as the measured distance increases. This value is entered in parts per million (ppm). The instrument above has been assigned a PPM uncertainty of 2.5 ppm. This implies that, at a measurement distance of 1000", the uncertainty contributed by the PPM component will be 0.0025".

The PPM and static uncertainties combine to determine the total distance uncertainty at a given point. At a measurement distance of 1000", the measurement has a total distance uncertainty of 0.0028" (0.0003" + 0.0025").

For portable CMM arms—as depicted in Figure 11-7—the uncertainty is entered at each link in the arm, in degrees.

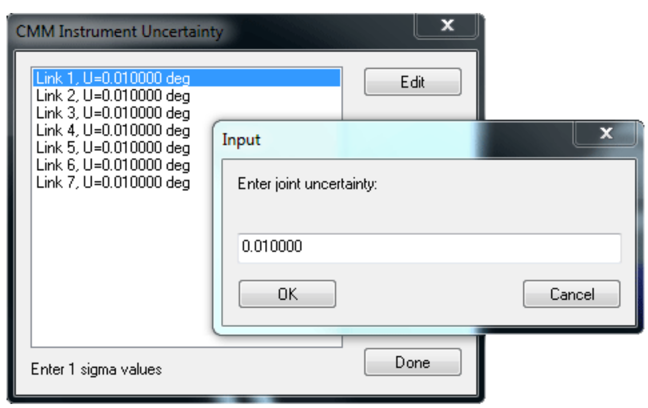


Figure 11-7. Uncertainty values associated with a portable CMM arm.

The theodolite uncertainty dialog is similar to that for other spherical measurement devices, however the distance uncertainty values do not apply (Figure 11-8).

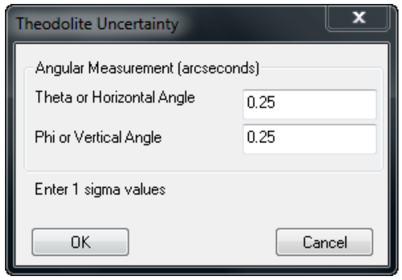


Figure 11-8. Theodolite uncertainty values (obviously) have no distance component.

Instrument Graphics (PCMM arms only)

PCMM arms have an Instrument Graphics button that appears directly under the Edit Uncertainty Variables button. This button calls up the ***Instrument Graphics*** dialog (Figure 11-9).

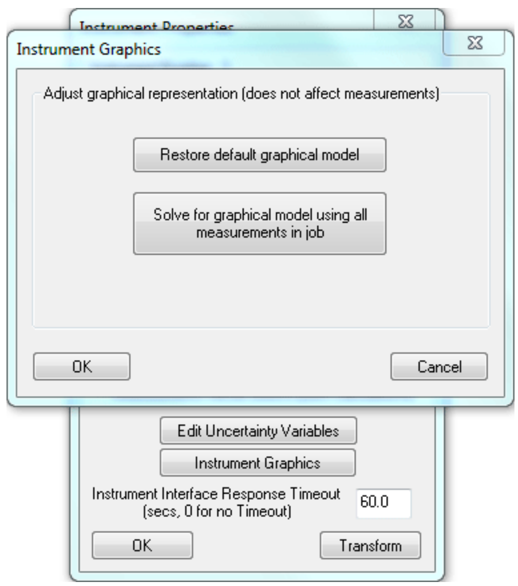


Figure 11-9. The Instrument Graphics dialog for portable arms.

Note: The Instrument Graphics options only affect how the PCMM arm is depicted in the graphical view. They have no effect on measured data.

- **Restore Default Graphical Model.** The kinematics for a PCMM arm are refined as points are measured. These kinematics only affect how the arm is depicted in the graphical view--not the accuracy of the measurement data itself. Clicking this button will restore the default graphical model that has been stored with the instrument definition.
- **Solve for Graphical Model Using All Measurements in Job.** This option will consider the measurements that the arm has observed, and recalculate the kinematics for the arm.
- **Instrument Interface Response Timeout.** This value determines (in seconds) how long an instrument interface will attempt to connect to the instrument until it times out with an error, stating that the instrument cannot be found. Enter zero

to continue checking indefinitely.

- **Transform.** The Transform button is used to transform (move and rotate) the instrument. More information about transformation can be found in the Move section.

IP-Based Instruments

IP Address Basics

An IP address is an identifier for a computer or device on a TCP/IP network. Networks using the TCP/IP protocol route messages based on the IP address of the destination. The format of an IP address (at least for the original, IPv4 version) is a 32-bit numeric address written as four numbers separated by periods. Each number can be zero to 255. For example, 1.160.10.240 could be an IP address. Within an isolated network, you can assign IP addresses at random as long as each one is unique.

The four sets of numbers can be thought of as a mailing address. State, City, Street and House number. Anyone can live in the same State and City and reside on the same street, however there can't be duplicate house numbers. So if an instrument has an IP address of 128.128.128.*100*, the computer must have a different "house number", such as 128.128.128.99.

Computer and Instrument Setup

Figure 11-10. A tracker connected directly to a computer via an ethernet cable.

- Host computer and instrument have unique IP addresses.
- It is typically easier to place the host computer on the same IP family as the instrument.
- As an example the instrument IP address is 192.168.0.168
- The host computer would need a unique IP address on the same IP family.
- Change the host computer to 192.168.0.100 for example.
- To test the connection use the Ping function in the SA network browser (Figure 11-11).

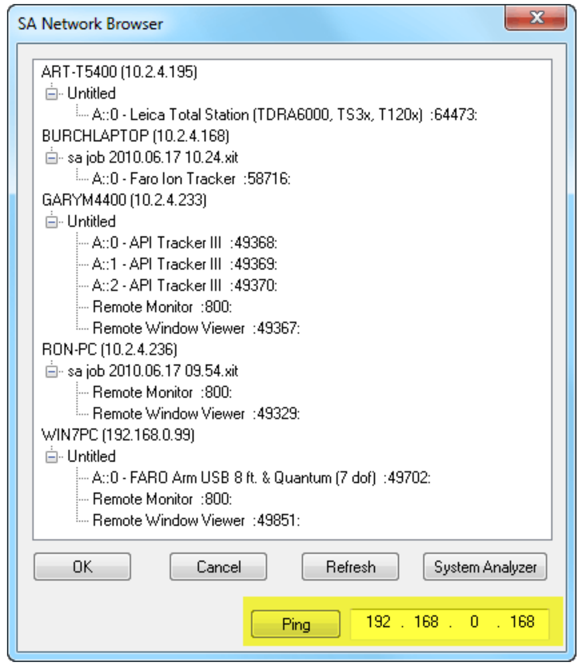


Figure 11-11. Pinging an instrument to check a network connection.

If multiple instruments exist on the network and they belong to the same family, no changes are needed. If they do not belong to the same family the host computer must be configured to have multiple IP addresses. See the section below regarding Multiple IP Setup.

Changing a Static IP Address

Windows XP

1. Navigate to the ***Control Panel*** via **Start>Control Panel.**
2. Select Network Connections to browse current connections.
3. Select the connection of interest and right-click to select properties (Figure 11-12).

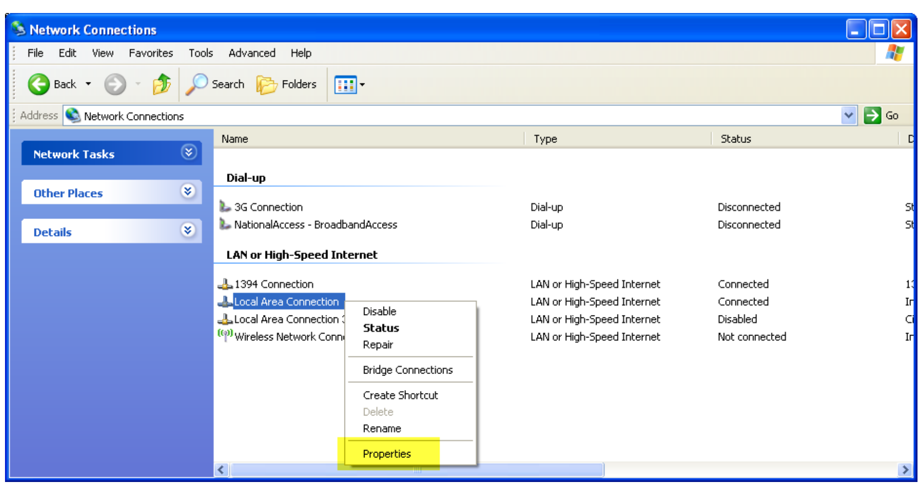


Figure 11-12. Selecting the connection properties.

4. Select **Properties**. Once inside of the ***Properties*** dialog choose Internet Protocol (TCP/IP) and select properties.

5. Enter in the desired static IP address and subnet mask.

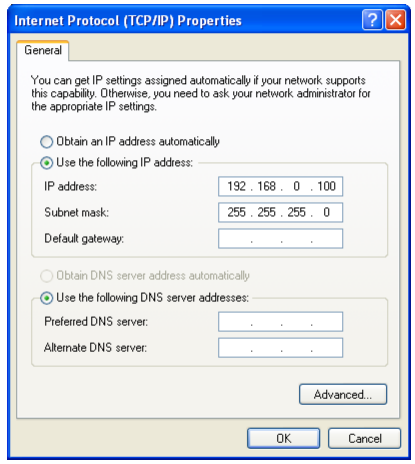
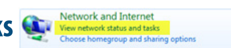


Figure 11-13. Specifying IP address settings.

Windows Vista/Windows 7

1. Navigate to the control panel via the Start Menu
2. Select **View network status and tasks**  to view the current connection.

3. Select the connection of interest.

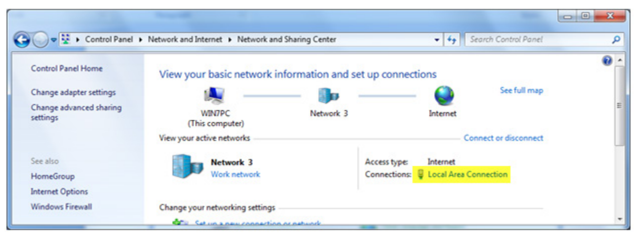


Figure 11-14. The Windows 7 Network Connections dialog.

4. Select **Properties**. In the ***Properties*** dialog, choose Internet Protocol Version 4 (TCP/IPv4) and click the Properties button.
5. Enter the desired static IP address and subnet mask.

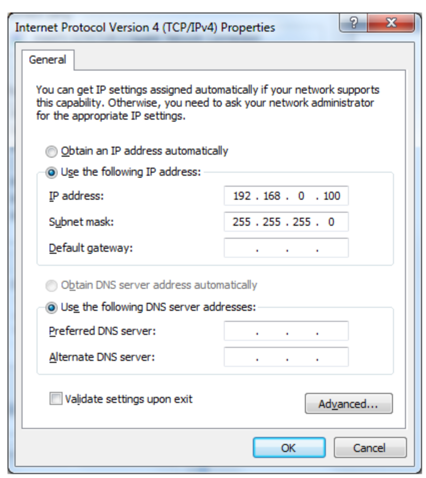


Figure 11-15. The Windows 7 IP settings dialog.

Multiple IP Setup (Dual IP)

Cases exist where the host computer and multiple instruments are unable to reside in the same IP family (subnet). In this case, the host computer can be configured to handle multiple IP addresses.

1. Follow the above directions to set a single static IP address.
2. Once the first static IP address is set, select the Advanced button at the bottom of the dialog.
3. Click the Add button and enter the additional IP addresses (Figure 11-16).

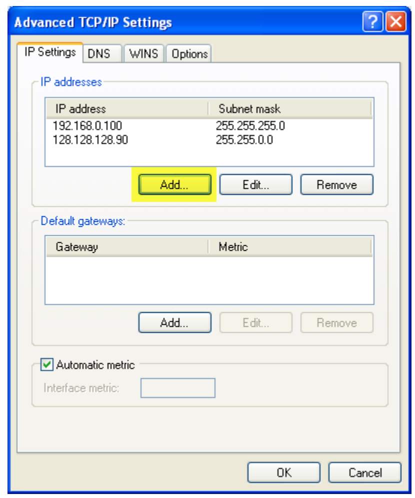


Figure 11-16. Assigning dual (or multiple) IP addresses to a single port.

Working With Surfaces

CHAPTER 12

Working with CAD surfaces is a discipline in itself. As more and more industries move toward computer model-based representations and away from paper drawings, the ability to work with surfaces grows every year.

Surface Grouping

Any set of surfaces may be combined into a single surface or split into separate surfaces. The process of combining surface faces is referred to as *merging surfaces*, and the process of separating surfaces is referred to as *dissecting surfaces*.

The advantage of have a merged surface is that it simplifies the tree. When surfaces are merged, they appear as a single surface in the tree, therefore there are less surface entities to wade through when moving around in the tree. This also can make selection easier, because selecting a single surface selects all of the faces that are part of that merged surface.

However, merged surfaces have some limitations. A merged surface cannot have its normals corrected—so, if you need to reverse some surface faces, the surface must be dissected first. Also, you cannot create queries, relationships or other entities involving specific subfaces of a merged surface. If you want a relationship, for instance, to only compare against a single face of a surface, that face must be extracted from the merged surface so that it appears as a separate entity in the tree.

One cannot definitively say whether a merged surface is better than a dissected surface, or vice versa. It really depends on the situation and what you'd like to do with the surfaces.

Merging Surfaces

The process of merging a surface involves selecting individual surfaces and combining them into a single surface in the tree that is treated as a single surface. When merging surfaces, you have the option of keeping the original surfaces in addition to the new merged surface, or deleting the original surfaces. In addition, you also have the option of hiding the original source surfaces as well.

To Merge Surfaces:

1. From the menu, select **Construct>Surfaces>From a Collection of Surfaces**.
2. Select all of the surfaces that you'd like to merge.
3. The ***Group Surfaces*** dialog appears (see Figure 12-1).
4. Select the desired options, then click OK. A new surface named **`Merged Surface xxx`** will be created.

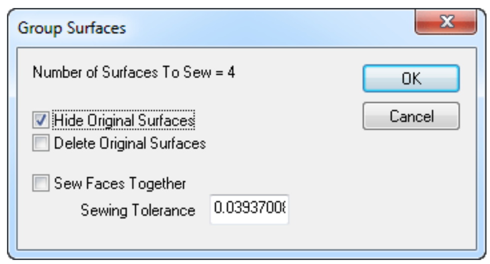


Figure 12-1. Merging a set of surfaces.

- **Hide Original Surfaces.** When selected, the original source surface faces will be hidden, but not deleted.
- **Delete Original Surfaces.** When selected, the original source surface faces will be deleted.
- **Sew Faces Together.** When selected, adjacent surface faces which lack tangency (they do not smoothly flow from one surface face to another) will be stitched together in accordance with the *sewing tolerance*.
- **Sewing Tolerance.** The maximum normal deviation from one surface face to the next in which those adjacent surface faces will be stitched together if *Sew Faces Together* is enabled.

Warning: Since sewing faces together can change the topology of a CAD model (usually undesirable in metrology), it is strongly recommended that you avoid this option unless you have a very valid reason for using it.

Dissecting Surfaces

Dissecting surfaces is the inverse operation of merging. Given a merged surface containing two or more surface faces, dissecting that surface will extract the selected faces and separate them as unique surfaces in the tree. Dissecting a surface leaves the original surface intact—it will be hidden for you in the tree.

To Dissect an Entire Surface:

1. From the menu, select **Construct>Surfaces>By Dissecting Surfaces>Entire Surface**.
2. Select the surface to dissect.

To Dissect Specific Surface Faces from a Surface:

1. From the menu, select **Construct>Surfaces>By Dissecting Surfaces>Select Faces**.
2. Select the individual faces to extract. The original surface will remain untouched, and the extracted faces will be created as separate surfaces.

Duplicate Surface Faces

Occasionally, you may encounter one or more surfaces that have overlapping surface faces. This may be a problem in the source model, a result of how the surface was imported, or simply a result of accidentally duplicating surface faces.

Duplicate surface faces can cause confusion in analysis, so it's best to clean these surfaces up before using them. Fortunately, SA has a command to detect and delete duplicate surface faces.

To Delete Duplicate Surface Faces:

1. From the menu, select **Edit>Remove Duplicate Surface Faces**.
2. Select the surfaces to examine.
3. Provide a coincidence tolerance. This indicates how close two overlapping surfaces must be to be considered duplicates.
4. If duplicate faces are found, confirm the deletion.

Surface Normal Conditioning

As explained in "SA Fundamentals", the normal direction of surfaces is critical to how offset compensation is applied to your measurements. An incorrectly-oriented surface normal can lead to dramatically incorrect analysis results. CAD surfaces should always have their normals facing away from the material of the part. That is, the side on which you approach the surface with a probe must be defined as the positive side of the surface.

The use of surface backside coloring (see "The Graphical View") enables you to determine if any surface normals are reversed. Whenever you import a CAD model, you should therefore check to ensure that surface normals are correct, and any incorrect normals should be fixed.

There are three methods for flipping surface normals: indicating point on positive side, reversing normals, and auto-reversing from viewpoint. All of these methods require a surface to be dissected into its individual surface faces before they are reversed.

Indicating Point on Positive Side

One method for correcting surface normals is to use a reference point as an indicator for whether that point should end up on the positive or negative side of a surface. This is similar to some metrology software that expects an additional measured point to indicate the direction of offsets for measured geometric surfaces.

To Condition Surface Normals by Indicating Point on Positive:

Note: This method currently compares the reference point to the middle of each surface face. Therefore, it is not 100% reliable.

1. Dissect the surface as described in "Dissecting Surfaces" on page 370.
2. From the menu, select **Edit>Surface Normal Conditioning>Indicate Point on Positive (or negative) side**.
3. Select the point to serve as the reference point.
4. Select the surfaces to consider.
5. At the ***Decision*** dialog, indicate whether the reference point should end up on the positive or negative side of the surface.

Reversing Surface Normals

The most straightforward method for reversing surface normals is to explicitly select the surfaces you'd like to reverse. The normal direction of each selected surface will be flipped.

To Reverse Surface Normals:

1. Dissect the surface as described in "Dissecting Surfaces" on page 370.
2. From the menu, select **Edit>Surface Normal Conditioning>Reverse Surface Normals**.
3. Explicitly select the surface faces you'd like to reverse.

Auto-Reversing Surface Normals

Note: You may need to perform this operation from several different viewpoints.

SA can use the current view orientation to automatically reverse a set of surface faces. This has the advantage of reversing a large number of surfaces at once, without requiring you to select the specific surfaces you'd like to fix.

With this command, you orient the view so that you can see the surfaces you'd like to correct. You then select these surfaces, and the key is that you can also select surfaces which are already facing the proper direction. SA will go through each selected surface, determine if it's visible from the current viewpoint, and then reverse it if its normal does not generally point toward the viewpoint.

To Auto-Reverse Surface Normals from Viewpoint:

Note: Occasionally, you may need to manually reverse a stray surface face.

Tip: If there is not a large number of surfaces, it's generally faster to just select all surfaces using rectangular selection or the F2 window versus trying to select only those surfaces that need to be fixed.

1. Dissect the surface as described in "Dissecting Surfaces" on page 370.
2. From the menu, select **Edit>Surface Normal Conditioning>Auto Reverse Surface Normals from Viewpoint**.
3. Orient the view so that you can see the surface faces you'd like to correct.
4. Select the surfaces you want the command to consider. (It's OK if these surfaces are already facing the correct direction).

This Page Intentionally Left Blank.

CHAPTER 13
Measurement

SpatialAnalyzer provides a large array of functions for performing measurement. The more familiar you are with the tools available at your disposal, the more efficient you will be when it comes to measurement.

Instrument Shots

It is often useful to draw instrument shots—that is, the line of sight from instruments to their corresponding measurements, as depicted in Figure 14-15. Instrument shots (also referred to as shot lines) help you visualize which instruments have measurements to which points, and also helps with visualizing the angles and distances to your shots, as well as the entire instrument network as a whole.

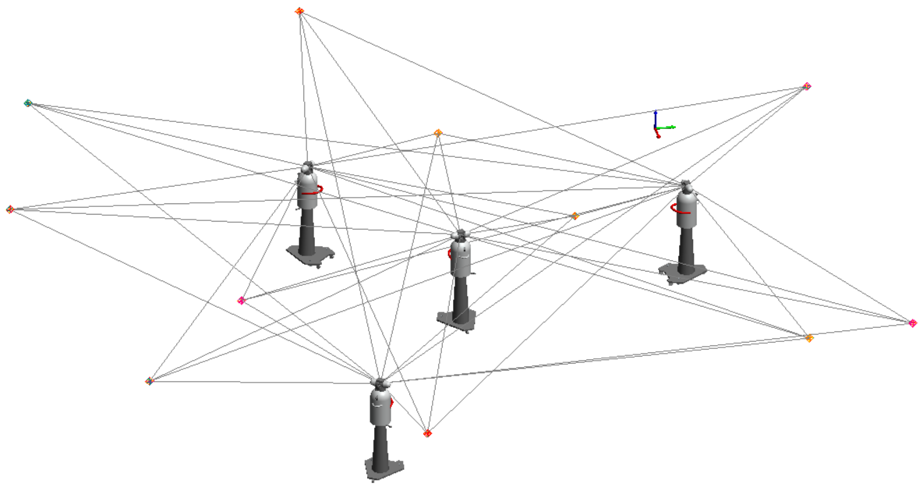


Figure 13-1. Shot lines for an instrument network.

To Control Whether Instrument Shots are Drawn in the Graphical View:

- Toggle the **View>Show Instrument Shots** setting.

Instrument Shot Modes

There are four modes for viewing instrument shots:

- **Draw only last shot of active instruments.** Draws only the most recent shot from any active instruments.
- **Draw all shots.** Draws every shot line from every instrument to every point (used in Figure 14-15).
- **Draw only active (enabled) shots.** Any shots which have been disabled will not have shot lines drawn—but all others will.
- **Draw only inactive (disabled) shots.** Any shots which are enabled will not have shot lines drawn—but all others will.

To Change How Instrument Shots Are Drawn:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, click the Shot Graphics Options button.
3. Select the desired mode from the *Mode* box.

Shot Colors

In addition to changing the instrument shot mode, you can also set the color used for active shots, inactive shots, or the latest (Current) update.

To Change Instrument Shot Colors:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, click the Shot Graphics Options button.
3. Click the Active Shots, Inactive Shots, or Current Update button

to indicate which color you'd like to modify.

4. Select a color from the dialog, then click OK.

Angle Only Shots

Instruments that do not measure distance (such as theodolites) cannot measure a point in 3D space, therefore the shot line length cannot be determined. A default length of 200 units is used for drawing these shots, although you can lengthen or shorten this value as appropriate to the scale of the measurement job.

To Change the Length of Angle Only Shots:

1. Click the User Options icon in the toolbar.
2. In the *Display* tab, click the **Shot Graphics Options** button.
3. Enter a length into the *Default Length* field.

Big Data Window

The Big Data Window (Figure 13-2) allows you to view various measurement data as it arrives. This includes:

- Target Name
- Target Coordinate
- Component Measurement Uncertainty
- Pointing Error
- Apex Angle
- Group Comparison
- Theodolite Data Observer information
- Error Sensitivity
- Distance Between Last Two Points

The window is customizable: you can turn different data elements on and off, resize and relocate their position in the window, change fonts, and control colors, borders, and other visual elements.

The window also allows you to step forward and backward through recent measurements to view the associated data, copy the data to the clipboard, or take a graphical snapshot of the window.

To View the Big Data Window:

- From the menu, select **View>Big Data Window**.

To Enable/Disable Elements of the Big Data Window:

1. In the Big Data Window's menu, click **Customize**.
2. The ***Customize*** dialog appears. Select different elements from

the dropdown, and check or uncheck their *Visible* option, then click Apply.

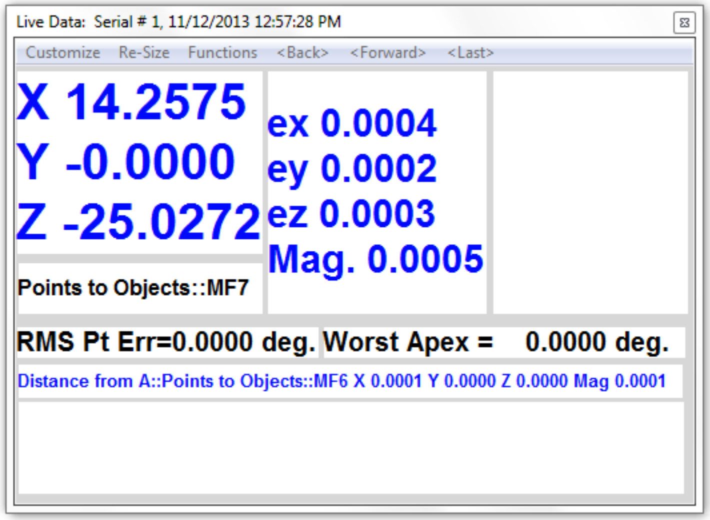


Figure 13-2. The Big Data window.

To Customize an Element of the Big Data Window:

1. From the big data window's menu, click **Customize**.
2. In the ***Customize*** dialog, select the element from the dropdown that you'd like to modify.
3. Choose among the below options to modify the element:

- **Text Font.** The font used to display the textual data.
- **Text Color.** The color of the font used to display textual data.
- **Draw.** When enabled, draws a border around the element.
- **Border Color.** Determines the color of the element's border.
- **Fill.** When enabled, fills the element's background with the selected color.
- **Fill Color.** The color to use for the element's background.
- **Highlight Color.** Sets the highlight color, if used by the element.

To Resize and Reposition Elements in the Window:

1. In the big data window's menu, select **Re-Size>Allow Sizing**. This puts the window into a resizing mode.
2. Click and drag an element to move it. Right-click and drag an element to resize it horizontally and vertically.
3. When finished, again select **Re-Size>Allow Sizing** to disable resizing mode.

To Copy the Big Data Window Text to the Clipboard:

- From the big data window's menu, select **Functions>Copy text to**

clipboard.

To Copy the Big Data Window to the Clipboard as an Image:

- From the big data window's menu, select **Functions>Copy graphics to clipboard**.

To Move Through Your Measurement History:

- Use the **<Back>**, **<Forward>**, and **<Last>** menu items in the big data window to navigate through historical measurement data.

Instruments

Before you can begin measuring with an instrument, you need to add one to your current job.

To Add an Instrument:

Do one of the following:

- From the menus, choose **Instrument>Add**, or
- Press **`Alt+I`**, the default keyboard shortcut, or
- In the Main Toolbar, click the Add Instrument button.

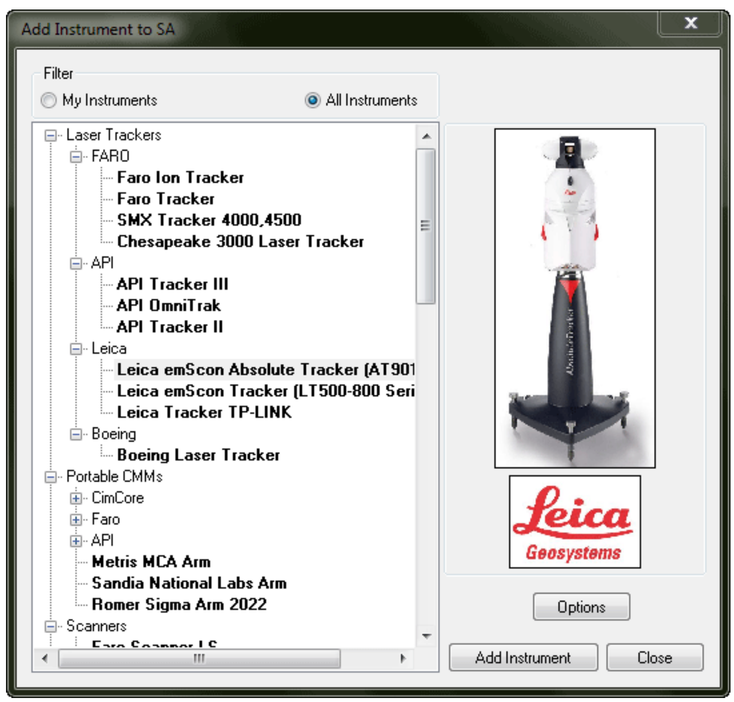


Figure 13-3. The Add Instrument to SA dialog.

The ***Add Instrument to SA*** dialog (Figure 13-3) will list all available instruments that can be used in SA. The instrument list can be filtered to only show the instruments that you have added in the past; each time an instrument is added to a job, it's added to this filtered list.

To Switch Between the Filtered Instrument List and the Full List:

- Select the **My Instruments** or **All Instruments** radio button.

To Reset Which Instruments Appear in the Filtered List:

- Navigate to the `Analyzer Data\Persistence` folder and delete the `My Instruments.bin` file.

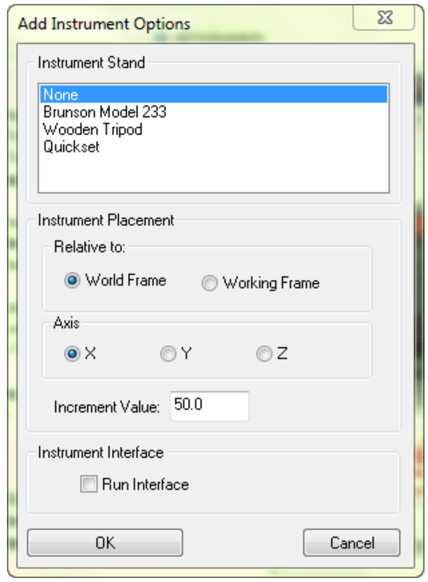


Figure 13-4. The options available for adding an instrument.

Clicking the Options button opens the ***Add Instrument Options*** dialog (Figure 13-4). In this dialog, you can specify an instrument stand (for visual representation purposes only), specify where to locate the newly added instrument, and specify whether to automatically run the instrument's interface when the instrument is added. These settings are optional conveniences and are not critical to the procedure of adding instruments. The settings persist and will apply to the addition of future instruments until the settings are changed.

To Specify an Instrument Stand:

- Choose **None**, **Brunson Model 233**, **Wooden Tripod**, or **Quickset** from the Instrument Stand list.

To Specify Where a New Instrument Should Be Located:

4. Choose **World Frame** (to add the instrument relative to the World frame) or **Working Frame** (to add it relative to the current working frame).
5. Specify whether to add the instrument along the chosen frame's X, Y, or Z axis.
6. Enter an Increment Value. This value specifies (in job units) how much further down the selected frame's axis each subsequent instrument should be added.

To Specify Whether a Newly Added Instrument's Interface Should Automatically Be Started:

- Check or uncheck the **Run Interface** option.

Instrument Options and Properties

When an instrument is added to SA, it is added to the graphical view as well as the tree. Each instrument is identified by an Instrument Index Number and the name of the instrument. In a given collection, the Instrument Index Number begins counting from zero.

Right-clicking an instrument in the tree will expose a number of commands that apply to instruments.

Instrument Properties

The instrument properties give access to several important properties. Here the user can set the scale of the instrument either using Scale Bars, Material CTE or a Manual entry. Instrument Properties are covered in SA Fundamentals | Entities | Instruments.

Instrument History

The instrument history (Figure 13-5) is a log of events that have occurred to that particular instrument. Events such as when an instrument was connected, instrument specific information, how the instrument was transformed, etc.

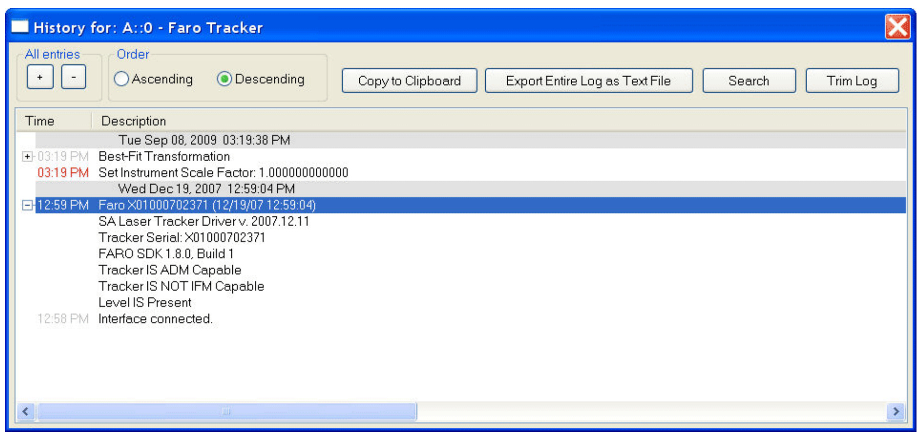


Figure 13-5. A tracker's history.

Instruments and Measurements

A unique characteristic to SpatialAnalyzer is the association of points to their respective instrument. SA treats an instrument and its points as a rigid body. If an instrument is transformed (moved), the instrument and its points will move. This is a key concept that is used extensively throughout SA. Objects can also be associated to an instrument, so when an instrument moves the associate points and objects will move.

When a point is measured it is treated as a measured point. Measured points contain all of the instrument specific measurement details. Please see the Point Properties section for more on measurement details.

The Instrument Interface

One of the most powerful aspects of SpatialAnalyzer is the modularity of the instrument interfaces and how they interact with the core analysis platform. These interfaces are able to work independently or together, on a single computer or across networked computers.

Instrument interfaces communicate with SpatialAnalyzer using the robust TCP/IP network protocol. This affords several distinct advantages. First, these connections are extremely fast and reliable. Second, since TCP/IP is a "network" protocol, the instrument interfaces may be run on either the same computer as the instance of SA, or on another computer accessible on the same network. Third, and perhaps most impressive: since it is the "Internet Protocol" (IP), the computers running the instrument interfaces could be in the same room, across the country, or around the world. Likewise, if activated, it is possible to remotely monitor a measurement job in progress using this same TCP/IP connection over the Internet or on a local network.

Modular instrument interfaces are the basis of the data acquisition aspect of SpatialAnalyzer. These provide an easy-to-use and friendly interface for the control of instruments during a measurement job.

Establishing the instrument interfaces as separate entities in the architecture affords several key advantages. First and foremost, this provides flexibility to adapt to tomorrow's instrumentation. The number of instrument interfaces available for SpatialAnalyzer is ever-expanding, much like printer drivers and other peripheral devices for the Windows operating system. Second, this approach enables SA to provide a common interface for each type or class of instrument. A portable CMM manufactured by Company A would has the same top-level interface as a portable CMM manufactured by Company B. This allows users to easily select the instrumentation best suited for a particular task. It also allows users to easily transfer the expertise they have gained with one instrument to another. A direct benefit of this capability is to improve a user's flexibility when moving from task-to-task and it reduces the training time for a user to become productive on a different instrument. Third, this allows for a more objective view when it comes time to replace or purchase new hardware, since retraining is minimized by the common instrument interface in SpatialAnalyzer.

In all cases, the instrument interfaces perform four key functions:

- They provide a common and simple user interface that can be used under SpatialAnalyzer control to guide a user through a particular measurement task.
- They communicate over a TCP/IP network to convey essential

measurement data to SpatialAnalyzer.

- They communicate via a native format to the instrument. For example: serial RS-232 connections for many theodolites, Network protocols for many laser trackers.
- They provide an efficient means to collect the raw measurements, the conditions and techniques under which they were collected, and to perform operational checks on the instrument to establish that it is meeting its performance specifications. It also supports simulating measurements and measurement tasks before the job is performed. By simulating a measurement process before the job is performed, accuracy and time estimates can be made to help you make the best choices regarding optimizing the placement of the instrument(s) relative to the subject of the survey.

Starting an Instrument Interface

To start an instrument interface, choose **Instrument>Run Interface Module** (or click the dropdown arrow next to the Run Instrument Interface icon). Then select the instrument interface of choice. Once selected, the ***Connect to SpatialAnalyzer*** window will display (Figure 13-6). This window displays all network-accessible and running instances of SA containing instruments. The network location (computer name and IP address) appears in the list, in addition to the open job filename. Any applicable instruments not already connected to an interface will appear in boldface.

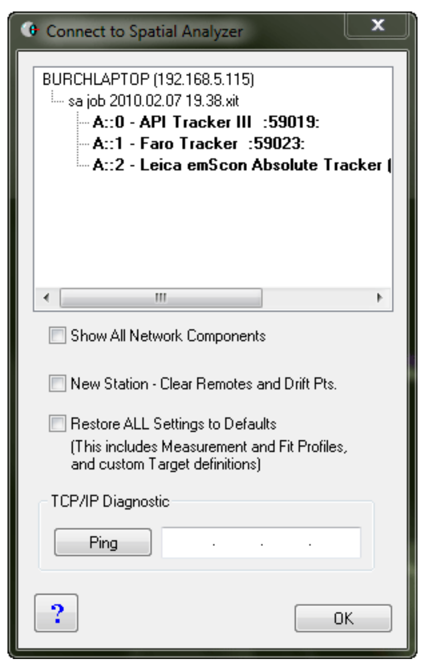


Figure 13-6. The list of available instruments to connect to.

In this window, you're choosing the instrument in an SA job file to which you'd like to send the measured data. While it is usually the file open on the same computer, this is not required. Choose the instrument to which you wish to connect and click OK.

The next dialog will display connection options that are specific to the instrument being added. In Figure 13-7, a Faro tracker is being added.

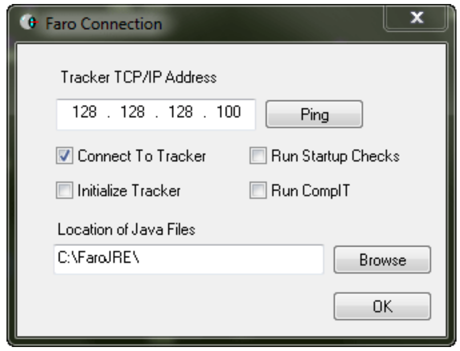


Figure 13-7. Connection options for this instrument (a Faro tracker).

Once accepted, the instrument interface will appear (Figure 13-8).

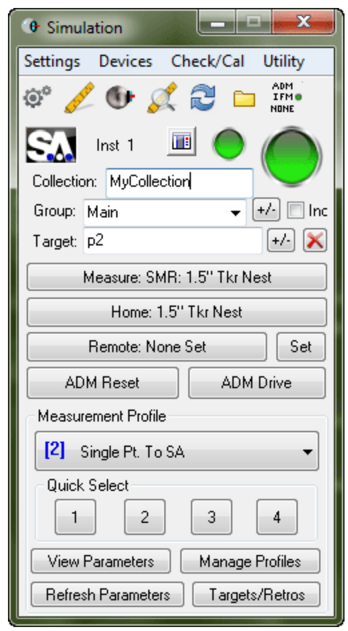


Figure 13-8. A simulated tracker interface.

See the Interfaces section for information on instrument-specific interfaces.

If a single instrument is in the job file, you can click directly on the Run Instrument Interface icon, and the instrument's interface will be connected and started up. This is equivalent to the **Instrument>Run Interface Module and Connect** command.

Running Multiple Instrument Interfaces

One of SpatialAnalyzer's strengths is its ability to gather data simultaneously from multiple instruments in the same job file. This can be done with any combination of instruments, and users may connect and disconnect instrumentation at will. The process for adding additional instruments is exactly the same as adding the first. Just follow the procedure outlined in earlier sections.

Measurement Simulation

SpatialAnalyzer allows for instruments and measurements to be simulated (Figure 13-9). This can aid in planning a job, analyzing instrument setup uncertainty, training, etc.

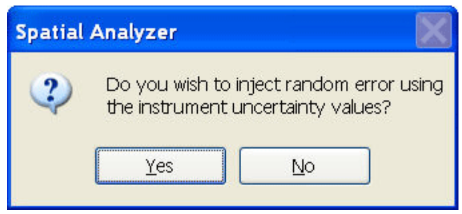


Figure 13-9. Measurement simulation.

The “Fabricate Measurement” command will create simulated measurements from the selected instruments and points.

Measurement error can be injected to simulate real world conditions. The error is based on the uncertainty model of the selected instrument.

Locating Instruments

Instrument Locate (Transform to Part)

An instrument can be located several ways. The basic instrument location methods can be found under **Instrument>Locate** or right-click an instrument and select **Locate**. Advanced methods are discussed in later sections and links are provided below.

Measure Nominal Points

Measure Nominal Points (Figure 13-10) is a great way to locate an instrument quickly. Simply select the nominal points and the instrument to locate.

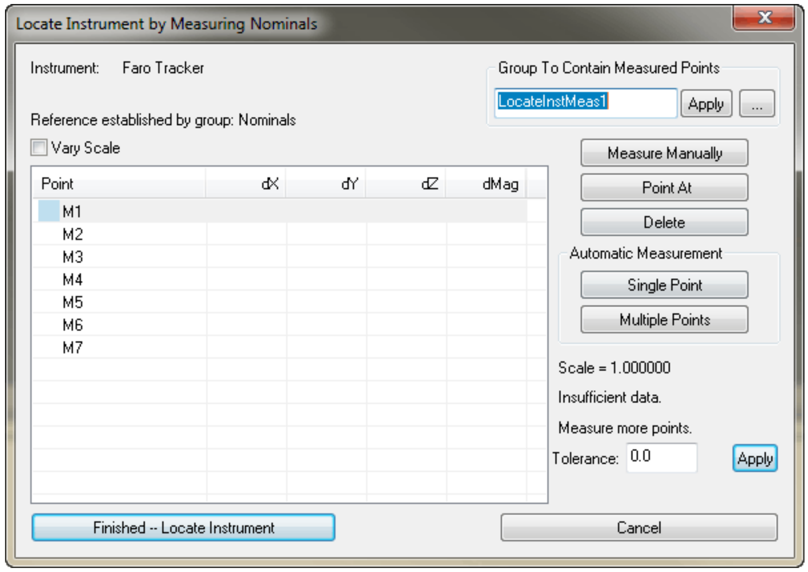


Figure 13-10. The Locate by Measuring Nominals window.

The ***Locate*** dialog will appear listing the points to measure. The user has the option for manual or automatic measurement. Once three points are measured the instrument is transformed via a Best-Fit. With the instrument roughly located the user can easily drive or point to each target. As measurements are recorded the Best-Fit is updated. A fit tolerance can be applied to notify user of an out of tolerance condition.

Quick Align to CAD

A quick and easy method to align to a CAD model. Simply click the model where the points will roughly be measured. A minimum of six points are needed to align to a CAD model. Red dots are used to guide the user where to measure. During the selection of the surface points the current view is saved. As measurements progress the views update.

Best-Fit

Locating an instrument via a Best-Fit is one of the most common location methods. It is used heavily during instrument location to nominal control points, instrument relocation and relocation due to drift. The user specifies the instrument to locate, the nominal point group and measured point group. The below fit dialog will appear allowing a view of the best fit. This Best-Fit Transformation dialog is covered in the Best Fit Points to Points section.

Plane, Line, Point (3-2-1)

A 3-2-1 fit consist of a locator point for the origin, line/vector for one axis and plane for the second axis. The Plane, Line, Point Transformation dialog allows control over the axis for the plane and line/vector. The user may also specify an alternate coordinate for the locator point. Once the transformation is accepted the instrument will be transformed with respect to the active frame.

Frame to Frame Transformation

Transform an instrument from a one frame to another. User will be prompted to select a source frame, a destination frame and an instrument to move.

3 Planes

The 3 Plane instrument locate uses three planes to define a coordinate system that is then transformed to the active frame. The user has control over the axis fit priority and position.

Analysis Best Fit Points to Points

Analysis>Best Fit Transformation>Points to Points is very similar to **Instrument>Locate>Best-Fit**. The main difference is that the user has the ability to move multiple instruments and additional objects. See Best Fit Points to Points for more information.

N-Point Full Fit

Finds the optimal transformation that minimizes the errors between the measured points and the CAD surface. This method contains many options for tolerance fits and other advanced fitting methods. See Best Fit Points to Surfaces for more information.

Locating By Minimizing Relationships

Instruments can be located by Minimizing Relationships. This is available in SA Ultimate and can found in the Minimizing Relationships section.

Unified Spatial Metrology Network (USMN)

USMN is a strong method for locating multiple instruments. It is usually used in a case where three or more instruments are present. This is available in SA Ultimate and can found in the USMN section.

Watch Windows

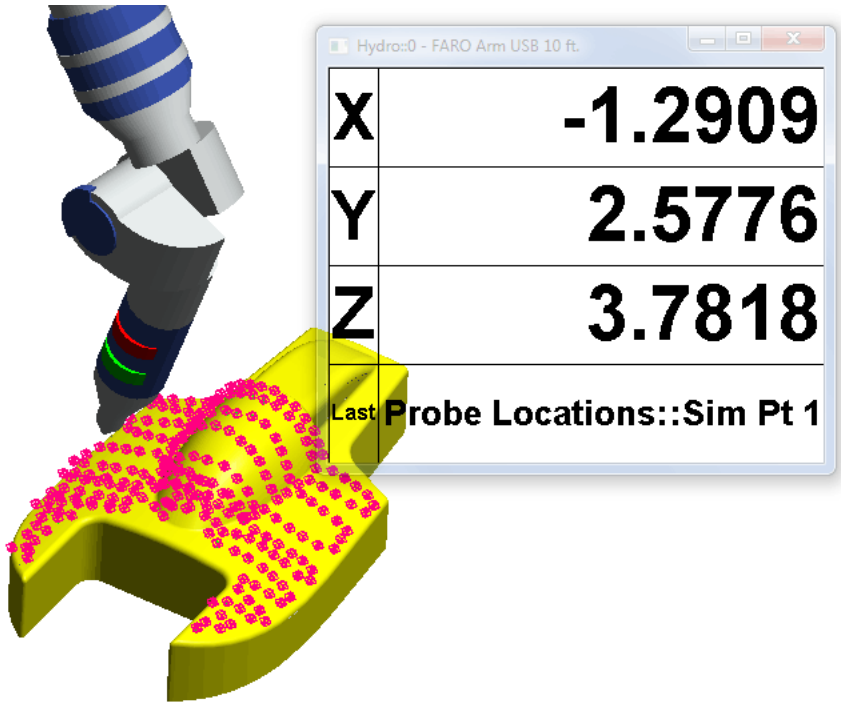


Figure 13-11. A watch window on an instrument.

SpatialAnalyzer uses watch windows (Figure 13-11) to display live data taken from measurement devices and relationships. These watch windows may be used to compare the current measurement from an instrument to a surface, another point, the closest point, or other entities.

These windows are re-sizeable to suit the user's preferences. In addition, when a tolerance is set, these windows will indicate out of tolerance conditions by changing the background color of the window.

Watch windows will update when the instrument performs a Query. This is different than when the instrument takes a measurement. Queries are sent to SpatialAnalyzer to update the watch windows and the graphics but are not saved as measurements. You can query an instrument from its instrument interface. In addition, where applicable, you can set the instrument to query at a given frequency. This is the mode you will usually place the instrument in when using watch windows.

How to Open a Watch Window

A watch window can be opened one of two ways:

- Navigate to the menu option **View>Watch Window** and select a Watch Window style, or
- Right-click the point, object, relationship or instrument of interest and select **Add Watch Window**.

Multiple watch windows can be opened and updated. This can be useful when monitoring several objects.

Watch Window Types

Point to Point

Add Point to>Point or right-click a point to add a watch window.

Point to Object

Add Point to>Object or right-click an object to add a watch window.

Add Instrument's Point

Add Instrument's Point or Right-Click instrument to add a watch window. Displays instrument's current target location in active frame.

Add Closest Point

Add closest Point or Right-Click a point group to add a watch window Will compare the instrument's target to the closest point.

Add Three Point Frame

Watch Two Point Axis Alignment

Watch Window Options

A watch window can be configured for size, precision, tolerance and more. Simply right-click the watch window to access watch window specific options.

Precision, Font, Text Color, Background Color and Highlight Color

Use these options to change the look and feel of the watch window. These are specific to the current watch window being changed.

Arrow Settings

Control guidance arrow components and length.

Hide/Show Row Items

Hide or show selected component.

Coordinate System Type

Display values in Cartesian, Cylindric or Polar.

Set Frame of Reference (FoR)

By default a watch window is expressed in the current active frame. However, it can be advantageous to display the coordinate values in an alternate frame. Use this option to select an alternate frame.

Default Tolerance

Use this option to apply a tolerance to the watch window.

Projection Options

Used for watch windows using an object. See the Vector Group section for further information on projection options.

Bundle Adjustment

A bundle adjustment may be used to refine measurements acquired using a spherical measurement system. These include theodolites, laser trackers, laser scanners, etc. Basically, a bundle adjustment will attempt to find the instrument transformations (positions and orientations) that yield the minimum combined pointing error.

Initial Guess

Since the bundle adjustment algorithms use the current locations of the instruments as a starting guess, it is best to "rough" position the instruments. To do this, either enter the instrument transformations manually, or use the **Drag Instruments** function from the Instruments menu. It is best to place the instruments near their final location, as this will speed the bundle solution. If using theodolites with level compensators, you will use the option to force instrument vertical during the bundle adjustment. If this is the case, the virtual instrument models should be perfectly parallel to each other before initiating the bundle adjustment.

Scale-Bar Database

For instruments that measure angles and distances, it is not necessary to use scale-bars in the bundle process. They are required for theodolite networks, however, because without them, the instruments will scale freely.

SpatialAnalyzer contains a scale-bar database. This is where you can enter the various scale-bars you have placed into the measurement area. Essentially what this database does is it associates target names with a scale-bar distance and uncertainty.

To modify the database, select the **Scale-Bars** tab from the User Options page. The next window will list all of the current scale-bars and their information.

To add a scale-bar, press the Add button and a new record will be generated. Complete the information required and press the Apply button. You should see the information in the list window update.

Since an accurate bundle solution for theodolites requires at least one scale-bar, make sure this database is populated before attempting a bundle. In addition, make sure the targets for the end of the scale-bar have been shot and included in the list of targets to be included in the bundle solution.

Note, however, that you may populate the scale-bar database at anytime before attempting to bundle adjust. It is not necessary to have this database in place before taking measurements.

Select **Bundle Adjust** from the Instruments menu. Next, select the instruments you wish to bundle using the dialog. The instruments that you select are the instruments that will be moved in order to refine the network solution. If you have 4 instruments, you will only have to bundle 3 because the first instrument may remain fixed while the others are adjusted relative to it. You may, however, select all of the instruments. This will make the solution take longer, however, because the entire network will be allowed to float in space. Next, the ***Bundle Adjustment*** dialog will appear.This dialog will allow you to configure the settings for the bundle, run the optimization, and view the results.

Selecting Targets to Bundle

Once the Bundle Adjustment window appears, you may select the targets to be included in the bundle. SpatialAnalyzer will default to including ALL of the targets. You may reduce the number of targets for computational efficiency or target accuracy reasons.

Scale-Bar Weight Factors

In the Bundle Adjustment window, you will notice a section for Scale Bar Weight factors. This section allows you to control the manner in which scale-bars are incorporated into the final solution.

You may change the state of the check-box to determine whether scale-bars are weighted based on their uncertainty. If this box is checked, the relative uncertainties in the Scale-Bar database will be used to weight those scale-bars with a higher accuracy more than those scale-bars with a lower accuracy (or a higher uncertainty). Basically, this check-box controls how multiplier scale-bars are compared to each other during the bundle solution.

In addition, you may enter a numerical weight that is applied to ALL scale-bars during the solution. This value defaults to 1.0, but may be increased or decreased at the user's discretion. Note, however, that this value is independent of the state of the check-box. You may weight scale-bars based on uncertainty and/or apply an overall weight. This is because the overall weight is applied after the uncertainty weight is applied.

Viewing System Errors

The Bundle Adjustment window displays both the Original shot errors and the adjusted shot errors. This gives you a measure of how much the bundle adjustment has improved the overall system error.

A maximum, average, and RMS error is displayed. These values characterize the combined pointing errors for each target that is included in the Bundle Adjustment. Each target has a pointing error, given in degrees. This is the amount that the measurements miss the best-fit computed target. The bundle adjustment attempts to minimize the combined pointing error by simultaneously adjusting both the position and orientation of the instruments.

Therefore, if there are 30 points in the bundle, the maximum error is the largest pointing error for all of the points, the average is the average of that set, and the RMS is the root-mean-square of the set.

If you wish to see the errors itemized by target, click the Show Details button, and you will see a list of the targets and their errors. You may wish to select this option before beginning the bundle to inspect the list.

Beginning Computation

Click the Begin Computation button, and the Bundle Adjustment will begin. As progress is made, the graphical display will be updated, and all of the numerical values in the Bundle window will reflect the system improvement.

When the algorithm has found a minimum, it will stop and issue a message to the user. Look at the details of the bundle to inspect the individual target errors as well as the scale-bar errors. If these are suitable, click the Accept Results button in the Bundle window, and the changes will be made active. The system is now bundled!

If, however, the results are not satisfactory, and you wish to start over where you began initially, select Cancel from the Bundle window. This will reset the instruments to their initial locations and re-compute the target locations.

Local Minima

Whenever you attempt to minimize a set of complex equations, the issue of local minima arises. Because you start from an initial guess and seek to minimize the error until you can minimize no more, you do not know that you are at the lowest possible minima for the entire solution space.

For this reason, it is possible for the bundle adjustment algorithms to find a solution that, while it is a minimum, is not the minimum you were searching for. SpatialAnalyzer contains a full 3-D graphical model of the measurement environment to help decipher some of the local minima. It is important, however, to start the instruments near to their optimal location (i.e. use a good guess) to minimize the chance of a local minima.

Weighting

There are several weighting factors described above. These control the importance of different parameters on the overall optimization process. There are several other weighting factors outside this dialog that control the process as well.

In the User Options dialog, under the Units tab, there is a button for Angle-Distance Weights. This button will allow you to control the relative weight of angular measurements compared to distance measurements. This will only affect the bundle if you are using instruments that can measure distance (laser trackers for example).

UDP Broadcast

SA's instrument interfaces have the ability to send a stream of UDP packets when their measurements are sent to SpatialAnalyzer. To enable this for trackers, for example, go to the **Utilities** menu and select **UDP Data Stream** (Figure 13-12). Then configure the settings:

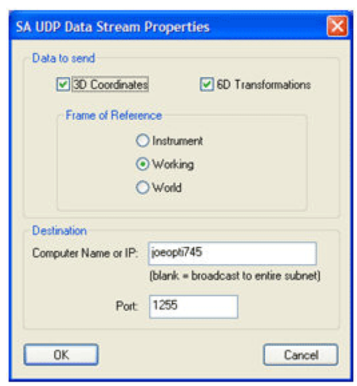


Figure 13-12. The UDP data stream properties.

Now, subsequent 3D and/or 6D measurements will also be sent as UDP packets to either the entire subnet or a particular IP address.

SA also contains a diagnostic utility so you can view this data stream (Figure 13-13). Navigate file explorer to the SA install directory and run `SpatialAnalyzer UDPMonitor.exe`. Set it to the same port and hit Start. When measurements or updates happen, the data will flow into the monitor, as pictured at right.

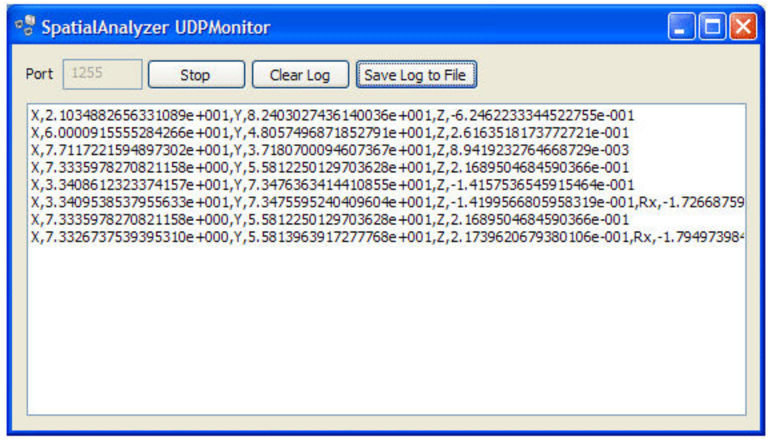


Figure 13-13. The UDP Monitor.

This output can be subsequently saved to a .CSV file.

SA Remote

SA Remote (Figure 13-14) is an iOS app that allows real-time viewing of 3-DOF and 6-DOF spatial data from SA and permits remote control of the SA Laser Tracker interface.

Figure 13-14. SA Remote.

SA Remote is available as a free app on the Apple App Store for iPhone® and iPod® touch devices. SA Remote will also run on an iPad®.

SA Remote will receive UDP data for display from watch windows in SA or the Laser Tracker interface on port 10000. If multiple sources of watch data are available, the app allows switching between the sources allowing selection of which watch source on which to focus.

SA Remote can also view/control the SA Laser Tracker interface. This requires entering the IP/hostname of the computer running the SA Laser Tracker interface into the app in the settings page. Once this is done, SA Remote will provide display and control over the interface remotely.

Setup

To download and install SA Remote, visit the app store, find the SA Remote app (http://itunes.apple.com/us/app/sa-remote/id473334282?ls=1&mt=8), and select install. Once installed, run the app.

The upper half of the app screen is used to display watch data while the lower half is used to view and control the SA Laser Tracker interface. Initially, the display will likely look like Figure 13-15 since no watch data is available yet and the SA Laser Tracker interface IP/hostname has not yet been entered.

Figure 13-15. SA Remote as it first appears on the iPhone.

Touch the small "i" info button (right) which is in the upper right corner of the screen to open the quick help screen (Figure 13-16).

Figure 13-16. The SA Remote quick help screen.

The quick help screen provides a brief summary of the setup tasks for the app. It also displays the device IP address which can be used when setting up the UDP watch transmission settings in SA.

When running SA v2011.10.20 or newer, add a watch window and enable UDP broadcast. Specify port 10000. If SA is not running on the same subnet as the iOS device, you may need to choose directed UDP transmission entering the IP address of the device. The device IP address is available in the quick help page mentioned above. Once correctly configured, the top area of the app screen should start showing watch data as in Figure 13-17.

Figure 13-17. Watch Window information is now streaming to the device.

You can also enable UDP broadcast or directed transmission from the SA Laser Tracker interface.

To configure communications with the SA Laser Tracker interface, touch the small detail button in the upper left corner of the screen to bring up the settings view. In this view (Figure 13-18), enter the IP or hostname of the computer running the SA Laser Tracker interface as well as the desired decimal precision and then tap Close.

Figure 13-18. Configuring the IP address.

Once the SA Remote app establishes communications with the SA Laser Tracker interface, the lower portion of the screen will enable and provide display and control of the interface as shown below.

Figure 13-19. Tracker control is now enabled.

When the SA Laser Tracker interface is busy, the lower portion of the screen will be disabled for input, a busy animation will show, and the measure button will change to stop, allowing you to cancel the current operation as shown at right.

If multiple sources of watch data are available (multiple watch windows and/or instrument UDPs enabled) the label at the top of the screen will turn into a button indicating that you can tap it to select which source of watch data to view (Figure 13-20).

Figure 13-20. The button in the top left indicates you can select a different watch source.

Tapping the button will open the watch source selection view shown in Figure 13-21.

Figure 13-21. Selecting the desired watch source.

Select the desired source of watch data and then tap CLOSE. The selected source will then be displayed on the main screen.

Available Status/Commands

- The COL/GRP/TGT fields show the current collection/group/target fields from the SA Laser Tracker interface and you can touch them to change the values; when changed the new values will be reflected in the interface as well.
- The + buttons to the right of GRP and TGT will auto-increment the corresponding fields (decrement is not currently supported).
- The status light will reflect what is shown in the interface.
- The X button can be used to remove the last measurement.
- The long button above the measure button shows the name of the current measurement profile. When touched, it cycles through the quick-select profiles (1 – 4) and the name updates to reflect the current profile.
- The ADM button commands an ADM Reset.
- The RH button commands a Remote Home, if one is set.
- The Measure button initiates a measurement using the current measurement profile and changes to Stop when the interface is busy and can then be used to cancel the current operation.

Transformation Tracking

For 3D instruments (e.g., Trackers) transforming objects in real time requires at least three on-line trackers. This functionality is also called Trans-Track (Figure 13-22).

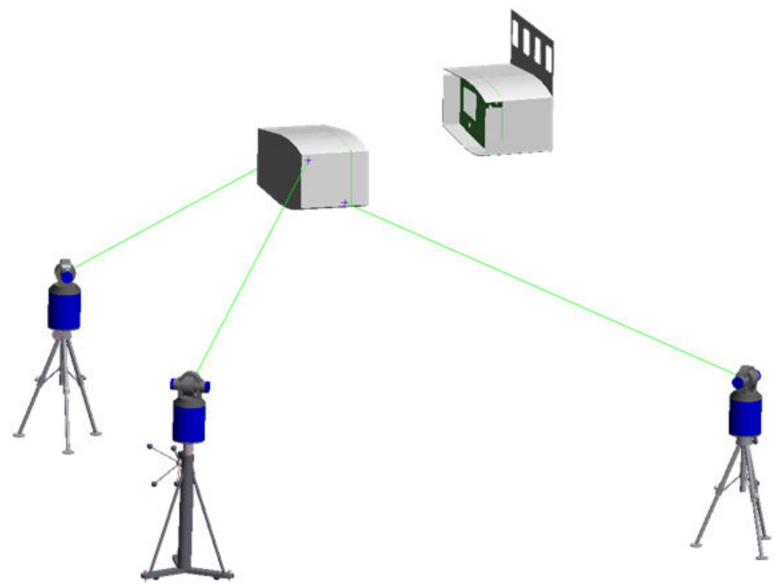


Figure 13-22. Transformation tracking.

Each instrument will monitor and update a point. The movements of the set of points are used together to update objects in 6D. Select one point per instrument. The point selection order must be consistent with the Instrument selection order e.g., select the points in the instrument sequence. After defining the instruments and tracking points; select the objects that need to move with the point set. The Trans-Track interface (shown below) indicates the instruments being monitored and the respective fit error for the point that it is tracking. The time lapse value indicates the time lapse between the measurements (i.e., synchronization error).

Relationships with Trans-Track

Relationships are dynamic by nature and work great with Trans-Track. Watch windows can be displayed for relationships which will allow real time monitoring while moving. Figure 13-23 shows an example of Trans-Track being used to join two cylindrical sections. A frame to frame relationship was used to monitor the position and orientation of the moving unit with respect to its final nominal location.

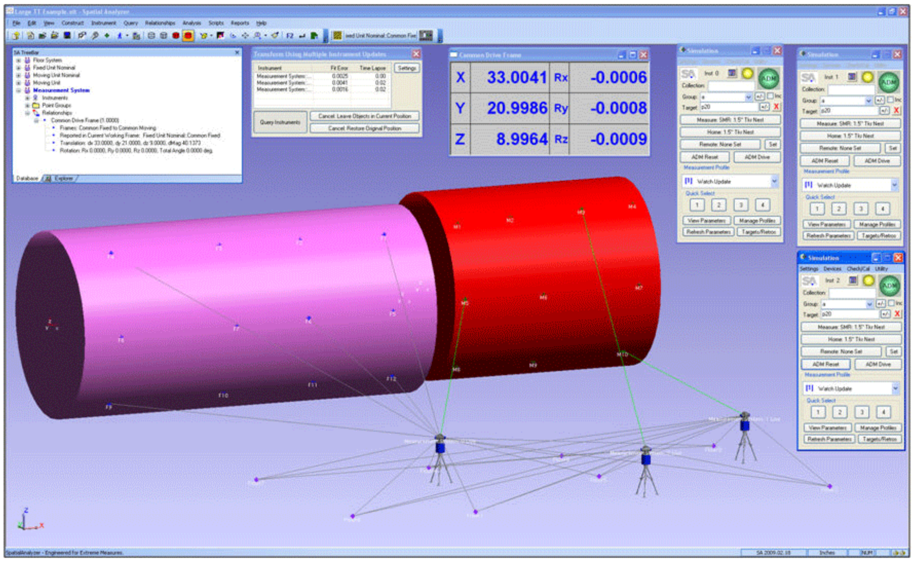


Figure 13-23. Using trans-track to join two cylindrical sections of an assembly.

Transform in 6D using Instrument Updates

Trans-Track can be accomplished with a single instrument if the instrument supports a 6D target. Examples are the Leica T-probe/T-Mac, API STS and Metris iGPS. This instrument can monitor and measure in 6D. The functionality is identical to Multiple Updates.

Laser Radar Interface

Add the Laser Radar Instrument by using or the menu item **Instrument>Add** (Figure 13-24).

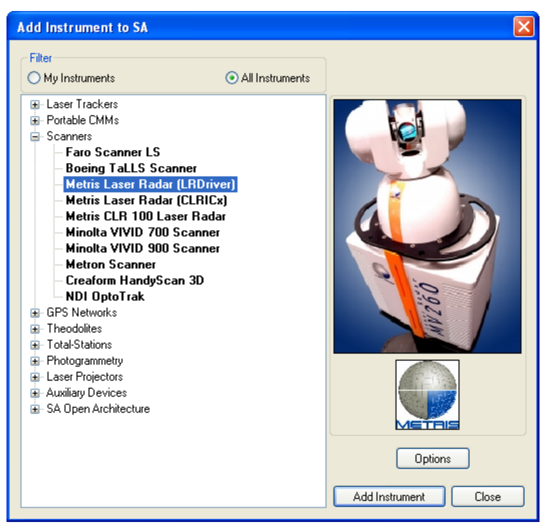


Figure 13-24. Adding the Laser Radar to the job.

Start the instrument interface using or the menu item **Instrument>Run Interface Module and Connect**.

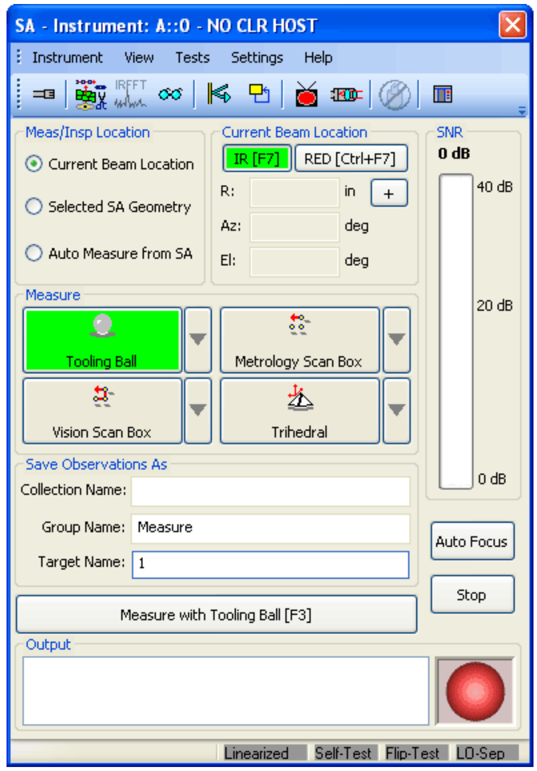


Figure 13-25. The Laser Radar interface.

Use the Connect button to display the Select CLR dialog. Select the respective instrument and press Connect (Figure 13-26).

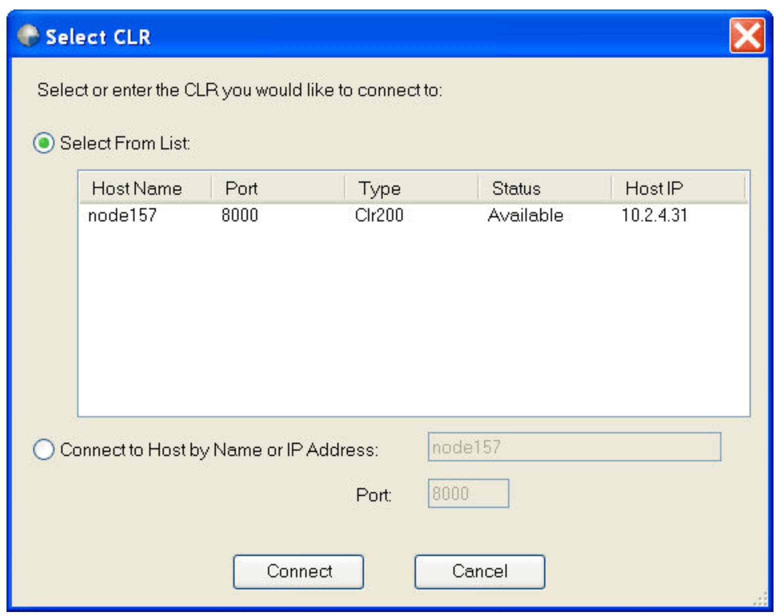


Figure 13-26. Selecting the instrument to connect.

Toolbar

Figure 13-27. The toolbar in the laser radar interface.

- **Connect/Disconnect.** Opens the LR Connection dialog (see above).

- **Target Manager - Advanced.** Displays the target manager interface (Figure 13-28).

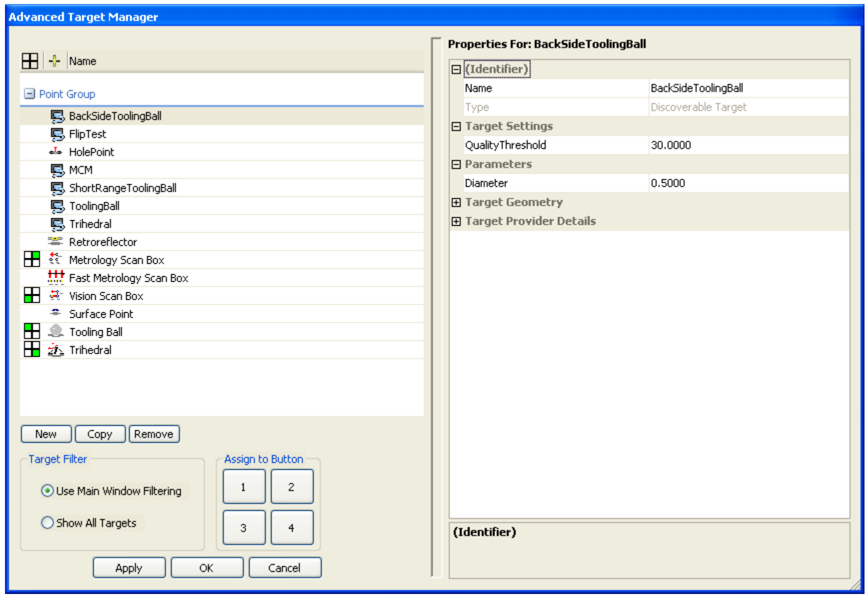


Figure 13-28. The Advanced Target Manager.

- **Show/Hide FFT Window.** Toggles the FFT window visibility (Figure 13-29).

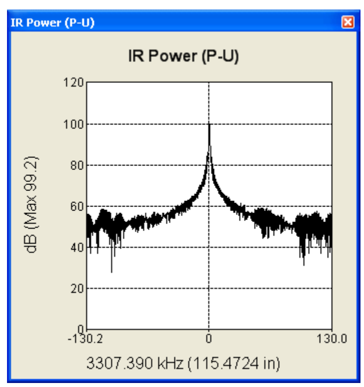


Figure 13-29. The FFT window.

- **Cloud Viewer.** Opens the cloud viewer interface (Figure 13-30).

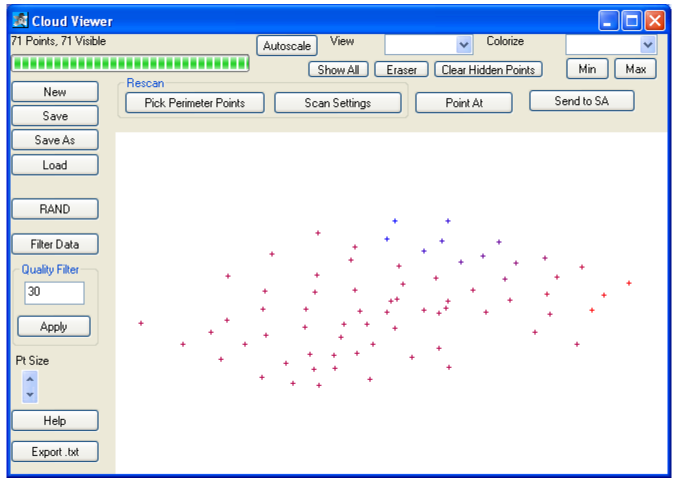


Figure 13-30. The Cloud Viewer.

- **Mirror.** Displays the mirror target dialog (Figure 13-31).

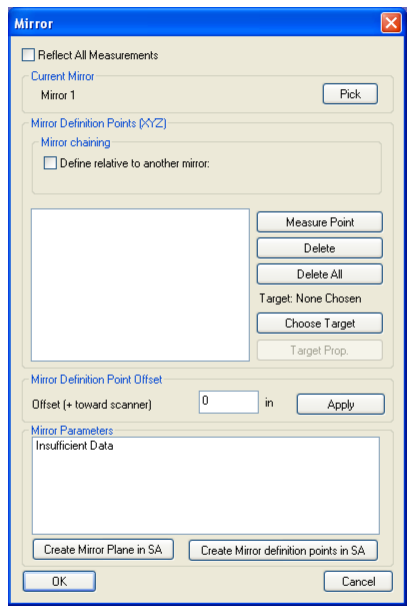


Figure 13-31. The mirror target dialog.

- **Continuously Repeat Command.** Sends updates by repeating the current measurement mode.
- **Video Window.** Toggles the video output window (Figure 13-32). Using **Ctrl** + **Shift** + **+** or **Ctrl** + **Shift** + "**-**" will increase or decrease the steering speed.

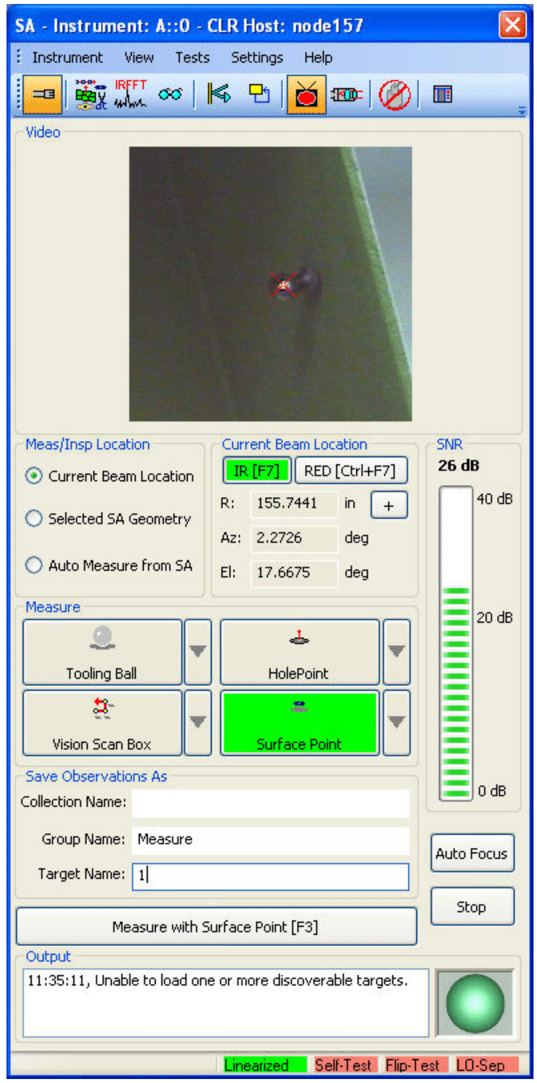


Figure 13-32. The video window.

- **Video Focus.** Displays slider for manual focusing of video camera.
- **Shutdown.** Disconnect and shutdown scanner.
- **Dock/Undock.** Docks or Undocks the instrument interface in SA.

Target Naming

Figure 13-33. Specifying a target name to measure.

- **Collection Name.** Specify the collection name for the group and target to be stored. If left blank, the group and target will be stored in the active collection.

- **Group Name.** Specify group name for target to be stored.
- **Target Naming.** Specify name for target, target name will automatically increment after each measurement.

Measure/Inspection Location

The LR interface supports three measure locations (Figure 13-34). When selected the user can configure the "measure" quick selects. When the measure operation is changed the "measure" quick selects will change.

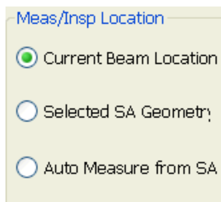


Figure 13-34. Measure quick-selects.

- **Current Beam Location.** Used to measure the current location of the beam.
- **Selected SA Geometry.** Used when a supported geometry is to be measure. An example would be a perimeter or a vector group.
- **Auto Measure from SA.** Used when SA sends an auto-measure request to the LR interface.

Current Beam Location, SNR and Auto-Focus Limits

This allows the user to view the IR SNR (signal to noise ratio) or the Red (pointing laser) focus limits (Figure 13-35). When selected the display to the right will change. When "Red" is selected the user will have quick access to the focus limits. **F7** can be used to toggle between the two displays. Auto-focus can be accessed via the interface button or use **F8**.

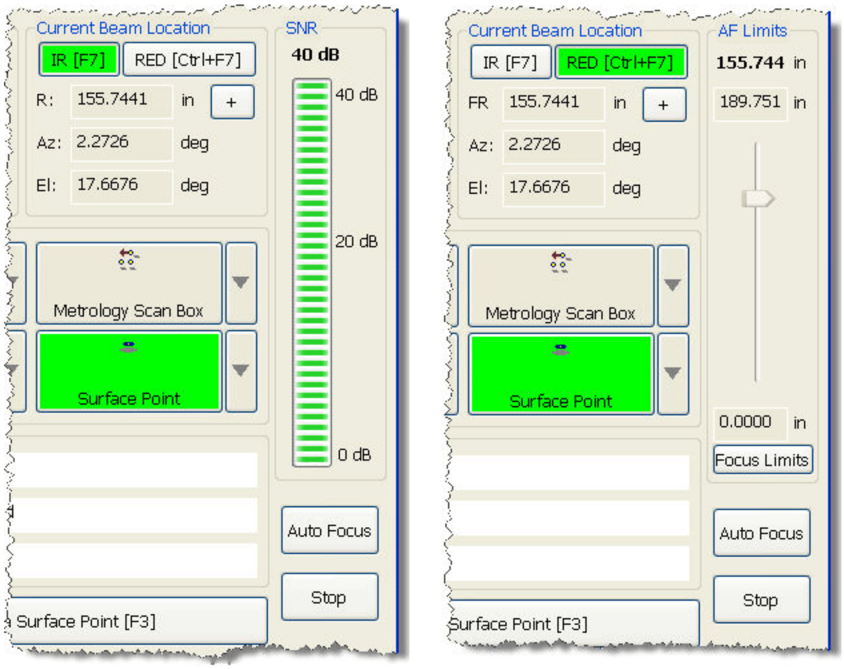


Figure 13-35. SNR/AF Limits.

Measure and Advanced Target Manager

To select a Measure mode for a particular quick select, press the arrow button the right of the desired button. A list of measure types will appear for selection (Figure 13-36).

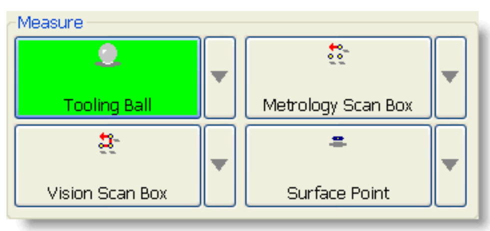


Figure 13-36. Measurement modes.

To access the parameters for the measure mode, double-left click and the ***Advanced Target Manager*** will appear (Figure 13-37). Here a user can tweak a particular profile to suit the measurement task at hand.

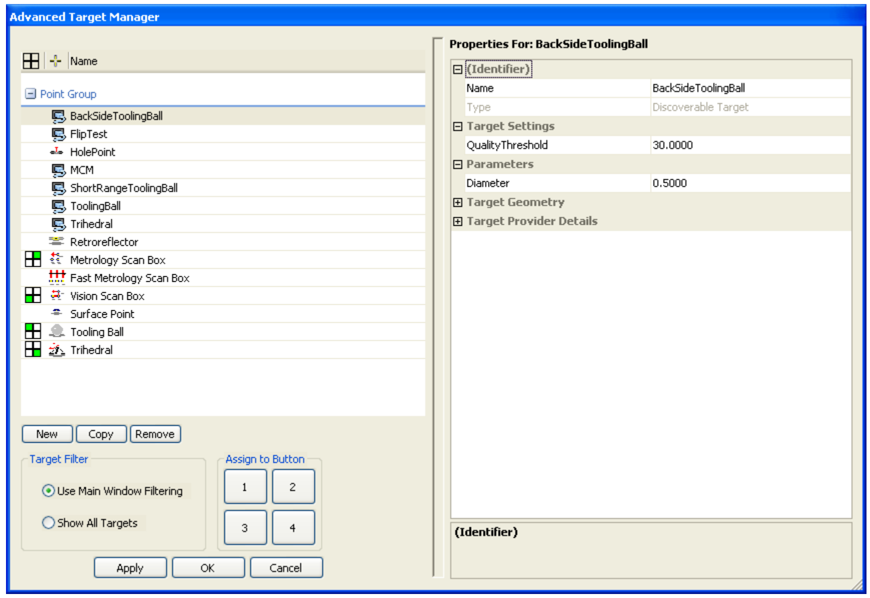


Figure 13-37. Specifying measurement details in the target manager.

Output

Displays information regarding the current measurement process (Figure 13-38).

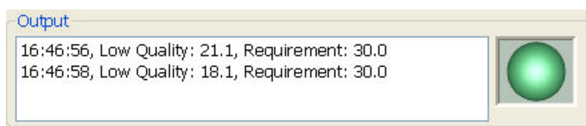


Figure 13-38. The output window.

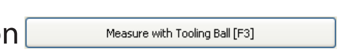
LR Operational Tests

Access the Metris LR Operational checks under the menu item Tests. Consult the Metris documentation for operational check procedures.

Single Point Measurements

This section will briefly cover some basic single point measurement types. For further information please consult Nikon Metrology.

Tooling Ball

1. Select the desired tooling ball measurement mode for selected quick select. Use the drop-down arrow to pick the desired measurement mode. If the needed measurement does not exist, create a new one in the target manager  .
2. Steer the laser to the tooling ball of choice. Point the laser to a surface near the tooling ball and press **F8** to perform an autofocus. Once complete aim the laser to the center of the tooling ball.
3. Press the measure button 

or **F3**.

4. The point will be sent to SA, if not check the output box to check measurement failure details.

Surface Point

1. Select the desired surface point measurement mode for selected quick select. Use the drop down arrow to pick the desired measurement mode. If the needed measurement does not exist, create a new one in the target manager .
2. Steer the laser to the surface point of interest. Press **F8** to perform an auto-focus.
3. Press the measure button [Measure with Surface Point [F3]] or **F3**.

Scanning Measurements

This section will briefly cover some basic scanning measurement types. For further information please consult Nikon Metrology.

Box Scanning

Box scanning allows the scanner to perform a scan bounded by a box with prescribe dimensions.

1. Point the laser at the center of the area of interest.
2. Select the appropriate scan mode. Double click the measurement mode to access scan parameters. Enter the scan box dimensions.
3. Press Measure or **F3**.
4. A scan will be performed inside the bounding box (Figure 13-39).

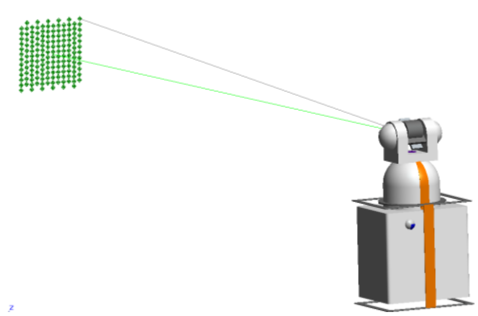


Figure 13-39. Scanning a box.

Perimeter Scanning

Perimeter scanning allows the scanning region of the LR to be constrained by a 3D perimeter.

1. In SA, select **Construct>Perimeter**. Select the measured or nominal points which describe the bounding region to be scanned. Two types of perimeters exist in SA, open and closed. A single perimeter can be changed from open to closed in the perimeter properties dialog.

2. In the LR Interface, change the *Meas/Inspec Location* to **Selected SA Geometry** (Figure 13-40).

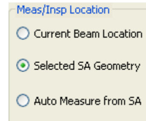
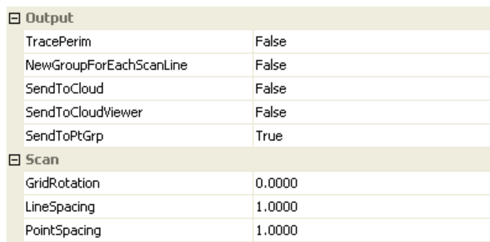


Figure 13-40. Choosing *Selected SA Geometry*.

3. In SA, select the desired perimeter. Once selected, the quick selects will populate with measurement modes applicable to perimeters.

4. Double-click the measurement mode to access scan parameters (Figure 13-41). Often times a user will change the output format, grid rotation and point spacing.



Output	
TracePerim	False
NewGroupForEachScanLine	False
SendToCloud	False
SendToCloudViewer	False
SendToPtGrp	True
Scan	
GridRotation	0.0000
LineSpacing	1.0000
PointSpacing	1.0000

Figure 13-41. Scan parameters.

5. Press Measure or **F3**.

Theodolite Manager

Theodolite manager provides a solution for interfacing all modern digital theodolites to SpatialAnalyzer. Theodolite Manager communicates through standard RS-232 serial ports to a wide variety of both conventional theodolites and total stations.

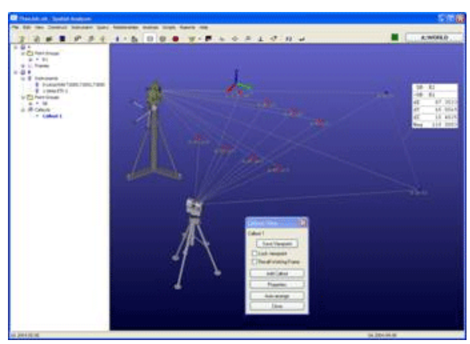


Figure 13-42. A job containing theodolites.

An image of an example SpatialAnalyzer job with Theodolites is shown in Figure 13-42. The instruments are shown with measurements and a scale bar. The actual distance between the scale bar points is shown with a Point Comparison callout view.

Cabling and Power Options

Fundamentally, communication to all current digital theodolites is done through a serial link. However, over time, different instrument manufacturers have selected slightly different flavors of serial communication to talk to their instruments. Some units use TTL (Transistor-Transistor Logic) level signals, but otherwise essentially follow RS-232 protocols. Additionally, some instruments require 6 volts for power, others 12 volts and still others 4 volts. Because of this non-uniformity among manufacturers some special cabling is usually required for a specific instrument.

Direct to the Instrument

This way of cabling an instrument is really most useful for applications where a single Total Station is to be used or where a limited network is configured. In this case, the instruments may be plugged directly into the computer or to a power augmenting device supplied by the instrument manufacturer. Most manufacturers supply cables and power units for this purpose.

Using a Serial Distribution Interface (SDI)

In instances where a user wishes to use a network of theodolites cabling issues for power and signals can become a major annoyance. The SDI unit manufactured by Brunson Instrument Company addresses these issues, by providing a simple USB device that provides all instruments with power and digital signals while still maintaining common cabling for each instrument. This is accomplished by using a signal conditioning device embedded in the Theodolite Junction Box in the last few feet of the cabling. The Theodolite Junction box performs three primary tasks:

- It alters the signal to levels suitable for each instrument.
- It provides a convenient "trigger" button remote from the instrument itself.
- It provides a junction point to which a short, theodolite-specific cable can be attached. This cable is necessary since there is no standard connector for all makes and models of theodolites.

Adding a Theodolite to an SA Job

Just as with other instrumentation in SA, the first step is to add the desired instrument to the SA job. This is done by selecting **Instrument>Add** from the main menu. Now you need to start the corresponding instrument interface. This can be done from the Operating System Start-Bar, or by selecting **Instrument>Run Instrument Module** from the main menu. After selecting the appropriate instrument, you will see the initialization dialog pictured in Figure 13-43.

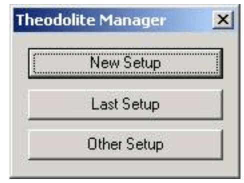


Figure 13-43. The Theodolite manager setup buttons.

These three buttons allow you to either make a new configuration of instruments, recall the last configuration, or recall a stored configuration by name. The first time you run the software you must make a new configuration.

You will then see the main ***Theodolite Manager*** dialog (Figure 13-44).

Figure 13-44. The Theodolite Manager.

The following sections will describe each component of the interface by section.

The area appearing as a large blank region is a listing of all active instruments being managed by this instance of the theodolite manager. It is possible to have multiple versions of theodolite manager tied into the same SA job at once.

Figure 13-45. The control buttons.

The control buttons (Figure 13-45) allow the operator to perform global tasks related to Theodolite Manager. In order from top to bottom these buttons provide the capability to save a Theodolite Manager configuration, recall a configuration, set user options for a configuration, and view information about Theodolite Manager.

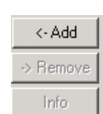


Figure 13-46. Buttons for managing the instrument list.

The buttons to the right of the instrument list are used to manage the instrument list (Figure 13-46). Here the user may add, delete, or display information about a specified theodolite connection.

The buttons at the bottom of the dialog are user to poll the instruments connected to Theodolite Manager (Figure 13-47). The top two only poll specifically highlighted instruments in the instrument list. If none are highlighted the buttons are deactivated.

Figure 13-47. Instrument polling buttons.

To add an instrument, click the Add button. A progress bar will display briefly while Theodolite Manager scans for available communication ports. The ***Add Instrument*** dialog will then appear (Figure 13-48).

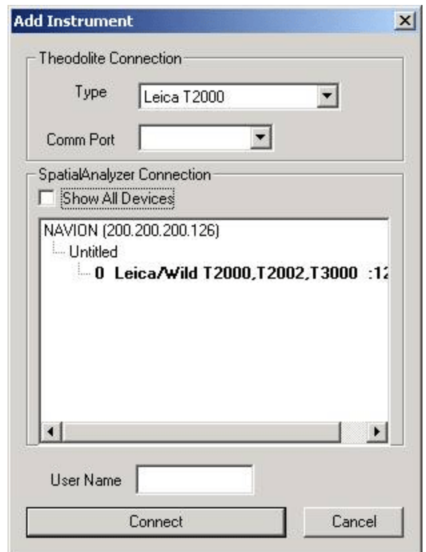


Figure 13-48. The Add Instrument dialog.

In the Theodolite connection area of the dialog users select the desired theodolite and its corresponding communications port.

The operator must then identify which SpatialAnalyzer job and which instrument within that job it corresponds to. By default the dialog limits its display to compatible and available instruments in all jobs on the network. If the show all devices checkbox is checked all instruments and SA jobs on the local network are displayed. The operator must simply highlight the desired instrument.

The operator also has the opportunity to optionally enter the user name for the operator of this instrument. If no name is given the instruments will be referred to as Station1, Station2, etc.

Once all selections are made the operator may click the Connect button. At this time the instrument is initialized and an item is added to the instrument list.

In this case it identifies that SpatialAnalyzer Instrument 0 (SP0) is a Leica T2000 connected to serial port Com4. After adding a second instrument the display might appear something like Figure 13-49.

Figure 13-49. This theodolite manager setup has two instruments added.

Notice that in this case we have highlighted one of the instruments and the other buttons in the interface have therefore been activated. Multiple highlights are possible and are accomplished using the standard windows selection methods of holding down either **Ctrl** to add select, or **Shift** to region select multiple instruments.

To demonstrate that the instruments are now connected, move the instruments to a new position and click the Query All button. You should see all the scopes move to their actual azimuth and elevation in SA.

Measurement Modes

Theodolite Manager allows operators to use theodolites in one of three primary measurement modes. These modes have been established based on a survey of common methods for applying theodolites. Some industries or applications may exclusively use one method while still others will vary their selection with each task at hand.

To change Measurement modes, click the User Options button. The dialog in Figure 13-50 will be displayed.

Figure 13-50. The Theodolite Manager's User Options.

The **Target Thickness** section will apply to all measurement modes. This is simply the target thickness (probe offset) used in SA for various analysis tasks such as point to surface queries.

Freeform Mode

The Freeform measurement mode is really designed for experienced crews who want and have the ability to enter target numbers at the theodolite. By far the biggest collection of these users are experienced operators of Leica T2000, T2002, and T3000 series theodolites. However, this methodology applies to other instruments as well. Newer Leica instruments and Zeiss Eth-2's for example, can use this method.

In freeform mode it is always necessary that the operator set the target number on the scope before recording. Recording may be accomplished at the instrument or the interface as desired.

Since SA uses both Target Names and Group Names the strategy employed is to arbitrarily divide the number into a group component and a target component. Several examples will be presented below to further illustrate the concept.

With the following settings (Figure 13-51):

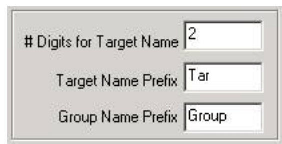


Figure 13-51.

the target numbers would convert to:

Target Number	Group Name	Target Name
1000	Group10	Tar00
2505	Group25	Tar05
2833	Group28	Tar33

With the following settings (Figure 13-52):

Figure 13-52.

The target numbers would convert to:

Target Number	Group Name	Target Name
1000	Fender1	T000
2505	Fender2	T505
2833	Fender2	T833

Single Target Mode

This mode is often used when building operations need to be performed or when a novice crew needs to be guided through the measurement process. The basic premise is that all theodolites will typically shoot the same target at the same time. This is particularly helpful for inexperienced users since sighting errors can be immediately identified and corrected.

After enabling the Single Target Mode in the user options dialog, the interface will appear as pictured in Figure 13-53.

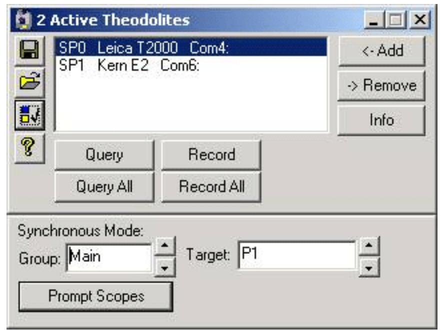


Figure 13-53. Single Target mode.

Note the extra section added to the bottom of the standard dialog. This enables users to enter alphanumeric group and target names. When the Prompt Scopes button is hit, the theodolites will display the desired target to prompt the scope operators to point at it. Any incrementing or decrementing of the group or target name will automatically update the instrument prompts. Recording may be accomplished through the standard user interface or by hitting the record button on the theodolite or its corresponding Theodolite Junction Box.

Prompted List Mode

This measurement mode is designed for tasks where the desired survey points are known right from the start. These names may either come from a CAD file, an ASCII file or may be entered at survey time by the operators.

When the Prompted list interface is initiated, the user interface appears as in Figure 13-54.

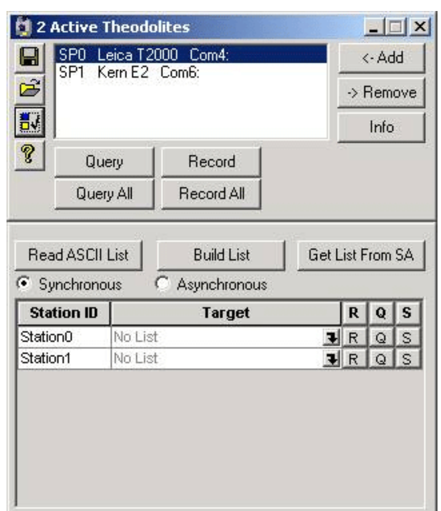


Figure 13-54. Prompted list mode.

Alternatively, if user names were entered as the instruments were added, this dialog might appear as in Figure 13-55.

Figure 13-55. User names in the dialog.

In this mode, each instrument has it's own list of targets to be surveyed. This list may be created on the spot using an ascii editor such as notepad, or by using the Build List option. Or it may be read from an existing file with the Read ASCII List option. Alternatively the user may hit the Get List From SA button and they will then be prompted in SA to select points. All points are identified by both group and target names with the following format (group::target).

When the Build List button is hit, the operator sees the Construct Target List dialog (Figure 13-56).

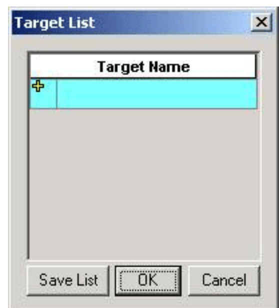


Figure 13-56. The Construct Target List dialog.

By typing a "group::target" name in the green field and hitting enter a new target is added to the list.

After several are entered our list might appear as follows (Figure 13-57).

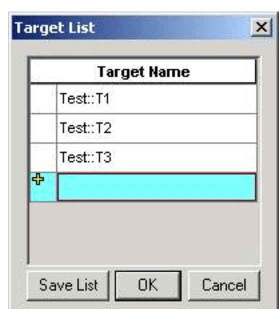
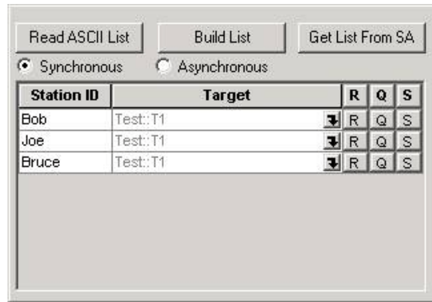


Figure 13-57. Targets have been added to the list.

Clicking OK will retain the list for this job. Clicking Save List will allow the operator to specify a filename so that this may be recalled later (always a good idea).

Returning to the main dialog, we see that the instrument list now has two lines. One line corresponding to each instrument station (or user).

Figure 13-58.



The buttons to the far right hand side in each row allow the operator to Record-R, Query–Q, or Skip–S a specific instrument (Figure 13-58).

The second column is the current target for a given user. When the dialog is first presented this column displays "No List" since a target list has yet to be loaded. After the target list is loaded, this column will display the current target for each instrument.

In Synchronous mode all instruments will be prompted to shoot a specific target. As each operator acquires data, they will be prompted to "Wait". When the last theodolite acquires data, all scopes will be prompted with the next target name. In Asynchronous mode all operators may proceed through the list at their own rate.

A target hot-spot box to choose the current target for that scope (more about that later). To the right are buttons for Record, Query, and Skip.

The down arrow in the second column is called the Target Hot Spot Control. The target hot spot pops up the target list for each instrument. When you hit the hotspot you get the list at right.

This is similar to the building list except the user cannot add new targets and the left column displays target status for that particular instrument.

The left column presents icons to indicate state of the target (Shot, Skipped, UnShot). The bold item is the current target for that user. You can click select what target you want to shoot next, and then either continue from there or (optionally) only prompt skipped or undone targets. When you shoot or skip the scope will proceed to the next target in the list. In Asynchronous mode all scopes use the same list but can progress through them at their own pace, skipping as required, and going back as required. Note that the operator will not normally view the list in this way, normally they would view all this information in the main dialog. The list is just a control to allow the user to move around the job freely.

Theodolite Data Observer

Theodolite Data Observer is activated from SA's Instrument menu. This dialog makes it easier to drive Theodolite systems from the graphic view in SA. The interface is shown in Figure 13-59.

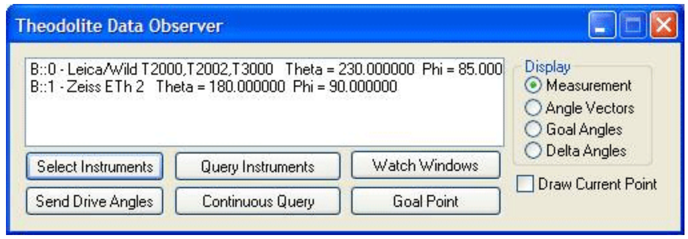


Figure 13-59. Theodolite Data observer.

This interface allows the display of Measurements, Angle Vectors, Goal Angle, and Delta Angles. Each view of the data is applicable for different applications. Use the Select Instrument button to choose the instruments to control from this control. Select the desired Goal Point then Send Drive Angles and Watch Window options to setup a real-time graphical build mode for the currently selected instruments. Continuously Query will repeatedly record angle measurements from the instruments. This mode is important for tracking the theodolites movements through a measurement process.

GSI V-Stars Photogrammetry Simulation

Add a GSI V-Stars Photogrammetry System to SA by using **Instrument>Add** or the Add Instrument button (Figure 13-60).

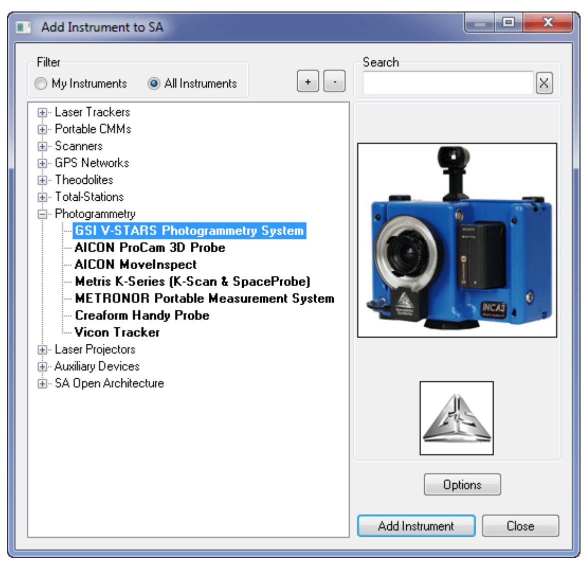


Figure 13-60. Adding the V-STARS system to an SA file.

Instrument Properties for Simulation

By right-clicking on the V-STARS instrument within the SA tree and selecting **Properties**, the properties are displayed (Figure 13-61).

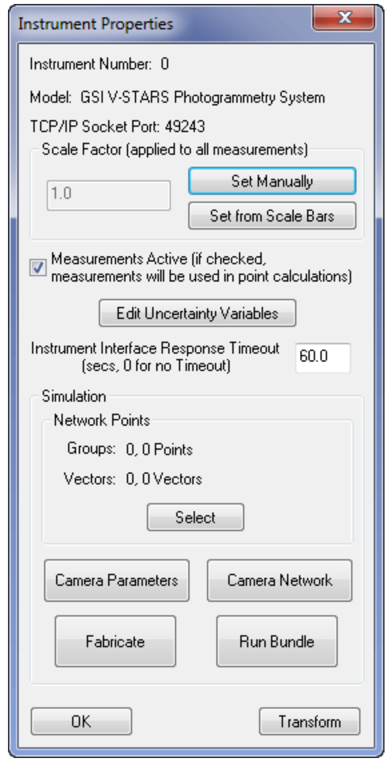


Figure 13-61. The properties of the V-Stars instrument.

Simulation

Contains all the functions and options to perform Photogrammetry Simulation.

Network Points

Defines the point and vector groups that will be used during the simulation process. Select defines the point and vector groups for analysis. The selection process will then show within the SA graphical view area. First the user will be prompted to select the desired point groups. Once all point groups have been specified, pressing the **Enter** key will cause prompting to continue for vector groups. If no items of a particular group are desired, simply press **Enter** to skip. Once the desired groups have been specified, the dialog will update to show the specified number for each group type.

Camera Parameters

The ***Camera Parameters*** dialog (Figure 13-62) provides a means of setting the necessary information for field of view display and determining what points are within the field of view. The **Incidence Angle** parameter is used when checking whether a vector is visible within a camera's field of view.

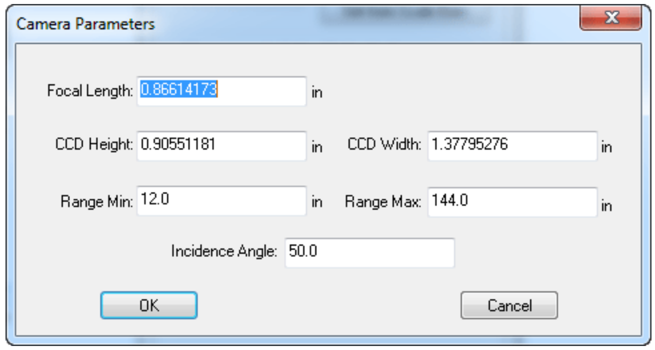


Figure 13-62. Camera Parameters.

Camera Network

The ***Camera Network Configuration*** dialog provides a means of setting the various options for each camera (Figure 13-63).

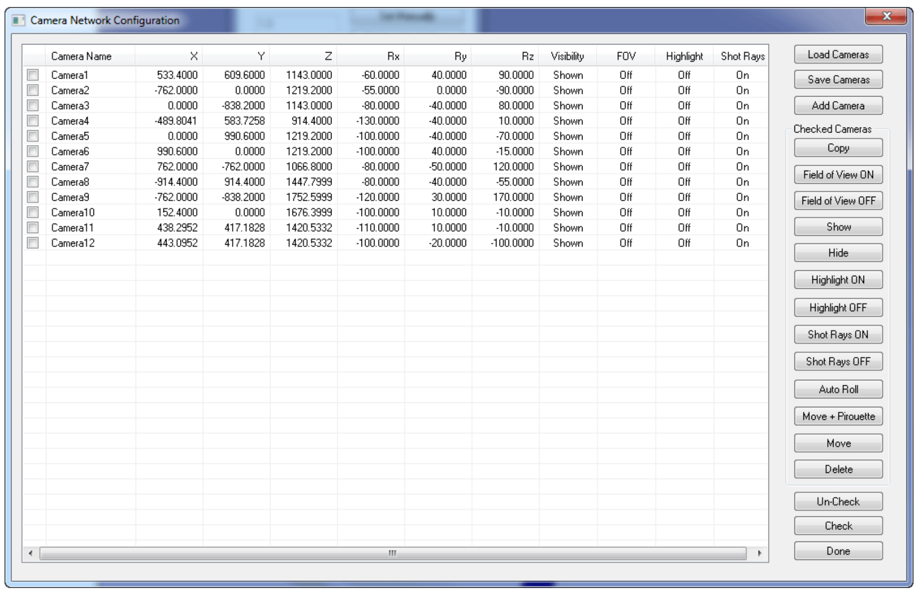


Figure 13-63. Camera Network Configuration dialog.

- **Load Cameras.** Load a predefined camera network.
- **Save Cameras.** Save an existing camera network.
- **Add Camera.** Places a new camera at the current working frame and display the transform dialog for the newly added camera.
- **Field of View ON.** Turn on field of view display for the selected cameras from the list.
- **Field of View OFF.** Turn off field of view display for the selected cameras from the list.
- **Show.** Cause the selected cameras to be displayed within the graphics view window.
- **Hide.** Cause the selected cameras to be removed from the graphics view display window.
- **Highlight ON.** Cause the selected cameras to become high-

lighted within the graphics view display window.

- **Highlight OFF.** Cause the selected cameras to become de-highlighted within the graphics view display window.
- **Shot Rays ON.** Cause the selected cameras to display shot rays to their visible measurements.
- **Shot Rays OFF.** Disable shot ray display for the selected cameras.
- **Move.** Move the selected cameras about the current working frame.
- **Delete.** Delete the selected cameras.
- **Un-Check.** Un-check ALL items within the list.
- **Check.** Check ALL items within the list.

Right clicking on the desired camera within the list will produce the menu options in Figure 13-64:

Figure 13-64. Menu options.

Fabricate

Executes the analysis of points/vectors within the field of view of the visible camera set. Once completed, the points found to be within the unobstructed view of the cameras will be highlighted (Figure 13-65).

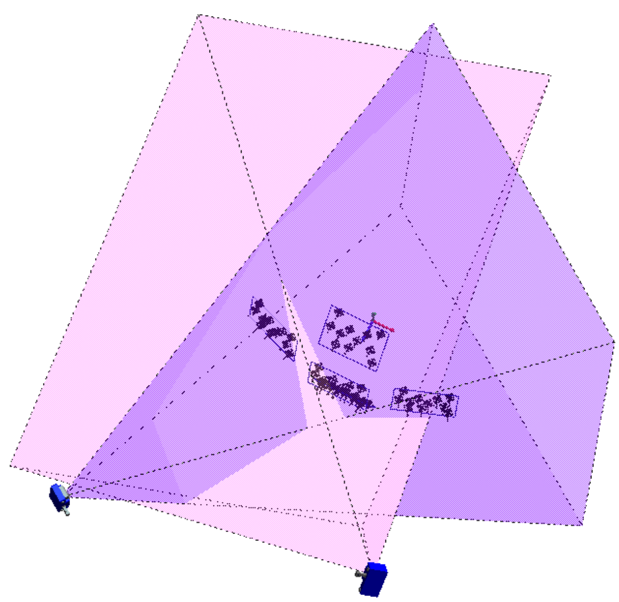


Figure 13-65. Fabrication.

Bundle

Performs uncertainty analysis and update the SA graphics and point data with the results. When the button has been actuated, the user will be prompted for a filename for the Common Photogrammetry File that will be created and used by V-Stars.

Note: V-Stars must be running before attempting to bundle. V-Stars also requires at least one camera station to be rotated 90 degrees about the lens axis to successfully bundle.

Once V-Stars has completed the analysis, the resulting uncertainty will be updated within the Common Photogrammetry File and displayed within SA (Figure 13-66).

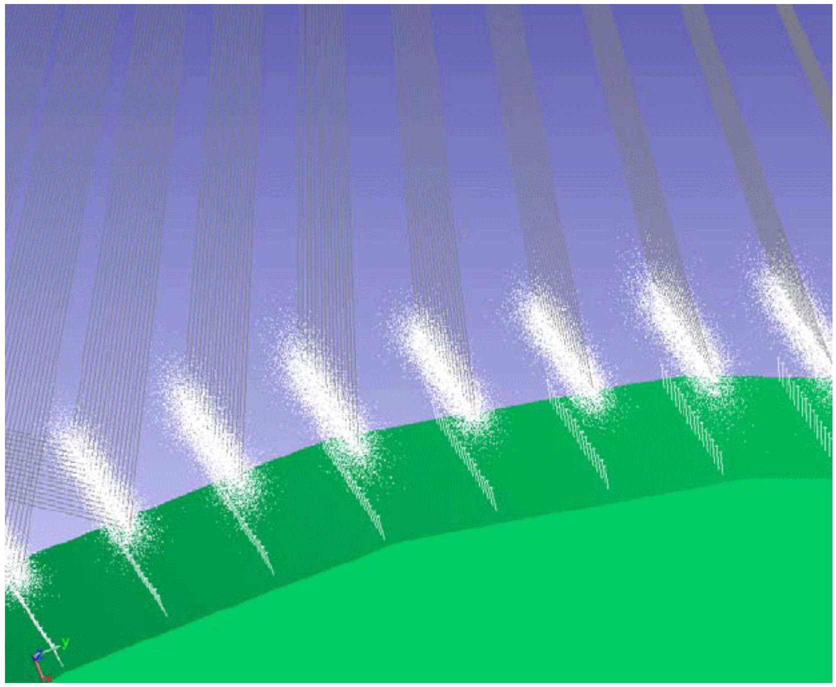


Figure 13-66. Photogrammetry uncertainties.

Camera Orientation

The X-Axis is shown in red, the Y-Axis in Green, and the Z-Axis in Blue (Figure 13-67).

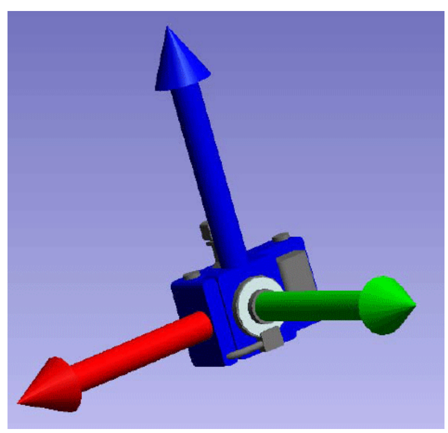


Figure 13-67. Camera orientation.

Measuring With Laser Trackers

CHAPTER 14

Laser trackers are prolific portable metrology instruments. They have many uses for both short- and long-range high-accuracy measurements of both discrete and scanned point data. This chapter discusses how to properly and effectively measure with laser trackers in SA.

There are two ways to measure with laser trackers in SA. The first is through the conventional laser tracker interface, which provides access to advanced functionality and settings and allows complete control over the instrument, target definitions, and measurement modes. The second method is through the simpler Instrument Toolbar, which provides an easy-to-use, streamlined approach to laser tracker measurement.

Note: For information on starting the laser tracker interface, refer to the Instruments chapter.

You'll find that most tracker measurement can be carried out in the Instrument Toolbar, which provides accelerated workflow for common tasks such as discrete and stable point measurement and spatial scanning. However, when more control is needed—or when detailed settings or profiles need to be created or modified—the traditional laser tracker interface provides the ability to work at a deeper level.

We'll first cover the laser tracker interface in detail, then cover the Instrument Toolbar.

Laser Tracker Interface

SpatialAnalyzer's laser tracker interface (Figure 14-1) provides the operator with a great deal of control when measuring with a laser tracker device. The interface has five main sections: the menu and toolbar, target naming, measurement control, measurement profile control, and additional controls.

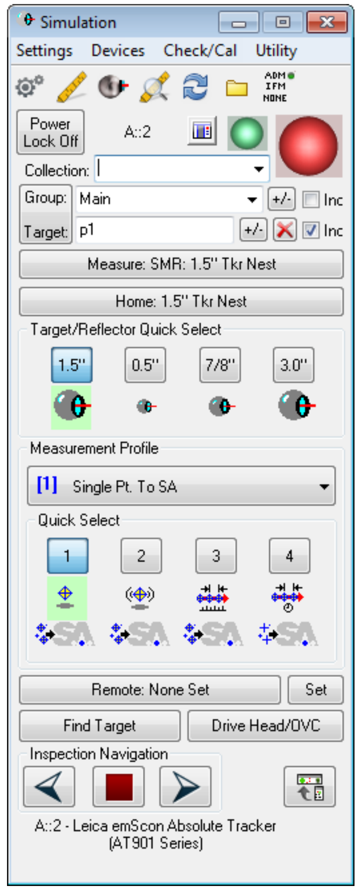


Figure 14-1. The Laser Tracker interface.

At the top of the interface, an interface docking button toggles between docking the interface to the SA window or letting it float as a separate window. If the instrument interface is opened without running SA, or if the interface is run on a machine other than the machine running SA, this button is unavailable (as there would be no instance of SA on the computer to dock with). If more than one interface is running, docking an interface will replace any currently docked interface.

Interface Overview

Menus

Use the menus to get to detailed tracker settings, instrument monitoring utilities, compensation check routines, and other instrument-related utilities.

Toolbar

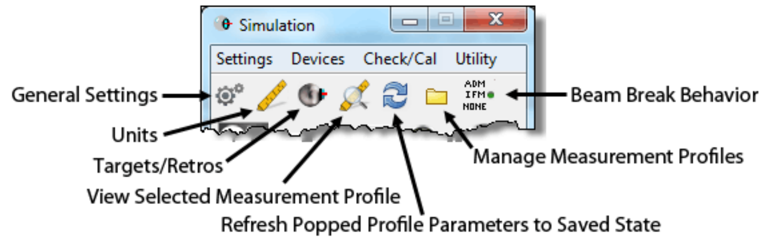


Figure 14-2. The tracker interface's toolbar.

The toolbar (Figure 14-2) provides buttons for commonly-accessed areas of the instrument interface.

Status Indicators

Two colored status indicators (sometimes referred to as traffic lights) indicate the current status of the weather station and tracker beam.

Target Naming

Figure 14-3. Naming for measured targets.

Controlling target names (Figure 14-3) is an important part of the tracker interface. Use this section to control how measured points are named and organized in the SA tree.

Measurement Control

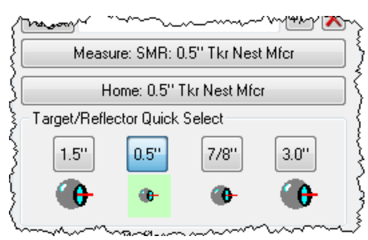


Figure 14-4. Measurement buttons.

These controls (Figure 14-4) provide access to the critical measurement commands required for operating a laser tracker, including taking a measurement, returning the instrument to the home position, and setting the active target.

Measurement Profile Control

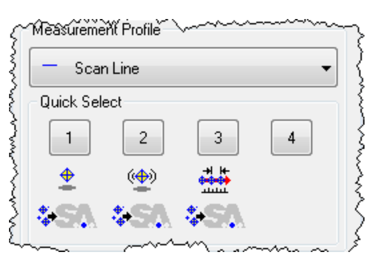


Figure 14-5. The measurement profile dropdown.

This section (Figure 14-5) allows you to quickly switch to any number of predefined measurement profiles. These profiles precisely control what's measured, how it's measured, and what's done with the measurements once they come into the interface.

Additional Controls

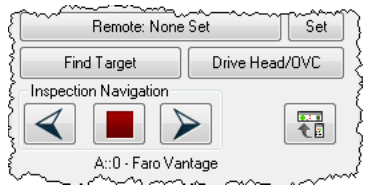


Figure 14-6. Additional tracker controls.

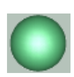
The additional controls (Figure 14-6) give you the ability to set and recall remote home positions, perform ADM locks, drive the tracker servos via a variety of different methods, switch to the Laser Tracker Toolbar, and navigate an inspection in the SA Toolkit.

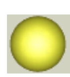
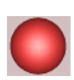
Collection/Instrument ID

When an instrument is added to an SA job, it is assigned an index number. The index (which, for a given collection, begins counting from zero) is used to associate measurements to instruments and also to differentiate one instrument from another, since multiple instruments (of the same type) are frequently added to a job to enable multiple station surveys. The instrument index number is incremented by one as each new instrument is added to a collection. This text displays the connected instrument's collection, index number, and name.

Beam Status

The Beam Status indicates both the current state of the laser tracker beam, as well as the distance measurement technology being used (IFM vs. ADM). There are four possible states:

-  **Green.** The tracker is locked onto the retroreflector, and the range data has been set using IFM (Interferometry) technology.
- 

Green/ADM. The tracker is locked onto the retroreflector, and the range data has been set using ADM (Absolute Distance Meter) technology.
-  **Yellow.** The tracker has locked onto the retroreflector, but a valid distance measurement has not yet been attained.
-  **Red.** The tracker is not currently locked onto the retroreflector, and valid range data is not available.

Power Lock

For instruments that support Power Lock, this functionality can be toggled on and off by clicking the Power Lock button (Figure 14-7).

Figure 14-7. The Power Lock toggle button.

Target Naming

Figure 14-8. The collection, group, and target name for a measured point is specified here.

Each measured point in SA has a collection, group, and target name (Figure 14-8). This naming system is used for organizing measured data into logical groups. In general, before taking a set of measurements the target naming section should be filled out in order to measure points into the proper collections and point groups.

Collection

Enter the destination collection name for the resulting targets or geometries, or select an existing collection from the dropdown. If this field is left blank, the measurements will be placed by default into the active collection.

Group

To place measured targets into an existing point group, either type in the group name or select it from the dropdown. To place measurements into a new group, enter a new name into the field.

- **Increment/Decrement.** The Increment/Decrement button +/- allows you to quickly increment or decrement the current group name by one. To increment a group, left-click this button. To decrement, right-click the button. If a group name does not have a numeric suffix, one will be added automatically.
- **Increment Checkbox.** The increment checkbox ☑ Inc specifies whether a group name should be automatically incremented at the completion of a measurement profile. If checked, then when a measurement profile completes the group name will automatically increment by one. Otherwise, the group name will not be changed at the completion of a measurement profile. This is perhaps most commonly used when measurement profiles are iterated.

Target

Enter a name for a specific target (measured point) into the *Target* field. This target name is automatically incremented after a measurement. If a target name does not have a numeric suffix, one will be added automatically. If the increment checkbox ☑ Inc is checked next to the *Group* field, then the target name will reset to 1 after the group name has been incremented.

- **Increment/Decrement.** The Increment/Decrement button quickly increments (left-click) or decrements (right-click) the target name by one. If incrementing and a target name does not have a numeric suffix, one will be added automatically.
- **Delete Last Point.** The Delete Last Point button will delete the most recently measured point, if it exists.
- **Increment Checkbox.** The increment checkbox Inc increments the point name when enabled, and leaves the point name as-is when disable. It is on by default (and is automatically turned back on every time you start the interface). Turn this off only if you intend to take multiple observations under the same point name.

Measurement Control

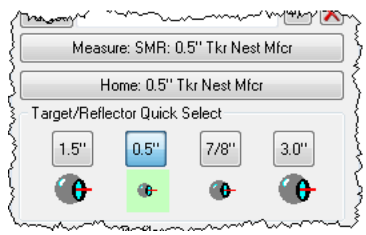


Figure 14-9. Controls for measurement.

Measure

Triggers the active measurement mode, which can do anything from take a single measured point to trigger a scan. The name of the currently active target in the interface is listed on the Measure button.

Home

The Home button sends the instrument back to its home position and attempts to lock on the active reflector. The instrument interface will wait until it is able to successfully lock onto the reflector or the operator aborts the process. A successful lock on the reflector sets the beam status indicator to a green state.

Some trackers do not have a home position defined. In this case, this button will be grayed out.

Target/Reflector Quick Select

Since changing target definitions happens somewhat frequently when measuring with a tracker, the target/reflector quick select buttons allow you to assign up to four target definitions to quick-select buttons. Once defined, a simple left-click activates the new target, avoiding the need to navigate into the targets/reflectors dialog to activate a different target. For more on target definitions, see Targets and Retros below.

To Assign a Defined Target to a Quick Select Button:

- Right-click the desired button and select the named target from the list.

Measurement Profile Control

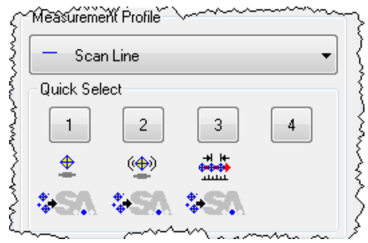


Figure 14-10. Measurement Profile control.

The *Measurement Profile* dropdown (Figure 14-10) allows you to select any of the saved profiles and set it as the active profile. After doing so, clicking the Measure button initiates that selected profile. There are also four profile quick select buttons, which allow you to easily assign a profile definition to each.

To Assign a Defined Measurement Profile to a Quick Select Button:

- Right-click a button and select the desired profile from the list.

For more on measurement profiles, see "Measurement Profiles" on page 439.

Additional Controls

Remote Home

Warning: When setting a Remote IFM position, it is very important that the position be stable and free from vibration. Any movement relative to the tracker will increase error in the IFM distance measurements.

A *remote IFM home* can be established in the laser tracker interface. A remote IFM home allows you to define a new "home" position which is used by the tracker for seeding the IFM distance value. This new position will have a stored IFM distance which will be used to initialize the tracker's range when locking onto this remote home position. The major advantage of this capability is that when working in a position that makes it difficult or time-consuming to access the tracker home position (such as when on scaffolding or far from the tracker), you can define a new home position. By pressing the Remote Home button, the tracker will point to the defined home position and wait for a retroreflector. Once a retro is acquired, the IFM distance value will be initialized by the stored value.

Warning: When setting remote home positions, the tracker should be in IFM distance measurement mode.

Note that when a remote home position is specfied the remote home button will contain the custom name of the remote home position.

To Define a New Remote Home Position:

1. Click the Set button. The ***Remote IFM Reset Positions*** dialog will appear (see Figure 14-11).

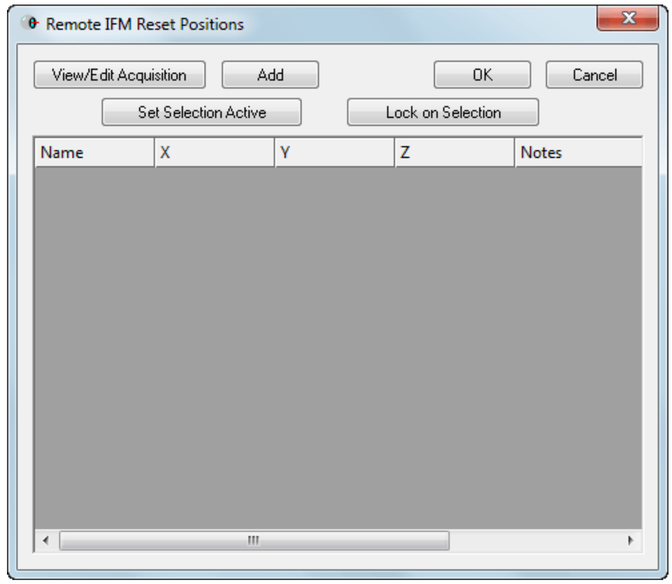


Figure 14-11. The dialog used for setting remote homing positions.

2. Place the retroreflector into a monument or other stable point of reference and click the Add button. The current position will be measured and added to the window.

3. Click the *Name* or *Notes* fields to add a custom name to the remote home position, or a custom note about the position (see Figure 14-12).

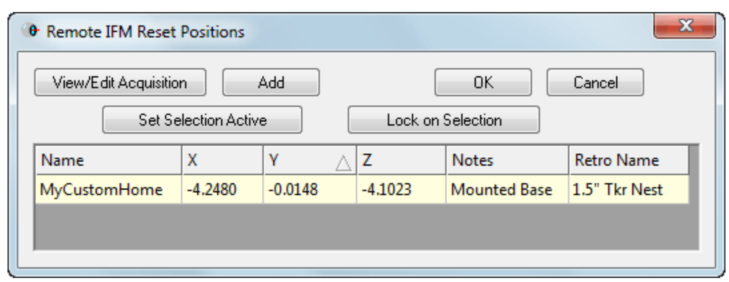


Figure 14-12. A remote home position has been set.

To Set a Remote Home Position as the Active Remote Home:

1. Click the Set button. The ***Remote IFM Reset Positions*** dialog will appear.

2. Select the remote home position you'd like to activate.

3. Click the Set Selection Active button.

4. Click the OK button. The Remote Home button will list the new active remote home position.

To Change How a Remote Home Position is Measured:

1. Click the Set button. The ***Remote IFM Reset Positions*** dialog will appear.

2. Click the View/Edit Acquisition button.

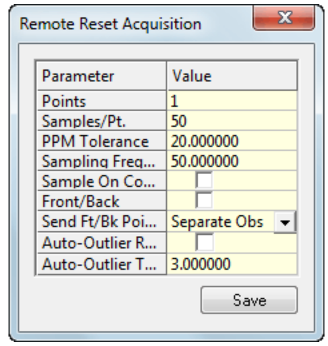


Figure 14-13. Changing the remote home acquisition method.

3. In the ***Remote Reset Acquisiton*** dialog (see Figure 14-13), choose the desired parameters for the remote home measurement. For more details on the specific parameters see "Measurement Profiles" on page 439.
4. Click the Save button.

To Point to and Lock on Any Specific Remote Home Position:

1. Click the Set button. The ***Remote IFM Reset Positions*** dialog will appear.
2. Select the row containing the desired remote home position to point at and lock on.
3. Click the Lock on Selection button.

To Delete a Remote Home Position:

1. Click the Set button. The ***Remote IFM Reset Positions*** dialog will appear.
2. Select any row containing the remote home position to delete.
3. Press the `Delete` key and confirm the deletion.

To Lock on the Active Remote Home Position:

1. Set the retroreflector into the nest/monument at the currently active remote home position.
2. In the tracker interface, click the Remote button. The tracker will point to the remote home position and lock onto the reflector.

Find Target

Note: Depending on manufacturer and proprietary methods, some trackers may not show the ADM indication after locking back on.

The Find Target button will force the tracker to spiral outward from the tracker's current pointing direction and lock onto the first reflector that it can find, using the Absolute Distance Meter (ADM) for the distance measurement. Once locked, the green beam status indicator with the "ADM" label will be displayed, indicating that range has been set via ADM.

To Lock Onto a Reflector Using ADM:

1. Point the tracker generally toward the reflector you'd like to lock onto. The closer the tracker is pointed to the center of the reflector, the faster the lock will occur.
2. Click the Find Target button and wait for the tracker to find and acquire the reflector.

To Adjust the Search Parameters for ADM Reset:

1. From the interface menu, select **Settings>Tracker>General Settings**, or click the Settings button in the toolbar.

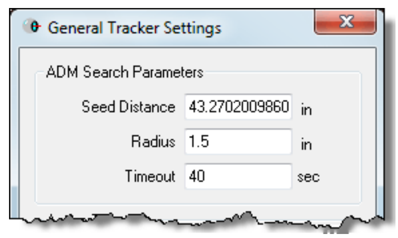


Figure 14-14. Changing the ADM Reset search parameters.

2. The ***General Tracker Settings*** dialog (Figure 14-14) will appear. The *ADM Search Parameters* section contains the applicable settings, which are described below.

- **Seed Distance.** Coupled with the **Radius** value, the **Seed Distance** is used to calculate the maximum horizontal and vertical angles for the tracker's spiral search. This value is typically set to the distance of the most recent measurement. As a general rule, if you have trouble locking onto a target, this value should be similar to the approximate distance from the tracker to the SMR.
- **Radius.** The outward distance (perpendicular to the line-of-sight) from the tracker's starting pointing direction that will be searched for a reflector.
- **Timeout.** If the tracker has not locked onto a reflector within this time limit (and if it is still searching), then the search will be aborted. This setting allows you to avoid a very long search operation if a large radius is used.

Drive Head/OVC

Clicking the Drive Head/OVC button opens a window giving you the capability to point the tracker using one of several manual methods, such as using the keyboard or mouse (Figure 14-15). If capable, camera-equipped trackers will display a video window when this mode is active.

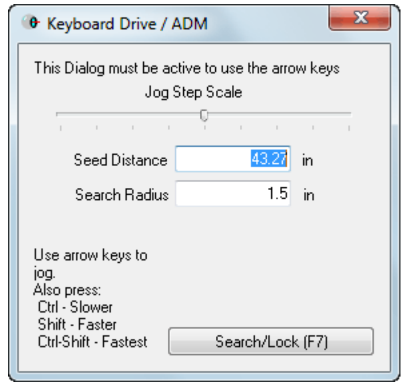


Figure 14-15. Driving a tracker with the keyboard.

To Steer the Tracker to a Target:

Note: This method may vary depending on instrument manufacturer.

- With the ***Keyboard Drive/ADM*** dialog open, use the arrow keys to move the tracker beam.

Inspection Navigation

Three Inspection Navigation controls are available at the bottom of the tracker interface (Figure 14-16). These controls are designed to be used while trapping measurements, either as part of an inspection routine using the SA Toolkit or via other measurement trapping methods.

Figure 14-16. The Inspection Navigation controls.

The left and right arrows jump to the previous or next feature for trapping, and the stop button stops all trapping operations.

Targets and Retros

When tracker measurements are taken, the planar and radial offsets are stored with the measured points. Since laser trackers aren't aware of the target or tooling being used, you need to keep the interface informed of which tooling you're using, so that the proper offsets can be assigned to measurements--and so those offsets can be accounted for when analyzing to surfaces or fitting geometry. The Targets/Retros button opens the ***Reflectors and Targets*** dialog (Figure 14-17) and provides the capability for defining new targets.

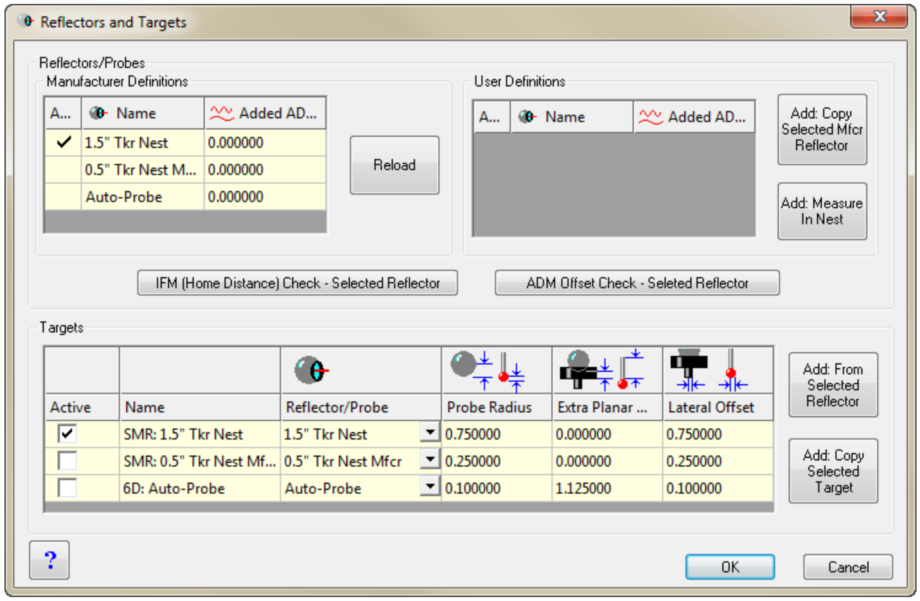


Figure 14-17. The Reflectors and Targets dialog.

This dialog is divided into three main areas: Manufacturer Definitions, User Definitions, and Targets.

Manufacturer Definitions

Every laser tracker controller has a set of defined reflectors that the instrument supports. For example, trackers commonly support 1.5", 0.875", and 0.5" SMRs, in addition to other peripheral devices. A tracker needs to know which defined reflector is being used for several reasons--one of which is that it tells the tracker which nest to return to when homing.

The list of manufacturer definitions is populated automatically when you open the dialog. In certain cases, the list may need to be refreshed. Clicking the Reload button will poll the controller and retrieve an updated list of manufacturer-defined reflectors. Note that this list may also include 6-DOF devices, mirrors, or other peripherals. Note that this list cannot be modified, and the selected reflector for the active target definition (see below) will be checked.

User Definitions

A user-defined reflector is essentially a manufacturer-defined reflector that has a custom measured ADM offset and/or a custom home position. ADM offset correction is applied when the difference in beam travel length when comparing the red laser to the ADM signal (due to refraction/dispersion) is significant. This is usually the case when the reflector has a clear windowed housing to protect the mirror, or when the reflective surface of the SMR is not the front face (the beam must travel through some material other than air before being reflected). ADM offsets apply a small correction to the measured distance value to account for the different behavior of ADM-wavelength signals.

There are two methods to create a user-defined reflector: Add: Copy Selected Mfcr Reflector or Add: Measure In Nest.

- **Add: Copy Selected Mfcr Reflector.** Creates a user-defined reflector using the same home position as the selected manufacturer reflector when the button is clicked.
- **Add: Measure In Nest.** Creates a user-defined reflector using a custom home position (nest).

Note: This option is not usable with FARO trackers.

To Create a User-Defined Reflector with a Custom Home Position (Non-FARO trackers only):

1. Lock the beam onto your custom reflector and place it into the custom nest.
2. From the ***Targets and Retros*** dialog, click the Add: Measure In Nest button.
3. When prompted, click Yes to measure.
4. A new user-defined reflector will be added to the table.

To Create a User-Defined Reflector Using an Existing Home Position:

1. Select the manufacturer-defined reflector whose home position you'd like to use from the ***Targets and Retros*** dialog.
2. Place your reflector in the appropriate nest, then click Add: Copy Selected Mfcr Reflector.
3. When prompted, click Yes to measure.
4. A new user-defined reflector will be added to the table.

To Delete a User-Defined Reflector:

1. Select the user-defined reflector.
2. Press the **`Delete`** key.

To Rename a User-Defined Reflector:

- Double-click the *Name* field for the user-defined reflector, and type in a new name, then press **`Enter`**.

To Measure an ADM Offset for a Custom Reflector:

1. Select the desired user-defined reflector in the ***Reflectors and Targets*** dialog and home on the reflector.
2. Click the ADM Offset Check - Selected Reflector button. The ***ADM Check/Cal*** dialog will appear (Figure 14-18).

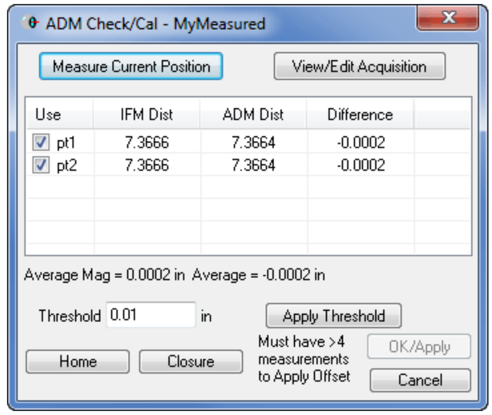


Figure 14-18. Measuring ADM offset.

3. If desired, set the measurement parameters for the ADM check by clicking on the View/Edit Acquisition button. These parameters are similar to the Discrete Point Measurement acquisition settings (see below).
4. Place the SMR in several positions (at least 5) around the desired measurement volume, and click the Measure Current Position button in each. The tracker will take an IFM and ADM measurement, recording the difference for each observation.
5. Applying a threshold (by setting the *Threshold* setting and clicking the Apply Threshold button) will highlight any observations in which the difference between the IFM and ADM measurements exceeds the threshold.
6. To ignore any specific measurement for the ADM offset calculation, uncheck its **Use** checkbox. The Home button will send the tracker home, and the Closure button will measure closure on the tracker.
7. When you feel you have taken sufficient measurements, click OK/Apply to apply the calculated ADM offset to the reflector, or Cancel to close the window and discard changes.

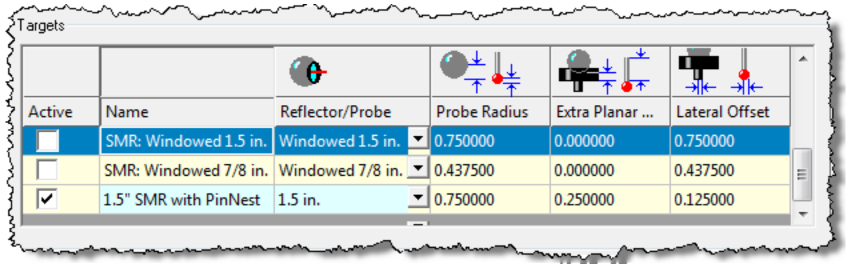
Targets

The targets section lists all of the defined target configurations for the tracker. Target configurations consist of a selected reflector and any combination of measurement tooling. This tooling, along with the reflector, determines the total offset of your measurements.

For instance, a simple 1.5″ SMR is a target configuration, as is a 7/8″ SMR. A 1.5″ SMR with pin nest is also a target configuration, as is a 6mm retro-probe or point probe. Each of these different configurations has a specified planar offset and lateral, or radial, offset.

After adding a target configuration, it will appear as a row in the *Targets* table at the bottom of the ***Reflectors and Targets*** dialog (Figure 14-19).

Figure 14-19. The new target definition is in the Targets table.



Targets

Active	Name	Reflector/Probe	Probe Radius	Extra Planar ...	Lateral Offset
☐	SMR: Windowed 1.5 in.	Windowed 1.5 in.	0.750000	0.000000	0.750000
☐	SMR: Windowed 7/8 in.	Windowed 7/8 in.	0.437500	0.000000	0.437500
☑	1.5" SMR with PinNest	1.5 in.	0.750000	0.250000	0.125000

This table has a number of columns that describe the attributes of the target:

- **Active.** The checked target is currently active, and its offsets will be applied to any measurements.
- **Name.** The custom name for the target configuration.
- **Reflector/Probe.** The manufacturer- or user-defined reflector used by the target configuration.
- **Probe Radius.** The radius of the spherical reflector or probe.
- **Extra Planar Offset.** An additional offset that is applied in addition to the probe radius when planar offsets are used (such as fitting a plane). The total primary (planar) offset is the sum of the probe radius and extra planar offset. For example, when using a 1.5" SMR with standard pin nest, the total offset is 0.75" (Probe Radius) + 0.25" (Extra Planar Offset) = 1.0".
- **Lateral Offset.** The lateral (radial) offset to use when fitting circles and ellipses for features such as holes and pins. When using a purely spherical probe, the lateral offset is equal to the probe radius. For an edge nest, it's zero--and for a pin nest, it's equivalent to the shank radius.

To Define a New Target Configuration:

1. In the ***Reflectors and Targets*** dialog, select the desired manufacturer-defined or user-defined reflector from one of the top two tables, as appropriate.
2. Click the Add: From Selected Reflector button. The ***Add Targets*** dialog will appear (Figure 14-20).

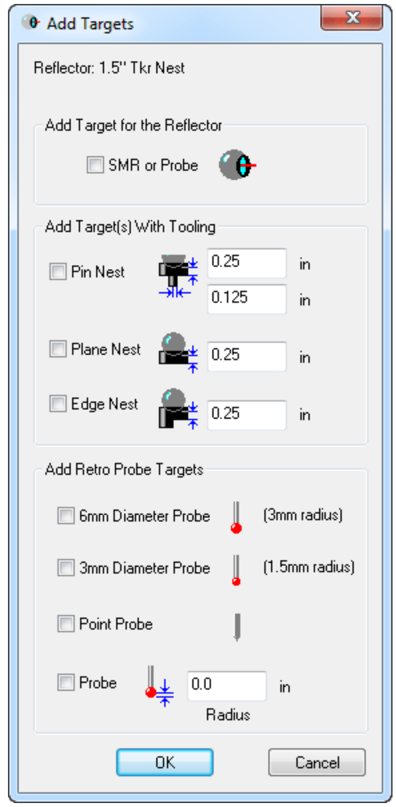


Figure 14-20. Adding pre-defined targets.

3. From the options, pick one or more targets to define, then click OK. (For a definition of these options, see below).
4. One or more new target definitions will be added to the *Targets* table, depending on the selected items.

When defining a new target, the ***Add Targets*** dialog provides several shortcuts for common tooling configurations.

- **SMR or Probe.** Adds a separate target configuration entry for the individual probe/reflector only.
- **Pin Nest.** Adds a target configuration for the selected reflector with a pin nest (you define the shank radius and planar offset.).
- **Plane Nest.** Adds a target configuration for the selected reflector with a plane nest (you define the planar offset).
- **Edge Nest.** Adds a target configuration for the selected reflector with an edge nest (you define the offset).
- **6mm Diameter Probe.** Adds a target configuration assuming the use of a retro probe with a 6mm diameter probe.
- **3mm Diameter Probe.** Same as 6mm Diameter Probe, except using a 3mm probe instead of a 6mm probe.
- **Point Probe.** Same as above, except assuming a point probe (no offsets).

- **Probe.** Allows you to enter a custom retro probe radius.

In addition to selecting the predefined options, you can also manually modify the *Extra Planar Offset* and *Lateral Offset* columns to use any desired numeric value. You may even see negative values. As long as you remember that the total planar offset it the sum of the probe radius and extra planar offset, the numbers should always make sense. For example, if 1.5" SMR is the selected reflector for use with a retro probe with a point probe, then the probe radius will be 0.75" and the extra planar offset will be -0.75".

To Rename a Target Configuration:

1. Double-click the *Name* field for the desired target in the *Targets* table of the ***Reflectors and Targets*** dialog.
2. Type a new custom name, then press **`Enter`**.

To Delete a Target Configuration:

1. Select the desired target to delete in the *Targets* table.
2. Press the **`Delete`** key and confirm the deletion.

To Make an Editable Copy of a Target Configuration:

1. In the *Targets* table, select the target configuration to copy.
2. Click the Add: Copy Selected Target button.
3. A copy of the selected target configuration will be added to the table.

To Make a Target Configuration the Active Target:

Tip: A faster way to activate a different target is to use the quick-select on the main interface.

- In the *Targets* table, check the **Active** checkbox next to the target configuration you'd like to activate.

Measurement Profiles

Measurement profiles are defined parameters for acquiring data (taking measurements), operating on that data, and then "doing something" with that data. For example, the most common and simplest measurement profile is one in which clicking the Measure button takes a single measured point. But there are many more capabilities, such as stable point or spatial scan acquisition modes, as well as many operations that can be performed on data (such as fitting geometry, creating cloud points, etc.). Measurement profiles are essentially custom presets. By defining your own custom presets for different tasks, you can easily switch from one profile to another and progress through pre-defined measurement profiles without taking time on the shop floor to adjust settings. This is most advantageous when performing repeated measurements of parts. Clicking the Measure button initiates a measurement profile, and it concludes when the profile is completed.

To edit or define new measurement profiles, click on the View Selected Meas Profile button. This will open the ***Meas Profile Parameters*** window (Figure 14-21).

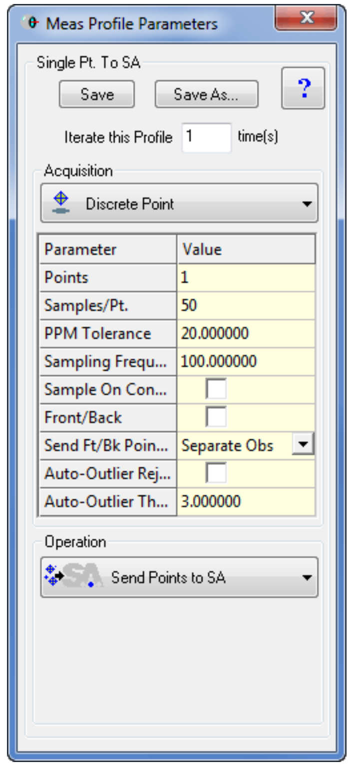


Figure 14-21. The Measurement Profile parameters.

If, after modifying the measurement profile parameters, you decide that you'd like to revert to the settings saved to the profile on disk, click the Refresh Popped Profile Parameters to Saved State button. At the top of the dialog is the name of the profile being viewed. In the screenshot above, it's *Single Pt. to SA*. There is a Save button to save the current settings and overwrite the existing profile, and a Save As button to save the settings as a new profile.

- **Iterate this Profile X times.** This setting will repeat the complete profile a given number of times.

To illustrate the profile iteration setting, consider an example in which you wish to measure a bolt hole pattern with 24 holes. Suppose you set a profile to measure points spaced by 0.1" around a bolt hole, then fit a circle and create a point at the center of that circle. Since you'd want to repeat that process for the remaining 23 holes, you'd set the interface to iterate the profile 24 times and measure the entire bolt hole pattern--all requiring you to press the measure button only once.

There are two major sections in the ***Measurement Profile Parameters*** dialog: *Acquistion* and *Operation*. The *Acquisition* section defines how measurement data is acquired, and the *Operation* section defines what is done with that measurement data once it's obtained. Details for these two sections are provided next.

Acquisition Modes

The acquisition section, as its name implies, determines how measurements are acquired by the tracker. You can measure a single point on command, measure points spaced by time or a distance, measure points on specified cross sections, and much more. The dropdown (Figure 14-22) allows you to specify the acquisition mode to use for the current measurement profile. When you select a mode, the parameters unique to that mode appear, which you can adjust as desired.

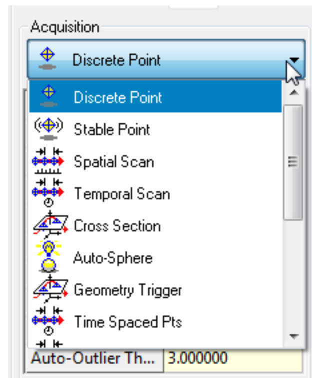


Figure 14-22. Choosing an acquisition mode.

Each of the default primary acquisition modes is described below. For each measurement mode, only the parameters not already discussed will be covered.

Discrete Point (Fast/Standard/Precise Point on Leica trackers)

This mode acquires a single point when the Measure button is clicked. (For Leica trackers, Fast/Standard/Precise controls the length of time that samples are taken, since the sampling frequency is not configurable).

- **Points.** Specify the number of points you would like to measure. Leave the value at "0" to continue repeating measurements.

Note: The number of samples cannot be directly configured on Leica trackers.

- **Samples/Pt.** The number of samples to average together in calculating a single discrete point. This is particularly useful for noisy environments, or cases in which vibration is encountered. This setting proportionately affects the time required for an individual measurement.
- **PPM Tolerance.** Sets the allowable RMS error for measurements based on the distance of the measurement from the instrument. For example, at the default value of 20 PPM, a measurement 7.21m from the instrument would have an allowable RMS error of 7.21m × (20/1,000,000) = 144.2 microns. If after collecting all samples for the point this allowable RMS is smaller than the actual RMS, a ***Tolerance Exceeded*** dialog will appear.

Note: Sampling frequency on Leica trackers is constant and cannot be set explicitly.

- **Sampling Frequency.** The desired rate at which to take samples, in Hz. If this rate is not achievable by the instrument, the closest value will be used. This rate inversely affects the time required to take a measurement. A higher frequency will result in faster measurement. Note that Sampling Frequency, in combination with the Samples/Pt setting, determines the duration of measurement. For example, a discrete point measurement at 50 samples/pt and 100 Hz will result in a measurement time of 0.5 secs.

- **Sample on Controller.** Specifies whether the samples are acquired on the tracker controller, or by the instrument interface. Acquiring samples on the controller will often permit a higher sampling frequency, but since SA does not have access to the individual sample data it cannot provide statistical information about the samples unless the controller makes it available.
- **Front/Back.** If checked, each measurement will consist of both front and back face measurements.
- **Send Ft/Bk Points As.** When measuring front face and back face, the individual face measurements can be sent as *Separate Obs*, in which case they become two separate observations under the target, or as *Single Avg Pt*, in which the front-face and back-face measurements are averaged together for a single observation.
- **Auto-Outlier Rejection.** If enabled, SA will perform outlier rejection on the samples (see *Auto-Outlier Threshold*, next).
- **Auto-Outlier Threshold.** The sigma value outside of which samples should be discarded. When measurement samples are collected, SA performs a statistical analysis and determines the standard deviation of the measurements. This value, specified in sigmas, indicates which samples should be thrown away and excluded from the samples used to calculate the average point coordinate. For example, a value of 3 indicates that all samples outside of 3 sigma should be tossed away.

Stable Point (Fast/Standard/Precise Stable Point on Leica trackers)

Stable Point acquisition mode waits for the reflector to move and then become stable again. Once stable for a specified period of time (2 seconds by default), a discrete measurement is taken. This cycle is repeated for as many points are specified in the profile. This acquisition mode is especially useful for single-operator environments, because the measure button only needs to be clicked once for the entire profile. (For Leica trackers, Fast/Standard/Precise controls the length of time that samples are taken, since the sampling frequency is not configurable).

- **Stable Space.** For stable point measurements, the diameter of a sphere that the probe center must remain inside to be considered stable for purposes of triggering measurement.

- **Stable Time.** For stable point measurements, the amount of time the probe center must remain inside the stable space to be considered stable for purposes of triggering measurement.
- **Measure When Beam Found.** This option is no longer used.

Spatial Scan

Spatial Scan measures points spaced by a specified distance. Each time the reflector moves the prescribed distance from the previous measurement, another measurement is triggered. Be aware that this distance will not be perfect, as hardware sampling rate, reflector speed, computer clock speed, network latency, and other factors introduce lag between when the prescribed increment is detected by the instrument interface and when a measurement is taken.

- **Start Trigger.** Indicates what trigger is to be used to start the measurement profile. After the Measure button is clicked, *Button/delay* delays the prescribed number of seconds (in *Delay Before Measuring* below) before measurement starts. *Button/stable* waits for the probe to move, then become stable again before triggering measurement.
- **Delay Before Measuring.** Used when *Start Trigger* is set to *Button/delay*, the number of seconds to delay before initiating the measurement profile.
- **Pause @ BeamBreak.** If active, the measurement profile will be paused if the beam is broken, and resumed when the beam is locked again.
- **Stop Trigger.** Indicates what event is used to signify the completion of the measurement profile. *# Points* stops measuring when the specified number of points is reached. *Beam break* stops measuring when the beam is broken. *Loop* stops measuring when the target loops back to its original measuring position, completing a closed loop.
- **Increment.** For spatial scanning modes, indicates the desired spacing between adjacent measurements.

Temporal Scan

Measurements are triggered at a specified time interval. Due to sampling frequency, latency, and other factors, there will be small discrepancies between actual and requested spacing.

Cross Section

Equally-spaced cross sections can be established along the X, Y, or Z axis of the working frame at a specified distance. These sections trigger a measurement as the probe crosses the plane.

- **Sampling Increment.** Indicates the spacing between adjacent samples.
- **Working Frame Axis.** The axis along which the planes should be located.
- **Axis Cross Value.** The position along the specified axis to locate the cross section.
- **Use Closest Point.** Indicates whether the closest point to the section should be used (raw measurement), or whether a point directly on the cross section should be used (interpolated between the closest point on each side of the plane).

Auto-Sphere

Spatially-spaced measurements will be taken, and when data is seen to be spherically distributed, SA will automatically fit a sphere to that data.

- **Fit Threshold.** A set of measured points must fit a sphere to better than this value to be considered a valid sphere.
- **Lock Radius.** If enabled, the resulting spheres must have the specified radius.
- **Lock Rad Value.** Specifies the radius to use when locking.

Geometry Trigger

Planes, cylinders, and spheres can be configured to act as measurement triggers such that when a probe crosses the specified geometry, a measured point is recorded.

- **Geometry Triggers.** Defines the geometry used to trigger measurement in *Geometry Trigger* acquisition. To define triggers, click the *None Set* box.

The ***Make Geometry Crossing Triggers*** dialog will appear when defining geometry triggers for this mode (Figure 14-23). There are three trigger types: planes, cylinders, and spheres. The Select button allows you to select existing geometry from the SA file to use as triggers. The Cartesian, Cylindric, and Concentric buttons provide options for using planes spaced along a Cartesian axis, Cylindrically rotated about an axis, or to create concentric cylinders. When *Draw In SA* is selected, the geometry will be physically created in the SA file.

There are two sampling methods for the geometry: *Closest Pt.* and *Interpolate*. Closest Pt. uses the closest sample to the geometry as the resulting measured point, whereas Interpolate creates a point directly on the geometry by interpolating between the closest samples on each side of the geometry.

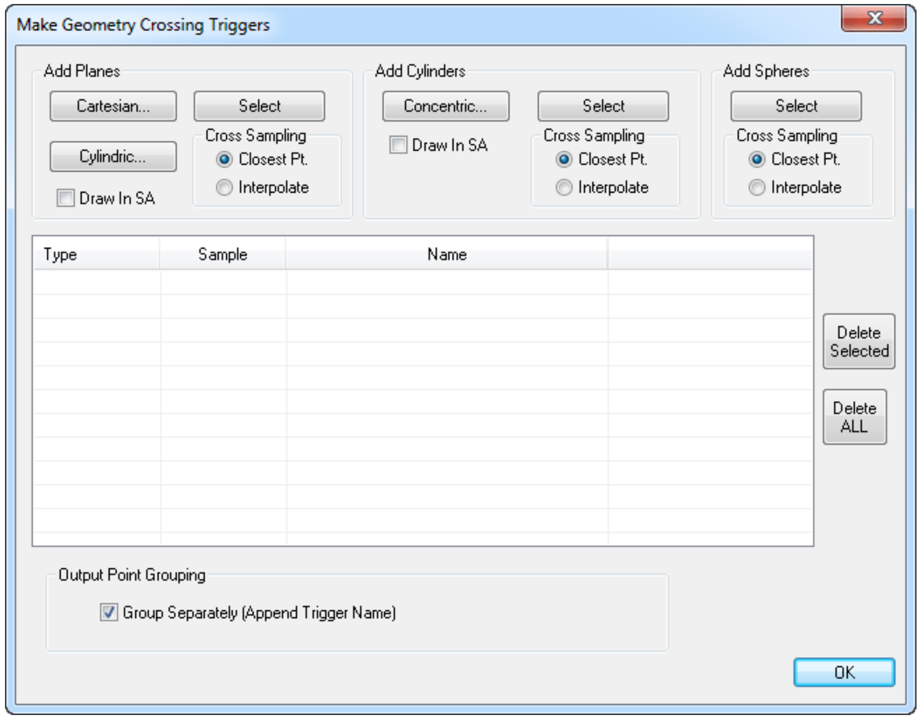


Figure 14-23. Defining geometry triggers in the tracker interface.

When *Group Separately (Append Trigger Name)* is selected, the measurements will be placed into separate point groups based on the specific geometry that triggered them.

Time-Spaced Points

Repeated discrete point measurements are taken spaced by a specified time. Note that this differs from Temporal Scan in that temporal scan has only a single sample per point and assumes that the target is in motion. Time-Spaced Points has a configurable number of samples and assumes that the target is stationary during the sampling process.

- **Time Between Points.** The time to wait between successive discrete measurements.

High Point

Warning: Offset compensation is automatically applied to the high point, and its planar offset is set to zero. Points are shifted toward the axis when using *From Axis* and in the negative axis direction when using *Along Axis*.

Measurement samples are taken, and upon completion of the measurement, only the highest point along a frame axis or the farthest point from a frame axis is recorded.

- **Working Frame Axis.** The axis (*X*, *Y*, or *Z*) to check for a high point, and whether the check should be performed along that axis or radially from that axis.

Hidden Point

Two points are measured (A & B), and a third (hidden) C point is constructed automatically based on the first two.

- **Measure Time (secs).** The time to measure each point.
- **Bottom Pt Offset.** The distance from the bottom (B) point to the hidden point.
- **Planar Offset.** The desired planar offset for the resulting hidden point.
- **Radial Offset.** The desired radial offset for the resulting hidden point.

Edge Points

Developed primarily for sheet metal work, this mode allows you to drag a probe along a flat surface, and when it begins to fall off the edge, will create a point on the edge of the surface.

- **Edge Plane.** Defines the plane that the probe will be slid along.

Operations

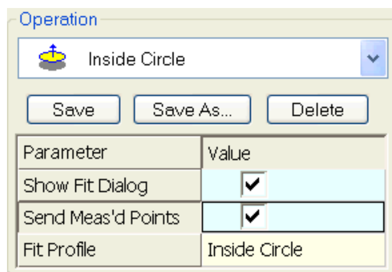


Figure 14-24. Specifying the tracker interface's operation.

The *operation* specifies (Figure 14-24) what to do with the measurements once they are obtained. Here a user can decide whether to send the points directly to SA or perform another operation. One example is using the "Inside Circle" operation: this will perform a circle fit and show the fit dialog prior to sending measurements to SA. The advantage of this type of operation is that real-time processing of measurements can take place. That is, you can fit geometry to measurements and identify potential problems right at the time of measurement.

The different basic operations are described below. For each operation, only the parameters not already discussed will be covered.

Send Points to SA

Measured points are sent on to SA to be created as normal measured points in the job file. There are no settings for this operation.

Send Updates to SA

SA is updated with the latest measured position and orientation of the tracker and probe, but no measurements are created in the job file. There are no settings for this operation.

Send Cloud Points to SA

Similar to "Send Points to SA", except that instead of creating standard measured points, SA creates cloud points. The major advantage of using this operation is seen when measuring a large number of points with a laser tracker (greater than ~20,000). For efficiency purposes, best practice when taking large volumes of data is to use cloud points, because the metadata associated with normal measured points consumes a large quantity of RAM.

- **Buffer Size.** The number of points to buffer in the interface before sending them over the network to SA. Only advanced users should modify this value, and only with good reason.

Poll Data

Note: The Poll Data operation is only useful for certain (usually older) trackers. You only need to use it if you find normal update rates to be too slow for your application.

Certain trackers take a relatively long time to return a measurement update when asked. For applications that operate best with data supplied as close to real-time as possible (such as Trans-Track), you may find it preferable to put the tracker into the "Poll Data" operation instead of "Watch Update", which may yield faster updates.

Make A Circle

A circle is automatically fit to the measured data.

- **Show Fit Dialog.** When enabled, the geometry fit dialog will be displayed at the completion of the measurement profile.
- **Send Meas'd Point.** When enabled, measured points will be sent to SA in addition to the resulting circle.
- **Fit Profile.** The geometry fit profile to use for fitting the circle. Click the box to modify the profile (see the chapter on Geometry Fitting). Right-click to use an existing profile.
- **Projection Plane.** The data can be automatically projected to a plane prior to the circle fit. This is particularly useful when fitting a circle to data that may not be measured along a cylinder's cross-section. For details, see "Projection Planes" on page 449.

Make A Plane

A plane is automatically fit to the measured data.

- **Patch Shift.** Shifts the resulting plane an additional amount after compensating for probe offset.

Make A Sphere

A sphere is automatically fit to the measured data.

Make A Cylinder

A cylinder is automatically fit to the measured data.

Make A Line

A line is automatically fit to the measured data.

Auto Proximity

Used for certain auto proximity automatic measurement modes. Sends data to a command in SA to be interpreted.

Single Point Circles

Note: This operation reduces accuracy when burrs or chamfers exist on the hole edge, or if the hole is out of round.

This operation is effective for measuring a hole with a single point. With a projection plane defined, place the probe into a hole and take a single measurement, and the resulting circle will be created based on the probe diameter and the degree to which the reflector sunk down into the hole.

- **Lock Radius.** Forces the circle to be locked to a specified radius.
- **Locked Radius.** The radius to use when *Lock Radius* is enabled.
- **Intersection Plane.** The plane defining the top surface of the hole can be selected from an existing plane by left-clicking this option, or measured as part of the operation by right-clicking this option. For details, see "Projection Planes" on page 449.

Patch Points

Projects all measurements to a specified plane.

3-Plane Point

Creates a point at the intersection of 3 measured planes (such as the corner of a box).

- **Plane1/Plane2/Plane3.** The three planes defining the corner to measure. Existing planes can be used by left-clicking this option, or measured as part of the operation by right-clicking. For details, see "Projection Planes" on page 449.

Send Frame Updates to SA

Updates SA on the position and orientation of a frame without actually creating any frames in the SA job. Offset frames and offset points can be sent as well. Sending frame updates requires a 6-DOF device.

- **Frames To Send.** You can send a *Raw Frame* (the frame from the device), an *Offset Frame* (a frame offset from the raw frame), or an *Offset Point* (the origin of an offset frame).
- **Offset Frame.** Click this parameter to define an offset frame to send. For more information, see "Offset Frames" on page 449.

Send Frames to SA

Constructs frames measured from a 6-DOF device in the SA job.

- **Frames To Send.** In addition to *Raw Frame*, *Offset Frame*, and *Offset Point* (described above), you can also send *Both Offsets* (offset frame and point), *Both Frames* (raw and offset frames), *Pt. & Raw Frame* (offset point and raw frame), and *All* (raw and offset frames & offset point).

Projection Planes

Certain operations (such as *Make a Circle* & *Make a Plane*) allow measurement data to be projected to a plane prior to the operation. Projection planes handle the settings for this.

To use an existing plane as the projection plane for a measurement operation:

1. Click the box next to *Projection Plane* in the operation's parameters.
2. Select an existing plane in the SA file.

The projection plane may also be measured as part of the operation.

To Measure The Projection Plane As Part Of A Measurement Profile:

1. Right-click the box next to *Projection Plane* in the operation's parameters. The ***Plane Options*** dialog will appear.
2. Pick the appropriate settings (see below), then click OK. The plane will be measured as part of the measurement operation.

- **Regenerate in SA.** The measured fit plane will be created in the SA file.
- **Remove from Profile.** If this option is checked and you click OK, the existing defined projection plane will be cleared.
- **Fit To Points.** The specified number of points will be gathered as part of measurement to define the plane.

Offset Frames

Offset frames allow you to measure a raw frame with a 6-DOF device, then send an offset frame or point that is spatially offset from the raw frame. The offset frame maintains its relative position and orientation to the raw measured frame.

To measure offset frames, you must first measure and construct the offset frame itself. This frame will be transformed as part of the operation. Then, you must also measure the raw frame, which defines the delta from the reference to the offset frame. As the raw frame is updated and moved, the offset frame will be updated as well.

To Measure Offset Frames:

1. In the ***Meas Profile Parameters*** dialog, select a measurement operation supporting offset frames (*Send Frame Updates To SA* or *Send Frames To SA*).
2. Select the desired feature to send from the *Frames To Send* parameter.
3. Measure the offset frame.

4. Click the box next to *Offset Frame* in the measurement operation's parameters. The ***Offset Frame*** dialog will appear (Figure 14-25).

5. Click the 1) Select Offset Frame From SA button and select the frame to become the offset frame.

6. Click the 2) Measure Reference Frame To Offset button to measure the reference frame. The target must be a 6-DOF target.

7. If creating an offset point and you'd like the offset point to have a defined probe radius, enter it in the *3) (Optional) Enter probe radius for offset point* field.

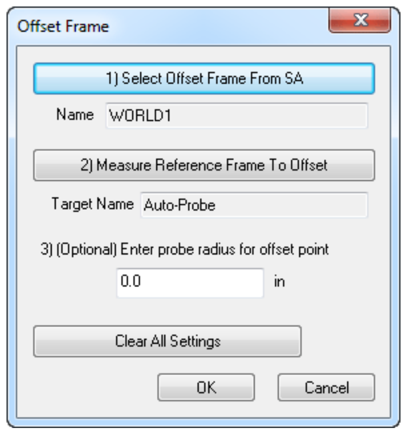


Figure 14-25. Defining an offset frame.

8. Begin measuring 6-DOF frames, and the offset frame will be updated automatically.

Managing Measurement Profiles

Measurement profiles can be managed (Figure 14-26) by clicking the Manage Profiles button in the Laser Tracker Interface. Here you can add and delete measurement profiles. Measurement profiles can also be imported and exported, allowing them to be transferred to another computer. This also serves as a great way to back up custom profiles.

To Delete A Custom Profile:

- Click a profile to select it, then click the Delete button.

To Create A New Profile Using Existing Acquisition/Operations:

1. Click the Add button, then enter a new profile name.

2. In the appropriate dropdowns, select the acquisition and operation to associate with the profile.

To Assign A Quick-Select Button To A Profile:

- Right-click a profile and select the desired button.

To Export Profiles To A File:

- Click the Export button, then select the desired filename and path for the .MSP file.

To Import Profiles From A File:

- Click the Import button, then select the .MSP file to import.

To Restore Measurement Profiles To Their Defaults:

- Click the Restore Defaults button.

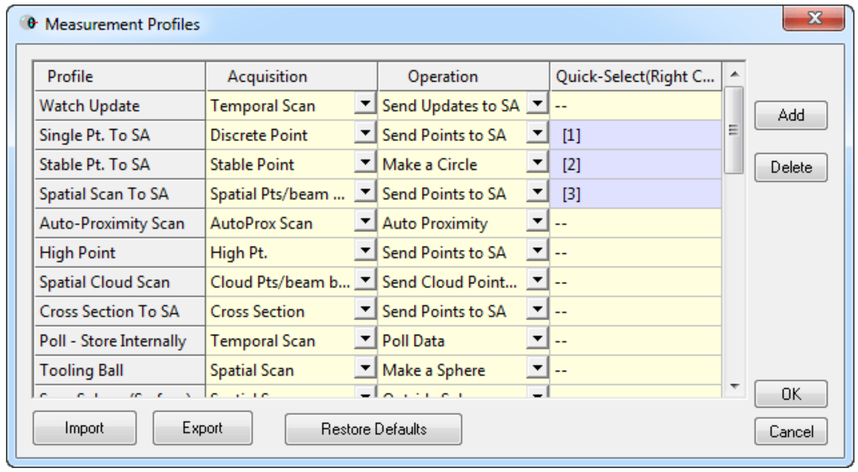


Figure 14-26. Managing measurement profiles.

Settings

General Settings

Access to Search Parameters, Action Time Out settings and Tracker-specific Settings can be reached by clicking the General Settings button or by selecting **Settings>Tracker>General Settings** from the interface menu. This opens the ***General Tracker Settings*** dialog (Figure 14-27).

ADM Search Parameters

- **Seed Distance.** Specifies the estimated distance to use when initiating an ADM search. This is generally updated automatically as measurements are taken and usually does not need to be manually modified unless there is large range variation relative to the last measured point and an ADM search to find a target is not successful.
- **Radius.** The radius from the starting pointing position (at a distance equal to the seed distance) at which to start searching. A larger value will initiate an ADM search over a larger cone.
- **Timeout.** If a target has not been found after this time has elapsed, an ADM search will be cancelled.

Action Time Out

- **Get Status.** After requesting the tracker's status, if a response hasn't been received after this time, the request will be cancelled.

- **Get System Info.** After requesting the tracker's system information, if a response hasn't been received after this time, the request will be cancelled.
- **Read Weather.** If a weather information request hasn't been received by this time, the request is cancelled.
- **Set Weather.** If a weather set operation isn't performed by this time, the operation is cancelled.

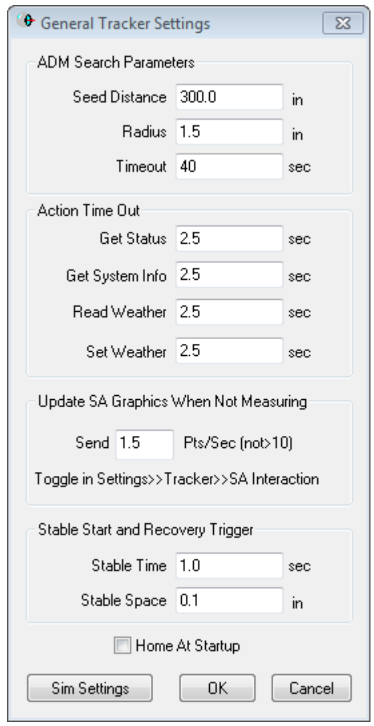


Figure 14-27. General tracker settings.

- **Send x Pts/Sec.** Indicates the frequency at which updates should be automatically sent from the tracker to the SA interface, even when not in a measurement.
- **Stable Time.** Sets the stable time (how long must the probe be stable before starting to measure?) used for stable measurement triggering (wait for move, wait for stable). This does *not* apply for Stable Point acquisition mode, which uses its own setting.
- **Stable Space.** Sets the size of a sphere within which the probe must remain to be considered stable for stable measurement triggering (not used for Stable Point acquisition mode, which has its own setting).
- **Home At Startup.** If disabled, the tracker will not go home when the interface is started.

Note: Some trackers may need to home at times, even if this option is disabled.

- **Sim Settings.** The label on this button will change depending on the specific instrument connected to the interface. Click it to access parameters specific to the connected hardware.

Units

Click the Units button to set the active units for the interface(Figure 14-28). When data is sent from the interface to SA, it is automatically converted into the active job units.

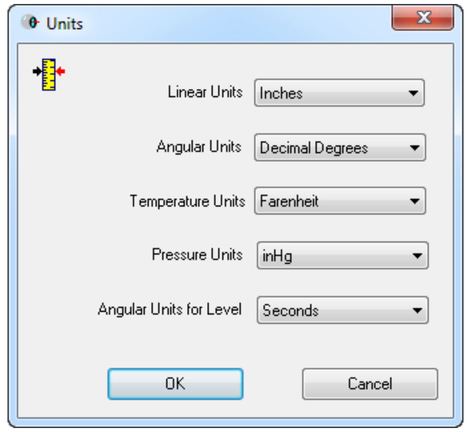


Figure 14-28. The tracker interface's Units dialog.

Function Keys

Note: The instrument interface will not see function key presses unless it has focus, so be sure to click on the interface prior to attempting to use a function key.

Function keys can be assigned for different operations in the tracker interface (Figure 14-29). The operations that can be assigned to a function key include:

- Measure/Resume
- Watch Data
- Increment Group
- Change Face
- Find Target
- Remote Home
- Home
- Check Closure
- Motors Off
- Motors On
- Decrement Group
- Increment Target
- Decrement Target
- End Acquisition
- Iterate Quick-Select Mode
- Navigation Next
- Navigation Previous
- Navigation Stop
- Remove Last Point

To modify the function keys assigned in the interface, select

Settings>Fcn Keys from the interface menu. The ***Function Keys*** dialog will appear (Figure 14-29).

To Change An Assigned Function Key:

- Select a new operation from the dropdown for the specified function key, then click OK.

To Restore Default Function Key Assignments:

- Click the Restore Defaults button.

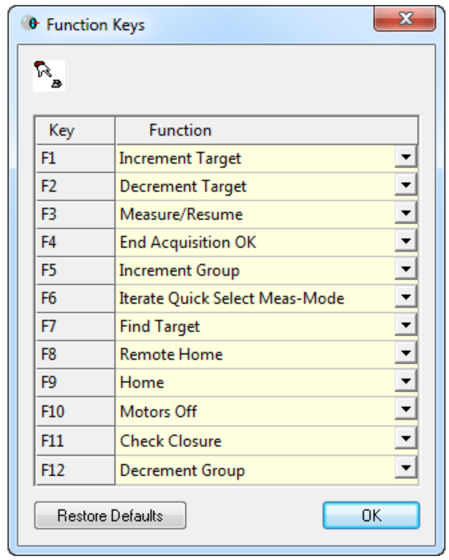


Figure 14-29. Defining function keys in the interface.

User Interface Settings

The verbosity of the tracker interface when displaying certain error message can be controlled using the ***User Interface Settings*** dialog (Figure 14-30), which is accessed by selecting **Settings>UI** from the interface menu.

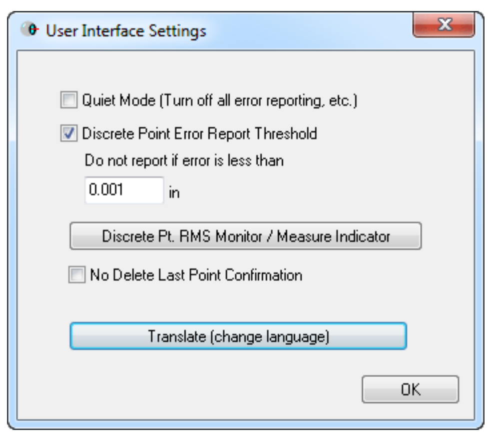


Figure 14-30. The User Interface settings dialog.

- **Quiet Mode (Turn off all error reporting, etc.).** When enabled, error messages such as RMS tolerance violations will be suppressed.
- **Discrete Point Error Report Threshold.** This setting controls the minimum limit for reporting an error indicating reflector

movement during measurement. This error is normally triggered by the *PPM Tolerance* setting in the measurement profile's acquisition mode, and is based on distance from the tracker. However, measurements close to the tracker may trigger this warning even though the data is valid. This lower limit suppresses errors below the specified value.

- **Discrete Pt RMS Monitor/Measure Indicator.** Turns on the RMS Monitor (see "RMS Monitor" on page 455).
- **No Delete Last Point Confirmation.** When enabled, you will not be prompted to confirm deletion of points via the *Delete Last Point* button.
- **Translate (change language).** The tracker interface can be translated using a custom language, just like the SA interface. For more information, (see "Language Translation" on page 150).

RMS Monitor

The ***RMS Monitor*** (see "User Interface Settings" on page 454) is a window that can remain open and will display atop the SA interface when docked. It reports the RMS error for measurements when that measurement involves averaging of samples (such as discrete point or stable point acquisition modes).

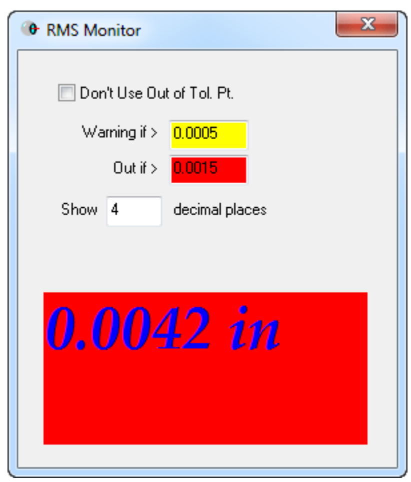


Figure 14-31. The RMS monitor indicating an out of tolerance measurement.

Two tolerances can be set: warning (yellow) and out (red). The RMS error of the previous measurement is displayed, along with the proper color code, to notify you that the previous measurement may have been noisy. The tolerances can be set to your preferred values.

- **Don't Use Out of Tol Pt.** When enabled, a point with an out-of-tolerance RMS value will not be recorded.
- **Show x decimal places.** Limits the displayed RMS value to the specified number of decimal places.

Beam Autoreset Mode

The behavior when the tracker loses a beam lock can be controlled by adjusting the *Beam Autoreset Mode*. There are three modes to choose from:

- **No Action.** When a beam lock is lost, the tracker will just stop and not attempt reset. This is generally only used with older legacy trackers.
- **Auto IFM.** When a beam lock is lost, the tracker will home and wait to lock onto the reflector in the home position using IFM distance measurement (not applicable to ADM-only trackers).
- **Auto ADM.** When a beam lock is lost, the tracker will stop head motion and wait for a beam lock, reacquiring using an ADM distance measurement.

To Change the Beam Autoreset Mode:

- From the interface menu, select **Settings>Tracker>Beam AutoReset>...**, or
- Click the Set Beam Break Behavior button in the interface to toggle between the three modes.

Tracker Pointing Behavior

When pointing a tracker at an existing point, the instrument can behave in one of two ways:

- The instrument can point to the point but not attempt an ADM reset, or
- The instrument can point and perform an ADM search to attempt to lock onto a nearby target.

To Change Tracker Pointing Behavior:

- From the interface menu, select **Settings>Tracker>SA Interaction>Point Laser>Point Only** or **Point and Attempt Reset**.

Updating Graphics While Not Measuring

Tip: This option persists primarily for older legacy trackers. In general, it's best to leave this option on unless you have a good reason to turn it off.

The probe position, current shot line, and instrument graphics are always updated while measuring, however if you are not in the midst of measurement, you can choose whether or not you'd like the graphical view to automatically update when new data is received from the tracker.

This allows you to view updated Watch Windows without requiring you to be in any sort of measurement mode, for instance.

To Update Graphics While Not Measuring:

- From the interface menu, enable **Settings>Tracker>SA**

Interaction>Update Graphics While Not Measuring. Upon doing so, SA will begin receiving updates from the connected tracker.

Running "Watch Update" Automatically

Tip: This setting is primarily for older legacy trackers. It is recommended to leave this setting off and leave "Update Graphics While Not Measuring" on unless you have a reason to do otherwise.

Older versions of SA required the tracker to be in an active measurement mode (such as "Watch Update") in order to receive updates to watch windows. This setting will automatically set the tracker to the "Watch Update" measurement profile (and initiate measurement) whenever a watch window is opened.

To Toggle Automatic "Watch Update" Mode:

- In the interface menu, select **Settings > Tracker > SA Interaction > Run "Watch Update" When Watch Window Opened In SA**.

Measurement Requests

Certain measurement commands, such as *Auto-Correspond With Proximity Trigger* and *Measure Nominal Points*, send measurement requests to the instrument interface. When these requests are received, the interface needs to know how to handle the request. These settings are controlled in one of two windows: *Single Point Parameter Measurement Requests* and *Auto-Proximity Profile Measurement Requests*.

Single Point Parameters

For commands that request discrete measurements (such as **Instrument > Locate > Measure Nominal Points**), the measurement settings are controlled by the single point parameters. These parameters are identical to those of the Discrete Point acquisition mode (see "Acquisition Modes" on page 441), but apply when measurements are initiated from SA itself.

To Access the Single Point Parameters:

- In the interface menu, select **Settings > Tracker > SA Interaction > Measurement Requests > Single Point Parameters**.

Auto-Proximity Profile

For commands that use auto-proximity modes, such as **Instrument > Automatic Measurement > Auto-Correspond With Proximity Trigger > Points**, the measurement parameters are controlled by the auto-proximity profile. These commands require some sort of temporal scanning acquisition mode. The specific measurement profile used for these commands can be modified.

To Change the Auto-Proximity Measurement Profile:

Note: Profiles not using a temporal scanning acquisition mode cannot be used.

- In the interface menu, select **Settings > Tracker > SA Interaction > Measurement Requests > Auto-Proximity Profile**. Select the desired profile from the list.

The Weather Station

The Weather Station Status indicator (the smaller of the two colored lights) gives an indication as to the current status of the weather data coming into the interface.

Laser tracker measurements must be compensated using weather data to account for factors such as atmospheric index of refraction. Without doing so, the tracker's measurements would not be as accurate. The possible status indicators and their meanings are presented below:

- **Green.** A weather station (whether internal or external) is connected and sending data.
- **Yellow.** The tracker interface is set for manual input. This setting compensates measurements using manually entered weather data. The yellow color serves as a reminder that this data may need to be updated by hand.
- **Red.** The tracker's controller does not have valid weather data from an internal or external weather station (whichever is selected). This most likely indicates that the interface is not receiving weather data at all. Measurements should not be trusted when the weather status is red.

You can indicate whether you'd like weather data to come from the tracker's internal weather station, an external weather station, or via manual input.

To Change the Source for Weather Data:

1. Double-click the weather station status indicator or choose **Devices>Weather>Set Up**. A ***Weather Source/Settings*** dialog will appear (Figure 14-32).

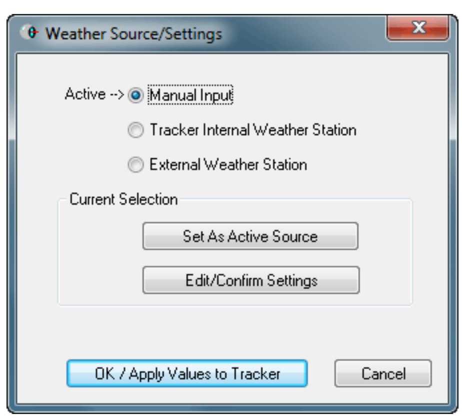


Figure 14-32. Setting the desired weather source.

2. Choose the desired source, then click the Set As Active Source button. Edit the settings for the current selection if necessary. Then click OK/Apply Values to Tracker. The options are listed below.

- **Manual Input.** Manually-entered weather data (Figure 14-33) will be sent to the tracker controller to be used for compensation. This weather data will continue to be used until updated manually again. When this option is selected and set as the active source, the ***Edit/Confirm Settings*** dialog will appear to allow you to set the weather values:

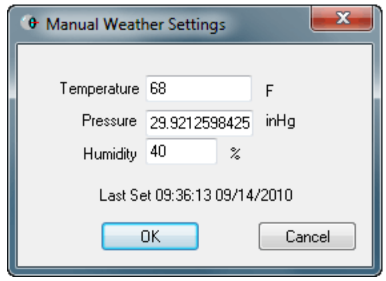


Figure 14-33. Entering manual weather data.

- **Tracker Internal Weather Station.** This setting will use the internal weather station integrated into the tracker. Typically, this setting is used when a weather station is either integrated into the tracker, or when a weather station connects directly to a tracker controller. This is the most common source for tracker weather data. When this option is selected, clicking the Edit/Confirm Settings button will allow you to view the weather station's values, but you cannot change them. After selecting this option, click the Set As Active Source button to apply the setting.
- **External Weather Station.** This setting allows you to use a separate, independent weather station device for weather data. After selecting this option, you must first click the Set As Active Source button. This will bring up the ***External Weather Source*** dialog (Figure 14-34).

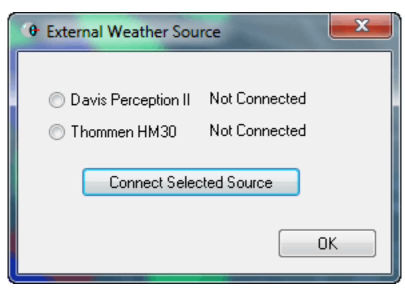


Figure 14-34. Specifying the external weather source.

In this dialog, pick the desired device, then click the Connect Selected Source button. If connected correctly, the external weather station's values will be fed to the tracker controller for compensation. As with the Internal Weather Station setting, the Edit/Confirm Settings button will allow you to view (but not edit) the weather station's values.

You can view the current weather data to monitor the change in temperature, pressure, and humidity since the window was first opened. This provides a warning if environmental conditions have changed enough to indicate potentially bad data. The ***Weather Values/Deltas*** dialog shows this information, and this window can be left open while working.

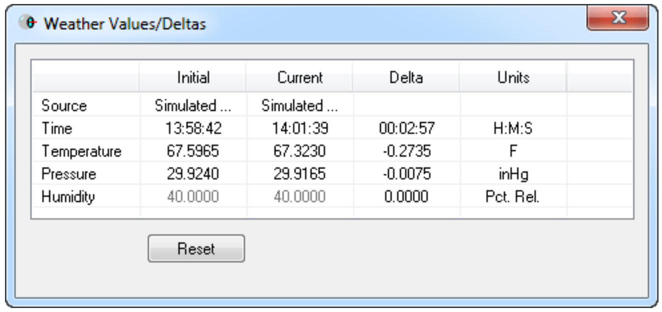


Figure 14-35. Viewing the initial and current weather data alongside the deltas.

To View Current Weather Information:

- In the interface menu, select **Devices > Weather > Current Values/ Deltas**.
- Click the Reset button to reset the initial values to the current observations.

Level Measurement

Note: Some trackers are not equipped with level measurement devices.

Different trackers have a wide variety of highly accurate levels, but all perform one function—to accurately read the gravity (level) direction at the instrument. This is often used for providing additional information for a fit, or constraining a network of instruments during alignment.

Note: An instrument's gravity frame will be associated with the instrument and will transform along with it.

When a level measurement is recorded, a coordinate frame is created on the instrument's *base*—the "origin" of the instrument. The Z-axis of this frame will be aligned along the measured gravity vector.

Trackers usually position their head in several positions and average multiple measurements in an attempt to limit the effect of noise and the weight of the tracker head on the level measurement.

Settings

Note: These settings vary from one instrument to another.

Several settings can be adjusted for level measurement (Figure 14-36) which affect how the level measurement is acquired.

To Adjust Level Measurement Settings:

- From the interface menu, select **Devices > Level > Set Up**.

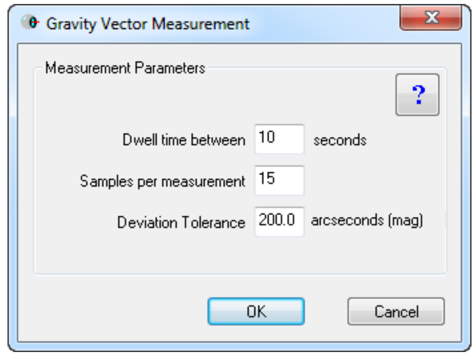


Figure 14-36. Modifying settings for level measurement.

- **Dwell time between.** Sets the amount of time the tracker waits after positioning the head before taking level samples. This gives time for the bubble level to stabilize before measurement.
- **Samples per measurement.** The number of samples to average when measuring gravity.
- **Deviation Tolerance.** Samples that exceed the specified tolerance (as a magnitude) are not included in the averaging operation.

Monitoring Level

Note: Some instrument OEMs have their own windows for monitoring level. SA will call these windows up in those cases.

Prior to measuring level, the instrument must be manually adjusted until the inclinometer is reading in-range for all azimuthal positions of the tracker head. The Level Monitor (Figure 14-37) permits viewing real-time output from the inclinometer to enable this adjustment.

- **Update Rate.** Sets the rate at which updates are displayed in the interface.
- **Show X Decimal Places.** Controls the number of decimal places displayed in the window.
- **Alarm.** When enabled, the computer will beep and a message will be written to the log file when the specified angular tolerances are exceeded. The alarm will not sound when this window is closed.
- **Set Baseline.** When clicked, the current values become the baseline or “reference” for establishing deltas for out-of-tolerance determination.
- **Display Level Values.** The display is updated with raw level readings.
- **Display Deltas (Threshold).** Shows angular deltas instead of absolute angular values. The threshold controls the limit that sets off the alarm.

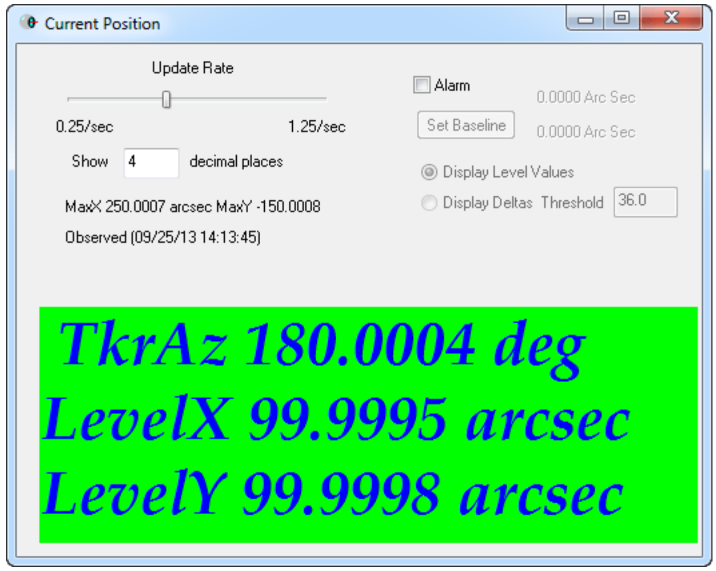


Figure 14-37. Monitoring a tracker's inclinometer.

To Monitor The Tracker's Level:

- From the interface menu, select **Devices > Level > Monitor**.

To Measure Level With A Tracker:

> **Warning:** Inclinometers are advanced bubble levels—they have angular limits beyond which the level is no longer accurate. Ensure that the inclinometer is within range by monitoring the level and manually leveling out the instrument prior to attempting to measure level from the interface.

> **Warning:** The tracker must be mounted to a stable, motionless base in order to acquire accurate level measurements.

1. Ensure the tracker is equipped with an inclinometer or other leveling device.
2. After setting up the instrument on a stable mount, monitor its level indication by selecting **Devices > Level > Monitor** from the interface menu.
3. Manually adjust the tracker (using body or tripod adjustments) until the monitor consistently reads in-range for all azimuthal positions of the tracker head.
4. Close the level monitor and initiate measurement by selecting **Devices > Level > Measure** from the menus.

Monitoring Part Temperature

Some trackers are equipped with part temperature sensors which allow you to compensate for thermal expansion or monitor the part temperature and take action if environmental conditions might generate invalid measurements.

The ***Part Temperature*** dialog (Figure 14-38) is used for this purpose, and can be kept on-screen if desired.

- **Update Rate.** Sets the rate at which updates are displayed in the interface.
- **Show X Decimal Places.** Controls the number of decimal places displayed in the window.
- **Alarm.** When enabled, the computer will beep and a message

will be written to the log file when the specified temperature thresholds are exceeded. The alarm will not sound when this window is closed.

- **Set Baseline.** When clicked, the current temperature reading become the baseline or "reference" for establishing deltas for out-of-tolerance determination.
- **Display Temperature Values.** The display is updated with raw temperature readings.
- **Display Deltas.** Shows the change in temperature from the baseline instead of the current temperature. The *Hi* and *Lo* thresholds control the limits that set off the alarm.

Warning: The thresholds are temperature values, not delta values.

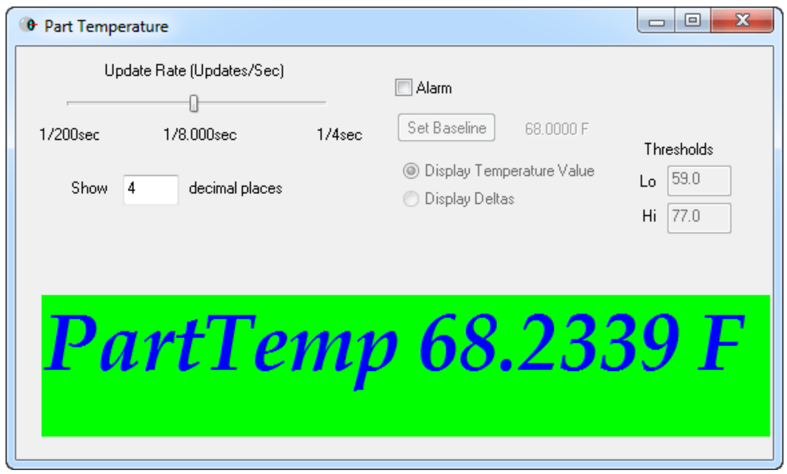


Figure 14-38. Monitoring a part's temperature.

Instrument Checks

Warning: SA tracker interface checks are purely to gauge performance of the tracker. No actual compensation is applied to the instrument based on the checks.

The SA tracker interface has a number of check routines to verify performance of the tracker. It is important to note that none of these checks in the interface actually modify the compensation values for the tracker. The instrument manufacturer knows best how to compensate their own instrument, so actual modifications are performed in the OEM's tracker compensation software.

Two Face

The Two Face check (Figure 14-39) gives an indication of orthogonality of the azimuth and elevation axes in the tracker. It measures discrete positions from both the front and back faces, and presents the angular discrepancies in each measurement pair.

It is generally recommended that this check be performed with the instrument using IFM measurement mode to minimize distance error, although the primary purpose of this check is to gauge errors in the azimuth and elevation readings between the front and back faces. Instrument manufacturers provide recommended passing thresholds for instrument operation.

- **Check Current Location.** Takes a front- and back-face measurement at the current reflector position.
- **View/Edit Acquisition.** Provides control over how the measured point is acquired.
- **Threshold.** Angular values exceeding the provided delta threshold will be marked as out of tolerance.
- **Apply Thresholds.** Click this button to apply the specified thresholds to the data.
- **Warning Band.** A percentage of the threshold that triggers a warning indication.
- **Home.** Homes the tracker.
- **Remote.** Sends the tracker to the defined remote home position (if defined).
- **Closure.** Initiates a closure check. (Checks the current position of the target against the home position).
- **OK/Log.** Writes results to the log and closes the window.

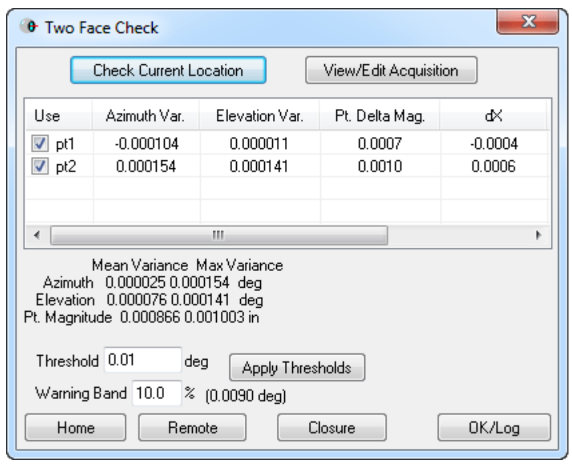


Figure 14-39. The Two Face check.

To Perform A Two-Face Check In SA:

1. From the interface menu, select **Check/Cal > Two Face.**
2. Home the tracker by clicking the Home button (IFM-enabled trackers only).
3. Track the target out to a puck and set it down.
4. Click the Check Current Location button. The tracker will acquire front and back measurements, then report the deviations between the two.
5. When finished, click OK/Log to log the results.

ADM Check

An ADM check is used to measure the difference between corresponding IFM and ADM distance measurements. A new ADM offset can be created and applied to a given reflector.

To Initiate an ADM Check:

1. From the interface menu, select **Check/Cal > ADM**.
2. For more information, refer to "User Definitions" on page 434.

IFM Check

IFM checks validate IFM distance measurements using a two-station method or scale bar. The check geometrically eliminates error based on expected birdbath position to give an indication of IFM distance error. All measurements must be obtained in IFM distance mode.

In the two-station check, two reference points are mounted in a fixed location. The tracker is placed between the two points (as close to the line connecting them as possible), and each reference point is measured. The tracker is then moved to a new position outside of both reference points, and the two points are measured again. Geometrically, one of these sets of measurements includes IFM error, whereas the other does not. The check involves determining the difference between both sets of measurements.

In the Scale Bar check, the tracker is only set up between the reference points, and the reference length of the scale bar is used as the baseline. The difference between the measurements and the scale bar length indicates IFM error.

To Perform an IFM Check:

1. From the interface menu, select **Check/Cal > IFM.**
2. Place the desired reflector in the home nest.
3. In the ***Reflectors and Targets*** dialog, selected the applicable reflector and click the IFM (Home Distance) Check - Selected Reflector button.
4. When the ***IFM (BirdBath) Check/Cal*** dialog appears (Figure 14-40), click the button appropriate to the check method you'd like to use.
5. Measure the points as indicated by the dialog.

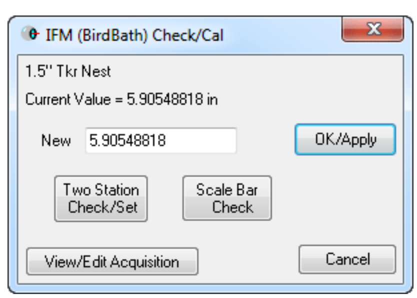


Figure 14-40. Performing an IFM check.

Drift Check

Use this check to create a database of points for monitoring drift. SA keeps track of the nominal points, and when remeasured the deltas (as well as the distance measuring mode) are displayed (Figure 14-41).

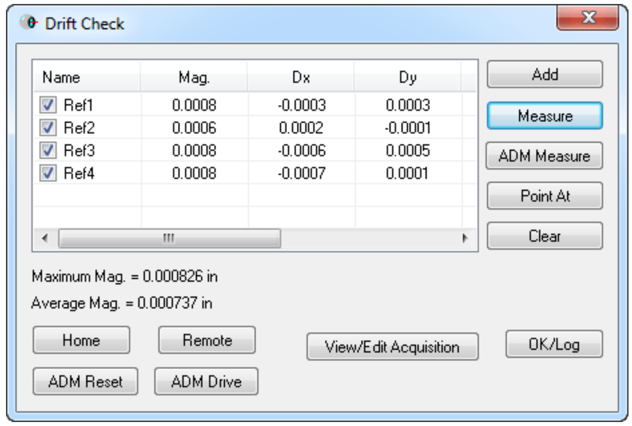


Figure 14-41. Performing a drift check.

- **Add.** Adds a drift reference point to the set and measures its baseline position.
- **Measure.** Measures the selected reference point.
- **ADM Measure.** Measures the selected reference point using ADM distance mode.
- **Point At.** Points at the selected reference point.
- **Clear.** Clears the reference points from the list.
- **Home.** Homes the tracker.
- **Remote.** Sends the tracker to a remote home (if defined).
- **ADM Reset.** Searches for the target using an ADM search.
- **ADM Drive.** Use to drive the tracker head to a specific position.
- **View/Edit Acquisition.** Modifies the parameters used for acquiring the drift measurements.
- **OK/Log.** Closes the dialog and writes the results to the log file.

To Perform a Drift Check:

1. From the interface menu, select **Check/Cal > Drift**.
2. Click Add to add each reference point to the list, entering a name and measuring the point in the process.
3. At some point in the future, reopen the dialog, select a point, and click Measure. The deltas and distance measurement modes will be displayed for inspection.
4. When finished, click OK/Log to write the results to the log file.

Closure Check

The closure check (Figure 14-42) compares the measured home position to the defined home position.

- **Recheck.** Measures the reflector in the nest and compares the measured position to the reference position.
- **Log.** Writes the results to the log file.
- **View/Edit Acquisition.** Provides access to the discrete point parameters for measuring the point.

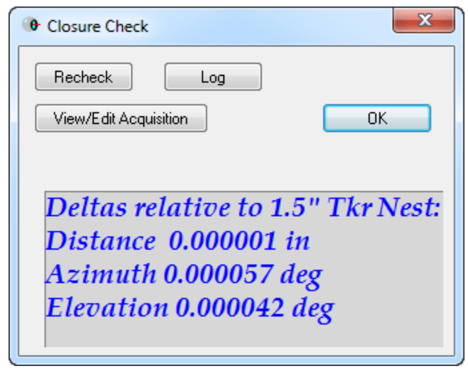


Figure 14-42. The Closure check.

To Perform a Closure Check:

1. Place the selected reflector into the home position (birdbath).
2. From the interface menu, select **Check/Cal > Closure**.
3. In the ***Closure Check*** dialog, click the Recheck button. The tracker will home on the reflector and take a measurement. Deviations should be very small.

Reflector Center Check

SMRs do not perfectly center the beam inside the sphere. Therefore, if the centering is slightly out, then taking multiple measurements while rotating the SMR in a nest will yield measurements that move slightly (runout). This test is particularly important if an SMR is dropped or jostled. It may appear fine, but the adhesive and reflective surfaces may have shifted on impact, putting the reflector out of tolerance.

Reflector center checks are carried out in the ***Reflector Center Check*** dialog (Figure 14-43). An initial measurement baseline point is acquired, then the test is started. Measurements are taken periodically as the SMR is rotated in the nest (ideally no rotation occurs while a specific point is being measured).

- **Set Baseline.** Measures the initial reference point for comparison.
- **Edit Baseline Acquisition.** Defines how the baseline (reference) point is acquired.
- **Edit Main Acquisition.** Defines how the comparison points

are acquired.

- **Show X decimal places.** Indicates the number of decimal places to show in the results.
- **Cartesian.** Displays the results in Cartesian Coordinates.
- **Polar.** Displays the results in Polar Coordinates.
- **Measure at X second intervals.** Indicates how frequently measurements are taken.
- **Save Results.** Stores the results to the log file.
- **View Results.** Shows the results of the test.
- **Clear Results.** Removes test results to prepare for a new test.
- **Start.** Initiates measurement of the reflector.

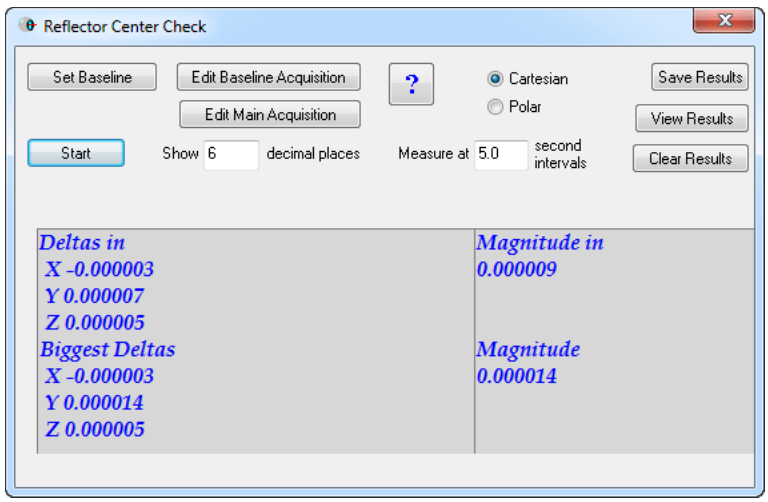


Figure 14-43. Performing a reflector center check.

To Perform a Reflector Center Check:

1. Place the reflector into a stable drift nest.
2. From the interface menu, select **Check/Cal > Reflector Center Check**.
3. In the ***Reflector Center Check*** dialog, click the Set Baseline button.
4. Click the Start button to begin measurement.
5. Between each measurement, rotate the reflector in the nest. The current and largest deltas relative to the baseline will be reported.
6. Click Stop when finished.
7. Click Save Results to log the results of the test.

Manufacturer-Specific Checks

The interface provides a convenient link to the manufacturer-provided checking and compensation routines. These compensation routines are the only place where true modifications to the instrument's compensation parameters are made.

To Access the OEM-Specific Checking & Compensation Routines:

- From the interface menu, select **Check/Cal > Tkr Specific (Mfcr)**.

Tracker Utilities

System Information

The ***System Information*** dialog (Figure 14-44) displays diagnostic tracker information, such as the SA tracker driver version, firmware and serial number, and OEM driver version information. This is primarily useful when diagnosing issues with SA customer support.

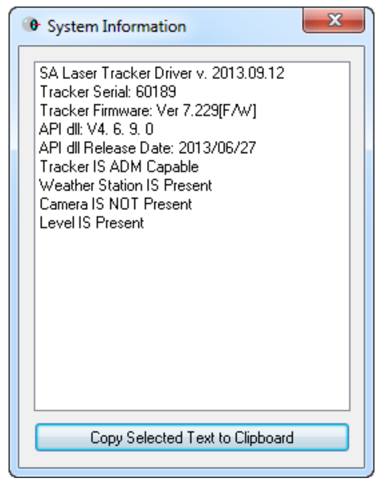


Figure 14-44. The tracker's vital system information.

To Display The Tracker's System Information:

- From the interface menu, select **Utility > System Info**.

Restarting the Laser Tracker

If you experience unexpected behavior, you can perform a warm (non powered-off) reboot of the tracker controller.

To Perform a Warm Reboot of the Tracker Controller:

- From the interface menu, select **Utility > Tracker Utilities > Restart Tracker**.

Reinitializing the Laser Tracker

Note: This is not available on all tracker brands.

You can issue a command from the tracker interface to reinitialize a laser tracker. The specific tasks performed during an initialization of a tracker are manufacturer-specific, but in general this can help resolve some accuracy and tracker performance issues.

To Reinitialize the Laser Tracker:

- From the interface menu, select **Utility > Tracker Utilities > Reinitialize Tracker**.

Manufacturer-Specific Utilities

You can access manufacturer-specific utilities directly through the tracker interface.

To Access OEM-Specific Utilities

- From the interface menu, select **Utility > Tracker Utilities > Tracker Specific**.

Tracker Status

Note: The information displayed in the tracker status dialog depends on the specific instrument in use.

You can view the overall status and health of the laser tracker by viewing the tracker's status (Figure 14-45). This typically includes information such as battery state of charge, ADM/IFM beam intensity, servo motor status, and other diagnostic functionality.

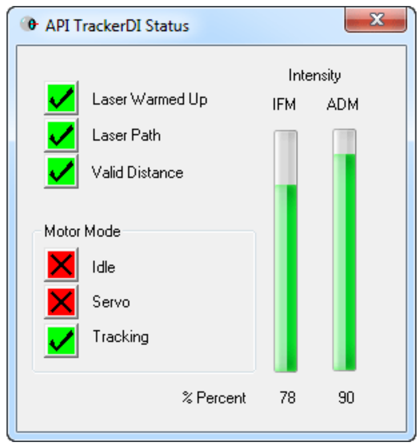


Figure 14-45. The tracker status dialog for an API Radian tracker.

To View a Tracker's Status:

- From the interface menu, select **Utility > Tkr Status**.

Current Position

In the ***Current Position*** dialog (Figure 14-46), you can view the current position of the reflector in both Cartesian Coordinates (expressed relative to the working frame in SA) or Spherical Coordinates (expressed in the tracker's native frame).

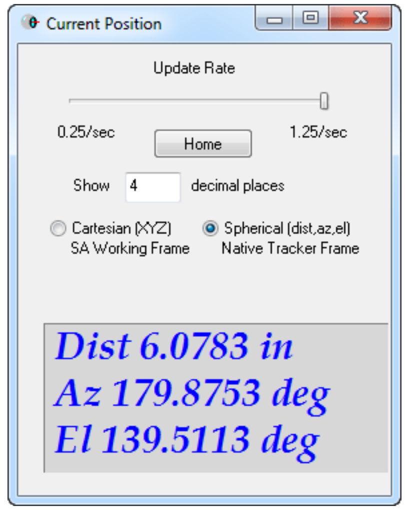


Figure 14-46. Live data from the tracker.

Importing Spherical Data

Note: Currently, if source data is delimited by commas, it will not import. The data must be delimited by spaces or other whitespace.

Occasionally, you may have measured data from a tracker that has been exported to an ASCII file. If this data is in spherical format (contains azimuth, elevation, and distance components), it can be imported and associated with a tracker. These imported points are automatically treated as measured points (although they will not have any measurement metadata). The import dialog is shown in Figure 14-47.

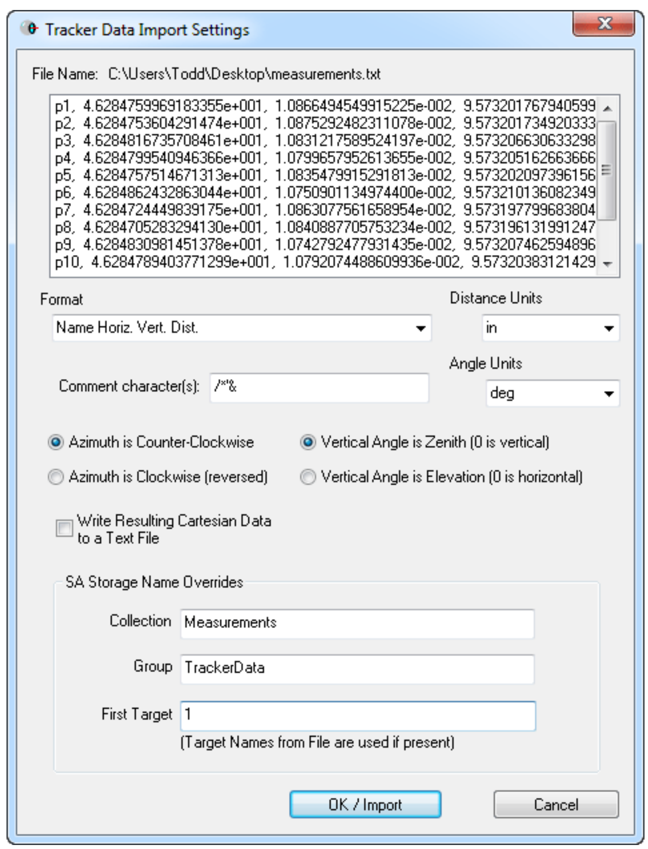


Figure 14-47. Settings for importing spherical data to a tracker.

- **Format.** Choose the appropriate format for the source data.

- **Distance Units.** The length units for the source data.
- **Comment Character(s).** Any lines starting with any of the provided characters are considered to be comments and are ignored.
- **Angle Units.** The angular units for the source azimuth and elevation angles.
- **Azimuth is Counter-Clockwise.** Specifies that the data was recorded on a tracker whose azimuth direction increases as the head rotates counter-clockwise as viewed from above.
- **Azimuth is Clockwise (reversed).** Specifies that the source data comes from a tracker whose horizontal angle increases as the head rotates clockwise as viewed from above (not common).
- **Vertical Angle is Zenith (0 is vertical).** Indicates that the source spherical coordinate system has 0° pointing straight up (zenith referenced).
- **Vertical Angle is Elevation (0 is horizontal).** Indicates that the source spherical coordinate system has 0° level with the horizon.
- **Write Resulting Cartesian Data to a Text File.** Upon import, the points will be written out to a text file in Cartesian coordinates.
- **Collection.** The destination collection for the points.
- **Group.** The destination group for the points.
- **First Target.** The name for the first target. This name will be incremented for following targets. If the source data already has point names, this field is ignored.

To Import Polar ASCII Data:

1. From the interface menu, select **Utility > ASCII Import**.
2. Select the source file from the browser.
3. Choose the appropriate format, units, and options for the import, then click OK/Import.

Streaming Tracker Data Over The Internet

The tracker interface has the capability of streaming real-time measurement data (3-DOF or 6-DOF) over the internet, using UDP (User

Datagram Protocol) broadcasting. Unlike TCP/IP network communications, UDP broadcasting does not require "handshaking" for data transmission. There is no guarantee that transmitted data is received, and it can be broadcasted "into the wild" so that any network device can listen to the broadcast.

UDP Data Streaming opens a vast array of possibilities for harnessing tracker data in custom ways. For example, you can write applications to intercept this tracker data and perform custom analysis for hardware control, or you could even write your own watch window.

There are a few limitations to keep in mind with regard to UDP Data Streaming:

- Data flows only in one direction—*from* the interface. You cannot transmit commands *to* the tracker interface.
- Data transmission is of course subject to some finite amount of latency. It will not be received "real-time" in a hardware sense.

Tracker data is transmitted as an ASCII-formatted packet. The format of the packet is dependent on settings in the ***SA UDP Data Stream Properties*** dialog (Figure 14-48).

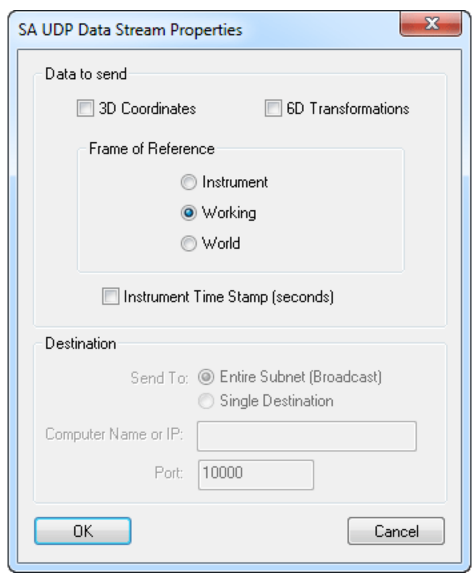


Figure 14-48. Properties for setting up UDP Data Streaming from the tracker interface.

- **3D Coordinates.** Transmits 3-DOF (XYZ) data.
- **6D Transformations.** Transmits 6-DOF (XYZ, $R_X R_Y R_Z$) data.
- **Frame of Reference.** Data can be transmitted relative to the *Instrument* frame, the *Working* frame, or the *World* frame.
- **Instrument Time Stamp (seconds).** If the instrument supports it, time stamps can be transmitted along with the data. This is often an approximately millisecond-accurate timestamp from the tracker hardware indicating when the measurement was taken.

- **Send To.** The broadcast packets can be sent to the *Entire Subnet (Broadcast)* or to a *Single Destination*. In the former case, data is transmitted "to the wild" for any and all devices on the appropriate subnet to receive. In the latter, the data stream is routed to a specific IP address on the network.
- **Computer Name or IP.** Enter the IP address or computer name for the destination device (if broadcasting to a *Single Destination*).
- **Port.** Every UDP broadcast must be directed to a destination port. This is the port on which the receiving device must listen for the appropriate broadcasts.

Viewing The Upcoming Group and Target

When working with a tracker a long distance from the computer, it is difficult if not impossible to read the name of the next target that will be measured. Although this problem is addressed by the SA HUD settings, you can also open up a ***Group and Target*** dialog.

Figure 14-49. Viewing the upcoming group and target to measure.

This dialog can be resized and positioned anywhere on screen, and will always display the group and target of the point that is about to be measured.

To View the Group and Target Window:

- In the tracker interface, click the button encompassing the **`Group:`** and **`Target:`** labels (Figure 14-50).

Figure 14-50. The Group and Target button.

Instrument Toolbar

The Instrument Toolbar (Figure 14-51) is a greatly-streamlined version of the laser tracker interface. Once properly configured, you'll likely find that the majority of your measurement work can be performed in this simpler interface.

Figure 14-51. The Instrument Toolbar for laser trackers.

To access the Instrument Toolbar from the tracker interface, click the Switch to Instrument Toolbar button. To switch from the Instrument Toolbar back to the tracker interface, click the Switch to Instrument Interface button.

The different functions of the Instrument Toolbar are described below.

Beam Status

A colored light (similar to the "traffic light" in the laser tracker interface provides feedback on beam state and measurement status. The different statuses indicated by the light are as follows:

Locked. The beam is locked on the target and valid distance information is being received.

Measuring. The tracker is currently measuring. (A glowing indicator spins in this mode).

No Valid Distance. The tracker is tracking the reflector, but a valid distance measurement is not available.

No Lock. The tracker does not have a lock on the reflector.

Waiting for Lock. A measurement mode has been initiated, but SA is waiting for a lock on the beam. (A glowing indicator spins in this mode).

Waiting for Move. A stable point measurement trigger is being used, and SA is waiting for the reflector to move. (A glowing indicator spins in this mode).

Waiting for Stable. A stable point measurement trigger is being used, and SA is waiting for the reflector to stabilize. (A glowing indicator spins in this mode).

Clicking on the beam status light will initiate a reflector search to lock on a target. It is equivalent to the Find Target button in the laser tracker interface.

Homing the Tracker

Click the Home button to send the tracker to the birdbath for the currently-active reflector. This icon not available for trackers that have no home nest.

Driving the Tracker

Click the Drive Beam button to drive the tracker head to the desired position. Drive operation is tracker-specific and described in "Drive Head/OVC" on page 432.

Toggling Motors

For instruments that support it, you can engage and disengage the tracker head motors by toggling the Motors button. (This icon is not shown for instruments that do not support motor toggling).

Tilt Compensation

Some instruments support tilt compensation. When turned on, the instrument automatically adjusts all measurements such that the standing axis of the instrument in SA is aligned with gravity. (That is, the instrument's internal reference frame is always oriented to gravity). For instruments that support this, the motor toggle button (see "Toggling Motors" on page 476) is replaced by a tilt compensator toggle button. Clicking this button toggles tilt compensation on and off.

Active Target

You can select the active target by clicking on one of the target quick-select icons (Figure 14-52).

Figure 14-52. The target quick-select buttons.

The reflectors are defined in the laser tracker interface (see "Targets and Retros" on page 433) or can be defined directly from the tracker toolbar. Right-clicking any of the four quick-select buttons will allow you to assign a different tooling definition to the button. You can also select **define new target** from the drop-down to create a new tooling definition.

Hovering the mouse cursor over a button momentarily will show a tooltip indicating the name of the target, and the offset (diameter if spherical, planar and radial if not) is displayed next to the quick-select buttons (Figure 14-53).

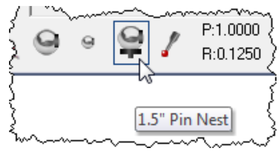


Figure 14-53. The planar and radial offsets are displayed, along with a target name tooltip.

Point Naming

The name of the next measured point is displayed above the target quick-select icons. To change the name of the next selected target, click this field. This will open the ***Instrument Interface Point Naming*** dialog, allowing you to type a new collection, group, and target name.

Measurement Modes

The Instrument Toolbar has eight buttons, each corresponding to a specific measurement mode (Figure 14-54).

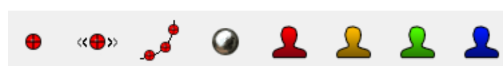


Figure 14-54. The measurement mode buttons.

The first four buttons are assigned to Single Point, Stable Point, Spatial Scan, and Tooling Ball measurement modes, respectively:

- **Single Point.** Measures a discrete point when clicked.
- **Stable Point.** Waits for the reflector to move—then become stable—before measuring the point.
- **Spatial Scan.** Initiates a spatial scan measurement. The increment can be set by right-clicking the button.
- **Tooling Ball.** Initiates a spatial scan measurement, then fits a sphere to the points and creates a center point on the sphere. Both the spatial scan increment and the radius of the sphere fit can be controlled by right-clicking the button.

The last four buttons (denoted by the silhouette icons) are user-defined measurement modes. You can assign any measurement mode you've created in the laser tracker interface to these buttons by right-clicking them.

Measurement modes can be initiated with or without a beam lock. If started without a beam lock, SA will wait for a beam lock before starting measurement. While in a measurement mode, press the button again to stop the measurement mode.

To access a few additional parameters related to the first four measurement modes, right-click the target button.

Alarms

The Alarm button allows you to set a number of thresholds that will trigger an alarm while measuring. An alarm is intended to notify you that a measurement may not be trustworthy. High and low threshold alarms can be set for the following parameters:

- **Air Temperature.** If the specified air temperature range is violated, the alarm will trigger.
- **Air Pressure.** If the specified pressures are violated, the alarm will trigger.
- **Humidity.** If the ambient humidity exceeds the specified thresholds, the alarm will trigger.
- **Part Temperature.** If the part temperature (if the instrument is equipped) exceeds the specified threshold, the alarm will trigger.
- **Level.** If the specified level range is violated (if the instrument is equipped with an inclinometer), the alarm will trigger.
- **RMS.** Turning on the *Discrete Point RMS Monitor* checkbox will display a warning if the *Warning* threshold is exceeded. The point will not be measured if the *Out Level* value is exceeded.

When an alarm is triggered, the alarm button will turn red and a blinking exclamation mark will appear.

Operational Checks

Clicking the Operational Checks button will take you to the manufacturer's operational checks.

Measuring With Arms

CHAPTER 15

Portable arms are ideal instruments when measuring small volumes where high accuracy is required and line-of-sight measurements are difficult to achieve. This chapter discusses how to properly and effectively measure with portable CMM arms in SA.

The Portable CMM Arms interface (Figure 15-1) provides unified access to all measurement parameters associated with measurement using a PCMM arm. The arm interface is divided into three primary sections: Target and Geometry Naming, Measurement Modes, and Settings.

At the very top of the interface, an interface docking button toggles between docking the interface to the SA window or letting it float as a separate window. If the instrument interface is opened without running SA, this button is unavailable (as there would be no instance of SA on the machine to dock with). If more than one interface is running, docking an interface will replace any currently docked interface.

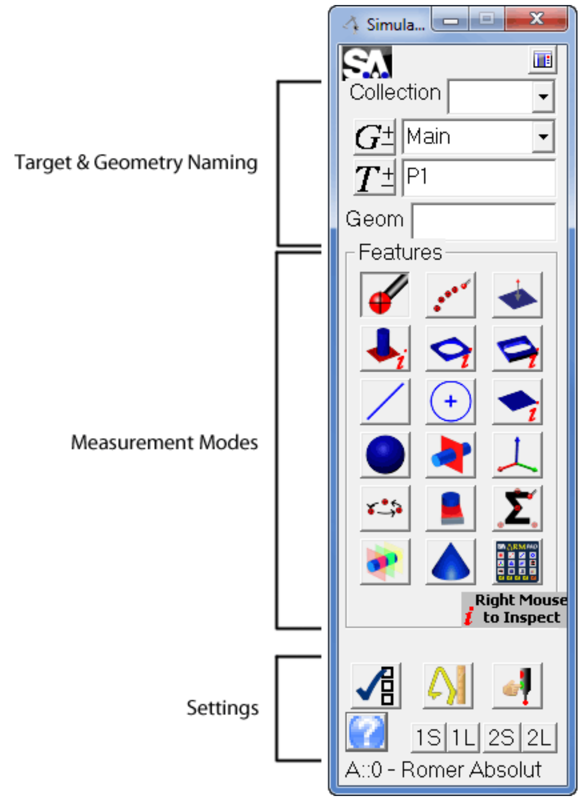


Figure 15-1. The portable CMM arm interface.

Target & Geometry Naming

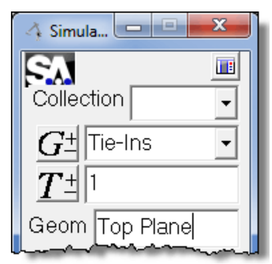


Figure 15-2. Target and geometry naming.

The Target & Geometry Naming section (Figure 15-2) is used to define how measured targets and constructed geometry will be named.

Collection

Measured targets and geometry constructed by the instrument interface are sent to SA. The *Collection* field is where you specify the destination collection for measured entities. If a collection name is not specified, entities are by default placed into the active collection.

Group Name

The point group name for measured targets is specified in the field next to the "G". To assign measured targets to a specific group, enter the name into the field. To assign measured targets to an existing point group, select it from the drop-down list, which shows all point groups in the active collection.

To Increment the Current Group Name:

- Left-click the G± button.

To Decrement the Current Group Name:

- Right-click the G± button.

Target Name

Target naming is managed in the field next to the "T". Enter the desired target name in this field. After each measured target, this name will automatically increment by one.

To Increment the Current Target Name:

- Left-click the T± button.

To Decrement the Current Target Name:

- Right-click the T± button.

Geometry Name

The arm interface supports real-time geometry construction. The *Geom* field is used to name this constructed geometry. After each geometrical entity is constructed, this name will automatically increment.

Button Control

The SA Arm interface supports the use of button commands from the buttons equipped on the Portable Arm. These buttons can be assigned different functions in the ***Edit Arm Settings*** window. One common use of the arm buttons is to move through the different measurement modes, which makes it possible to change measurement methods without the need to return to the computer keyboard. (The SA ArmPad enhances this functionality even further).

Measurement Modes

Measurement modes are extremely powerful and functional. They allow a user to take measurements and process data in one step. The arm interface supports 17 different measurement modes: Discrete Points, Stream Points, Patch (Projected Pt.), Pin, Hole, Slot, Line, Circle, Plane, Sphere, Section Cross, Frame, Batch (Guided Points), Scanner, Average Discrete Points, Geometry Trigger, and Cone. A special method for selecting measurement modes can be accessed as well. These modes are discussed below.

Discrete Points

Discrete Points mode is used to take individual measurements (Figure

15-3). When in this mode (signified by the appearance of the Points Measurement window below), the assigned Record button will take a measurement, and the assigned Delete Last button will delete the last measured point (if applicable). Point names will be incremented and decremented automatically.

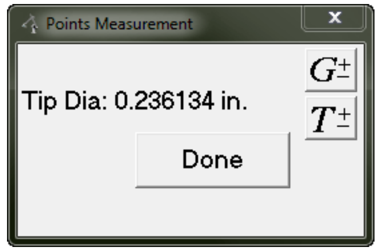


Figure 15-3. This window indicates you are in discrete point measurement mode.

To manually increment/decrement the group or point name, left- or right-click the G± or T± buttons.

When finished with this mode, press the assigned Accept or Cancel buttons on the arm--or click the Done button--and the Points Measurement window will close, returning you to the interface.

Stream Points

Stream Points mode is used to take measured data at specific spatial or temporal increments (as specified in the ***Edit Arm Settings*** window). It is extremely important to recognize that the actual measured data will not be spaced by the specified increments exactly--the increment will be limited by such factors as the hardware's measurement update speed, the connection speed between the instrument and SA, the computer's performance, and the speed at which the probe is moved.

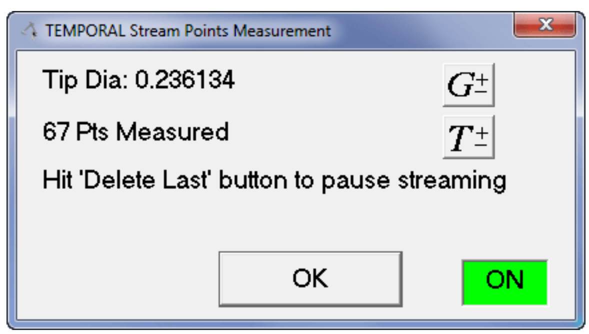


Figure 15-4. Streaming points temporally to the interface.

Note: The title of the measurement mode window will either say TEMPORAL or SPATIAL, depending on the currently selected mode.

When in this mode (signified by the appearance of Figure 15-4), pressing the assigned Record button will begin or continue measurement, and the assigned Delete Last button will pause measurement.

To manually increment/decrement the group or point name, left- or right-click the G± or T± buttons.

When finished with this mode, press the assigned Accept or Cancel button--or click the OK button--and the Stream Points Measurement window will close, returning you to the interface.

Patch (Projected Pt.)

Patch (Projected Pt.) mode (Figure 15-5) will project measured points to a plane, with an optional offset (set in the Edit Arm Settings window). After starting this measurement mode, you can either select a plane to project to (by clicking the Select button) or you can measure plane points yourself by pressing the assigned Record button. Statistics for the measured plane will be displayed as you measure points. Note that you can delete a measured point by pressing the assigned Delete Last button, and the plane will automatically re-fit.

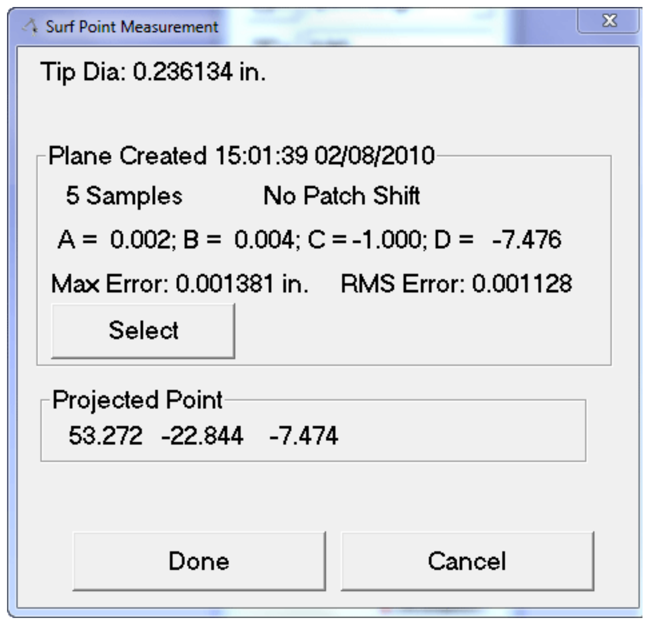


Figure 15-5. Patch projected point measurement.

When you've defined the plane, press the default Accept button to accept it. Every subsequent measurement will be projected to the selected plane (or offset plane). The projected point's coordinates will be displayed in the window with each measurement. When finished, press the assigned Accept or Cancel button--or click the Done button--and you will be returned to the interface.

Pin

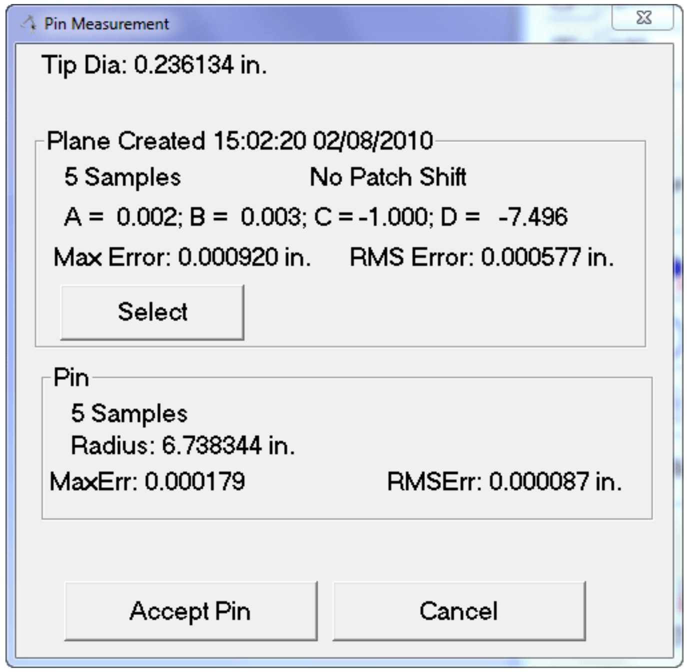


Figure 15-6. Pin measurement mode.

Pin mode (Figure 15-6) will project measured points on the outside of a cylinder to a plane, and create a circle representing the pin diameter. This is useful for determining, for example, the diameter of a pin. When in this mode, you first define the plane to which you'd like to project. As in Patch (Projected Pt.) mode, you can either select a plane to project to (by clicking the Select button) or you can measure plane points yourself by pressing the assigned Record button. Statistics for the measured plane will be displayed as you measure points. Note that you can delete a measured point by pressing the assigned Delete Last button and the plane will automatically re-fit.

Once the plane is defined, press the assigned Accept button to accept it. Subsequent measurements should be taken around the outside of cylindrical geometry (such as a pin or shaft), and these measured points will be projected to the defined plane. Once 3 measurements have been taken, statistics for the constructed circle (radius, fit error, etc.) will be displayed. To accept the Pin measurement, press the assigned Accept button or click the Accept Pin button. To cancel geometry creation, press the assigned Cancel button or click Cancel.

You can right-click this measurement mode button in order to inspect a pin feature that already has nominal geometry in your file. The inspect mode will compare your measured feature to the geometry selected as the reference. To inspect a pin, first right-click on the Pin button. When prompted, choose the circle to inspect. You'll then be prompted to measure plane points. After measuring and accepting the plane points, you'll be prompted to measure the pin itself. When complete, and after pressing the assigned Accept button, you'll get a comparison of feature characteristics (Figure 15-7).

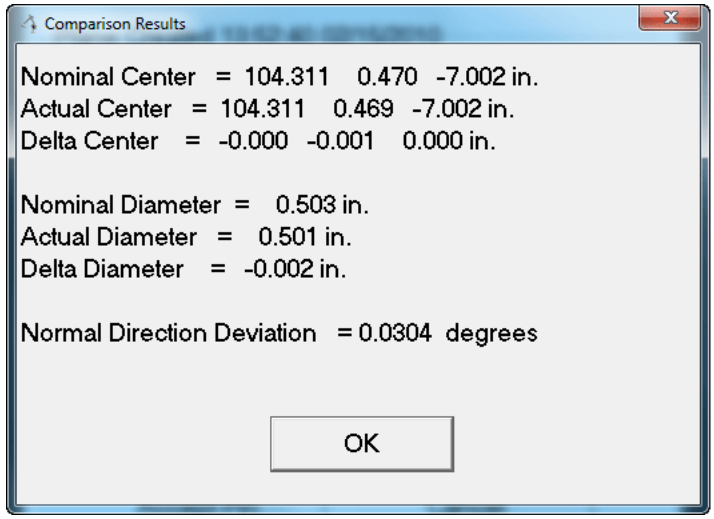


Figure 15-7. Comparison results.

Hole

Hole mode (Figure 15-8) will project measured points on the inside of a hole to a plane, and create a circle representing the hole diameter. When in this mode, you first define the plane to which you'd like to project. As in other modes, you can either select a plane to project to (by clicking the Select button) or you can measure plane points yourself by pressing the assigned Record button. Statistics for the measured plane will be displayed as you measure points. Note that you can delete a measured point by pressing the assigned Delete Last button, and the plane will automatically re-fit.

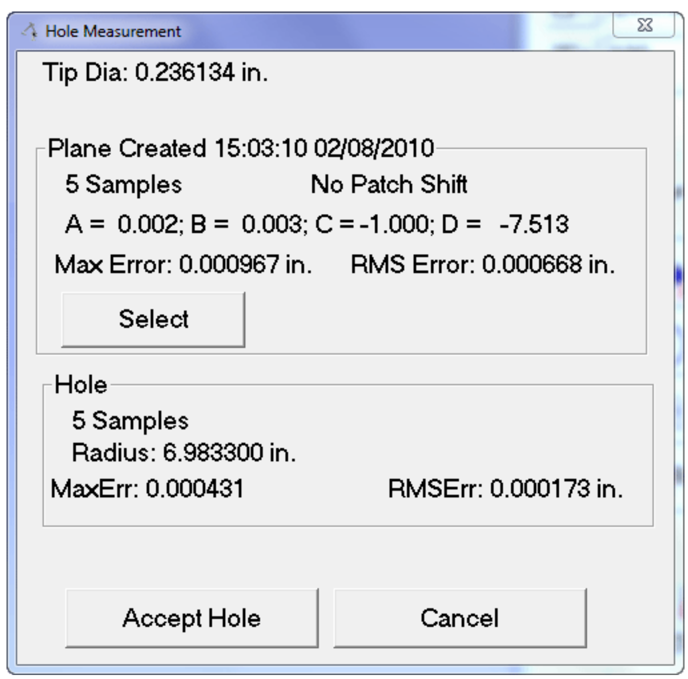


Figure 15-8. Hole measurement.

Once the plane is defined, press the assigned Accept button to accept it. Subsequent measurements should be taken around the inside of cylindrical geometry (such as the inside of a hole or pipe), and these measured points will be projected to the defined plane. Once 3 measurements have been taken, statistics for the constructed circle (radius, fit error, etc.) will be displayed. To accept the Hole measurement, press the assigned Accept button or click the Accept Hole button. To cancel geometry creation, press the assigned Cancel button or click Cancel.

You can right-click this measurement mode button in order to inspect a hole feature that already has nominal geometry in your file. The inspect mode will compare your measured feature to the geometry selected as the reference. To inspect a hole, first right-click on the Hole button. When prompted, choose the circle to inspect. You'll then be prompted to measure plane points. After measuring and accepting the plane points, you'll be prompted to measure the hole itself. When complete, and after pressing the assigned Accept button, you'll get a comparison of feature characteristics (Figure 15-9).

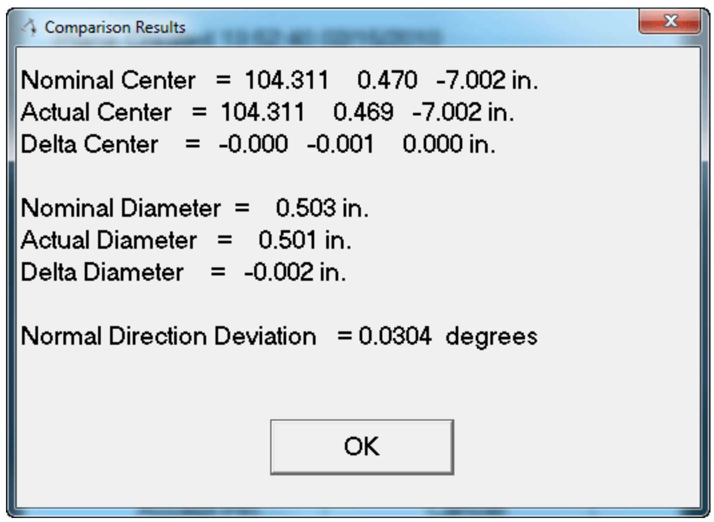


Figure 15-9. Comparison results.

Slot

Slot mode (Figure 15-10) will project two measured inside diameters of a slot to a plane, and construct circles and a slot midpoint (if specified). When in this mode, you first define the plane to which you'd like to project. As in other modes, you can either select a plane to project to (by clicking the Select button) or you can measure plane points yourself by pressing the assigned Record button. Statistics for the measured plane will be displayed as you measure points. Note that you can delete a measured point by pressing the assigned Delete Last button and the plane will automatically re-fit.

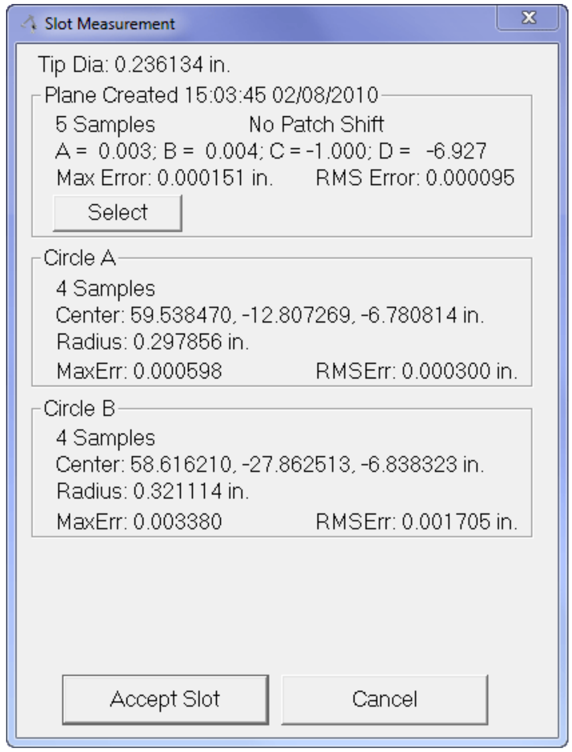


Figure 15-10. Slot measurement mode.

Once the plane is defined, press the assigned Accept button to accept it. You should then measure the points defining the inside diameter of one end of the slot (Circle A). Once 3 measurements have been taken, statistics for the constructed circle (radius, fit error, etc.) will be displayed. To accept the measurement, press the assigned Accept button. Next, measure the points defining the inside diameter of the other end of the slot (Circle B). To accept the slot measurement, press the assigned Accept button again, or click the Accept Slot button. To cancel geometry creation, press the assigned Cancel button or click Cancel.

Line

Line mode (Figure 15-11) creates a best-fit line through a series of measured points. Once two points have been measured, statistics for the line and the fit will be displayed. The assigned Record button will take additional measurements, while the assigned Delete Last button will delete the previously measured point and re-fit.

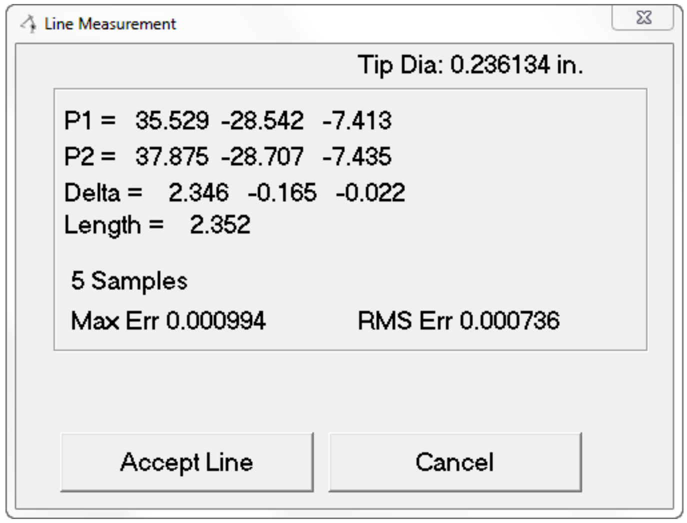


Figure 15-11. Line measurement mode.

When finished measuring, press the assigned Accept button or click the Accept Line button. This will create a line in the SA job. To cancel the measurement mode, press the assigned Cancel button or click the Cancel button.

Circle

Circle mode (Figure 15-12) creates a best-fit circle to the measured points. The difference between this mode and Pin/Hole modes is that the measured points are not first projected to a plane. This mode allows you to specify whether you're measuring an inside radius (hole), outside radius (pin), or face radius (no offset). Probing direction is used to define the normal direction of the resulting circle. After 3 points have been measured, statistics for the fit of the resulting circle will be displayed. You can remove the previously-measured point from the fit by pressing the assigned Delete Last button.

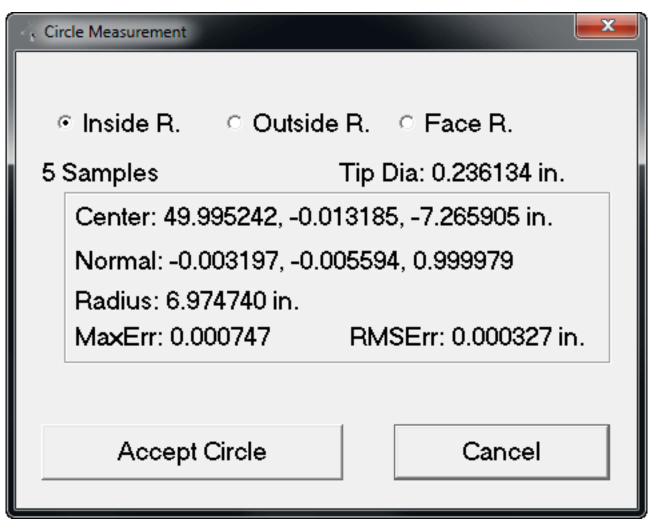


Figure 15-12. Circle measurement mode.

When finished measuring points, press the assigned Accept button or click the Accept Circle button. To cancel geometry creation, press the assigned Cancel button or click the Cancel button.

Plane

Plane mode (Figure 15-13) creates a best-fit plane to a series of measured points. Once 3 points are measured, the Plane Measurement window displays statistics for the fit and the resulting plane. The normal direction of the plane is determined by the probing vector. You can remove the previously-measured point from the fit by pressing the assigned Delete Last button.

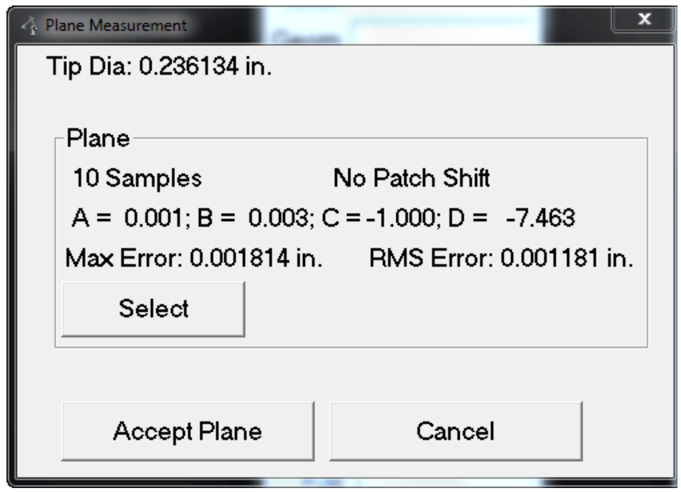


Figure 15-13. Plane measurement mode.

When finished measuring points, press and hold the assigned Accept button or click the Accept Plane button. To cancel geometry creation, press the assigned Cancel button or click the Cancel button.

You can right-click this measurement mode button in order to inspect a plane that already exists in your file. The inspect mode will compare your measured feature to the geometry selected as the reference. To inspect a plane, first right-click on the Plane button. When prompted, choose the plane to inspect. You'll then be prompted to measure plane points. After measuring and accepting the plane points, you'll get a comparison of feature characteristics (Figure 15-14).

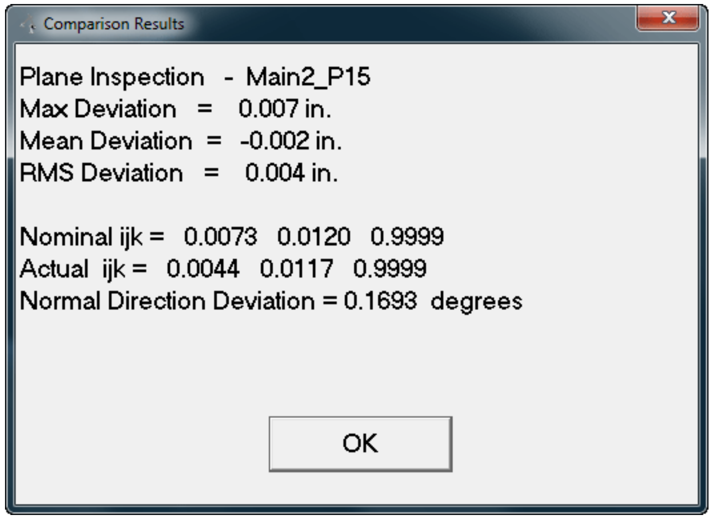


Figure 15-14. Comparison results.

Sphere

Sphere mode (Figure 15-15) creates a best-fit sphere to a series of measured points. Once 4 points are measured, the Sphere window displays statistics for the fit and the resulting sphere. You can remove the previously-measured point from the fit by pressing the assigned Delete Last button.

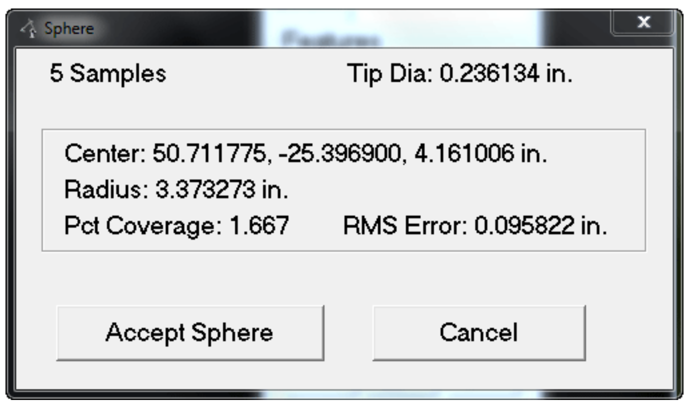


Figure 15-15. Sphere measurement.

When finished measuring points, press the assigned Accept button or click the Accept Sphere button. This will create a sphere and a point representing the center of that sphere. To cancel geometry creation, press the assigned Cancel button or click Cancel.

Cross Section

Cross Section mode (Figure 15-16) will automatically trigger measurements when the probe crosses a specified coordinate frame axis value (plane). This is useful, for example, when creating measured cross-sections of complex geometry. After initiating this mode, the Section Specification dialog will appear, in which you specify the axis (X, Y, or Z) and plane on that axis that you would like to use to trigger measurements:

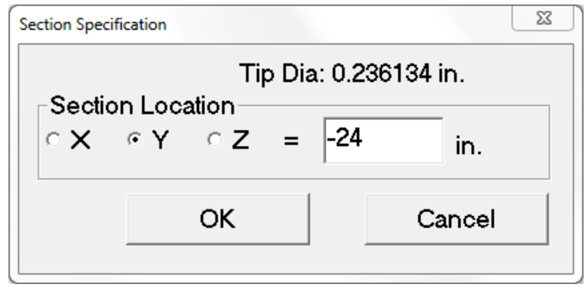


Figure 15-16. Cross section mode.

Once the plane is defined, click the OK button or the assigned Accept button. This will bring up the ***Section Measurement*** dialog (Figure 15-17).

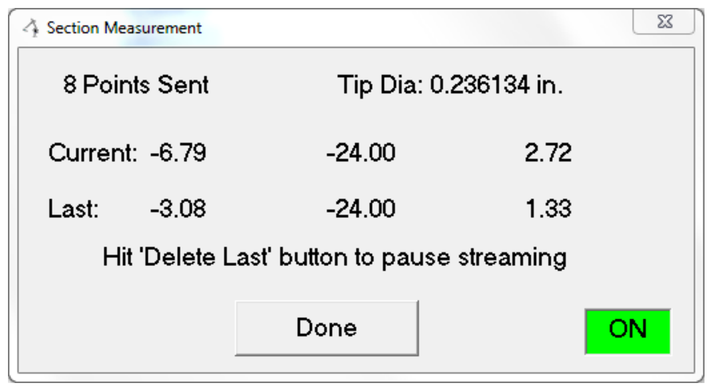


Figure 15-17. The section measurement dialog.

When ready to begin measurement, press the assigned Record button . The arm will begin streaming data to SA (Figure 15-18). When the probe crosses the specified plane, a measurement will be fired. Notice that the result of this measurement will be an interpolated point that lies directly on the specified plane. (Due to limits of hardware sampling frequency, network speed, computer speed, and other restrictions, it is highly unlikely that the instrument's measurement will lie perfectly on the specified plane).

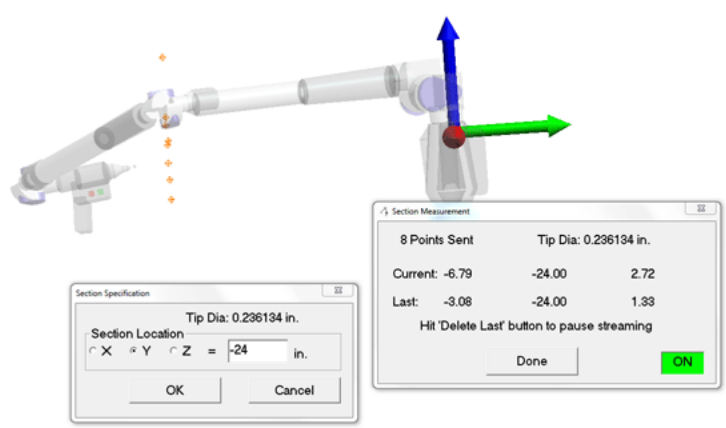


Figure 15-18. Measuring at a specific cross-sectional value.

To pause measurement, press the assigned Delete Last button, which will put the arm back into a mode in which the graphical view will be continuously updated with the arm's position, but no measurements will be taken. When finished measuring, press the assigned Accept or

Cancel button, which will return you to the interface.

Frame

Frame mode (Figure 15-19) constructs a frame based on the probing direction (final joint position) of the arm. The origin is placed at the probe position, and the constructed frame's Z-axis is along the direction of the probe.

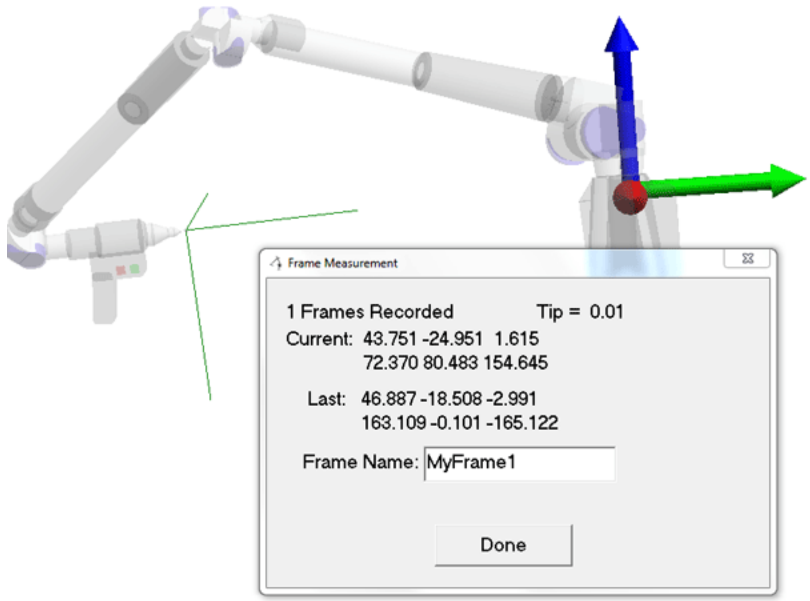


Figure 15-19. Measuring a frame with the arm.

To specify a name for the constructed frame, fill in the *Frame Name* field. To measure a frame, press the assigned Record button. When finished measuring, press the assigned Accept or Cancel button.

Batch (Guided Points)

Batch, or Guided Points mode, guides the user to a series of selected points using dynamic zooming and audio cues. When entering this mode, you are first prompted to select the points to guide to (usually nominal points). Select these points in the order in which you'd like to measure them, then press **Enter**. Next, you will be presented with the ***Measure a Batch of Points*** dialog (Figure 15-20).

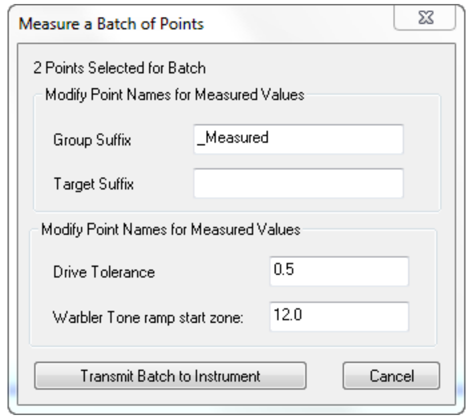


Figure 15-20. Settings for measuring a batch of points.

- **Group Suffix.** Attaches the specified suffix to the group of measured points.
- **Target Suffix.** Attaches the specified suffix to each measured target.
- **Drive Tolerance.** Specifies the radius around the current point that is considered to be "good" or "in-tolerance". When the probe is within this radius, it turns green and the tone reflects an in-tolerance position (see below image).
- **Warbler Tone ramp start zone.** Specifies the radius at which the audio tone begins to ramp up in pitch to signify that the probe is approaching the desired position.

When the desired parameters are entered, click Transmit Batch to Instrument to begin measurement. While in the actual measurement mode (Figure 15-21), press the assigned Record button to measure the point and the assigned Delete Last button to delete the previous point. Note that as the probe is moved closer to the desired position, the tone increases in pitch and the view zooms in closer to the reference point.

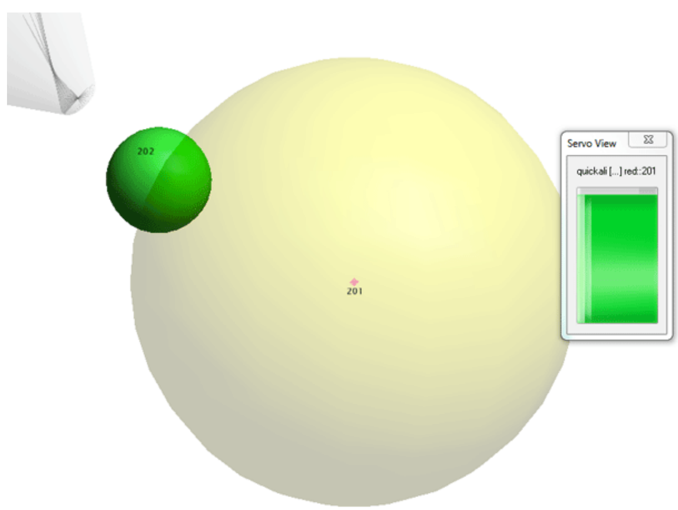


Figure 15-21. Measuring a batch of points.

When all of the batch points have been measured, this mode exits automatically. You can also exit the mode prematurely by pressing the assigned Accept or Cancel button.

Scanner

Scanner mode activates the laser line scanner on the arm, if equipped. Typically, the assigned Record button starts and stops scanning, while pressing the assigned Accept or Cancel button will end scanning mode. The specific display for this mode is dependent on the hardware being used to execute the scan.

Average Discrete Points

Average Discrete Points (Figure 15-22) mode creates an averaged point from a number of measured points. This mode is useful when you want to average out a series of potentially noisy measurements (perhaps from vibration or other issues).

When in this mode, press the assigned Record button to begin taking samples. The number of samples that are averaged to determine the ultimate measurement position is determined in the ***Edit Arm Settings*** window.

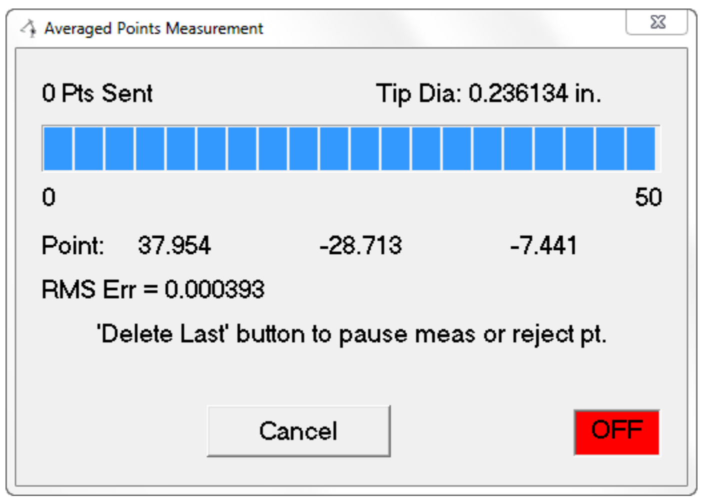


Figure 15-22. Measuring averaged points.

While taking samples, the assigned Delete Last button will pause sampling. Press the assigned Record button to resume. At the completion of the desired number of samples, the resulting point (and RMS error) will be displayed. Press the assigned Delete Last button to reject the point, the assigned Record button to take another measurement, and the assigned Accept button to complete the measurement mode.

Geometry Trigger

Geometry Trigger mode allows you to define planar, cylindrical, or spherical cross sections that, when crossed by the probe, trigger measurement. By default, these geometric triggers are not defined. You first need to define them by right-clicking on the Geometry Trigger button, which will take you to the ***Make Geometry Crossing Triggers*** dialog (Figure 15-23). The top of this dialog (separated into *Add Planes*, *Add Cylinders*, and *Add Spheres* sections) is where the geometry triggers are defined. The active geometry triggers are displayed in the list in the center of this dialog.

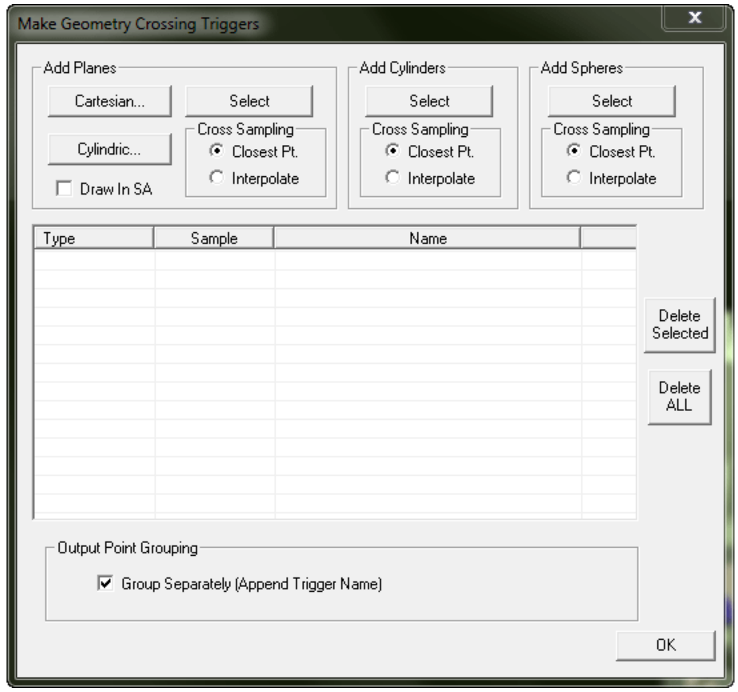


Figure 15-23. Defining geometry triggers.

Add Planes:

- **Cartesian.** Creates planar triggers (Figure 15-24) along the X, Y, and/or Z axes:

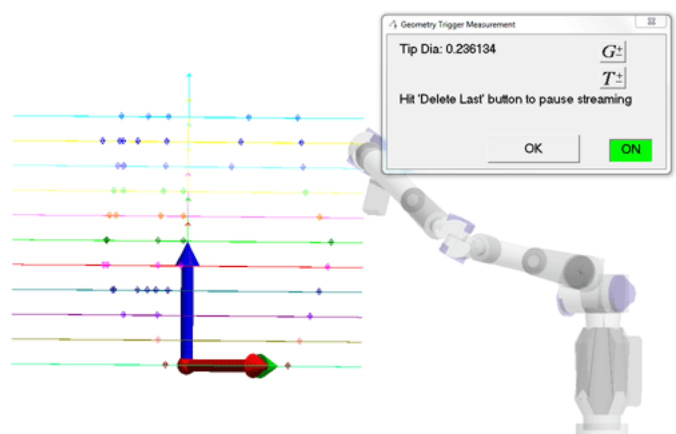


Figure 15-24. Planar geometry triggers.

After clicking this button, you are presented with the ***Cartesian Along Working Axes*** dialog (Figure 15-25).

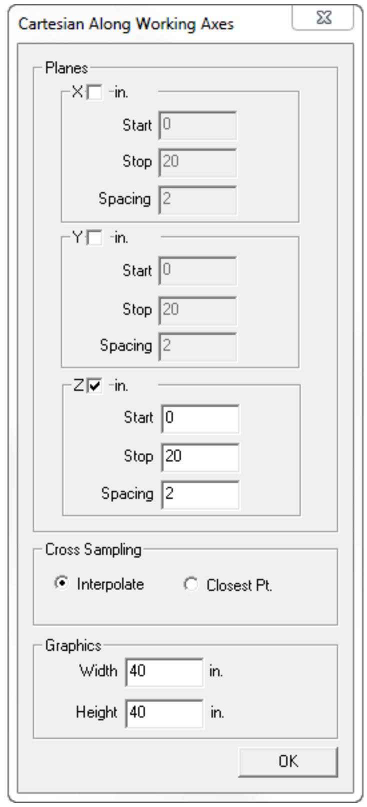


Figure 15-25. Defining the extent of planar geometry triggers.

In this dialog, check the axes along which you'd like to create planes, and also specify their start and stop position (in active units), as well as their spacing. In this example, planes will be placed along the active Z axis, from Z=0" to Z=20", spaced by 2" (see image above).

It is important to realize that, due to a number of limiting factors such as hardware sampling speed and computer speed, the sampled data will almost never be taken exactly on the geometry's boundary. In the Cross Sampling section, choose Interpolate to have SA interpolate between two sampled points on each side of the geometry trigger, and Closest Pt. to have SA use the closest sampled point to the trigger. The Interpolate option will result in a point lying directly on the geometry trigger (although it will not have been an actual sampled point), whereas Closest Pt. will be a true sampled point, but will likely not lie directly on the defined trigger.

You can also specify the width and height of the planes (for drawing purposes only) if they are going to be drawn in the graphical view.

- **Cylindric.** Creates planar triggers that pivot about the working frame's Z axis. Clicking this option brings up the ***Cylindric About Z in Working*** dialog (Figure 15-26). Here, you can define the start angle, stop angle, and angular spacing of the planes. Note that the X-axis defines the 0-degree plane for these values.

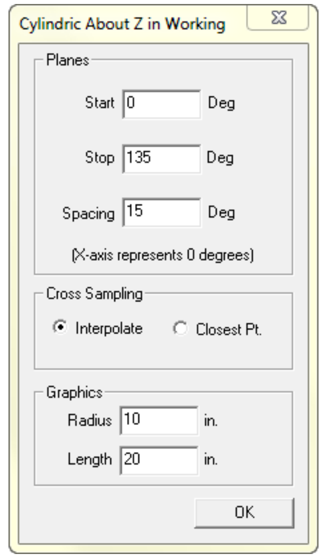


Figure 15-26. Defining cylindric geometry triggers.

It is important to realize that, due to a number of limiting factors such as hardware sampling speed and computer speed, the sampled data will almost never be taken exactly on the geometry's boundary. In the Cross-Sampling section, choose Interpolate to have SA interpolate between two sampled points on each side of the geometry trigger, and Closest Pt. to have SA use the closest sampled point to the trigger. The Interpolate option will result in a point lying directly on the geometry trigger (although it will not have been an actual sampled point), whereas Closest Pt. will be a true sampled point, but will likely not lie directly on the defined trigger.

Figure 15-27. Cylindrically oriented planar triggers.

You can also specify the radius (width) and length of the planes (for drawing purposes only) if they are going to be drawn in the graphical view. The settings in Figure 15-26 were used to create the trigger planes in Figure 15-27.

- **Select.** Allows you to add an existing plane to the list of triggers by clicking in the treebar or the graphical view.

- **Cross-Sampling.** This applies to triggers added via the Select method (see previous). It is important to realize that, due to a number of limiting factors such as hardware sampling speed and computer speed, the sampled data will almost never be taken exactly on the geometry's boundary. Choose Interpolate to have SA interpolate between two sampled points on each side of the geometry trigger, and Closest Pt. to have SA use the closest sampled point to the trigger. The Interpolate option will result in a point lying directly on the geometry trigger (although it will not be an actual sampled point), whereas Closest Pt. will be a true sampled point, but will likely not lie directly on the defined trigger. Note that the desired option must be chosen prior to selecting the plane.
- **Draw In SA.** This option will physically create the trigger planes in the job upon clicking the OK button for the ***Make Geometry Crossing Triggers*** dialog.

Add Cylinders

This section is used to add cylindrical geometry triggers.

- **Select.** Allows you to add an existing cylinder to the list of triggers by clicking in the treebar or the graphical view.
- **Cross-Sampling.** It is important to realize that, due to a number of limiting factors such as hardware sampling speed and computer speed, the sampled data will almost never be taken exactly on the geometry's boundary. Choose Interpolate to have SA interpolate between two sampled points on each side of the geometry trigger, and Closest Pt. to have SA use the closest sampled point to the trigger. The Interpolate option will result in a point lying directly on the geometry trigger (although it will not be an actual sampled point), whereas Closest Pt. will be a true sampled point, but will likely not lie directly on the defined trigger. Note that the desired option must be chosen prior to selecting the cylinder.

Add Spheres

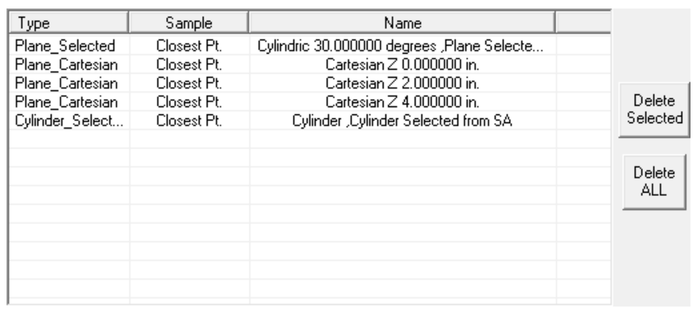
This section is used to add spherical geometry triggers.

- **Select.** Allows you to add an existing sphere to the list of triggers by clicking in the treebar or the graphical view.
- **Cross-Sampling.** It is important to realize that, due to a number of limiting factors such as hardware sampling speed and computer speed, the sampled data will almost never be taken exactly on the geometry's boundary. Choose Interpolate to have SA interpolate between two sampled points on each side

of the geometry trigger, and Closest Pt. to have SA use the closest sampled point to the trigger. The Interpolate option will result in a point lying directly on the geometry trigger (although it will not be an actual sampled point), whereas Closest Pt. will be a true sampled point, but will likely not lie directly on the defined trigger. Note that the desired option must be chosen prior to selecting the sphere.

Trigger List

The ***Make Geometry Crossing Triggers*** dialog displays a list of the currently active triggers (Figure 15-28). You can single or multi-select active triggers, then remove them from the trigger list using the Delete Selected button. Or, you can delete all active triggers using the Delete ALL button.



Type	Sample	Name
Plane_Selected	Closest Pt.	Cylindric 30.000000 degrees ,Plane Selecte...
Plane_Cartesian	Closest Pt.	Cartesian Z 0.000000 in.
Plane_Cartesian	Closest Pt.	Cartesian Z 2.000000 in.
Plane_Cartesian	Closest Pt.	Cartesian Z 4.000000 in.
Cylinder_Select...	Closest Pt.	Cylinder ,Cylinder Selected from SA

Figure 15-28. A list of the defined geometry triggers.

- **Group Separately (Append Trigger Name).** When this option is checked, measured points will be separated into point groups based on their respective triggers (Figure 15-29). The trigger name will be appended to the end of the group name as appropriate. Once the geometry triggers have been defined, click OK to return to the interface. At this point, left-clicking the Geometry Trigger button will take you into the measurement mode. Pressing the assigned Record button will begin (or continue) streaming points, and only those triggered by the geometry triggers will be recorded. To pause streaming, press the assigned Delete Last button. To exit measurement mode, press the assigned Accept or Cancel button.

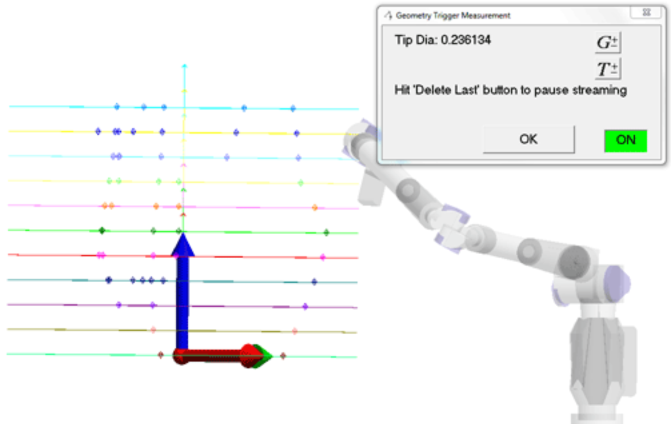


Figure 15-29. Separately grouped measurements on different geometry triggers.

Cone

Cone mode (Figure 15-30) creates a best-fit cone to a series of measured points. Once at least 12 points are measured, the Cone window displays statistics for the fit and the resulting cone. You can remove the previously-measured point from the fit by pressing the assigned Delete Last button.

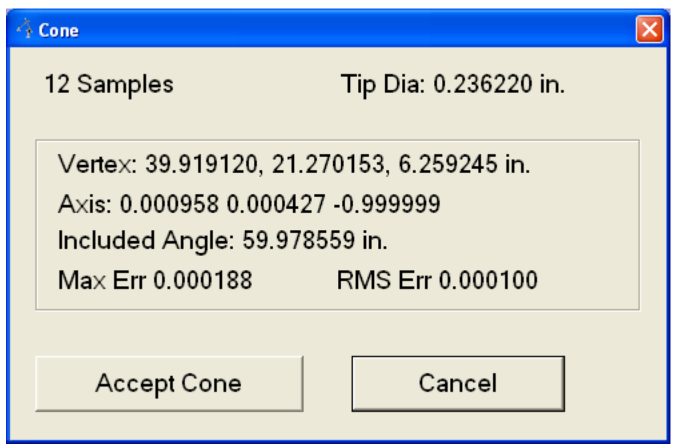


Figure 15-30. Cone measurement mode.

Fit parameters (Figure 15-31) can be accessed by right-clicking on the Cone icon.

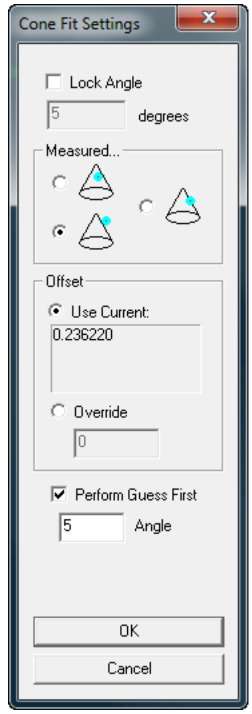


Figure 15-31. Fit settings for a cone.

- **Lock Angle.** Allows you to specify the apex angle and lock it, so that the resulting cone matches the specified angle.
- **Measured.** Indicate whether you measured the outside of the cone, inside of the cone, or have zero offset.
- **Offset.** Use the current probe offset, or override it with a custom value.
- **Perform Guess First.** When checked, as each additional point is measured the fitting algorithm will re-evaluate all of the measured points. If unchecked, the initial guess parameters will not necessarily be re-evaluated as additional points are measured. In general, leave this option checked, unless computation times become unreasonable.
- **Angle.** Define the apex angle used for the initial cone fit guess.

When finished measuring points, press the assigned Accept button or click the Accept Cone button. To exit the dialog, press the assigned Cancel button.

SA ArmPad

The SA ArmPad (Figure 15-32) is a patent-pending input method that takes advantage of a calibrated pad (like a mousepad) that can be placed on any work surface. Once calibrated, you can tap any function on the pad with a probing device, instantly performing an action or entering the selected measurement mode. You can then perform various interactive actions with SA without using the computer or setting down the arm/probe.

By measuring three calibration points (A, B, & C below), SA will recognize the position and orientation of the pad, as well as which pad is being used. (Multiple variations of ArmPads, each with different functionality, are possible). As long as the pad is not moved relative to the instrument, its functions can be activated.

Figure 15-32. The SA ArmPad.

Calibrating the ArmPad is a straightforward process. It only needs to be performed once for a given instrument/pad orientation. That is, as long as the relative position between the instrument and the pad remain fixed, there is no need to recalibrate. Calibration results are stored with the instrument interface persistence files, so if an interface is shut down and restarted, the calibration is stored and recalled.

To Calibrate the ArmPad:

1. In the instrument interface (but without any measurement modes active), click the SA ArmPad button, or hold button 2 on the arm for a brief moment.

Figure 15-33. The ArmPad calibration dialog.

2. The ***ArmPad Calibration*** dialog will appear (Figure 15-33). Follow the prompts and touch the probe to each calibration point on the pad, measuring a single point at each.

Contact NRK Support to request a free pad. Or, click the Print Pad button to bring up a PDF of the pad which you can print out yourself on a letter-sized sheet of paper to use. (Note that the PDF should be printed at full scale, ensuring Page Scaling is set to NONE.

The Cancel and Done buttons have various uses (which are intuitive) based on context--they are equivalent to the Cancel or Accept buttons on the arm itself. The Prev and Next buttons have several meanings depending on context as well. For example, they switch to the previous or next feature when performing a GD&T Inspection, and they move

to the previous or next relationship when trapping measurements.

To Select a Measurement Mode Using the ArmPad:

- With the arm interface running, tap the probe onto the desired measurment mode on the pad. (Note: you do not need to press a button to select the mode).

To Change Views Using the ArmPad:

- With the arm interface running, tap the probe to the Top, Front, Side, or Oblique boxes on the pad.

To Autoscale the View Using the ArmPad:

- With the arm interface running, tap the probe to the Autoscale button (similar to the button) on the pad.

To Delete the Last Measured Point Using the ArmPad:

- With the arm interface running, tap the probe to the Delete Last Point button on the pad.

Edit Arm Settings

To access the SA Arm interface options, press the Edit Arm Settings button in the bottom left-hand corner of the interface. This takes you to the ***Options*** dialog (Figure 15-34).

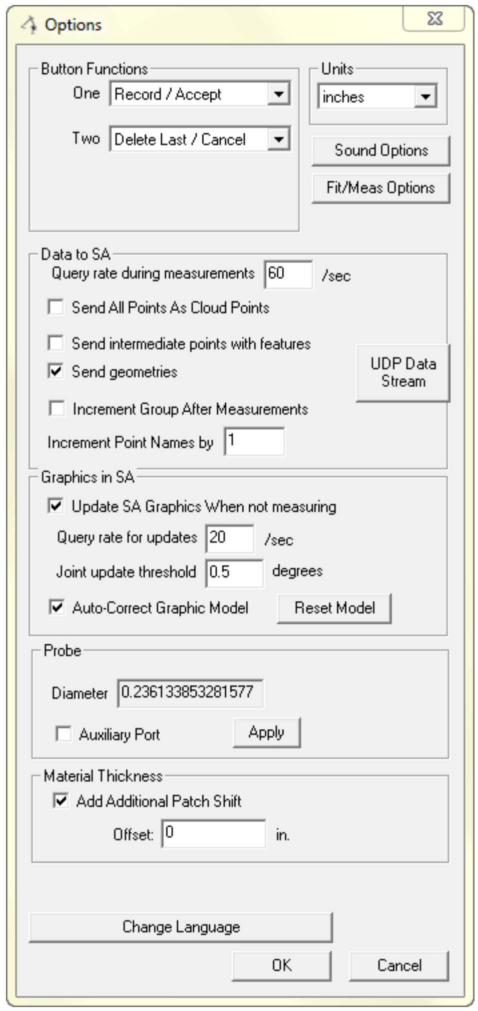


Figure 15-34. The Arm interface options dialog.

Button Functions

Arm buttons 1, 2, and 3 (if available) can be set to perform a series of functions:

- **None.** Disables the associated button so it performs no action.
- **Record/Accept.** The associated button will record a point or begin streaming points if pressed momentarily, and accept a dialog, accept geometry creation, or close a measurement mode if held for a few seconds.
- **Delete Last/Mouse Toggle.** The associated button will delete

the last point or pause streaming if pressed momentarily, and if held for a few seconds, will toggle the use of the arm to control the mouse cursor.

- **Delete Last/Cancel.** The associated button will delete the last point or pause streaming if pressed momentarily, and cancel a dialog, cancel geometry creation, or close a measurement mode if held for a few seconds.

Units

Instrument interfaces have unit settings in addition to SA itself. Choose the active units (inches, meters, millimeters, centimeters, or feet) in this dropdown.

Sound Options

Clicking the Sound Options button will bring up the ***Sound Properties*** dialog (Figure 15-35).

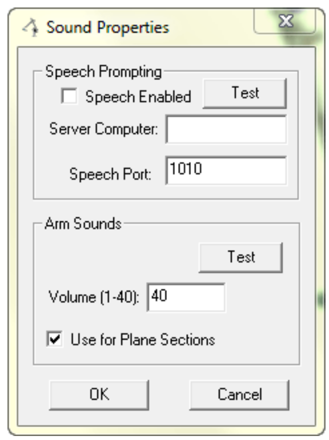


Figure 15-35. The Sound Properties dialog.

Speech Prompting

- **Speech Enabled.** If the SA Speech Server is running, this option will add speech prompting to the arm interface.
- **Test.** This button can be used to verify that the speech feature is operating correctly.
- **Server Computer.** The IP address of the computer running the speech server (leave blank if running on the current computer).
- **Speech Port.** The network port used by the speech server (set in the Speech Server's settings).

Arm Sounds

- **Test.** Play a series of arm sounds to check the volume setting and verify that arm sounds are audible.
- **Volume (1-40).** Sets the volume of the arm's speaker (if applicable). If the arm is not equipped with an internal speaker, the

computer's speaker will be used instead, and this setting will have no effect.

- **Use for Plane Sections.** When enabled, the arm will sound when the probe crosses a plane section or geometry trigger that results in a measurement.

Fit/Meas Options

The Fit/Meas Options button brings up a dialog (Figure 15-36) where you can specify a series of options related to arm measurement and geometry fitting.

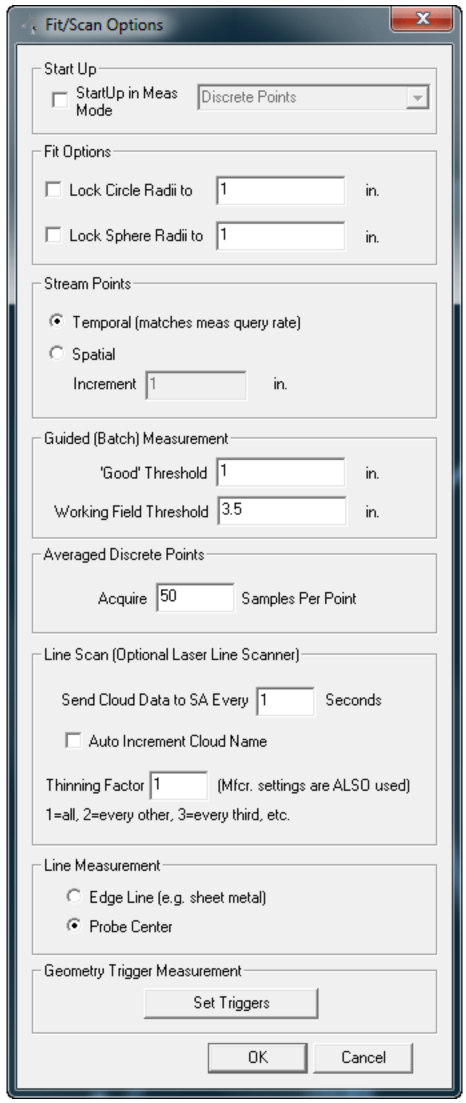


Figure 15-36. The Fit/Scan options dialog.

Start Up

- **Startup In Meas Mode.** Check this option and choose a measurement mode to have the interface automatically enter a specific measurement mode after being started.

Fit Options

- **Lock Circle Radii to.** Check this option and enter a radius to lock the resulting circle from a circle fit to a specific radius.
- **Lock Sphere Radii to.** Check this option and enter a radius to lock the resulting sphere from a sphere fit to a specific radius.

Stream Points

- **Temporal.** When in Stream Points mode, points will be recorded at the rate specified under the Arm Settings in the Query Rate During Measurements field. Note that due to hardware, network, and other limitations, the rate will not match the requested rate exactly.
- **Spatial.** When in Stream Points mode, points will be recorded spaced by the specified increment. Note that due to hardware, network, and other limitations, the spacing will not match the requested increment exactly.

Guided (Batch) Measurement

- **'Good' Threshold.** Set the default radius for measurements to be considered "good" (in the "green zone").
- **Working Field Threshold.** Set the default radius in which the mode begins zooming and emitting an audio cue.

Averaged Discrete Points

- **Acquire X Samples per Point.** Specify the number of samples that will be averaged to produce the resulting measured point while in Average Discrete Points mode.

Line Scan (Optional Laser Line Scanner)

- **Send Cloud Data to SA Every X Seconds.** Point cloud data acquired from scanners is sent to SA in batches. This option allows you to control how frequently those cloud point batches are sent to SA.
- **Auto Increment Cloud Name.** When selected, upon completion of a scan, the point cloud name will automatically be incremented for the next set of scan data.
- **Thinning Factor.** This setting is applied in addition to the manufacturer's thinning settings in order to determine what portion of the scanned points are actually sent to SA. For example, if the manufacturer's settings are set to send every 2nd point, and the setting here is to send every 2nd point, then SA will receive every 4th point.

Line Measurement

- **Edge Line (eg sheet metal).** This option is used for measuring an edge, typically of a thin plate (such as sheet metal) when in Line measurement mode. For this mode, you first define the plane of the flat surface whose edge you'd like to measure. Then, start measuring and slowly drag the probe tip along the surface until it begins to fall off the edge. This will define one end of the line. Repeat on the other end of the edge, and a line will be created. It is important to move the probe tip very slowly (depending on the measurement query rate) and to ensure that the probe remains in contact with the surface at all times.

- **Probe Center.** When in Line measurement mode, the line is fit to the probe center points.

Geometry Trigger Measurement

- **Set Triggers.** This is equivalent to right-clicking on the Geometry Trigger measurement mode button, and allows you to define geometry triggers.

Data to SA

- **Query Rate During Measurements.** Specify the rate at which the instrument is polled for data while measuring. This controls, for example, the rate at which measurements are recorded during the temporal Stream Points measurement mode, and the rate at which data is transmitted for UDP Broadcasts. Keep in mind that the requested data rate is limited by hardware, network, and other restrictions.
- **Send All Points as Cloud Points.** All measured points will be sent to SA as cloud points instead of measured targets.
- **Send Intermediate Points with Features.** For measurement modes that create geometry, if checked this option will also send the measured points that created the geometry. By default, these points are not sent.
- **Send Geometries.** If disabled, geometries resulting from measurement modes that create geometry will not be sent to SA.
- **Increment Group After Measurements.** When selected, the group name will increment upon the completion of a measurement mode.
- **Increment Point Names By X.** Each subsequent measured point will be incremented by the specified amount.

UDP Data Stream

The UDP Data Stream feature (Figure 15-37) enables the instrument interface to broadcast real-time probe position and orientation over the User Datagram Protocol (UDP). This allows the interface to send this data over a network without requiring a specific data connection to be established beforehand. This means that other clients on the network can be set up to receive this data and, perhaps, do something useful with it.

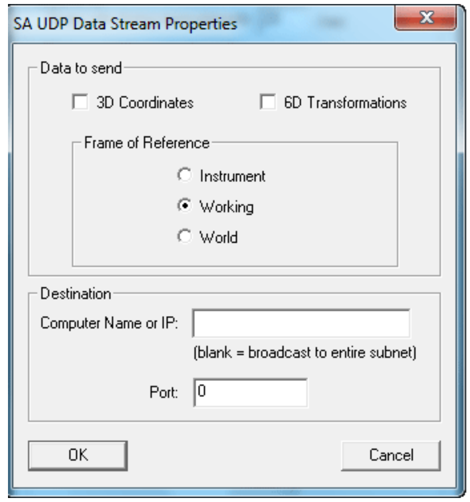


Figure 15-37. The arm interface's UDP data stream properties dialog.

For example, you might set up UDP data streaming to broadcast real-time 6D probe information to a custom client application that is listening on the proper port for UDP data packets. This client application could take this 6D information and process it, performing a useful task (such as using the 6D information to control a motorized pan/tilt head camera mount).

- **Data to Send.** Choose whether to send 3D coordinates (X, Y, and Z probe position) or 6D Transformations (X, Y, Z, and probing vector). You must also choose whether to send this data relative to the instrument, working, or world coordinate frame.
- **Destination.** Enter the network computer name or IP address to target for the UDP broadcast, or leave it blank to broadcast to the entire local network. You must also specify a port number to target.

To test the data being transmitted via UDP packets, use the SpatialAnalyzer UDP Monitor (available in the SA Install directory as `SpatialAnalyzer UDPMonitor.exe`. Set it up with the appropriate port, then click the Start button to view the data as it arrives over the network port, as seen in Figure 15-38.

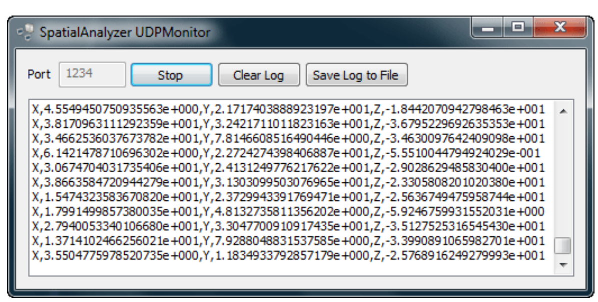


Figure 15-38. The SpatialAnalyzer UDP Monitor.

Graphics in SA

- **Update SA Graphics When Not Measuring.** When enabled, the depiction of the arm in the graphical view will update, even when a measurement mode is not active.
- **Query Rate for Updates.** The rate at which graphical updates will occur when not measuring. While the interface will ask the instrument for this rate, the actual update rate is limited by hardware, network, and other restrictions.
- **Joint Update Threshold.** If a joint encoder has not rotated by the angle specified in this field, the joint's graphical view will not update. Larger values will result in less frequent updates.
- **Auto-Correct Graphic Model.** Due to the kinematics involved, and that fact that some manufacturers do not have fixed encoder orientations (or they do not expose this information), sometimes the model of the arm in the graphical view may appear to be oriented incorrectly. When this option is enabled, the interface will attempt to automatically correct the kinematics of the model of the arm in the graphical view. This option is exclusively for graphical purposes, and has no effect on the accuracy of the actual measurement data coming from the instrument.
- **Reset Model.** If the model continues to be incorrect, click this button to attempt to reset the graphical representation.

Probe

- **Diameter.** You can override the arm's probe diameter by entering a different value in this field (generally NOT recommended). Click the Apply button to apply the changes.

Material Thickness

- **Add Additional Patch Shift.** Select this option to add an additional thickness for the Patch (Projected Pt.) measurement mode. This can be used, for example, to account for extra material thickness on a measured surface.
- **Offset.** Enter the desired patch shift offset amount in this field. A positive offset implies that there is extra material on the measured part.

Choose Language

This button is used to load or create a translation file for the arm interface. See Language Translation for more details.

Arm Calibration

This will display hardware-specific calibration routines and settings. The options provided by this button are strictly controlled by the instrument manufacturer.

Toggle Arm Buttons

Press this button to toggle the arm buttons on and off. Toggling the arm buttons off will render them inoperable until toggled back on again.

Blue Question Mark

This button will display the MP string commands that can be used to set the arm measurement modes or apply additional settings via Measurement Plans.

Instrument Simulation

In simulation mode, you can simulate the different measurement modes, as well as record points to simulate a measurement process. When simulating, you will come across a series of buttons in various parts of the interface (Figure 15-39).

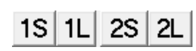


Figure 15-39. The simulation buttons.

These buttons refer to "First button, short", "First button, long", "Second button, short" and "Second button, long" and refer to simulating pressing the arm buttons momentarily or pressing and holding them.

Instrument Index Number

At the very bottom of the interface is the collection, instrument index number, and name of the active instrument. This index number (which for a given collection begins counting from zero) is used in several places in SA--one of which is Measurement Plans.

Instrument Toolbar

The instrument toolbar is a simpler, easier to learn and use interface for arm measurement than the full arm interface. It is well suited for performing simple point measurement and scanning tasks, for new users, and when performing inspection routines using the SA Toolkit. It does not have any of the geometry measuring routines that are found in the traditional arm interface.

The instrument toolbar is displayed in Figure 15-40. It is accessible from the main arm interface by clicking the icon. (Similarly, the icon switches back to the standard interface).

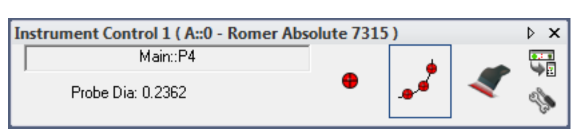


Figure 15-40. The arm instrument toolbar.

Setting the Point Name

To explicitly set the point name for the next measured point, click the text field above the probe diameter listing. This will call up the ***Instrument Interface Point Naming*** dialog (Figure 15-41), which allows you to specify the collection, group, and target names. If performing an inspection routine, the SA toolkit will name your data for you.

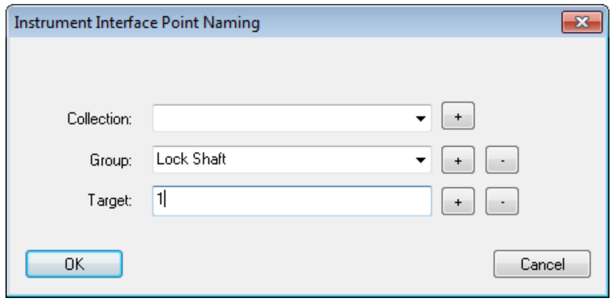


Figure 15-41. Naming a point in the arm toolbar.

Measurement Modes

There are three measurement modes available to you in the toolbar: single point measurement, stream measurement, and scanning mode (Figure 15-42).

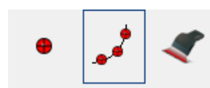


Figure 15-42. (Left to right) Single point mode, stream mode, and scanning mode.

Click the single point or streaming measurement mode to activate it. For arms that have a scanner device with a switch attached, flip the switch to activate scanning mode. Some instruments may require you to click the scanning measurement mode button.

Measuring In The Toolbar

The following buttons control measurement using the arm toolbar.

- **Button 1.** A short press moves to the next feature in the SA Toolkit, whereas a long press iterates the measurement mode.
- **Button 2.** A short press records a measurement, whereas a long press deletes the last measurement.
- **Button 3.** A short press advances to the next feature in the SA Toolkit, whereas a long press iterates the measurement mode.

Checks/Utilities

By clicking the icon, you can access calibration, scanner, and arm settings (these are the same destinations as found in the traditional arm interface).

Alignment & Transformations

CHAPTER 16

One of the key areas of portable metrology is alignment: transforming an instrument into the proper position and orientation such that it is aligned to the desired coordinate system. This chapter explores methods for transforming into a desired coordinate system.

Typically, metrology data is of limited use until it has been aligned to some set of reference datums, nominal data, or other measurements. Aligning to a coordinate system is sometimes referred to as *bucking in*, *tying in*, *locating*, or *aligning*. Some of the reasons this is done include the following:

- Governing dimensions and measurements (acceptance criteria) are typically expressed on drawings with respect to a specific datum or frame of reference. Often, evaluating features with respect to other datums is not permitted when evaluating design conformance.
- When comparing as-built measurements to a nominal design, the data must be aligned to the nominal design—otherwise, the comparison cannot be performed.
- Multi-station measurements (measurements taken from more than one position in space) must be "tied" together so that the data is continuous.
- Digital alignment and assembly (virtual fit-up) requires that measured components be "virtually" assembled using some sort of alignment in order to check the assembly for fit and function.

In some situations, this involves creating a coordinate frame from the measured points. In other (less common) situations, it involves transforming CAD data to the measurements. But most often, it involves aligning measurements to a coordinate system of interest. Since measurements are rigidly fixed to the instrument that measured them, this typically involves transforming an instrument in space until its measurements end up in the desired position.

In this chapter, we will explore the tools available for performing all different types of alignment.

Best-Fit Transformations

Best-fit transformations are widely used in coordinate metrology to consistently and confidently align measurements into a coordinate frame that is meaningful to a user. Measurements made with an instrument should be presented to users in the part coordinate frame—that is, aligned to the part. The best-fit transformation routine in SpatialAnalyzer is one way to do that.

A best-fit transformation solves for the difference between a group of points and the expected (or reference) coordinates for those points. The difference between the current position and orientation of the points and the ending position/orientation is referred to as the *transformation*. After the solution is reached, SpatialAnalyzer shows you the difference and allows you to apply the transformation to your measurements of the points and to other objects as well. Once applied, SpatialAnalyzer is able to report those measured points and any new measurements in a part coordinate frame that is meaningful to you.

The best-fit alignment technique has many applications; for example, to combine sets of coordinate data or locate an instrument in the part coordinate system using surface features imported from a CAD system. Simply stated, this alignment technique helps you determine the transformation that when applied to one set of coordinates causes the match-up between the two sets to be ideal.

The term "best fit" is the colloquial term given to a mathematical procedure known as a *Linear Least Squares* fit. The transformation solved for and applied is determined using least-squares optimization methods. A least squares optimization seeks to minimize the square of the distance between corresponding points, or between points and corresponding surfaces.

Point to Point Best-Fit

A common need for performing a point to point best-fit is when trying to locate to a known coordinate system in which discrete points already exist. For instance, you have a network of control points (fiducials) from a previous station measurement and you'd like to relocate the instrument and remeasure these points in order to align back into the coordinate system. When performing this type of alignment, the group of points being aligned *to* are referred to as the *reference group*, and the points being moved are referred to as the *corresponding group*. They may also be referred to as the *nominal group* and *measured group*, respectively, although if the term *nominal group* is being used, it does not necessarily mean that the points are true nominal design points (Figure 16-1).

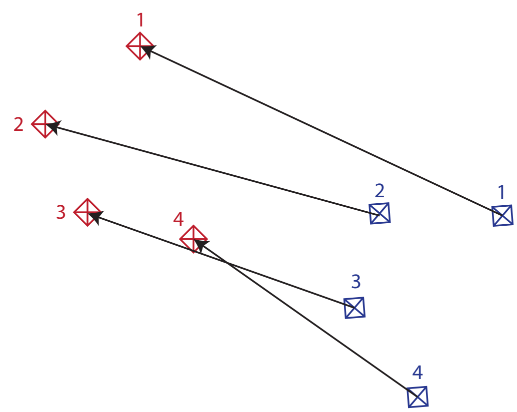


Figure 16-1. A group of four points (blue), when best-fit to a set of reference points (red), will move as illustrated. The four blue points move together as a rigid body to try to align with the reference points. In this case, a rotation is involved.

In order to perform a point-based best-fit transformation, the points involved must meet two prerequisites:

- There must be at least 3 points in both the reference and corresponding groups that have corresponding names.
- The reference and corresponding points must be in separate point groups.

Note that the two groups need not have exactly the same number of points. Any points that don't have corresponding pairs are simply ignored in the command, although if they are in the moving point group they will be transformed along with the rest of the points in the group. There are three ways to perform a point-to-point best fit transformation:

- **Instrument-based best-fit.** In this method, you pick an instrument to align, and then select the reference and measured group. The transformation will be applied to the instrument, bringing all of the measured points along with it.
- **General best-fit.** This is nearly identical to the first method, however it's not instrument-centric, and provides a little more flexibility when needed. You can choose to transform anything, including instruments, point groups, and other objects. There are also a few additional options (discussed below).
- **Locate an instrument by measuring nominal points.** This al-

lows you to measure a set of nominal points, and let SA handle the point naming and alignment for you.

These three methods are described below, but to use them (well, at least two of them) we first need to understand the ***Best-Fit Transformation*** dialog.

The Best-Fit Transformation Dialog

The ***Best-Fit Transformation*** dialog (see Figure 16-2) is a simple interface allowing a high degree of control over the fit. Here you can review the fit, apply tolerances, restrict degrees of freedom, edit point constraints, and more. Any time one or more changes is made, the current fit is no longer valid (indicated by a dark blue background on the fit button and the fit results) and needs to be re-solved. The dialog is divided into three main parts: the controls (top left corner), the fit results (top right corner), and the point list (bottom).

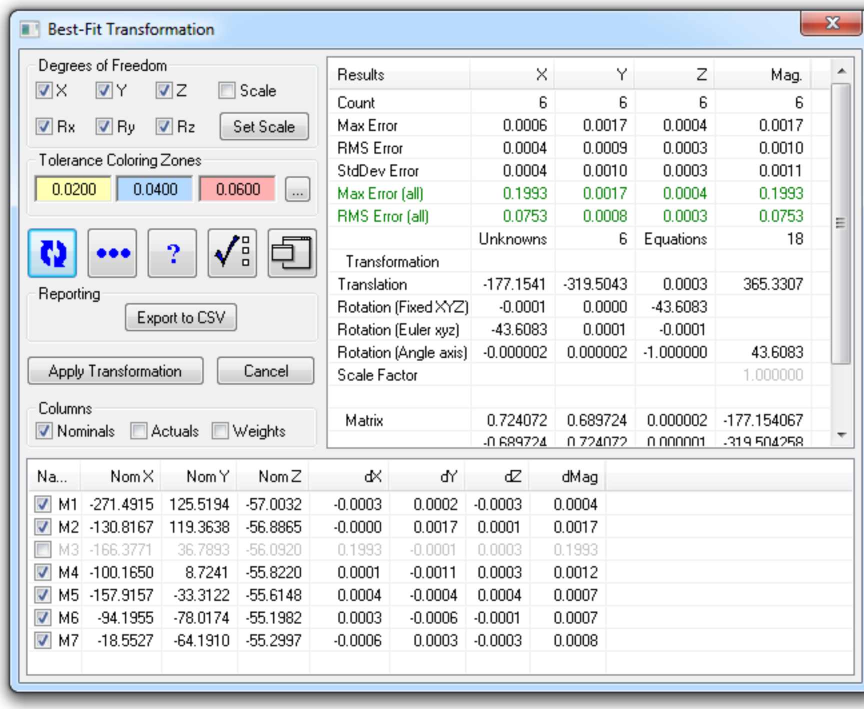


Figure 16-2. The ***Best Fit Transformation*** dialog.

Fit Results

The fit results are listed in the table in the top right corner of the dialog. These results will be invalid (and therefore will have a dark blue background) whenever the best-fit solution has either not been solved or a change has been made since it was last solved. The fit results table contains, on a component-by-component basis, a number of statistical values related to the fit that help you determine how well the fit performs. These values include:

- **Count.** The number of points included in the fit for the specified component.

- **Max Error.** The maximum error between a pair of corresponding points. This is an indicator of the "worst fit" for any pair of corresponding points.
- **RMS Error.** The root-mean-square error of the fit as a whole. This is a good indicator of the overall error of the fit. This number is always less than or equal to the max error (although usually it's noticably smaller than the max error).
- **StdDev Error.** The standard deviation of the errors. A small value indicates that each pair of points had similar errors. A large value indicates that one or more error values differed significantly from the group as a whole.
- **Max Error (all).** This is the same as **Max Error**, except all points are included (even those which are deselected from the fit).
- **RMS Error (all).** This is the same as **RMS Error**, except all points are included (even those which are deselected from the fit).
- **Translation.** The x, y, and z instrument translation needed for the solution.
- **Rotation (Fixed YZ).** The instrument rotations required for the solution, in Fixed XYZ form.
- **Rotation (Euler XYZ).** The instrument rotations required for the solution, in Euler XYZ form.
- **Rotation (Angle axis).** The instrument rotation required for the solution, in Angle axis form (first three values describe the rotation axis, and the last value describes the rotation angle).
- **Scale Factor.** The scale factor applied for the fit. This will be 1.0 unless the **Scale** degree of freedom is checked (allowed to float).
- **Matrix.** This is the 4x4 transformation matrix that will be applied to the instrument.

Point List

The point list is a table that contains all points that were found to have matching pairs between the reference and corresponding groups. Immediately above this list, in the *Columns* section, are three checkboxes: **Nominals**, **Actuals**, and **Weights**. Checking any of these boxes will display the corresponding values in the table. Any column can be sorted in ascending or descending order by clicking the column header. The columns in the point list are described here:

- **Name.** The name of the point.
- **Nom X.** The nominal x-coordinate of the point. This is the x-coordinate of the reference (non-moving) point.

- **Nom Y.** The nominal y-coordinate of the point. This is the y-coordinate of the reference (non-moving) point.
- **Nom Z.** The nominal z-coordinate of the point. This is the z-coordinate of the reference (non-moving) point.
- **Act X.** The actual x-coordinate of the point. This is the x-coordinate of the measured (moving) point.
- **Act Y.** The actual y-coordinate of the point. This is the y-coordinate of the measured (moving) point.
- **Act Z.** The actual z-coordinate of the point. This is the z-coordinate of the measured (moving) point.
- **Wt X.** The fit weight for the x-coordinate of the point.
- **Wt Y.** The fit weight for the y-coordinate of the point.
- **Wt Z.** The fit weight for the z-coordinate of the point.
- **dX.** The current x-deviation between the reference and corresponding point.
- **dY.** The current y-deviation between the reference and corresponding point.
- **dZ.** The current z-deviation between the reference and corresponding point.
- **dMag.** The magnitude of the deviation between the reference and corresponding point.

Each point in the fit has a checkbox next to its name in the point list. Deselecting this checkbox will remove the point from the fit, which means that it will have no influence in the final solution. It is standard procedure to remove erroneous points from the fit so as not to corrupt the overall solution. If a point is removed from the fit, it will be grayed out in the point list.

Best-Fit Item Dialog

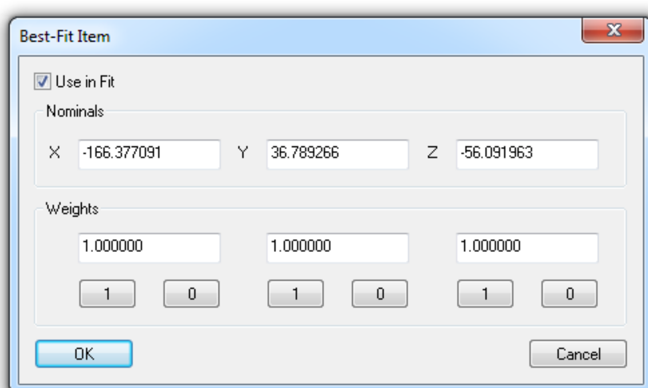


Figure 16-3. The ***Best-Fit Item*** dialog.

Double-clicking a point in the list will bring up the ***Best-Fit Item*** dialog (Figure 16-3). This dialog provides another way to remove the point from the fit (**Use in Fit**), and conveniently allows you to change the reference point's coordinates, as well as the per-component fit weight for the point. The 1 and 0 buttons are simply shortcuts for setting the corresponding fit weight value.

Fit Weights

Fit weights give you control over defining whether a certain component of a point has any influence in the fit process. A value of zero indicates that the component has no influence in the fit, whereas a value of 1 indicates the component has full influence. Any intermediate value may be chosen. If a component is weighted to zero, it is displayed in gray in the point list. If an intermediate value is chosen, it is displayed in blue, and if the weight is set to 1, it is displayed in black.

One reason for reducing the weight of a specific component of a point is to reduce directional errors. Suppose you have a measured point that was performed on a floor that was flexing in the z-direction during measurement. It is reasonable to assume that the measured point has significant error in the z direction when compared to the x or y direction, therefore reducing the weight of the z component may be a wise choice. Another reason is when using discrete points to control specific degrees of freedom, an advanced technique that, while doable using best-fit, is much easier to perform using relationship fits.

Controls

You'll notice that you have a significant amount of control in the fit. Access to changing the fit parameters is provided via the controls in the upper-left hand corner of the dialog.

Degrees of Freedom

When solving a best-fit transformation, you can specify that the transformation may only be permitted in certain unlocked degrees of freedom, relative to the working coordinate frame. You can lock any combination of X/Y/Z translation, X/Y/Z rotation, and scale. Any degree of freedom that is checked is unlocked—unchecking a degree of freedom locks it. Keep in mind that these degree of freedom restrictions are applied with respect to the working coordinate frame, therefore rotations are applied about the working frame origin.

As an example, suppose you have a bolt pattern measurement on a flange and you'd like to fit this measured pattern to a mating, nominal pattern. In this case, you would likely align the face of the measured points to the nominal flange face using perhaps a relationship fit, then you might wish to use the nominal bolt pattern to spin the measured flange about its axis until the bolt holes are aligned (Figure 16-4).

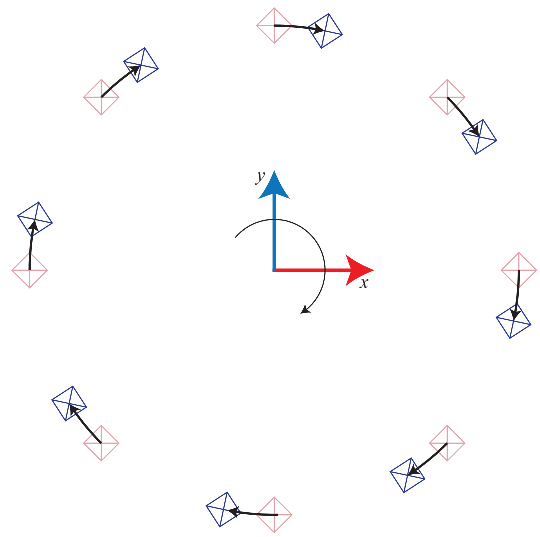


Figure 16-4. Performing a best-fit transformation allowing only rotation about the Z axis.

By locking all degrees of freedom except for rotation about the Z axis—and by placing the working frame at the center of the flange—the fit would simply clock the measured pattern to the nominals.

You might also restrict degrees of freedom if you have a leveled instrument (its axis is parallel to the Z axis) and you do not want it to rotate about the X or Y axes.

Notice that you can also set the instrument to a specified scale value for the fit, which allows you to apply temperature compensation to the measurements in situ.

Tolerance Coloring Zones

You can apply tolerance coloring to the deviations in the point list by setting the tolerances in the *Tolerance Coloring Zones* section.

Fit Controls

A series of buttons controlling the fit is displayed below the *Tolerance Coloring Zones* section:

- **Re-fit button.** Recalculates the best-fit transformation based on the current settings. A blue background on this button indicates that the current fit is not valid based on the current settings and a re-fit is needed.
- **Edit Selected Items.** Brings up the ***Best-Fit Item*** dialog (page 518) for any selected items. More than one point can be selected at one time in the point list. This button allows you to change all of their properties at once.
- **Quick Tips.** Displays a series of tips related to the dialog.
- **Advanced Settings.** Displays the ***Advanced Best-Fit Options*** dialog, discussed below.
- **Advanced View.** Toggles to a simplified version of the inter-

face, hiding less frequently-used parameters.

Advanced Settings

The ***Advanced Best-Fit Options*** dialog (Figure 16-5) provides some advanced, less commonly-used settings for best fit transformations.

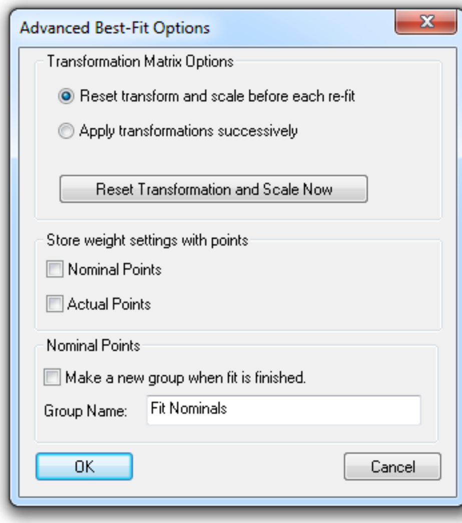


Figure 16-5. The ***Advanced Best-Fit Options*** dialog.

Descriptions of these parameters follow:

- **Reset transform and scale before each re-fit.** The transformation is solved from the same initial starting position and orientation each time the fit is performed. This is the default and is applicable to most fitting operations.
- **Apply transformations successively.** Each fit operation solves using the previous calculated transformation as a starting point. This allows you to apply iterative best-fit operations with different degree of freedom settings for each. For instance, you might wish to first solve the best-fit with some degrees of freedom locked, then solve again from that position with a different set of locked degrees of freedom.
- **Reset Transformation and Scale Now.** Resets the current transformation back to the "starting point".
- **Store weight settings with points.** Fit weights can be stored with nominal or actual points such that if the points are used in another best-fit operation, the weights are recalled.
- **Make a new group when fit is finished.** Makes a copy of the reference (non-moving) points and puts them into a separate group with the specified name at the completion of the operation. If nominal coordinates were changed in the ***Best Fit Item*** dialog, this group will use those updated coordinates. This is not a common need.

Export to CSV

The Export to CSV button will export the fit results to a comma-separated .CSV file and import it into Excel (if installed).

Instrument-Based Best-Fit

To reiterate, an instrument-based best-fit transforms just an instrument and (of course) its measurements. Only one instrument can be transformed. As such, it is most appropriate when locating a single instrument to a set of reference points, or when relocating an instrument to a new station.

1. Measure a set of points into a point group. These will be the "measured" points.
2. Ensure that the measured point names match the corresponding point names in the reference group.
3. From the menu, select **Instrument>Locate (Transform to Part)>Best-Fit**.
4. If there is more than one instrument in the job file, select the instrument to locate (as prompted).
5. When prompted for the "nominals" group, select the reference group. This is the group containing the points that will not move during the transformation.
6. When asked for the "measured" group, select the group containing the measured points. These points should be measured by the instrument selected in step 4.
7. Choose the desired options from the ***Best-Fit Transformation*** dialog and perform the fit as described, then click Apply Transformation.

Note: If your point names don't correspond, use **Edit> Rename> Points by Auto-Corresponding 2 Groups** to automatically rename them.

General Best-Fit

A general best-fit can be performed in which the calculated transformation is not simply applied to a single instrument that measured the corresponding points. This is useful for applying point to point best-fit transformations when instruments are not involved, or when you'd like to move several instruments together as part of the fit. In the general best-fit, you can apply transformations to instruments and other objects, in addition to a few more options.

To Perform a General Best-Fit Operation

1. Select **Analysis>Best-Fit Transformation>Points to Points**, or press `Ctrl + B`.
2. Select the reference (non-moving) group.

3. Select the corresponding (moving) group.
4. In the resulting ***Best-Fit Transformation*** dialog, select the desired options for the fit and solve it (see "The Best-Fit Transformation Dialog" on page 516).
5. In the ***Best-Fit Options*** dialog (Figure 16-6), select the desired options (described below), then click OK.

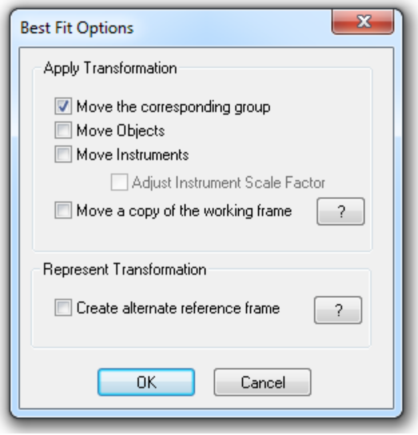


Figure 16-6. The options for a general best-fit.

Best-Fit Options

Toward the completion of a general best-fit operation, you are provided with a series of options that are not displayed in the single instrument version of the best-fit command. These options are described below.

- **Move the corresponding group.** Determines whether the transform is applied to the corresponding (moving) group. If unchecked, the corresponding group is not moved.
- **Move Objects.** If you'd like to transform other objects, select this option. After clicking OK, you'll be prompted to pick the additional objects to transform.
- **Move Instruments.** If you'd like to transform instruments, select this option. After clicking OK, you'll be provided the opportunity to select instruments to transform.
- **Adjust Instrument Scale Factor.** This option is only available when the scale degree of freedom is unlocked during the fit. When checked, the scale factor calculated during the fit operation is applied to the moving instruments.
- **Move a copy of the working frame.** When selected, the working frame will be copied, and the transform will be applied to the copy. This creates a frame that can be used to report the coordinates of the points as they were prior to the transformation.
- **Create alternate reference frame.** When selected, a frame is

created that represents the inverse of the applied transformation. This allows you to transform the moving objects back to their position prior to the best-fit operation through the use of a frame to frame transformation.

Locate By Measuring Nominal Points

While locating an instrument by fitting to a set of nominal points (see "Instrument-Based Best-Fit" on page 522) is not a difficult task, you may find that using **Instrument>Locate (Transform to Part)>Measure Nominal Points** is preferable for typical in-process alignment in which more control is not needed.

Locating by measuring nominal points provides a few benefits over the instrument-based best-fit method:

- Point naming is handled automatically, eliminating the potential for typographical errors in measured point names.
- If the instrument is roughly aligned to the points already (for instance, it is being re-fit into the reference system after being bumped), it can easily be automatically pointed at the targets (if the instrument supports pointing). Single points or the entire set of reference points can be automatically measured with a single button click.
- The quality of the fit is displayed at the time of measurement—therefore any blunders in measurement (resulting in errors in the fit) can be discovered much sooner than if the fit were being performed later in the process.

There are a few things to keep in mind when using this command:

- It must be used at the time of measurement. If the measured points already exist, you must use one of the other two best-fit commands.
- You have much less control over the fit. You cannot control degrees of freedom (with the exception of scale), weighting, or advanced fit operations—and you cannot remove points from the fit unless you delete them.
- You're presented with much less detailed information about the fit results.

For the first two measurements, insufficient measurements exist to perform the fit, so the delta deviations are not displayed. However, as each successive measurement is obtained, the fit is recalculated and the deviation values are updated. The ***Locate Instrument by Measuring Nominals*** dialog is presented in Figure 16-7.

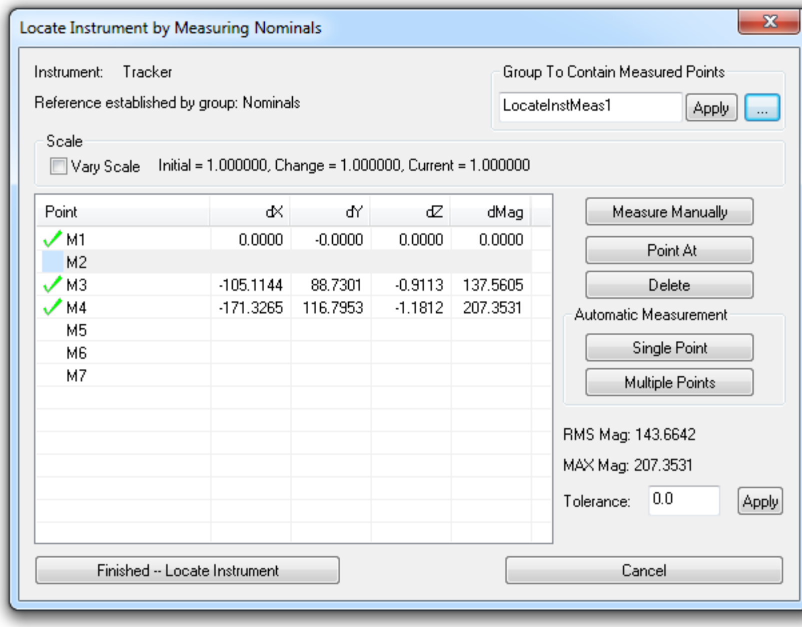


Figure 16-7. The ***Locate Instrument by Measuring Nominals*** dialog is useful when performing a best-fit points to points transformation during the measurement process.

A description of the options follows:

- **Group to Contain Measured Points.** This is the name of the group in which you'd like to measure the corresponding points. It is best to type this in before measuring any of the points, because if you change it after measuring any points, the already-measured points will not move into the new group. The Apply button will update the table based on the entered group, and the ellipsis button allows you to select an existing group to use.
- **Vary Scale.** When checked, scale is allowed to float during the fit. The initial, delta, and final scale values are listed next to this checkbox.
- **Measure Manually.** When clicked, the selected target in the list is measured. If the target already exists, an additional observation is added.
- **Point At.** Points the instrument at the selected target and attempts to lock on. (Applies only if the instrument supports pointing/target acquisition).
- **Delete.** Deletes the selected target.
- **Automatic Measurement - Single Point.** Attempts to point at, acquire, and measure the selected target.
- **Automatic Measurement - Multiple Points.** Attempts to consecutively point at, acquire, and measure all points in the list.
- **Tolerance.** You can specify a tolerance in this field. Any deviation magnitudes that exceed the tolerance will be highlighted red. Click the Apply button to apply the specified tolerance to

the table.

To Locate an Instrument by Measuring Nominals

1. Ensure the instrument interface is running.
2. From the menu, select **Instrument>Locate (Transform to Part)>Measure Nominal Points**.
3. If more than one instrument exists in the file, select the instrument to transform, as prompted.
4. Select the nominal (reference) points.
5. The ***Locate Instrument by Measuring Nominals*** dialog is displayed.
6. In the *Group to Contain Measured Points* section, type in a name for the point group in which to place the measured points.
7. Measure points manually or automatically using the dialog as described above. Once three or more points have been measured, click the Finished -- Locate Instrument button at any time to accept the fit and complete the command.

Best-Fit Points to Surfaces

Performing a linear least-squares fit between a set of points and one or more surfaces is a common task. Often, you want to know the deviation of an as-manufactured part or assembly from its perfect design condition, and it is common to have the nominal design represented in a CAD package as a set of surfaces. In cases where specific discrete reference datums for aligning measurements to the nominal design are not specified (or not desired), best-fit points to surfaces is one method for aligning your measurements to the reference system.

Best-Fit Points to Surfaces uses optimization methods to find the transformation that, when applied to the instrument (and therefore its points), minimizes the RMS error between the measurements and the surfaces, taking into account your specified constraints. Typically, the goal is to drive all the points (compensated for probe offset) to be exactly on the surface. SpatialAnalyzer extends that capability by allowing the operator to specify high and low tolerances for points and assign weights to those constraints.

Initial Condition

Like several other optimization methods, fitting points to surfaces

Basic Operation

For a standard best-fit, just leave the options (Figure 16-8) set to the defaults and then the points will be brought to the surface using a best-fit taking into account the target offsets of the points. Make sure all the surfaces you are using have their normals pointing in the right direction.

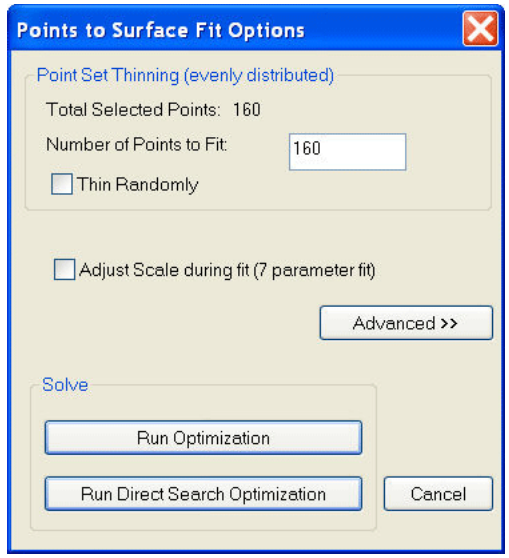


Figure 16-8. The Points to Surface Fit options.

Point Set Thinning

This option allows you to easily thin selected point sets for purposes of fitting to a surface. Suppose you have a scan with 10,000 points, but don't want to wait a long time for the best-fit. You can specify that you only want to fit 400 of the points and it will evenly sample 400 points from the selected point set.

Run Optimization

This is the default mode of operation. The optimization method performs a version of the steepest-descent algorithm on the system. This means it assembles a matrix that is 6 (unknowns) x N (number of points), inverts it and determines move directions. Though quick for reasonable data sets, it can take a long time when using a large set of points depending on the amount of RAM on the computer. The stopping tolerance is the amount that the objective function (RMS error) must repeat by in order to indicate a minima has been reached.

Run Direct Search Optimization

This optimization provides a powerful method for searching the solution space. It is also required when using the toleranced best-fitting options (in the advanced area) since they introduce non-linearities into the solution space. This method basically steps in a direction, evaluates the fit, then steps in another direction, etc. There is a lot of advanced logic built into the search, but at the core, it is a searching method. The advantage is that it does not have to assemble a large partials matrix and invert it. This means that it can effectively deal with huge point sets. The disadvange is that since it is trying to exhaustively search, it can sometimes take a long time to complete. At any time during the fit, however, you can cancel the operation and use the current location. To use this mode, you set step sizes. They are set as a ratio of the position and angular perturbation values in the SA units options page. The step size starts at the starting value then decays when it detects no improvement in any direction. Once it decays to the Ending step size, a minima is reached.

Recommendations

We have several recommendations for setting these options based on past experiences. As with many things, you may need to experiment to determine the best set of options for a particular combination of points and surface geometry.

The most important recommendation is to start with a good initial condition. The easiest way to do this is to use Quick Align to CAD.

- **Reasonable point sets without toleranced fitting.** Use only the conventional solution. It is not necessary to waste the time on the exhaustive search.
- **Huge point sets with a reasonable starting guess.** Use only the exhaustive search with appropriate start and stop perturbations.
- **Reasonable point sets with tolerances or unusual geometry.** Use both fit options.
- **Reasonable point sets, no tolerances, but very high accuracy required.** Use both fit options.

Advanced Options

By pressing the Advanced button, you can open the advanced side of the dialog (Figure 16-9).

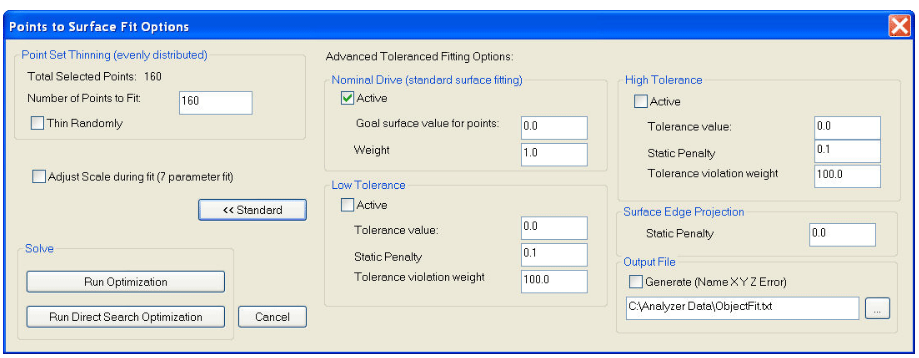


Figure 16-9. Advanced Best-Fit options.

These allow you to modify the constraints of the fit to achieve more complex fit ups. This may be used to avoid part to part interference in build operations, or other processes where the interaction between parts is of interest.

Suppose we want to perform a standard best fit to the CAD surface, but we want to make sure none of the points are more than 0.030

below the surface. The parameters could be set as follows:

- Nominal Drive. Use default settings.
- Low Tolerance 0.030, and default weight values.

The result is that the optimization will drive the points to the nominal CAD surface but also exert extra pressure on points that fall below 0.030 on the negative side of the surface. Because the weight for the tolerance is fairly high, it will push the set of points so that the offending low tolerance points lie within the -0.030 band.

Nominal Drive (Standard Surface Fitting)

To perform a standard surface fit, use this option by checking **Active**. Set the goal value to 0.0 if you wish to drive the points to the surface, or you may use another value to use the offset surface. You do not need to set the weight unless you are using other tolerances in the fit.

Low Tolerance

These four variables control the low tolerance value. The tolerance value does not have to be on the negative side of the surface. You could, for example, have a low tolerance of 0.25" and a high tolerance of 1". The low tolerance is simply the lowest tolerance value.

- **Tolerance Active.** This flag allows you to easily turn on and off the low tolerance. Turning the tolerance off is equivalent to settings its weight to 0.0, except that is skips the computations necessary to determine the toleranced objective function value.
- **Tolerance Value.** This value is the tolerance for the point to surface fit of this FitPoint. If 0.0, then the point will be driven to the surface (after compensating for target thickness)
- **Static Penalty.** This value is added to the objective function in addition to the weighted tolerance violation. Think of this as a static offset applied any time a point moves beyond the tolerance. Initially, leave this value set to 0.0 and only use it when needed. When you use it, set its value low so it does not unnecessarily increase the nonlinearity of the system. Using large static offset penalty values will have the effect of stopping the optimization prematurely.
- **Tolerance Violation Weight.** This value is multiplied by the amount that the point violates the low tolerance. If the low tolerance is –0.030 and the points projected distance is –0.040, the violation is 0.010 multiplied by the Weight.

High Tolerance

The parameters are identical to the Low tolerance except that they apply to the high tolerance.

Edge Projection Penalty

This value is a fixed penalty added to a point's objective function contribution if the point projects to the edge of a surface.

Watch Windows

CHAPTER 17

Viewing live measurement data is essential for tasks such as real-time assembly and tool-building. SA's Watch Windows provide you with tools for not only viewing live measurement data, but also for viewing other live spatial relationships.

SA's watch windows are live readouts of measurement data. You can view the coordinates of an instrument's measurement, watch the live delta transformation between two objects as they are moved, watch the deviation of a probe from a nominal surface, and more. This enables real-time assembly operations, in which you can watch a moving object's positional error in real-time and make corrections live. It also permits easy tool and fixture building, because you can watch a feature's position in real-time and check it as you make adjustments to the tool.

Tip: If possible, you can use SA Remote (an iOS app for iPod/iPad/iPhone) to view watch window data remotely. See the chapter "Measurement" for more information.

A typical watch window is shown in Figure 17-1. Depending on the type of watch window being viewed, you may be seeing raw coordinates (X/Y/Z values) or deviations (dX/dY/dZ values). Each value is displayed on a separate row in the window. The window may be resized or even maximized as necessary for easier viewing from a distance—the text will expand to fill the available window. You can open as many simultaneous watch windows as you'd like--they will all update automatically.

By default, watch windows (like almost everything in SA) show their data relative to the active coordinate frame. So an X value is the value in the working frame's X axis, and a dZ value is the delta along the working frame's Z axis. Settings for a given type of watch window are saved—meaning that if you, for instance, hide the X delta in a specific type of watch window, that value will still be hidden the next time you display that type of watch window.

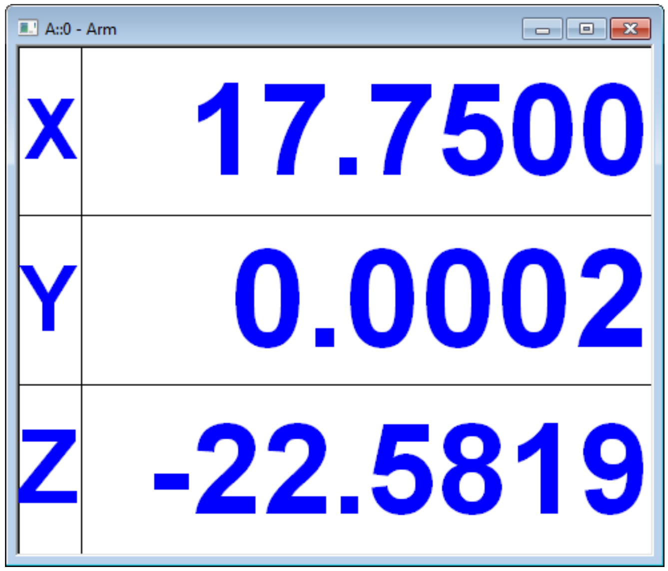


Figure 17-1. A typical watch window.

Watch Window Types

There are a number of different types of watch windows that can be displayed in SA. They include:

- **Instrument's Point.** Shows the position of a live instrument's probe in real-time.
- **Point to Point.** Shows the deviation of the instrument's probe from a specific point.
- **Point with View Zooming.** Compares the instrument's probe to a specific point, zooming in on the point as the deviation decreases.
- **Closest Point.** Compares the instrument's probe to the closest of a set of points.
- **Point to Objects.** Compares the instrument's compensated probe position to the closest of one or more objects.
- **Three Point Frame.** A point measurement from each of three different instruments is used to define a coordinate frame, which is then viewed relative to the working frame.
- **Two Point Axis Alignment.** Two measured points are used to calculate pitch and roll from a given coordinate frame.
- **Theodolite Observer Point to Point.** Compares a point derived from two or more theodolite observations to any other point.

- **Theodolite Observer's Point.** Displays a point derived from two or more theodolite observations.
- **Theodolite Observer's Data.** Displays the data from the Theodolite Observer.
- **Relationships.** Any relationship can have a watch window added in order to view that relationship's deviation.

Watch Instrument's Point

This watch window displays the coordinates of the *center* of the instrument's probe. It will update whenever updated data is received from the instrument.

To Display an Instrument's Point Watch Window:

- Right-click an instrument and select **Add Watch Window** from the context menu, or
- From the menu, select **View>Watch Window>Add Instrument's Point**.

Watch Point to Point

This watch window displays the deviation between the center of an instrument's probe and the selected point. Offsets are not used in this comparison.

To Watch Point to Point:

- From the menu, select **View>Watch Window>Add Point to>Point**, then select the point to compare, or
- Right-click a point and select **Add Watch Window** from the context menu.

Watch Point to Point With View Zooming

This watch window functions the same as a point to point window, but automatically zooms the graphical view and plays a sound whose pitch is proportional to the proximity of the target to the point. Offsets are not used in this comparison.

To Watch Point to Point with View Zooming:

1. From the menu, select **View>Watch Window>Add Point to>Point with View Zooming**.
2. Select the point to compare.

Watch Closest Point

A closest point watch window will automatically compare an instrument's probe position to the closest of a set of points. This set of points can be entire group, a subset of a group, or a subset of multiple groups. This watch window will automatically display the name of the point to which it is comparing. Offsets are not used in this comparison.

To Watch Closest Point:

1. From the menu, select **View>Watch Window>Add Closest Point**.
2. Select the point(s) to compare against.

To Watch a Point Group:

- Right-click the group and select **Add Watch Window** from the context menu.

Watch Point to Objects

Warning: When comparing to a line, ensure that your projection options are correct. If you want to compare the center of the probe to a line, offsets should be overridden to zero.

This watch window displays deviation between the probe and the closest of a set of provided objects. It's important to recognize that offsets *are* accounted for in this comparison—that is, probe compensation is applied.

However, if the object you are comparing to is a line, SA will by default override target offsets to zero, since comparing the center of a probe to a line is far more common than comparing a compensated point to a line.

When multiple objects are selected, the closest object to the probe's current position is compared. As the probe moves, the object compared against may change.

To Watch Point to Objects:

1. From the menu, select **View>Watch Window>Add Point to>Objects**.
2. Select the objects to compare against.

To Watch Point to a Specific Object:

- Right-click the object and select **Add Watch Window** from the context menu.

Watch Three Point Frame

Watching a three point frame is a fairly unique operation. It involves watching three 3-DOF devices simultaneously, wherein the device's points define a frame's origin, X axis, and XY plane. The watch window then shows the transform of this frame in the working frame.

For this command to be useful, the three instruments must be aligned to the same coordinate system—that is, they must be located to each other. Otherwise, the spatial positioning of the three points won't make sense.

If you have an object in which three points define a repeatable frame, you can use this command to easily determine the position and orientation of the object. If you then create and activate a corresponding coordinate frame representing a desired end state, this watch window represents the deviation needed to get to the active frame.

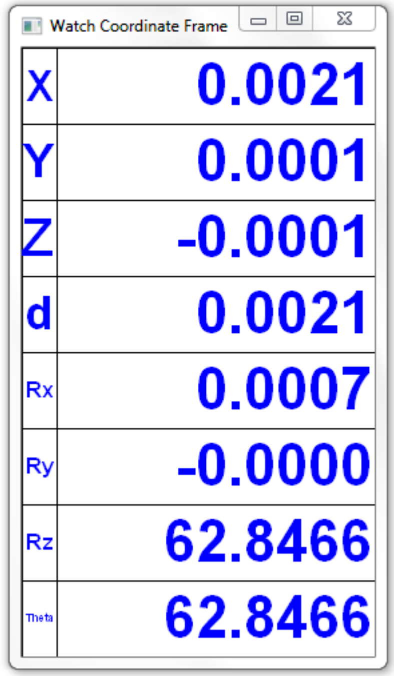


Figure 17-2. A three point frame watch window.

To Watch a Three Point Frame:

1. Ensure that the interfaces for three separate 3-DOF devices are running.
2. From the menu, select **View>Watch Window>Add Three Point Frame**.
3. Select the instrument whose point will define the origin of the frame.
4. Select the instrument whose point defines the X axis of the frame.
5. Select the instrument whose point lies in the XY plane.

Watch Two Point Axis Alignment

Often, two points defining the axis of an object must be measured and adjusted to either match a nominal orientation, or to match a prescribed pitch and yaw.

This watch window allows you to measure two discrete points with a single instrument. The watch window then indicates the pitch (elevation) and yaw (azimuth) of the measured axis relative to the prescribed axis (Figure 17-3).

The working coordinate frame defines the vertical direction—therefore its Z axis is used in the calculations.

To Watch an Axis Using Two Point Axis Alignment:

1. Measure or construct two points along the axis that serve as the nominal or "goal" points.
2. From the menu, select **View>Watch Window>Watch Two Point Axis Alignment**.
3. Select the instrument to use for measurements.
4. Select the first and second points, defining the X axis (from the first to the second). The measured X axis is along the direction from the first to the second points. The Z axis is clocked along the working Z axis, therefore if pitch to gravity is desired, this axis should be along gravity.

Figure 17-3. Two Point Axis Alignment.

5. The ***Monitor Axis*** dialog appears.
6. Specify the group name in the text field, if desired.
7. Click Target 1 and Target 2 to measure the points and associate them with the ends of the axis. The pitch and yaw values will update.
8. To automatically measure both targets (assuming you have at least two targets installed and a steerable instrument), click the Cycle Targets button. The instrument will point, search for, lock on, and measure each target.
9. As repeated measurements come in, SA will automatically increment the group name and put each repeated measurement in a separate group.

Note: Cycle Targets will not work if the axis points have moved beyond a reasonable search region for the instrument.

Watch Theodolite Observer Point to Point

When using the Theodolite Data Observer (see the "Measurement" chapter), this watch window compares the latest computed point to a selected point. For instance, if you have three bundled theodolites, and they are all focused on a common target, you can query the instruments with the Theodolite Data Observer to calculate the position of the target. This watch window will compare this position to a specified point. Note that target offsets are not used here.

To Watch Theodolite Observer Point to Point:

1. Ensure that the Theodolite Data Observer is up and running, and that your instruments are bundled (see the "Measurement" chapter).
2. From the menu, select **View>Watch Window>Add Theodolite Observer Point to>Point**.
3. Select the point to compare against.
4. As updated data arrives from the theodolite data observer, the watch window will automatically update.

Watch Theodolite Observer's Point

Similar to "Watch Theodolite Observer Point to Point" on page 536, except the point's coordinates are displayed—no comparison is performed to another point. Note that target offsets are not used here.

To Watch the Theodolite Observer's Point:

- From the menu, select **View>Watch Window>Add Theodolite Observer's Point**.

Watch Theodolite Observer's Data

This watch window displays all data from the Theodolite Data Observer, depending on the settings in the Theodolite Data Observer. For instance, it will display instruments and theta/phi angles from those instruments.

To Watch Theodolite Observer's Data:

- From the menu, select **View>Watch Window>Add Theodolite Observer's Data**.

Watch Window Options

Right-clicking a watch window will present a context menu containing a number of options for changing how data is represented and displayed. These options are discussed below.

Precision

The number of decimal places used to display watch window data can be controlled. Depending on the necessary accuracy and the user's tolerance for quickly-changing values, the default values may provide too much precision (revealing noise and resulting in quickly-changing numbers) or too little (yielding insufficient detail for the required accuracy).

By default, SA uses the default length and angle precision set in the User Options. However, you can use any preferred precision. The options when changing decimal precision are as follows:

- **Use SA Significant Digits.** When selected the watch window will use the software defaults. Deselect this to use custom precision settings for the watch window.
- **Length.** The number of decimal places to use for displaying linear values.
- **Angle.** The number of decimal places to use for displaying angular values.

To Change a Watch Window's Decimal Precision:

5. Right-click the watch window and select **Precision** from the context menu. The ***Watch Window Significant Digits*** dialog will be displayed (Figure 17-4).

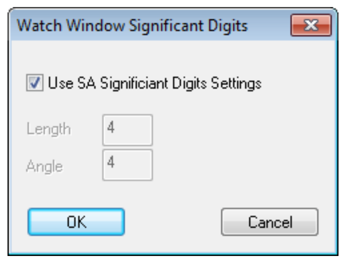


Figure 17-4. Changing decimal precision in a watch window.

6. Apply your desired settings, then click OK.

Font

Watch windows can be displayed in a custom font. You can change the font and style. (Although you can change the font size, this setting is ignored and the size of the window itself is used for determining text size).

To Change a Watch Window's Font:

1. Right-click the watch window and select **Font** from the context menu.
2. In the font dialog, select the desired typeface and style, then click OK.

Text Color

You can change the text color for the watch window on a row-by-row basis.

To Change Watch Window Text Color:

1. Right-click on the row in the watch window that you'd like to modify.

2. From the context menu, select **Text Color**.
3. In the color dialog, choose a new color, then click OK.

Background Color

You have row-by-row control over the background color used in the watch window.

To Change Watch Window Background Color:

1. Right-click on the row in the watch window that you'd like to modify.
2. From the context menu, select **Background Color**.
3. In the color dialog, choose a new color, then click OK.

Highlight Color

If a prescribed tolerance for a watch window is exceeded, the out-of-tolerance values will be highlighted. You can control the color used for this highlight.

To Change the Watch Window Highlight Color:

1. Right-click on the row in the watch window you'd like to modify.
2. From the context-menu, select **Highlight**.
3. In the color dialog, choose a new color, then click OK.

Arrow Settings

When a watch window is displayed that is comparing to another entity, arrows are displayed in the graphical view as a way to visualize the deviations. You can control how these arrows are depicted using the options in the ***Watch Window Settings*** dialog (Figure 17-5).

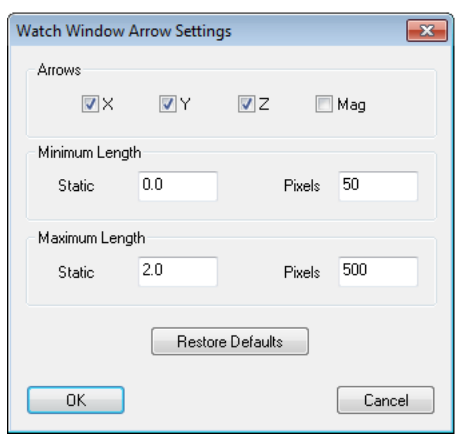


Figure 17-5. Watch window arrow settings.

- **Arrows.** Specifies which arrows should be displayed: X, Y, Z, and/or magnitude. The X/Y/Z arrows show the dX/dY/dZ values. Magnitude shows the shortest distance between the watched item and the entity it's being compared against.

- **Minimum Length (Static).** The real-world minimum deviation below which an arrow does not get any shorter.
- **Minimum Length (Pixels).** The shortest length that any arrow can be drawn in the graphical view.
- **Maximum Length (Static).** The real-world maximum deviation above which an arrow does not get any longer.
- **Maximum Length (Pixels).** The longest length that any arrow can be drawin in the graphical view.

To Change Watch Window Arrow Display Settings:

1. Right-click the watch window and select **Arrow Settings** from the context menu.
2. Choose the desired options, then click OK.

Group Name

When viewing a watch window to closest point, by default the group name for the watched point is displayed in the window. You can turn this off, if desired.

To Toggle Display of Group Names:

- Right-click a closest point watch window and toggle the **Show Group Name** context menu item.

Showing/Hiding Rows

Individual rows in a watch window can be shown or hidden. For instance, if you are only interested in adjusting the XY position of a feature, you can remove the Z component from the window to prevent confusion and simplify the display.

To Show or Hide Rows in a Watch Window:

- Right-click the desired row and select **Hide Row Items** from the context menu to hide a row, and select **Show All Items** to restore all rows.

Coordinate System

Watch windows can display data in one of three coordinate system types: Cartesian, Cylindrical, or Spherical. In the cylindrical/spherical case, the XY axes of the coordinate frame define the θ plane. In the spherical case, the Z axis of the coordinate frame defines the φ=0 direction.

To Change the Coordinate System for a Watch Window:

Right-click the watch window and select **Cartesian**, **Cylindric**, or **Spherical** from the context menu.

Frame of Reference

By default, watch window data is displayed with respect to the working coordinate frame. Sometimes, you may wish to display the data in a specific coordinate frame, regardless of the working frame (Figure 17-6). For instance, you may have the **`World`** frame active but want to view a specific watch window in **`Local`** coordinates.

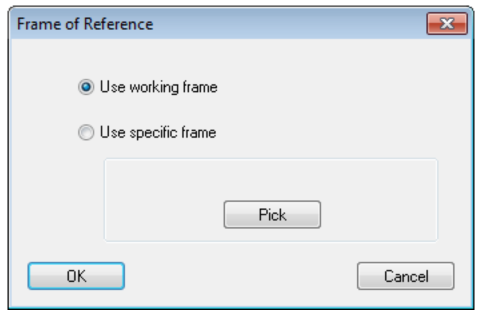


Figure 17-6. Changing the frame of reference for a watch window.

Warning: This setting (along with others) is saved for a particular watch window type. If you forget to set this back, you may view a watch window at a later time with respect to the incorrect frame.

- **Use working frame.** Always represents the values relative to the working coordinate frame. This is the default.
- **Use specific frame.** When selected, a specific coordinate frame is used for data representation, regardless of the active frame. The selected frame can be changed using the Pick button.

To Change a Watch Window's Frame of Reference:

1. Right-click a watch window and select **Set Frame of Referece [FoR]** from the context menu.
2. Select the desired option, then click OK.

UDP Broadcast

Watch windows can be configured to transmit their data over TCP/IP using the UDP Protocol (Figure 17-7). This is useful for intercepting watch window data with your own custom code.

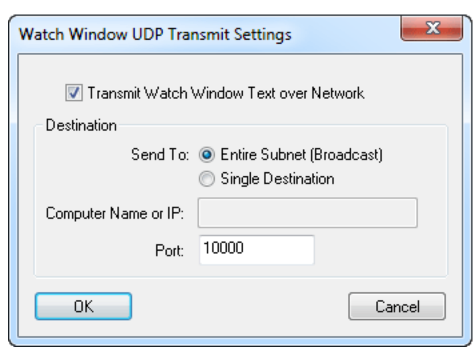


Figure 17-7. UDP Settings.

Options for UDP transmitting are as follows:

- **Transmit Watch Window Text over Network.** When enabled, data is transferred over TCP/IP.
- **Send To.** UDP packets can be transmitted to the entire subnet, which is referred to as a broadcast. Or, it can be directed to a specific destination (*Single Destination*).
- **Computer Name or IP.** If transmitting to a specific destination, the IP address or computer name of the recipient.
- **Port.** The port over which the data should be transmitted.

To Transmit a Watch Window's Data over UDP:

1. Right-click a watch window and select **UDP Network Transmit Settings** from the context menu.
2. Select the desired options, then click OK.

Tolerances

You can set tolerances for watch windows that provide deviation values. Depending on the type of watch window, this may just be a magnitude tolerance (for watching objects) or individual component tolerances (when watching points). The ***Vector Tolerance*** dialog, which allows setting component tolerances, is shown in Figure 17-8.

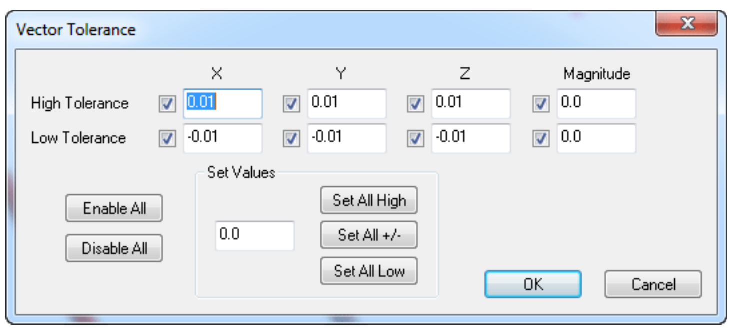


Figure 17-8. Setting component tolerances for a point to point watch window.

In this dialog, you can enable or disable individual component tolerances, set all high or low tolerances to a given value, or set them all to a symmetrical value.

To Set Tolerances on a Watch Window:

1. Right-click the watch window and select **Default Tolerance** from the context menu.
2. Set the desired options, then click OK.

Projection Options

As is the case with relationships and queries, you can control how deviations are depicted in a watch window. This is handled through the watch window's projection options.

The projection options for a point-based watch window are shown in Figure 17-9:

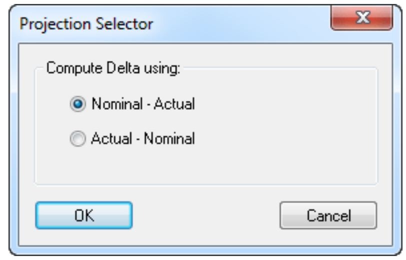


Figure 17-9. Projection options for a point to point watch window.

- **Nominal-Actual.** Deviations show the delta from the "nominal" or reference point to the measured point. This presents the deviation from an inspection point of view—that is, how far is the measurement from nominal?
- **Actual-Nominal.** Deviations show the delta from the measured point to the reference point. This presents the deviation from a build point of view—that is, how far do I need to move the probe to get to nominal?

Projection options for an object-based watch window are shown in Figure 17-10. There are four available indicator styles:

-  Deviations are from the measurement to the design—that is, they reflect the "build" point of view.
-  Deviations are from the design to the measurement—that is, they reflect the "inspect" point of view.
-  Deviations are the same as the "build" option above, but the arrows are drawn with their tails at the center of the probe.
-  Deviations are the same as the "inspect" option above, but the arrows are drawn with their heads at the center of the probe.
- **Ignore Edge Projections.** This option is ignored.
- **Offset Value.** The desired offset value when overriding the probe's offset.
- **Override Target Value.** When enabled, the probe's offset is overridden by the provided value.
- **Add Extra Material Thickness.** Adds virtual thickness to the objects being compared against. This is useful, for example, when you have an OML surface in your CAD model but you're actually comparing to an IML, assuming a certain thickness for the material.

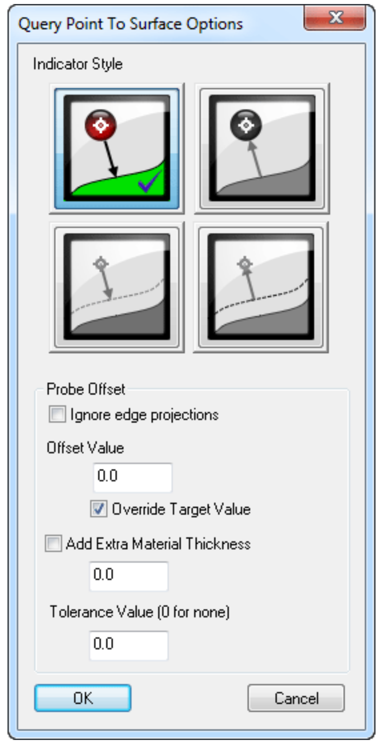


Figure 17-10. Projection options for an object-based watch window.

- **Tolerance Value.** This value is ignored.

To Set Projection Options for a Watch Window:

1. Right-click the window and select **Projection Options** from the context menu.
2. Enter the desired settings, then click OK.

Transparency

Watch windows can be made transparent, so that their backgrounds are invisible. The window border can also be made partially transparent. This allows you to see what's going on in the graphical view while a watch window is being displayed.

To Make a Watch Window's Background Transparent:

- Right-click the watch window and enable the **Transparent Background** option.

To Make a Watch Window's Border Partially Transparent:

1. Ensure that **Transparent Background** is enabled (see above).
2. Right-click the watch window and select **Adjust Frame Transparency** from the context menu.
3. Adjust the transparency to the desired level, then click OK.

Google Glass App

NRK has developed an app for Google Glass that allows you to view watch window data in a heads-up-display directly on your Google Glasses (Figure 17-11).

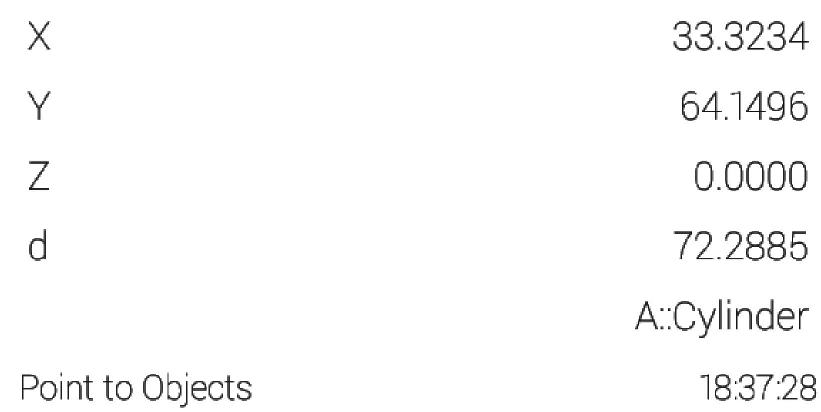


Figure 17-11. The watch window on the Google Glass app.

Since the app is not yet available on the Google App store, the following provides instructions for "side-loading" the app onto a Google Glass device—this is not necessary when adding apps from the Google store.

To Install the Google Glass App:

1. Download the Android SDK ADT Bundle for Windows: http://developer.android.com/sdk/index.html
2. Extract the downloaded .zip file anywhere on your machine (the resultant directory will be referred to as **`<SDKDir>`**).
3. Run the SDK Manager (**`<SDKDir>\SDK Manager.exe`**), it will download the latest list of available packages and might immediately conclude that updates are available, if so click the Install packages button and then exit and restart the SDK Manager when it is finished.
4. In the SDK Manager in the **Packages** menu, verify that *Sort by API Level* is selected. Under *Android 4.4.2 (API 19)* verify that the following are either installed or checked to be installed:

- SDK Platform
- Glass Development Kit Preview

5. Under *Extras* verify that the following is either installed or checked to be installed:

- Google USB Driver

6. Click Install Packages if enabled to install/update the neces-

sary components.

7. Add the SDK platform-tools directory to your system PATH: `<SDKDir>\sdk\platform-tools`
8. Verify the SDK is setup by entering the below from a command prompt:

```
adb version
```

9. This should provide you with output similar to the below:

```
Android Debug Bridge version 1.0.31
```

10. On the Glass device, under settings, there's an option to enable USB debugging. Make sure this option is turned on to allow communications with the computer.
11. Plug the Glass into your computer's USB port.
12. The Glass device may prompt you to allow this computer to debug, if so, allow it.
13. From a command prompt, enter the following:

```
adb wait-for-device
```

This command will wait until communications with the device are available. If it just sits there and never returns you back to the command prompt, then something is not working. If it returns quickly with no output, then the installation is correct. It will eventually time out if a connection cannot be established.

14. From a command prompt, enter the following (this assumes the .apk file is in the current directory). Replace the example .apk filename below with the actual filename:

```
adb install SAWatch-YYYY.MM.DD.apk
```

To Run the Google Glass Watch Window App:

1. Wake up the Glass device (tap the side or angle your head up 30 degs) so that the main screen is displayed with the time and "ok glass".
2. Either say "ok glass" and then say "watch measurements" or tap to get a command menu and then tap to select "watch measurements".
3. Configure SA with a watch window that has UDP broadcast enabled to port 10000 and you should see the data displayed in the Glass app.
4. If there are multiple watch windows open, you can switch between them by swiping left and right.
5. To exit the app, either swipe down, or tap then tap again to

select Stop.

Note that the app is setup as a Glass "immersion" and it is configured to force the screen to stay on while its running. Because of this, you'll want to exit the app when done to preserve battery life.

Uninstalling the Google Glass App:

1. You can remove the app from your Glass device by entering the following from a command prompt:

```
adb uninstall com.kinematics.sawatch
```

Troubleshooting

ADB unable to communicate with device, "adb wait-for-device" hangs:

It is possible the USB driver `.inf` file does not have the correct entries for your specific Glass device. Editing the `.inf` file and then updating the driver in device manager (and specifying the edited `.inf` file) may fix this.

1. Open Device Manager (right-click My Computer and select **Manage > Device Manager** and find the entry for the Glass device—it may be under "Other" with a yellow icon indicating a problem.
2. Right-click the Glass device and choose **Properties > Details**. Select **Hardware IDs**. You'll see something similar to Figure 17-12.

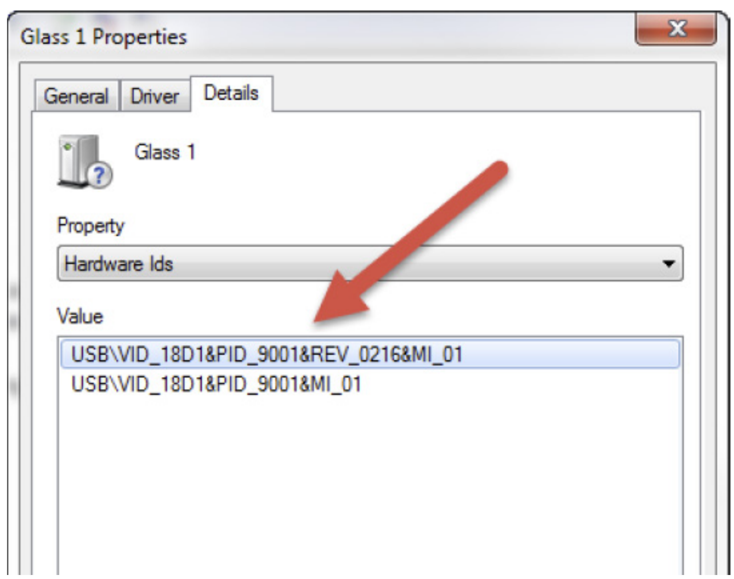


Figure 17-12. The Glass device in the Hardware IDs dropdown.

3. Open the `android_winusb.inf` file in notepad (this file can be found in `<SDKDir>\sdk\extras\google\usb_driver`.
4. In this file, there will be two sections, Google.NTx86 and Google.NTamd64 which must both contain entries matching the hardware IDs shown in the above screenshot. In these sections add entries like the below based on the info shown

in the device properties window. You can leave other, existing entries alone:

```
[Google.NTx86]
;  Google Glass
%SingleAdbInterface%=USB_Install, USB/VID_18D1&PID_9001&REV_0216&MI_01
%CompositeAdbInterface%=USB_Install, USB\PID_18D1&PID_9001&MI_01

[Google.NTamd64]
; Google Glass
%SingleAdbInterface% = USB_Install, USB\VID_18D1&PID_9001&REV_0216&MI_01
%CompositeAdbInterface% = USB_Install, USB\VID_18D1&PID_9001&MI_01
```

5. Save the edited file and close Notepad; then go back to device manager, right-click the device and pick the option to update hardware drivers; when asked, browse the local computer and find the edited `.inf` file.
6. Retry the **`adb  wait-for-device`** command in the install section.

App Install Error, "INSTALL_FAILED_OLDER_SDK"

This suggests the Glass device on which you're trying to install the app has an older SW version than what was used when building the app. To fix this, update the Glass SW; see the below link for more info:

https://support.google.com/glass/answer/3226482?hl=en

Glass turns off after about a minute and can no longer be used

• Software update XE17 introduced a problem with the on-head detection feature.

1. Try turning off this feature in settings.

Chapter 18
Measurement Uncertainty

Under development.

CHAPTER 19

Unified Spatial Metrology Network

Under development.

Analysis

CHAPTER 20

This chapter covers a myriad of analysis operations that can be performed in SA, from Geometry Fit to fitting surfaces to measurements.

Geometry Fitting

Fitting geometry to measured points is a fundamental and critical aspect of almost any metrology analysis task. SA's Geometry Fit Interface has the capability to fit the following geometric shapes to any set of measured points or cloud points:

- Planes
- Circles
- Spheres
- Cylinders
- Lines
- Paraboloids
- Slots
- Cones
- Ellipses

Note: When fitting slots, there are a few rules to follow. If measuring 5 points, obtain 3 in the middle (2 one side, 1 on the other other), and one on each end. If measuring 6 points, measure an extra point in the middle. For more than 6 points, distribute your points evenly around the slot.

Fitting geometry in SA is completely interactive. You can see and interact with the results of your fit settings in real-time as you modify them. You can interact with the graphical view, re-select points, change geometry types, and reverse offset and geometry directions, all while watching your geometry update in the graphical view.

Geometry can be fit to any number of measured points, constructed points, or cloud points--and it's fast. For example, you can fit a sphere to 640,000 cloud points in about 12 seconds. Unlike some metrology software, when you select large numbers of points the geometry is fit to all selected points rather than a subsample.

You can easily construct interactive vector groups while tweaking settings to visually see how they affect the fit. Easily create cardinal points, offset points, or graphs of fit deviations with a single button click.

Another powerful feature is the ability to project points to a plane before fitting the geometry. This allows you to measure several off-plane features and project them to the same planar surface.

The dialog (Figure 20-1) is also designed with potential repeated use in mind. It enables you to fit multiple geometric features without forcing you to re-select the menu command. This permits you to fit a lot of geometry to measured data quickly. When satisfied with your fit, simply click the Next button and you're set to select points for the next fit.

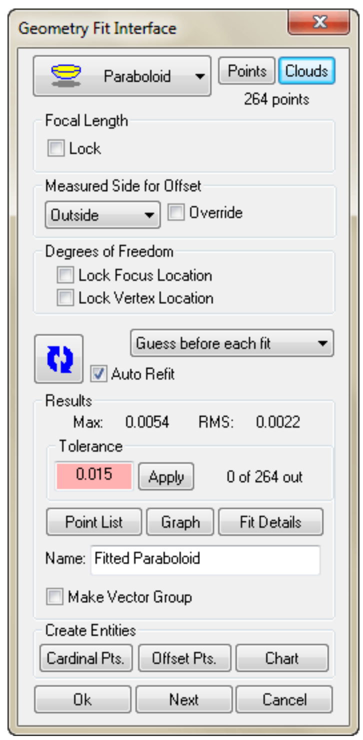


Figure 20-1. The Geometry Fit interface.

To Open the Geometry Fit Dialog:

- From the menus, select **Construct>Geometry by Fitting**, or
- Press the default keyboard shortcut of **Ctrl+G**, or
- To fit geometry to an entire point group, right-click the point group in the tree and select **Fit Geometry**.

About Planar Offset Methods

Planar offset methods determine what is considered the "positive" side of resulting geometry. The positive side of planar geometry (planes, circles, and ellipses) is the side toward which the normal faces, and in metrology it is typically thought of as the "outside" of the part or the side of the surface that was measured (Figure 20-2).

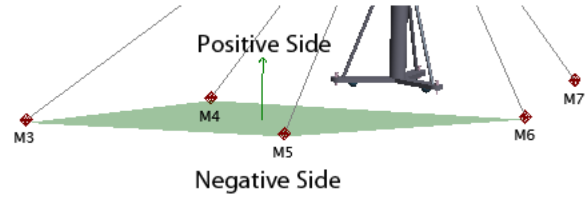


Figure 20-2. Geometry has a positive and negative side.

When fitting geometry to points, there are three offset methods that can be used to determine the positive side of geometry: Probing Direction, Working Origin Positive, and Right-Hand Rule.

Probing Direction

This is the direction from which the point was measured (Figure 20-3). For a line-of-sight instrument such as a laser tracker, total station, or laser radar, probing direction is the line-of-sight itself, as depicted by the red arrows below.

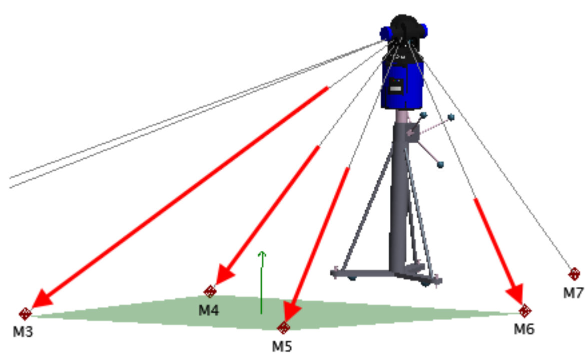


Figure 20-3. Probing directions.

For non-line-of-sight devices, the probe direction is the vector of the device on which the probe is attached, as depicted by the red arrow in Figure 20-4.

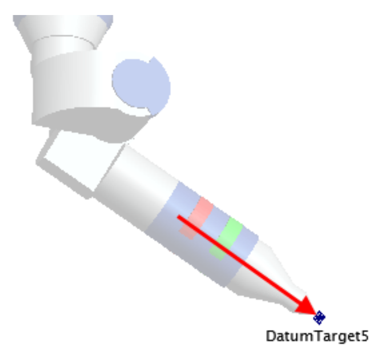
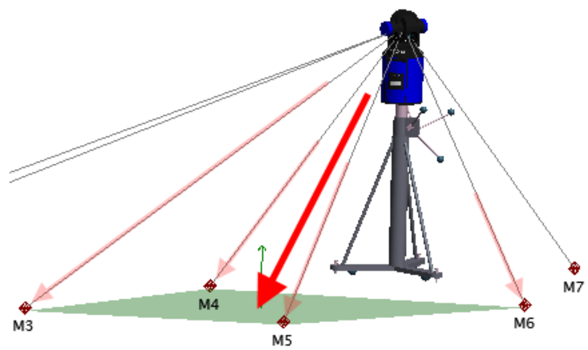


Figure 20-4. Probing direction for a portable arm.

For the purposes of determining the positive side of geometry, the overall probing direction is the average direction of the probing directions for all measured points in the fit (Figure 20-5). Put another way, it is simply the average of all of the individual probing directions. The figure at right depicts the average probing direction for the plane fit from the laser tracker depicted earlier.

Figure 20-5. Average probing direction.



Note: Constructed points do not have probing directions. If no measured points are included in a fit, SA will fall back to using the Working Origin Positive method (described below). If at least one measured point is included in the fit, then the probing direction is used.

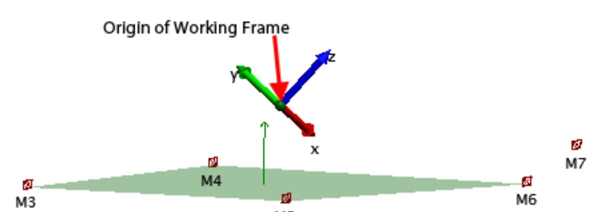
When probing direction is selected as the planar offset method, the positive side of the resulting geometry points generally in the opposite direction as the average probing direction. Put another way, the average probing direction points in the same direction as the negative direction of the geometry. For line-of-sight instruments, the result is that the positive side of the geometry points back toward the instrument, which is usually the desired result. This can be seen in the previous figure: The thin green arrow depicting the positive side of the plane points back generally in the opposite direction as the average probing direction.

The advantage of this method is that for most situations, it describes the resulting geometry correctly--that is, the "outside" of the geometry is the side from which the measurement was approached.

Working Origin Positive

With this method, the geometry is defined such that the origin of the working frame lies on the positive side of the resulting geometry, as depicted in Figure 20-6.

Figure 20-6. Working origin positive.



This is often used as the planar offset method in situations in which probing directions do not exist (for example, when using constructed or imported points).

Right-Hand Rule

This method allows you to define the direction of the resulting geometry through selection order--it uses the traditional "Right-Hand Rule" standard. If you curl the fingers of your right hand in the direction from the first selected point, to the second, then to the third, your thumb will define the positive direction. When using this method, the first three points in the fit should be individually clicked to define the selection order--then the remaining points can be selected as a whole using the marquee selection tool.

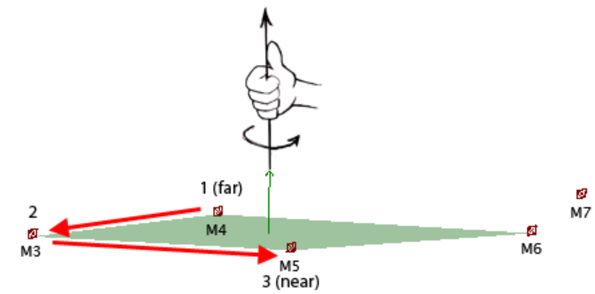


Figure 20-7. Defining offset direction using the right-hand rule.

In Figure 20-7, point M4 was selected first, followed by M3, then M5. M6 was selected after the first three points. Because of the orientation of the first three sequentially selected points, the resulting geometry points up in this image, as dictated by the Right-Hand Rule.

About Measured Side Settings

In order to properly offset geometry, SA must know on which side a feature was measured. For planar geometry (planes, circles, and ellipses), the planar offset direction has one of three options: Above Plane, Probe Center, or Below Plane.

The radial analogs to these settings, used in geometry such as circles, ellipses, spheres, and cylinders, are Inside, Probe Center, and Outside.

Above Plane

With this setting, the line-of-sight to the target (or the probing direction) points in the direction in which offsets should be applied. This is by far the most common measurement situation. In Figure 20-8, the tracker measured several points on the floor. Since the resulting plane representing the floor should be offset away from the tracker, the Above Plane setting is used. In other words, the SMRs were located between the instrument and the measured feature.

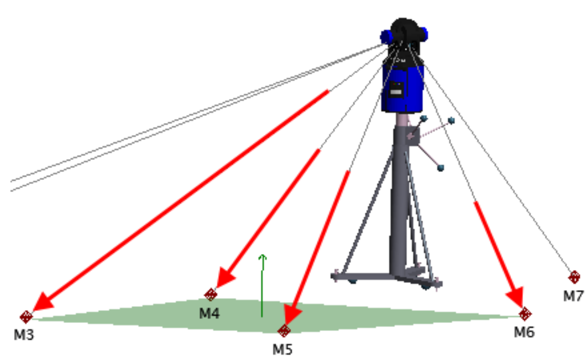


Figure 20-8. Offsetting along the probing direction using Above Plane.

Probe Center

This setting is used when it is preferred to ignore offsets and instead fit geometry along probe centers. This is not as common a setting. In Figure 20-9, the Probe Center setting is active. Notice that the plane lies along the centers of the measured points--no offset is applied.

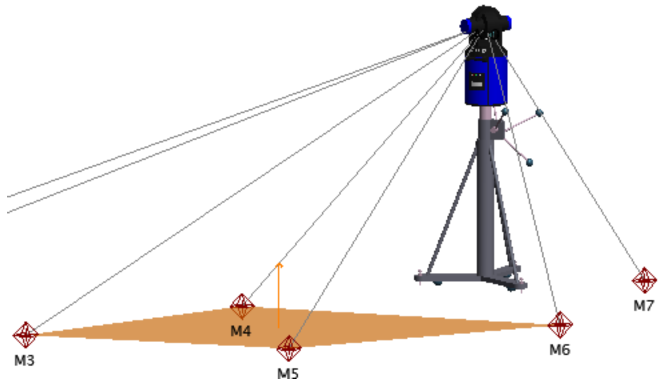


Figure 20-9. No offsets are applied when the Probe Center method is used.

The radial version of Probe Center is equivalent--it implies that no offset is applied in the radial (inward or outward) direction.

Below Plane

This setting is used when the desired offset direction is in the opposite direction as the probing direction. With this setting, the geometry is shifted back toward the instrument (Figure 20-10).

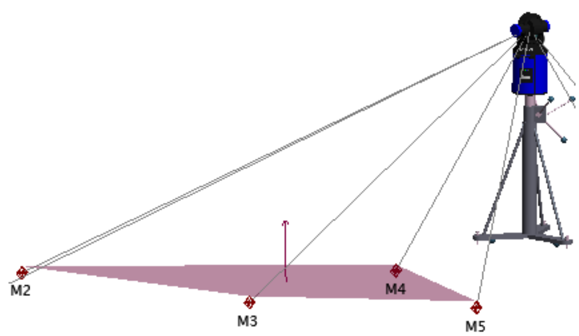


Figure 20-10. Below Plane applies offsets opposite the probing direction.

This is most commonly encountered with line-of-sight instruments in measurement situations where obtaining a proper line-of-sight to a surface is not obtainable, perhaps due to lack of space or the inability to move an instrument. As a result, the measured surface faces away from the instrument, and the probe is placed on the edge of the feature to obtain the measurement. This is often referred to as a backside measurement.

Inside

This is a radial setting for geometry such as circles, spheres, cylinders, and paraboloids. Inside (Figure 20-11) implies that the probe was placed on the inside of the geometry, and therefore the geometry should be shifted outward from the probe centers. The following figure depicts a sphere with the *Measured Side* set to ***Inside***:

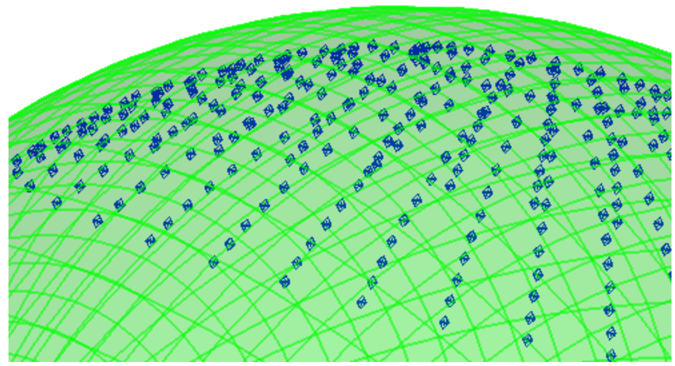


Figure 20-11. Measured side is set to ***Inside***, since the measurements were taken on the inside of this sphere.

Outside

This is a radial setting for geometry such as circles, spheres, cylinders, and paraboloids. Outside implies that the probe was placed on the outside of the geometry, and therefore the geometry should be shifted inward from the probe centers. Figure 20-12 depicts a sphere with the *Measured Side* set to ***Outside***:

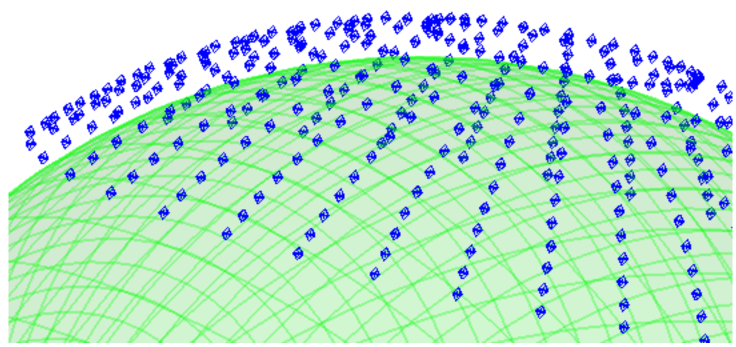


Figure 20-12. Measured side is set to ***Outside***, since the measurements were taken on the outside of this sphere.

The Fit Interface

The Geometry Fit Interface for a circle is pictured in Figure 20-13 (the available options change slightly depending on the selected geometry).

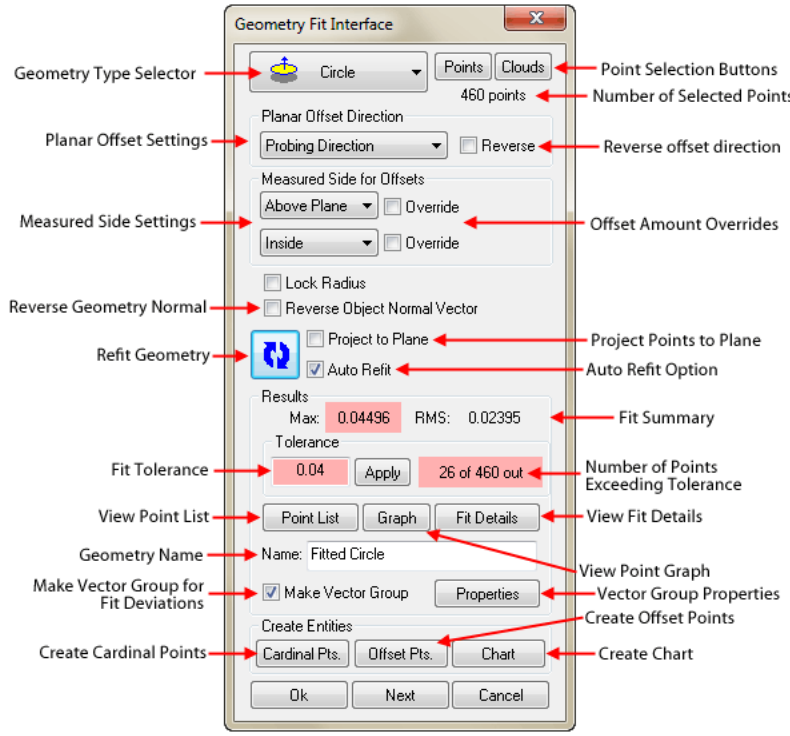


Figure 20-13. The Geometry Fit Interface

The options common to most geometry types will be discussed below, followed by geometry-specific options.

Common Settings

- **Geometry Type Selector.** Allows you to select the type of geometry to fit. This dropdown can be changed at any time to switch to a different geometry type.
- **Point Selection Buttons.** Choose from either Points or Clouds. Click the button, then select the points to include in the fit. The number of selected points will be displayed below these buttons--choose as many points as desired.
- **Planar Offset Direction.** Determines the planar offset method of the resulting planar geometry, using one of three possible methods: Probing Direction, Working Origin Positive, and Right-Hand Rule. This setting is used for plane, circle, and ellipse fits.
- **Reverse.** Reverses the current offset direction. If the selected planar offset method does not offset in the correct direction, click this checkbox to reverse it.
- **Measured Side Settings.** These settings allow you to specify on which side of the feature the specified geometry was measured. For planar geometry (planes, circles, and ellipses), there are planar settings of Above Plane, Probe Center, and Below Plane. For solid geometry, there are radial settings of Inside, Probe Center, and Outside.

Warning: Reversing the offset direction does two things: it reverses the positive direction of the geometry (so that the normal faces the opposite direction), and it changes the direction in which the geometry is offset from the points.

- **Override.** Checking this checkbox allows you to enter a value to override the selected probe radii and therefore use a different offset value (not applicable to the Probe Center option).
- **Reverse Object Normal Vector.** Flips the normal direction of the resulting geometry, without changing other characteristics (such as position or radius).
- **Refit Geometry.** Click this button to update the fit. If this button has a blue background, it indicates that the fit is not currently valid and needs to be recalculated.

Warning: Changing some parameters, such as removing a point from the fit, will not cause an automatic recalculation, even if *Auto Refit* is checked. In these cases, the Refit Geometry button will have a blue background, indicating that the current fit is invalid.

- **Auto Refit.** When checked, changing fit parameters will cause the fit to automatically recalculate.
- **Project to Plane.** When checked, you can select a plane to project points to prior to performing the fit.
- **Fit Summary.** The maximum absolute error and RMS error of the fit is displayed in this section.
- **Fit Tolerance.** The current user-specified tolerance for the fit. To enter a new tolerance, type a new number into the field and then click Apply.
- **View Point List.** Opens the Point List for the fit (see below).
- **Graph.** Displays the Geometry Fit Graph for the fit (see below).
- **Fit Details.** Displays a detailed, geometry-specific summary of the fit. For example, it shows absolute and signed errors, calculated radii or sphere centers, and other important information.
- **Name.** Use this field to specify the name for the resulting geometry. The geometry will be placed into the active collection.

Note: The vector group is updated dynamically. Changing fit parameters will update the vector group in real-time.

- **Make Vector Group.** If checked, a vector group will be created that represents a comparison between the points and the resulting geometry from the fit. This is an excellent way to visualize errors in the fit.
- **Properties.** The Properties button opens the properties from the vector group representing the fit. This button is only displayed when **Make Vector Group** is checked.

Warning: Cardinal points, offset points, and charts are not dynamic. Once created they are not updated if fit parameters are changed.

- **Cardinal Pts.** When clicked, creates cardinal points (centroid, axis points, etc.) for the geometry.
- **Offset Points.** When clicked, creates points offset toward the geometry by an amount equal to the probe radius.
- **Chart.** When clicked, creates a histogram and run chart depicting point errors for the fit.

Geometry-Specific Settings

- **Lock Radius (Circles, Spheres, & Cylinders).** Allows you to specify a radius for the resulting geometry. SA will orient the specified geometry to minimize errors in the fit.
- **Lock Focal Length (Paraboloids).** Allows you to specify a focal length for a paraboloid. SA will orient the paraboloid to minimize errors in the fit.
- **Lock Included Angle (Cones).** Allows you to specify the apex angle for a cone. SA will orient the cone to minimize errors in the fit.
- **Standard [Geometry]/Max Inscribed/Min Circumscribed (Cylinders, Circles, and Slots).** Specifies a different type of fit for cylinders, circles, or slots. *Standard* performs a traditional fit by minimizing errors. *Max Inscribed* finds the largest geometry that fits inside the specified points, whereas *Min Circumscribed* finds the smallest geometry that fits around the specified points.
- **Round/Square (Slots).** Used to specify whether a slot should be round or square.
- **Guess Before Each Fit/Use Last Fit As Guess/Use Object As Guess (Cylinders, Paraboloids, Cones).** Specifies how the initial condition for a fit is determined. Due to the nature of these fit types, the initial condition for the fit can have an effect on the resulting generated geometry. For example, if there is not great axial or radial coverage of a cylinder, it is possible for the cylinder fit algorithms to get "confused". This option helps the fit algorithms find the proper solution. **Guess Before Each Fit** asks the algorithm to take a guess as to the initial size and orientation of the geometry before it progresses through the fit to find the ideal solution, and is the default setting. **Use Last Fit As Guess** uses the current geometry fit result's size and orientation as the initial condition for the next fit recalculation. **Use Object As Guess** allows you to pick existing geometry of the same type to specify the starting condition geometry for the fit algorithm. In this way, you can provide a "hint" as to the resulting orientation of the geometry. If you find that your cylinder, paraboloid, or cone fits are not what you expect, try using this setting to help the algorithm with an initial guess.
- **Lock Focus Location (Paraboloids).** Prevents the current focus location from changing if a fit recalculation occurs.
- **Lock Vertex Location (Paraboloids).** Prevents the current vertex location from changing if a fit recalculation occurs.

Point List

The ***Geometry Fit Point Listing*** dialog (Figure 20-14) allows you to view individual points and their errors in a geometry fit.

The table contains all points selected for the fit, in addition to their individual deviation from the best-fit geometry. For circles and ellipses, planar and radial component error is displayed as well. To sort the list by a specific attribute (point name, magnitude error, radial error, etc.) click the associated column header. Click it again to sort the list in the opposite direction.

Any point can be excluded from the fit by unchecking its checkbox, and any excluded point can be included again by re-checking the checkbox. Unchecked (excluded) points appear with other excluded points in the list, so it is easy to find all excluded points.

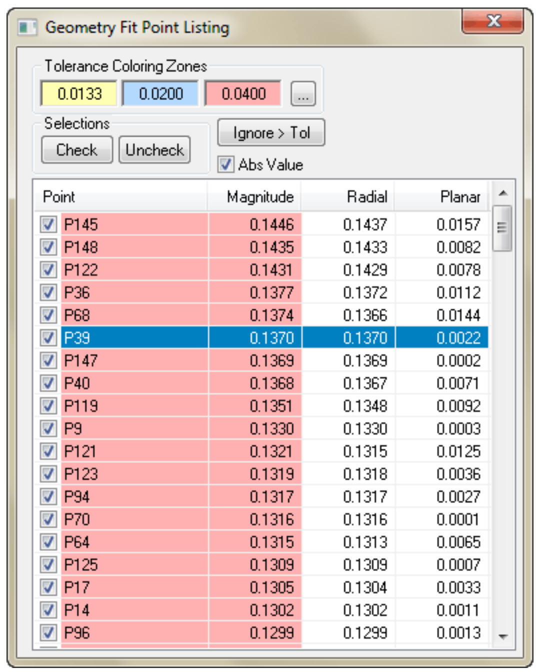


Figure 20-14. The Geometry Fit Point List.

Points that lie outside the specified tolerance in the main dialog are colored red. Points in the intermediate tolerance zones are yellow and blue, and points inside the first tolerance zone have a white background.

To Toggle Between Absolute Error and Signed Error:

- Check or uncheck the Abs Value checkbox.

To Remove Selected Points From the Fit:

1. Select one or more points in the list.
2. Do one of the following:

- Click the Check or Uncheck button to include or exclude points from the fit, or

- Right-click any of the selected points and choose **Check** or **Uncheck** from the context menu.

To Toggle the Included/Excluded State for the Selected Points in the Fit:

1. Select one or more points in the list.
2. Right-click any of the selected points and choose **Toggle**.

To Permanently Remove Selected Points From the Fit:

1. Select one or more points to remove.
2. Right-click any of the selected points and choose **Remove From List**.

To Exclude Points Beyond the Specified Error Tolerance:

- Click the Ignore > Tol button.

To Delete Selected Points from the SA Job:

1. Select one or more points to delete.
2. Right-click any of the selected points and choose **Delete From SA Job**.

To Change Limits or Colors for the Tolerance Zone Coloring:

- Click the ellipses ... button next to the zone color swatches.

Tolerance Zone Coloring

The ***Tolerance Zone Coloring*** dialog (Figure 20-15) allows you to change the specified limits and coloring for three tolerance zones.

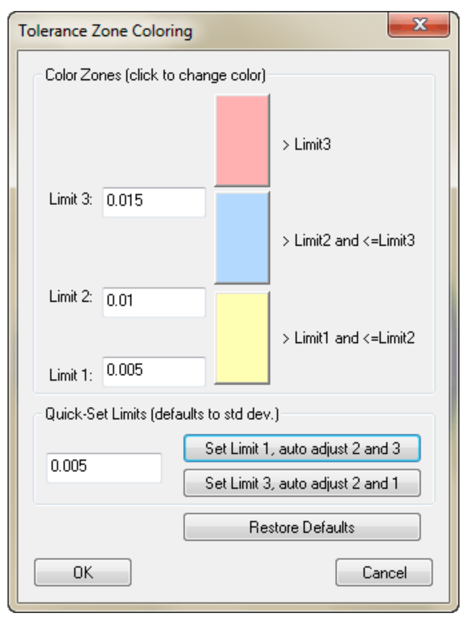


Figure 20-15. The Tolerance Zone Coloring dialog.

These zones allow you to characterize errors with three coloring zones. This is often useful for users who have a 1, 2, and 3-sigma tolerance. Each zone is bounded by limits. The first coloring zone starts at greater than *Limit 1* and ends at *Limit 2*. The second zone starts at greater than *Limit 2* and ends at *Limit 3*. The last zone starts at greater than *Limit 3*.

In the figure at left, points with errors >0.005" and <=0.01" are colored yellow; points with errors >0.01" and <=0.015" are colored blue, and points with errors >0.015" are colored red. For users who only need one coloring zone, simply pay attention to the red zone and ignore the other two, or set all three zones to the same value.

The tolerance specified in the main Geometry Fit Interface is applied as the *Limit 3* value.

The *Quick-Set Limits* section allows you to set a tolerance for Limit 1 or Limit 3, and then set the remaining limit values based on 1, 2, and 3-sigma values. Click Restore Defaults to return the values to their default limits.

Geometry Fit Graph

The ***Geometry Fit Graph*** (Figure 20-16) allows you to view an interactive graph of the errors for all points in the fit. It is an excellent way to visually find outliers.

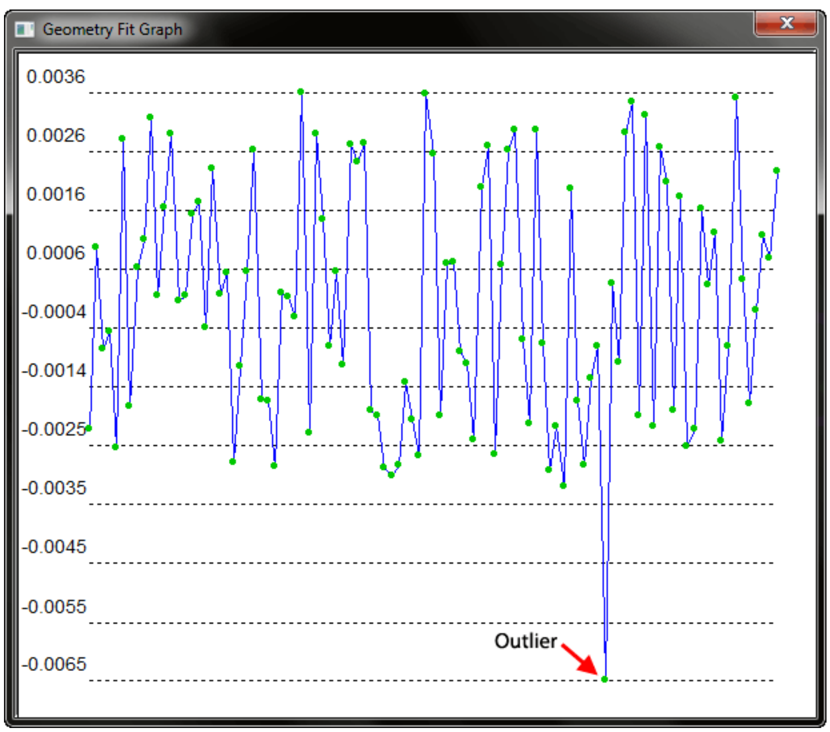


Figure 20-16. The Geometry Fit Graph makes detection of outliers easy.

Any point (outliers in particular) can be removed from the fit by way of the graph, and you can also find the point name and error for every point.

To Remove an Outlier from the Fit Using the Graph:

- Left-click the point on the graph and choose Ignore Point.

To View a Point's Name and Error:

- Left-click a point in the graph. The point name and error will appear at the top of the context menu.

In addition to a run chart of the data, you can also view a statistical histogram of the data (Figure 20-17).

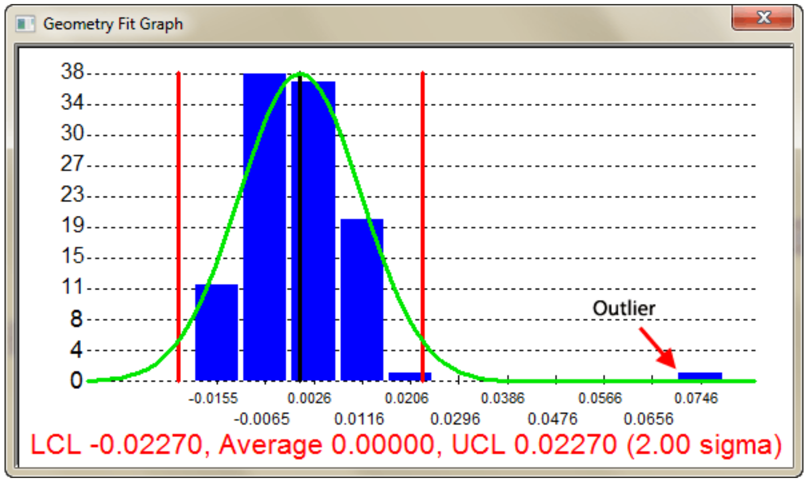


Figure 20-17. Viewing fit data in histogram form can also point to outliers or inconsistencies in data.

The histogram also includes the Lower Control Limit (LCL), average, and Upper Control Limit (UCL) at 2 sigma. As seen above, the histogram can also be used to detect an outlier, in this case located far outside the UCL.

By removing the outlier from the fit, the histogram reveals a more natural bell-shaped curve (Figure 20-18), as real-world data would typically indicate.

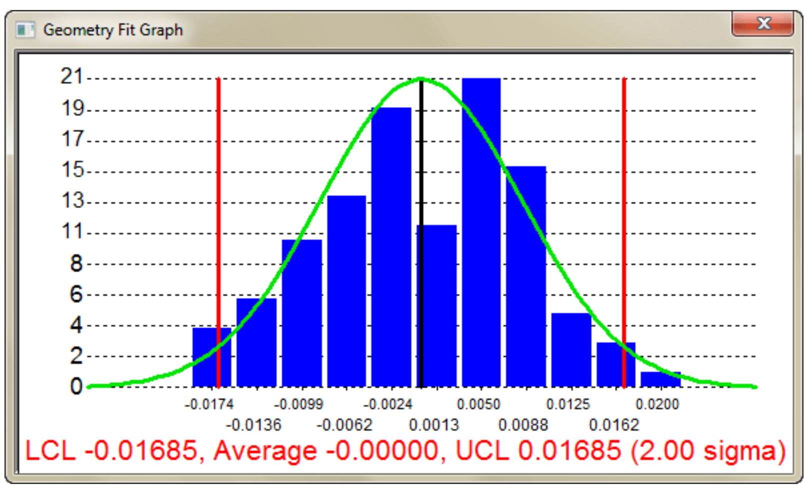


Figure 20-18. The outlier has been removed from the histogram, revealing a more natural Gaussian distribution of data.

Both the histogram and the run chart can be easily copied to the clipboard for pasting into a report or graphics program.

To Copy a Chart to the Clipboard:

- Display the desired chart, then right-click the chart and select **Copy to Clipboard**.

Queries

The **Query** menu contains numerous methods for comparing points and objects. A bulk of your analysis needs can be found under this menu item. All query commands automatically account for target offset unless otherwise noted.

The output of most query commands is either a ***Query Results*** dialog (Figure 20-19) or a vector group. The result dialog will contain query specific results and can be used to create a relationship by selecting **Create Relationship**.

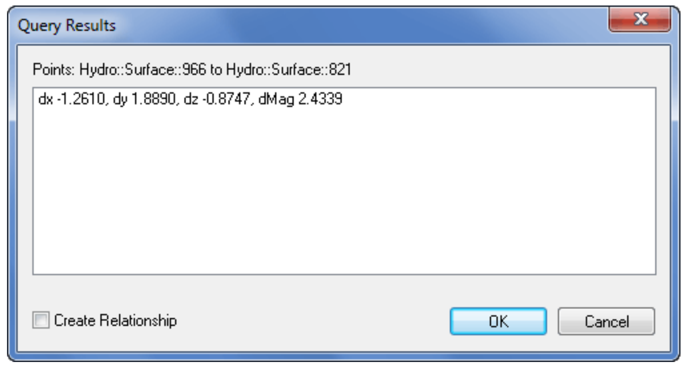


Figure 20-19. The Query Results dialog.

A vector group is a graphical depiction of the magnitude between points and objects (Figure 20-20). It is often displayed with arrows (whiskers) showing the deviation from the object to the point. Please refer to the Vector Group section for further details.

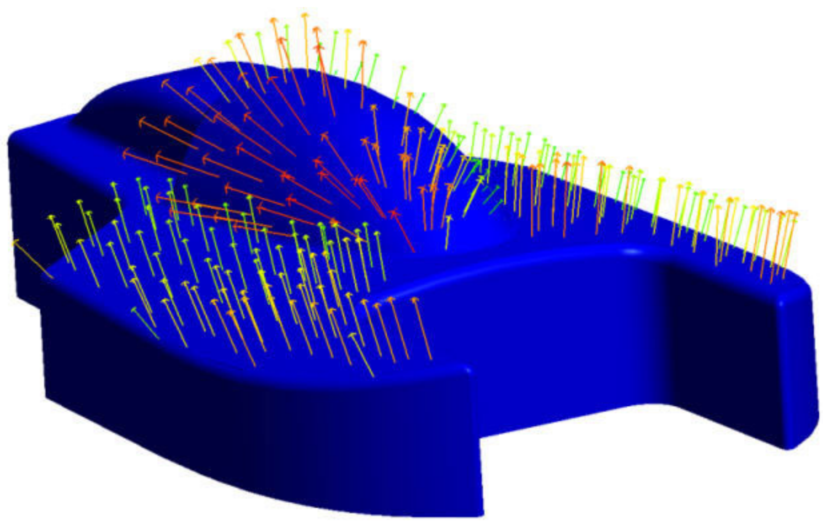


Figure 20-20. A vector group depicting error magnitude and direction.

Query Basics

When a point is compared to an object it is projected to the object using a local normal and then the difference is calculated (Figure 20-21). The below example is the result of a point to object query. The dx, dy and dz values represent the difference between the point and its projected point in working coordinates. dMag represents the magnitude. Conventionally magnitude does not carry a sign but since objects have a positive and negative side the magnitude sign can be used to determine which side of the object the point lies. Also, the objects positive side is used for determining target offset direction. The target offset is simply subtracted from the calculated distance.

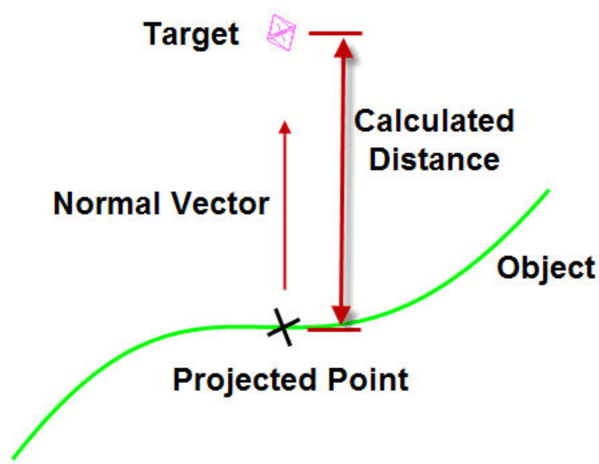


Figure 20-21. The anatomy of a query.

Query Point To Object

Displays the deviation of a point to an object. Results displayed in Working Coordinates.

Object (Ignore Offsets)

Displays the deviation of a point to an object without compensating for target offset. Results displayed in Working Coordinates.

Point

Displays deviation between two points in Working Coordinates.

Circle (ignore offsets)

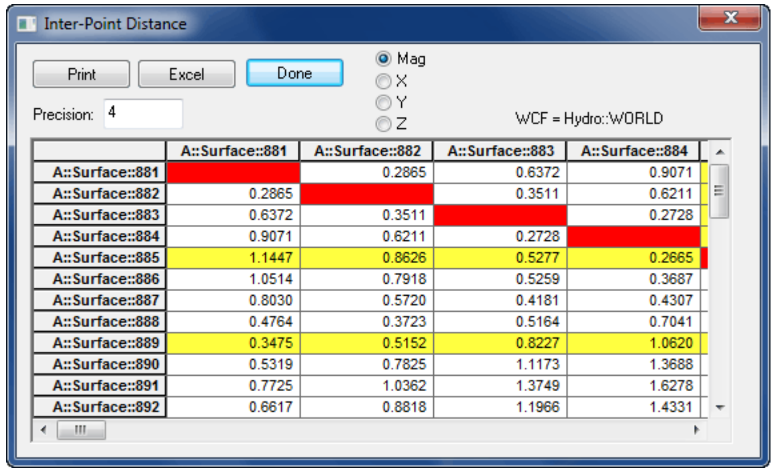
Displays the deviation of a point to a circle without compensating for target offset.

Query Points To Single Point

Creates a vector group contain deviation between selected points and single point.

Points

Points to points query results in a distance map between each combination of points (Figure 20-22).



	A::Surface::881	A::Surface::882	A::Surface::883	A::Surface::884
A::Surface::881		0.2865	0.6372	0.9071
A::Surface::882	0.2865		0.3511	0.6211
A::Surface::883	0.6372	0.3511		0.2728
A::Surface::884	0.9071	0.6211	0.2728	
A::Surface::885	1.1447	0.8626	0.5277	0.2665
A::Surface::886	1.0514	0.7918	0.5259	0.3687
A::Surface::887	0.8030	0.5720	0.4181	0.4307
A::Surface::888	0.4764	0.3723	0.5164	0.7041
A::Surface::889	0.3475	0.5152	0.8227	1.0620
A::Surface::890	0.5319	0.7825	1.1173	1.3688
A::Surface::891	0.7725	1.0362	1.3749	1.6278
A::Surface::892	0.6617	0.8818	1.1966	1.4331

Figure 20-22. An Inter-Point Distance query finds distances between all combinations of a set of points.

Curves/Objects

Generates a vector group with deviations from the selected points and the closest object (if more than one object is selected).

Circle (Radial and Planar)

Generates two vector groups, one for radial deviation and one for planar deviation.

Query Group to > Multiple Groups > Organize By Groups

Use this command to instantly compare several groups back to a reference group. This command will create multiple vector groups: One comparing the reference group to group A, another comparing the reference group to group B, and so on. This is useful for comparing drift checks or doing repeatability studies, enabling you to view deviations of multiple measured groups over time.

Multiple Groups > Organize By Points (Gage R&R)

Similar to the "Organize by Groups" command described above, except the vector groups contain the same point as seen over time. This is particularly useful for viewing the variation of a specific point over time (Figure 20-23).

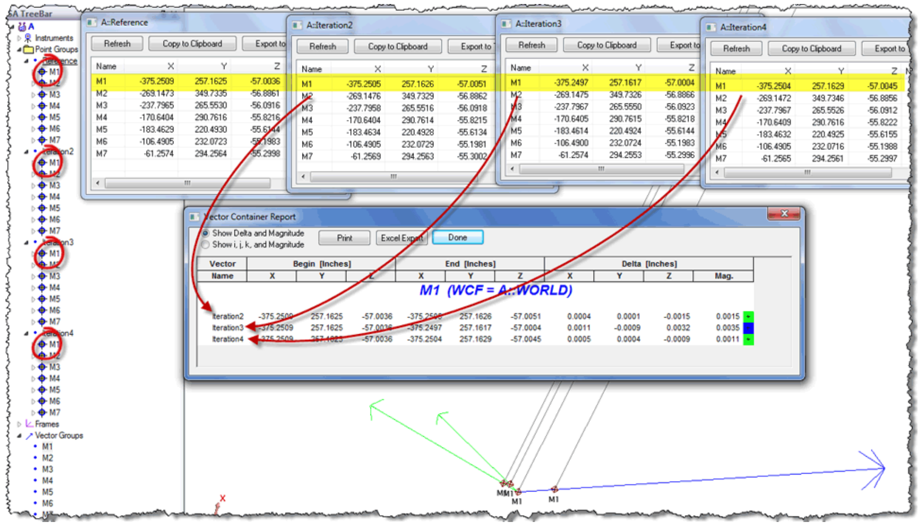


Figure 20-23. A multiple-group query organized by points.

Query Point Clouds to Objects

Generates vector group containing deviation of individual points in a point cloud to object(s).

Query Line to Line Angle

Computes angle and mutual perpendicular between two lines.

Plane Angle

Computes angle and mutual perpendicular between line and the normal vector of the plane.

Query Plane to Plane

Computes angle and mutual perpendicular between the two plane normals.

Query Frame to Frame

Computes position and rotational difference between two frames expressed in the working coordinates.

Query Object to Object Direction

Computes angle and mutual perpendicular between the two object normals.

Query Sensitivity Cloud to Sensitivity Cloud

Using the SpatialAnalyzer coordinate uncertainty computation capabilities, you can create a cloud of points around a measured point. These points represent the "cloud of uncertainty" of the coordinate. If you have clouds created for at least two points, you can determine the uncertainty of the line between them. The results are presented in Figure 20-24.

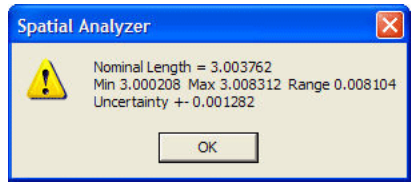


Figure 20-24. Querying one sensitivity cloud to another.

Query Auxiliary Data

Generates a vector group using auxiliary data (Figure 20-25). This allows the user to view and chart auxiliary data as you would with 3D data.

Figure 20-25. The Auxiliary Data Vector Group dialog.

Best Fit Points to Surfaces

Vector Groups

A vector group is a graphical depiction of the magnitude between points and objects. It is often displayed with arrows (whiskers) showing the deviation from the object to the point (Figure 20-26).

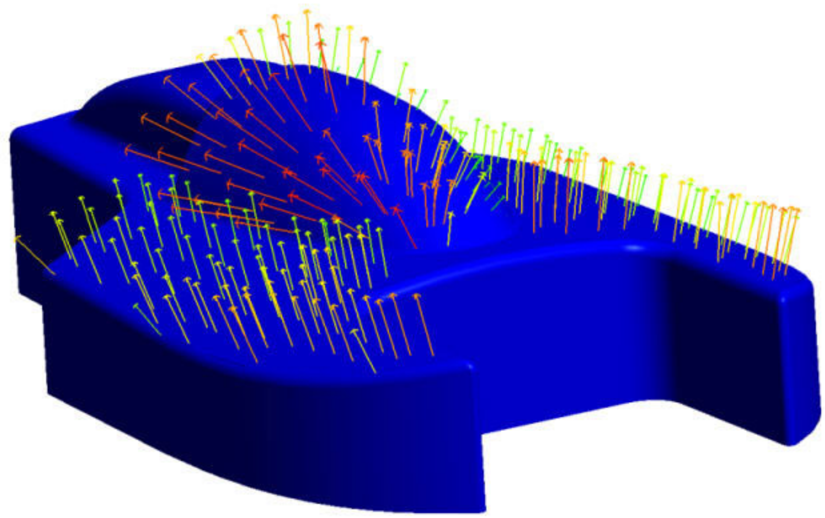


Figure 20-26. A vector group.

By Clicking Surfaces

Creates a vector group from clicking on surface face.

From Existing Vectors

Creates a new vector group from selection of individual vectors.

From Existing Vector Groups

Creates a new vector group from selection of existing vector groups.

Scale Existing Vector Groups

Creates a new vector group scaled about the selected frame. User may select a scale factor for each component of the vectors (Figure 20-27).

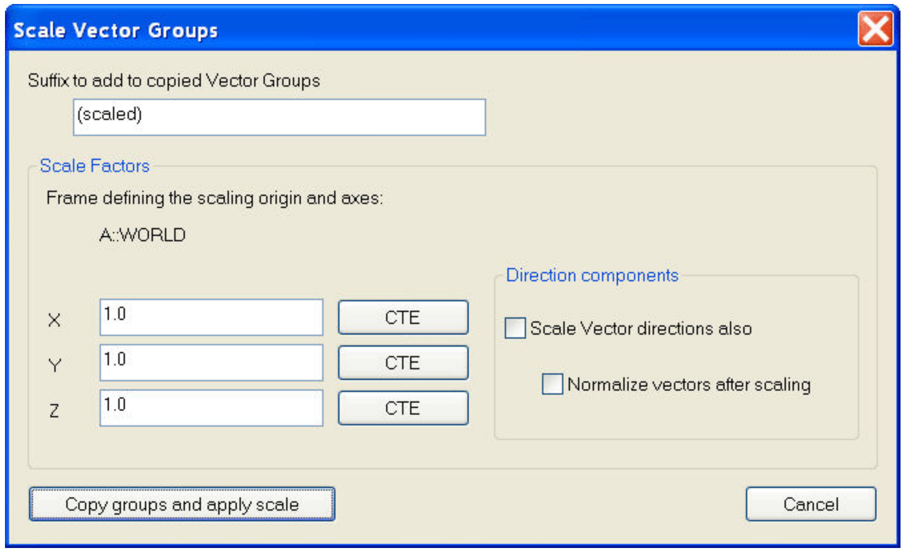


Figure 20-27. The Scale Vector Groups dialog.

Copy Vector Groups Excluding Obscured Vectors

Creates new vector group only coping visible vectors. Vectors blocked from current point of view are not copied.

Area Profile Check From Point Measurement Probing Directions

Creates a vector group representing the probing direction stored for each selected point.

Projection Options

The projection options dialog (Figure 20-28) can be seen in several places in SA but is mainly used in Query Points to Objects and relationships. The projection options allow the user to control vector group direction, add material offsets, create offset point groups, and change probe offset compensation.

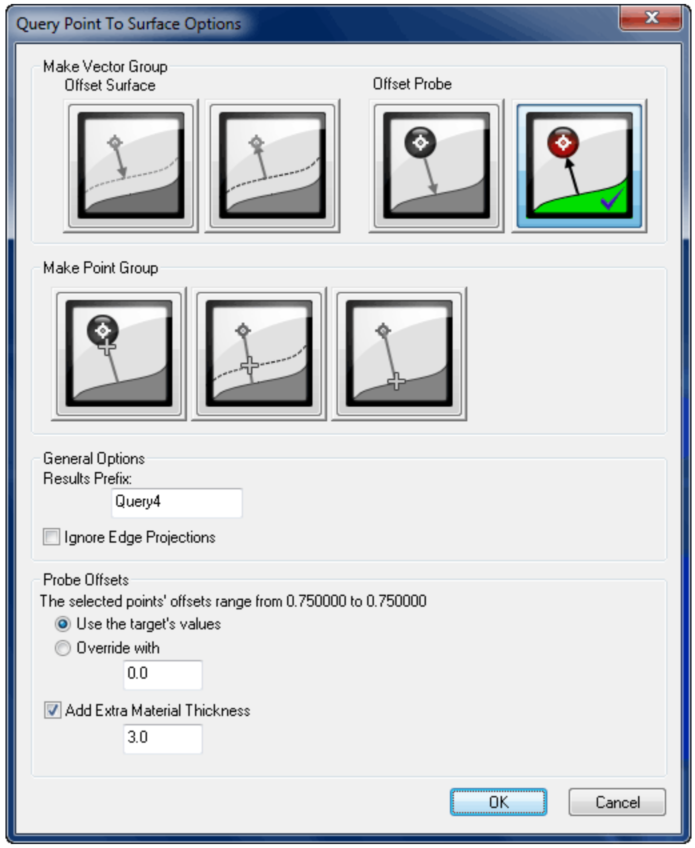


Figure 20-28. The Query Point To Surface Options dialog.

Offset Probe

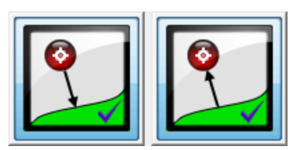


Figure 20-29. The Offset Probe vector group options.

These two options (Figure 20-29) create vector groups depicting deviation of the surface from the probe (left) and of the probe from the surface (right, the default option). Both of these options create vectors with the exact same magnitude but exactly opposite in direction. The left option therefore depicts a typical "build" perspective (how far do I need to go to get to nominal?), whereas the right option depicts a typical "inspect" perspective (how far am I from nominal?).

Offset Surface

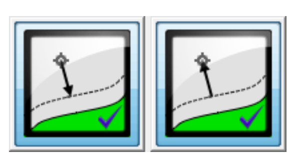


Figure 20-30. The Offset Surface vector group options.

These options (Figure 20-30) create vector groups with the head or tail on the center of the target and all offsets applied at the object. This is sometimes used with thin parts to prevent the vector from disappearing behind the nominal surfaces.

Make Point Group

Figure 20-31. Projection options for making point groups.

These options (Figure 20-31) create point groups instead of vectors. The first option applies any offset at the probe center. Unless additional material thickness is added, this represents the probe's contact point on the surface. The second option applies all offsets at the surface, and the third option creates projected points on the object to which the query is being performed.

Ignore Edge Projections

The **Ignore edge projections** option is used when you would like to ignore points whose projections onto objects lie on a surface edge (Figure 20-32).

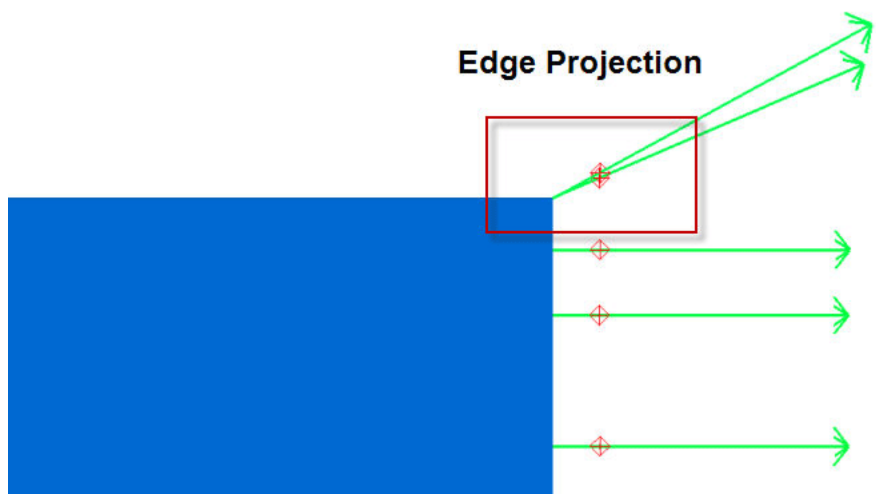


Figure 20-32. Two points which are edge projections.

Probe Offsets

The geometry offset allows the user to use the point specific offset or override the offset (Figure 20-33). The "Use the target's values" will display the offset of the selected points.

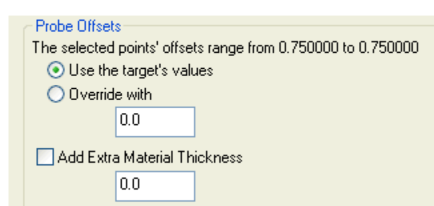


Figure 20-33. Probe offset options.

Vector Group Properties

The vector group properties allow the user to view vector group statistics as well as change the graphical appearance (Figure 20-34).

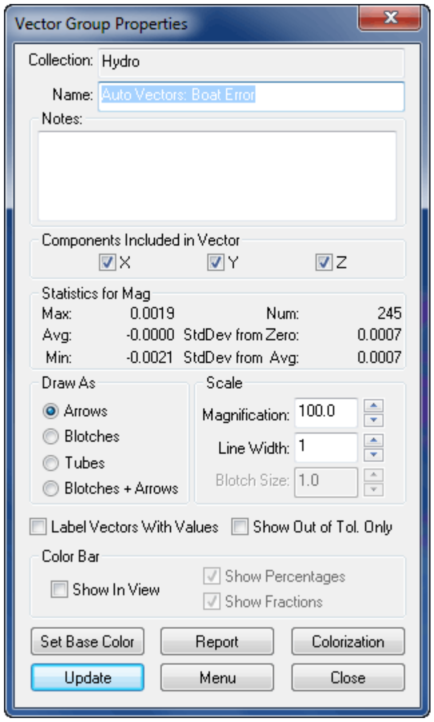


Figure 20-34. The properties of a Vector Group.

Display Settings

- **Label Vectors With Values.** Display each vector's magnitude (Figure 20-35).

Figure 20-35. A set of vectors that have been labeled with values.

- **Vector Magnification.** Magnifies the graphical length of the vector. Improves viewing when deviation is small.
- **Line Width.** Increase the graphical width of the vector for improved viewing.
- **Show Out of Tol Only.** Only displays vectors that are out of tolerance.
- **Draw As.** Draws the vectors as arrows, blotches, tubes, or blotches with arrows (Figure 20-36).

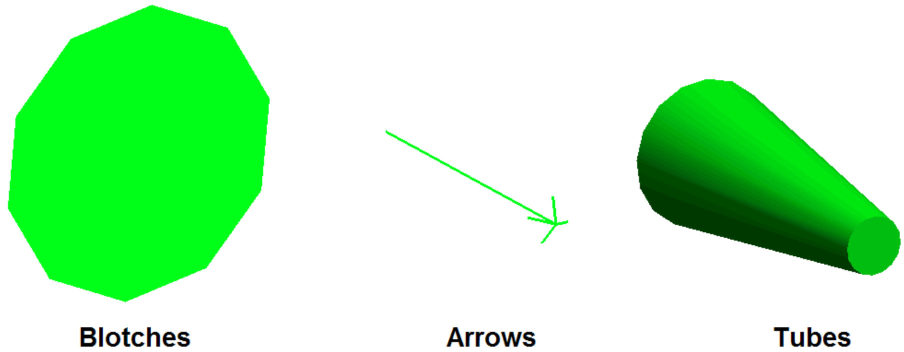


Figure 20-36. Different methods for depicting error.

- **Color Bar Appearance.** Displays color bar and options for color bar display (Figure 20-37).

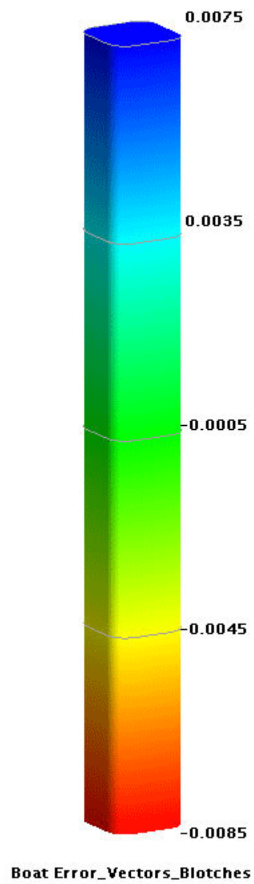


Figure 20-37. A vector group color bar.

Edit Colorization

- **Tolerance.** Set the High and Low Tolerance as needed. Use Auto-Range Limits to set the High and Low Saturation Limits.
- **Color Ranging Method.** Pick from five preset color ranges.

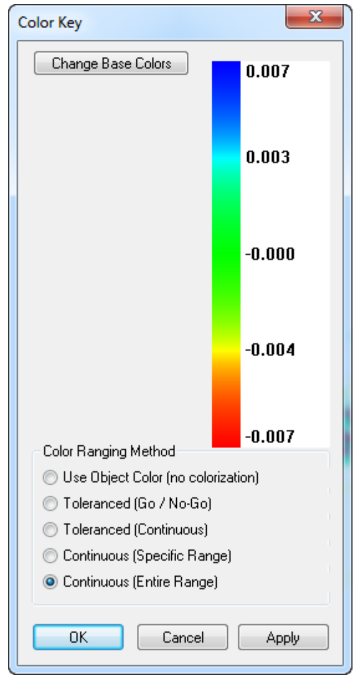
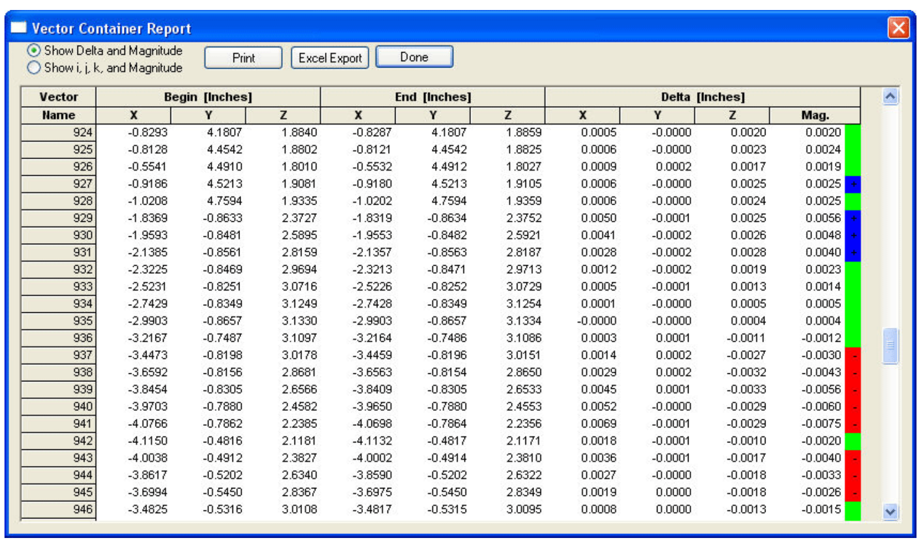


Figure 20-38. Vector group colorization options.

Report

The Report button will generate a report showing each vectors begin, end and delta coordinates (Figure 20-39). It also displays magnitude with the respective colorization. This report can be printed or exported from this dialog. This report can also be generated in other formats such as Quick and Composite reports.

Each column can be sort ascending or descending when name value is double left-clicked.



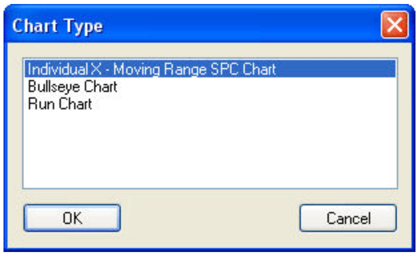
Vector	Begin [Inches]			End [Inches]			Delta [Inches]				
Name	X	Y	Z	X	Y	Z	X	Y	Z	Mag.	
924	-0.8293	4.1807	1.8840	-0.8287	4.1807	1.8859	0.0005	-0.0000	0.0020	0.0020	
925	-0.8128	4.4542	1.8802	-0.8121	4.4542	1.8825	0.0006	-0.0000	0.0023	0.0024	
926	-0.5541	4.4910	1.8010	-0.5532	4.4912	1.8027	0.0009	0.0002	0.0017	0.0019	
927	-0.9186	4.5213	1.9081	-0.9180	4.5213	1.9105	0.0006	-0.0000	0.0025	0.0025	+
928	-1.0208	4.7594	1.9335	-1.0202	4.7594	1.9359	0.0006	-0.0000	0.0024	0.0025	
929	-1.8369	-0.8633	2.3727	-1.8319	-0.8634	2.3752	0.0050	-0.0001	0.0025	0.0056	+
930	-1.9593	-0.8481	2.5895	-1.9553	-0.8482	2.5921	0.0041	-0.0002	0.0026	0.0048	+
931	-2.1385	-0.8561	2.8159	-2.1357	-0.8563	2.8187	0.0028	-0.0002	0.0028	0.0040	+
932	-2.3225	-0.8469	2.9694	-2.3213	-0.8471	2.9713	0.0012	-0.0002	0.0019	0.0023	
933	-2.5231	-0.8251	3.0716	-2.5226	-0.8252	3.0729	0.0005	-0.0001	0.0013	0.0014	
934	-2.7429	-0.8349	3.1249	-2.7428	-0.8349	3.1254	0.0001	-0.0000	0.0005	0.0005	
935	-2.9903	-0.8657	3.1330	-2.9903	-0.8657	3.1334	-0.0000	-0.0000	0.0004	0.0004	
936	-3.2167	-0.7487	3.1097	-3.2164	-0.7486	3.1086	0.0003	0.0001	-0.0011	-0.0012	
937	-3.4473	-0.8198	3.0178	-3.4459	-0.8196	3.0151	0.0014	0.0002	-0.0027	-0.0030	-
938	-3.6592	-0.8156	2.8681	-3.6563	-0.8154	2.8650	0.0029	0.0002	-0.0032	-0.0043	-
939	-3.8454	-0.8305	2.6566	-3.8409	-0.8305	2.6533	0.0045	0.0001	-0.0033	-0.0056	-
940	-3.9703	-0.7880	2.4582	-3.9650	-0.7880	2.4553	0.0052	-0.0000	-0.0029	-0.0060	-
941	-4.0766	-0.7862	2.2385	-4.0698	-0.7864	2.2356	0.0069	-0.0001	-0.0029	-0.0075	-
942	-4.1150	-0.4816	2.1181	-4.1132	-0.4817	2.1171	0.0018	-0.0001	-0.0010	-0.0020	
943	-4.0038	-0.4912	2.3827	-4.0002	-0.4914	2.3810	0.0036	-0.0001	-0.0017	-0.0040	-
944	-3.8617	-0.5202	2.6340	-3.8590	-0.5202	2.6322	0.0027	-0.0000	-0.0018	-0.0033	-
945	-3.6994	-0.5450	2.8367	-3.6975	-0.5450	2.8349	0.0019	0.0000	-0.0018	-0.0026	-
946	-3.4825	-0.5316	3.0108	-3.4817	-0.5315	3.0095	0.0008	0.0000	-0.0013	-0.0015	

Figure 20-39. Viewing a vector group report.

Chart

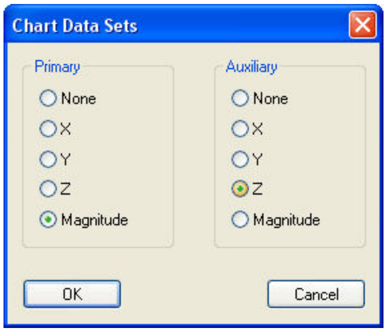
The chart button will generate one of three predefined charts (Figure 20-40).

Figure 20-40. Chart Type.



The user can select the inputs for the chart type (Figure 20-41):

Figure 20-41. Axis settings for the charts.



Once the chart is generated (Figure 20-42) it is saved in the SA Treebar.

Figure 20-42. The resulting generated chart.

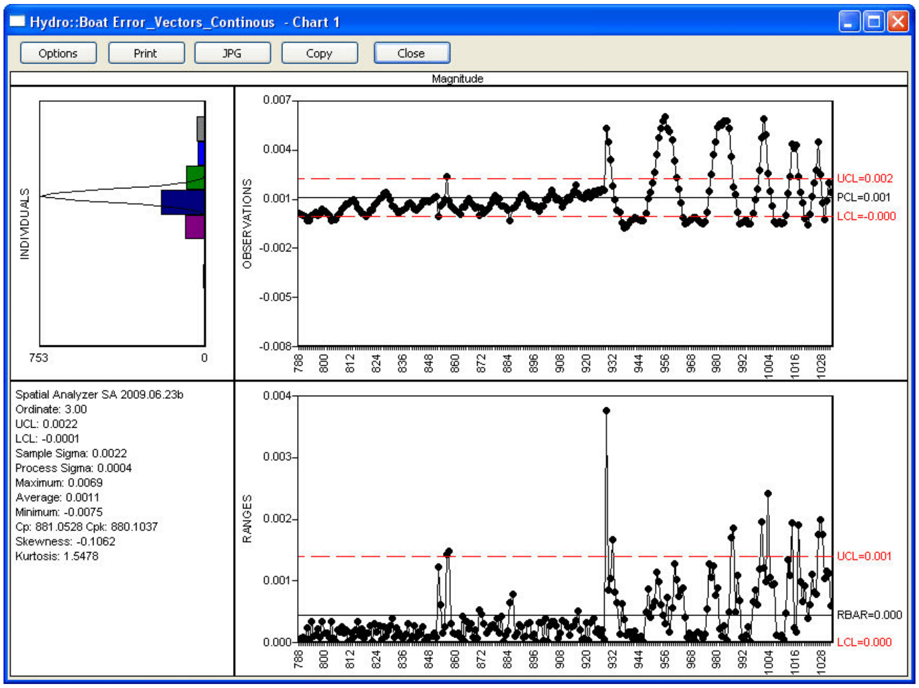


Chart Options

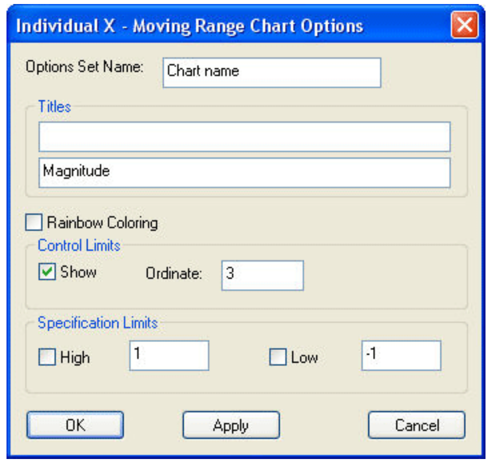


Figure 20-43. The options for a moving range chart.

Relationships

Relationships can be used for automatically updating dynamic changes between objects, make creating vector groups for analysis faster, provide a quick overview of the status of build operations and enable you to optimize your measurement directly against design constraints.

Relationships can also be used as a fitting tool. By optimizing relationships a user can transform points and objects to an optimal position. Please see the Minimizing Relationships section as it is a feature of SA Ultimate.

Relationship Types

Relationships can cover a combination of point, points and object comparisons.

Point to Point

Reports deviation between two points.

Point to Object

Reports deviation between a point and object.

Points to Objects

Report deviation between multiple points and multiple objects.

Point Clouds to Objects

Reports deviation between point clouds and multiple objects.

Groups to Objects

Report deviation between multiple groups and multiple objects.

Frame to Frame

Reports a delta transform between two frames expressed in the working frame.

Group to Group

Report deviation between points in two point groups.

Object to Object Direction

Reports the angle and mutual perpendicular between two object normals.

Geometry Comparison

Functionality includes comparing measured points to nominal object, comparing object to object or fitting measured points. See the Geometry Relationships section below for more information.

Relationship Properties

Most relationships have properties that can effect how a relationship is used for fitting and reporting. Here the user can control tolerances, projection properties, etc. (Figure 20-44).

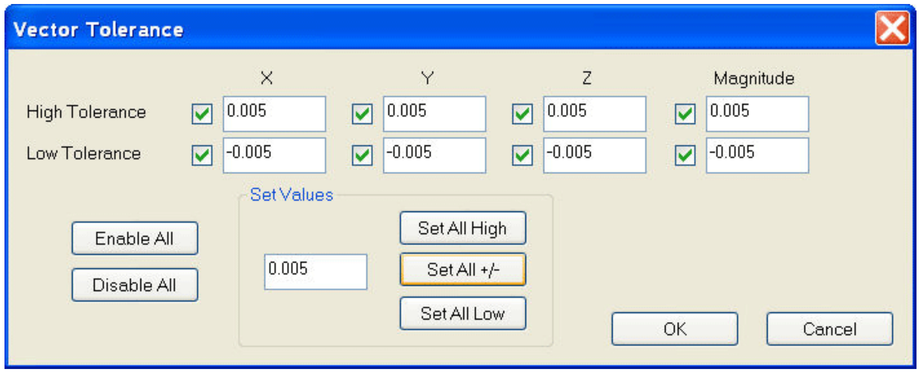


Figure 20-44. The Vector Tolerance dialog.

Report Options control how the deviations will be reported (Figure 20-45).

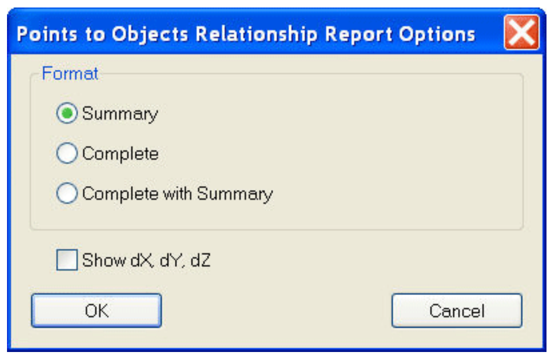
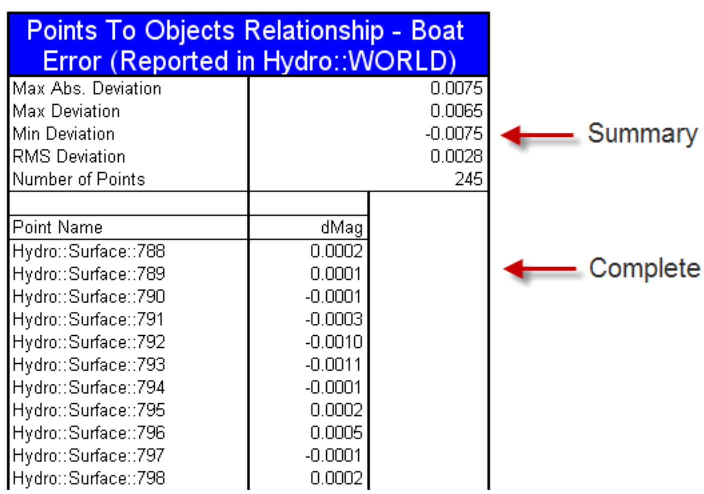


Figure 20-45. Relationship report options.

The summary contains a snapshot of the relationship (Figure 20-46). The complete options will display all the individual deviations.



Points To Objects Relationship - Boat Error (Reported in Hydro::WORLD)

Max Abs. Deviation	0.0075
Max Deviation	0.0065
Min Deviation	-0.0075
RMS Deviation	0.0028
Number of Points	245

Point Name	dMag
Hydro::Surface::788	0.0002
Hydro::Surface::789	0.0001
Hydro::Surface::790	-0.0001
Hydro::Surface::791	-0.0003
Hydro::Surface::792	-0.0010
Hydro::Surface::793	-0.0011
Hydro::Surface::794	-0.0001
Hydro::Surface::795	0.0002
Hydro::Surface::796	0.0005
Hydro::Surface::797	-0.0001
Hydro::Surface::798	0.0002

Figure 20-46. Relationship summary.

Fit settings

The Relationship Fit settings are covered under the Minimizing Relationships section.

Auto Update Vector Group

This option allows a dynamic vector group to be created. Any time the relationship updates the vector group will update as well.

Geometry Relationship Overview

Geometry Relationships allow for dynamic fitting and comparison of geometry. Three relationship types allow the user to "Fit and Compare to Nominal", "Fit Only" and "Compare Only". Relationships are dynamic in nature and update when associated data moves. So as points are measured, moved, updated, etc…the relationship will re-compute, updating geometry and comparisons.

Geometry Relationship Properties

The Relationship Properties consist of the geometry specific fit options. Right-click the relationship and select properties, the respective fit options will appear (Figure 20-47).

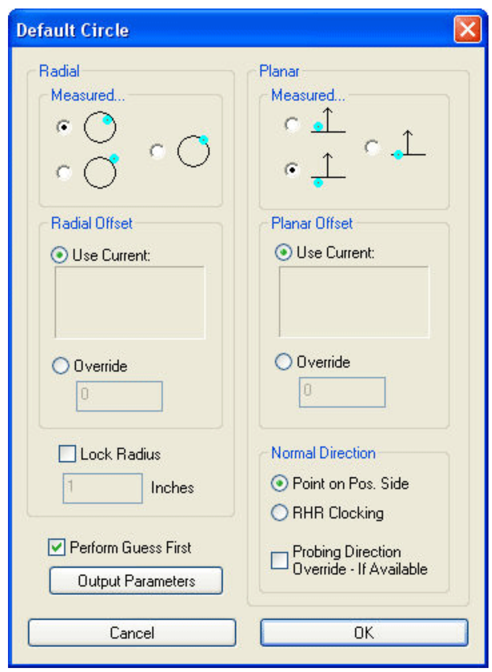


Figure 20-47. Geometry relationship properties.

Report Options

The report options for the relationship will allow the user to set many parameters (Figure 20-48). The interface will allow the user to choose which components of the fit or comparison to report. In addition, tolerance and relationship optimization settings can be accessed.

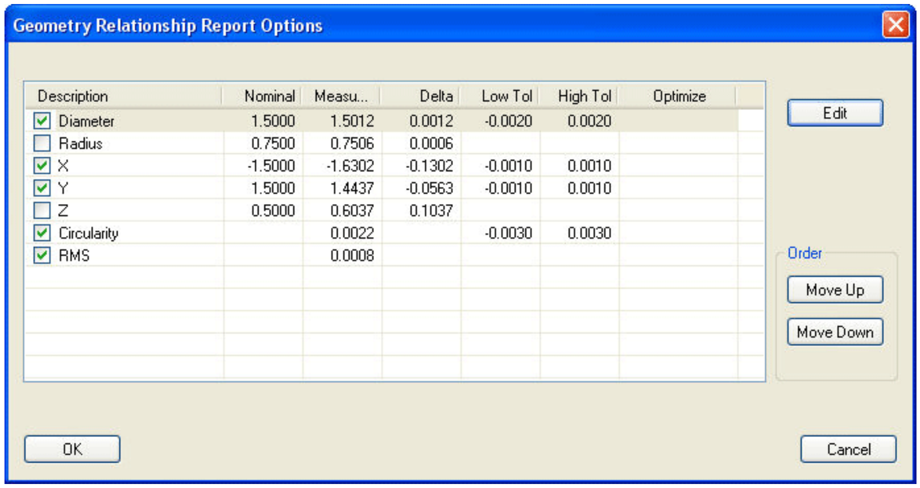


Figure 20-48. Geometry relationship report options.

Each component has the ability to be used in a relationship minimization, allowing a user to fine tune a relationship fit based on geometry components (Figure 20-49). The user has the ability to minimize based on delta and/or the amount out of tolerance. This allows the fit to drive the delta to zero and/or bring the amount out of tolerance to zero.

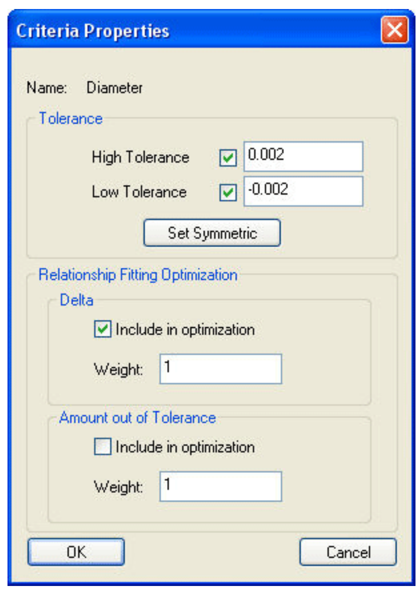


Figure 20-49. Adjusting geometry relationship criteria.

Fit and Compare to Nominal

Specify a nominal geometry and points as inputs and the respective geometry will be fit and compared. For example, specify a nominal circle and measured points. A measured circle will be fit and compared to the nominal (Figure 20-50). Adding a callout for the relationship will yield the comparison.

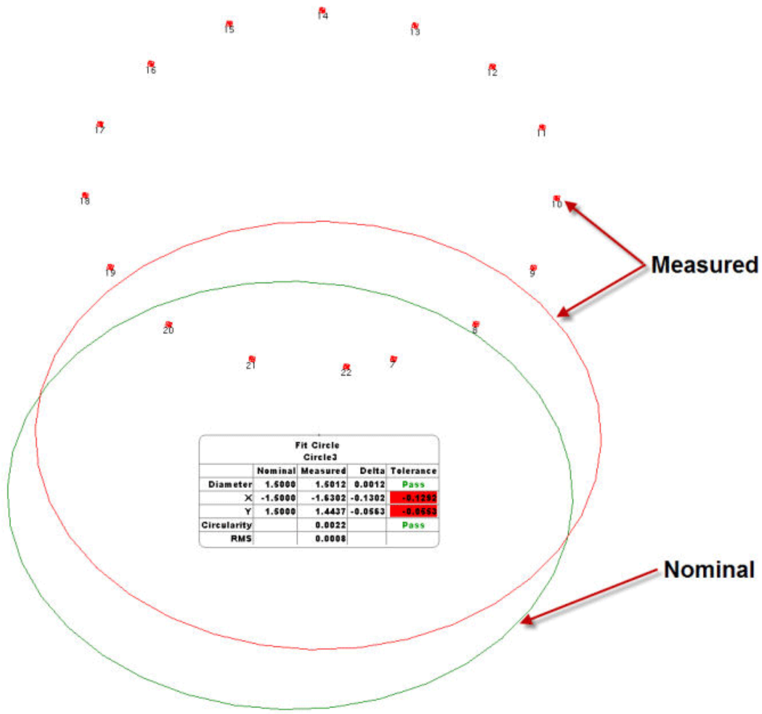


Figure 20-50. A Fit and Compare to Nominal geometry relationship.

Fit Only

Specify points for fit and choose geometry type to fit. New geometry will be created (Figure 20-51).

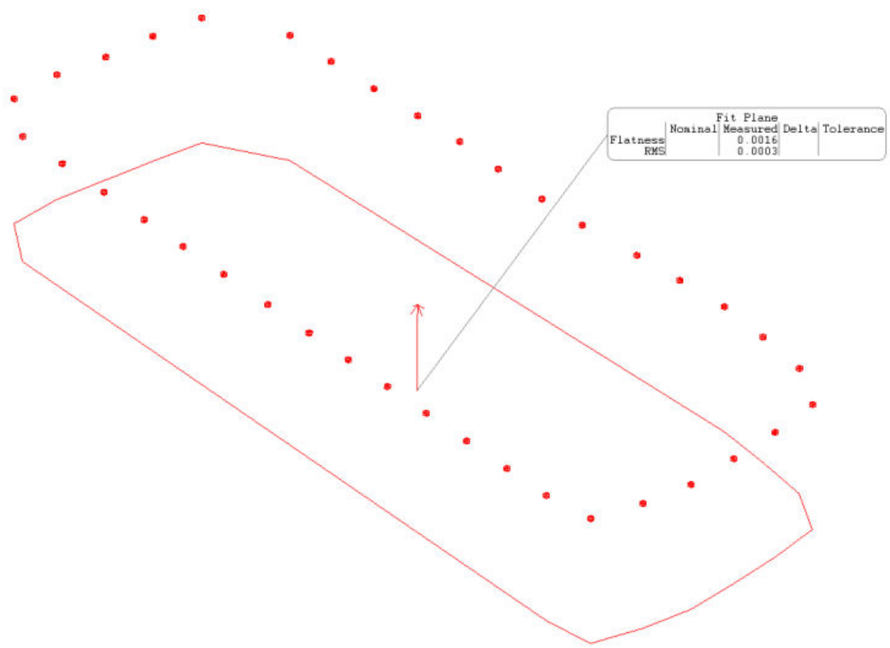


Figure 20-51. A Fit Only geometry relationship.

Compare Only

Select two geometries of the same type to compare (Figure 20-52).

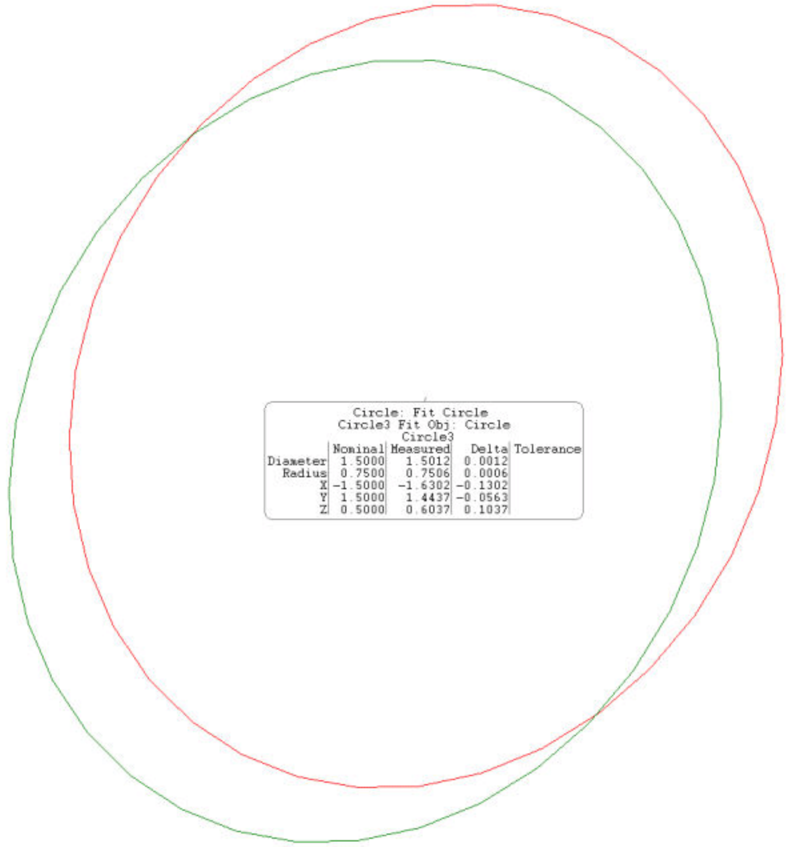


Figure 20-52. A Compare Only geometry relationship.

Minimizing Relationships

See section Minimizing Relationships.

Uncertainty Analysis

Coordinate Uncertainty

SpatialAnalyzer provides powerful coordinate sensitivity analysis functions. Essentially, uncertainty is determined by injecting normally distributed, random error into the measurement system and evaluating the effect of the error on the coordinate data. The amount and type of error that is inserted is based on the precision of the measuring device.

Uncertainty Analysis and Visualization

SpatialAnalyzer is able to compute and report measurement uncertainty estimates for each of the instruments it interfaces to. Looking through a long list of computed uncertainty estimates often provides little insight into the specific effects that it might be having on the measurement job. To enable users to make tangible use of the uncertainty estimates, SpatialAnalyzer can generate a cloud of points (around a target) that represents the target's region of uncertainty. The point cloud is called a sensitivity cloud.

Creating Point Uncertainty Fields

Uncertainty fields can be created for selected points (Figure 20-53). The uncertainty is calculated by using the respective instrument uncertainty model and the number of samples entered.

Figure 20-53. Uncertainty fields for several measurements.

The uncertainty values can be accessed via the point properties or a report. The uncertainty cloud display properties can be changed under the User Options.

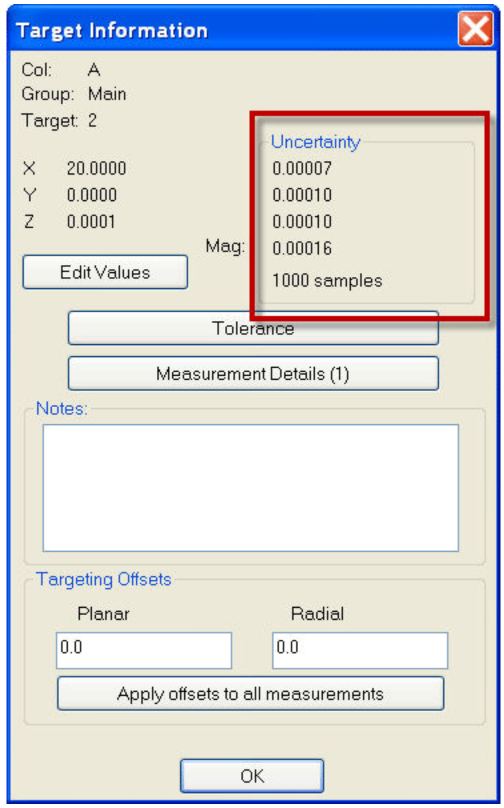


Figure 20-54. Uncertainties as listed in the Target Information dialog.

Deleting Uncertainty Fields

Use **Delete selected Point Uncertainty Fields** to delete individual point uncertainties or use **Delete All Point Uncertainty Fields** to delete all calculated point uncertainties for the SA job.

Thinning Point Uncertainty Fields

Reduced the amount of samples for selected points.

GD&T Inspection

This feature of SA supports the engineer's ability to read and implement GD&T design from CAD drawings. The GD&T design gives the measurements relativity, and once those points are known, the data can be arranged. SA allows the technician to encode CAD with annotations indicating curvature, diameter or true position. In SA, you go measure then validate the code's pass or failure. The feature checks assign and validate measurements, and you can also generate reports. Another feature is that in addition to process flow, manual creation is allowed with imported CAD files that need to be annotated. So it is not only compatible with GD&T notations, a user can create them on files when necessary, making it easier to implement GD&T.

With SA, engineers and mechanical technicians can more easily implement and integrate GD&T into their daily work, using the tools it provides, as well as work on their own customized annotations. GD&T has been around for the past 36 years, and it will continue to change the industry and the way that measurement is conveyed.

Import CAD with GD&T annotations

To the import a CAD model with annotations use **File>Import>Direct CAD Access**. See Direct CAD Access for supported formats.

In the ***Direct CAD Access Settings*** dialog (Figure 20-55), check the **Annotations** option.

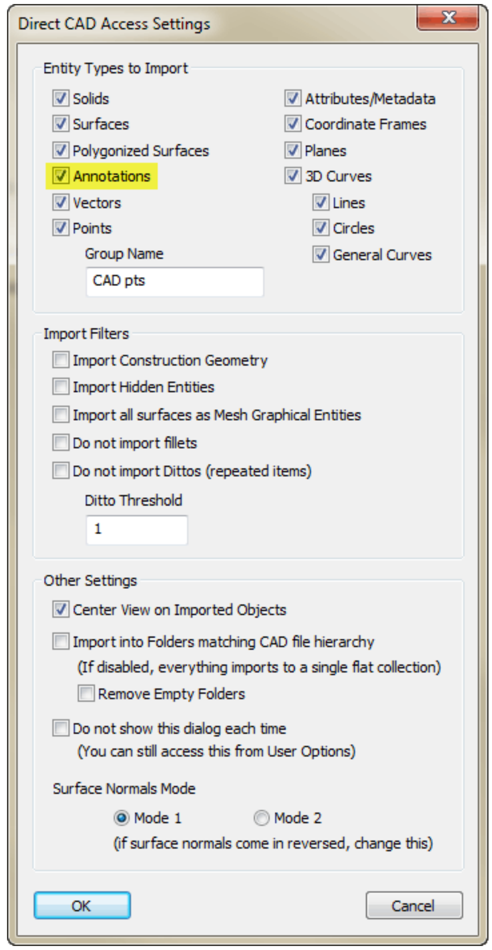


Figure 20-55. Direct CAD Access import settings.

Once imported, the annotations will appear graphically and in the tree under Annotations (Figure 20-56).

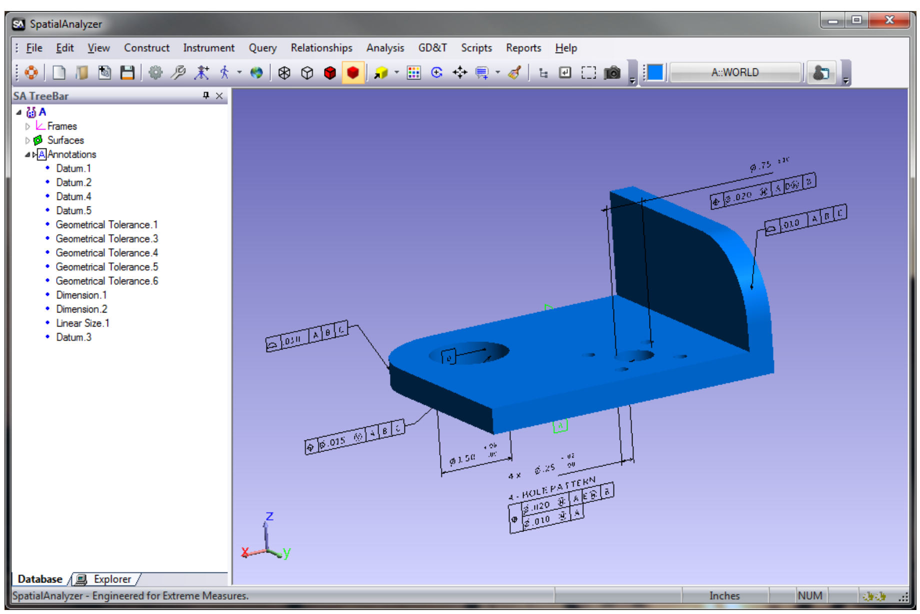


Figure 20-56. Annotations have been successfully imported.

Creating GD&T annotations

If a CAD file with GD&T annotations is unavailable GD&T annotations can be created/modified inside of SA using the **GD&T** menu, or the GD&T Toolbar can be used.

When creating a Datum or Feature check from the tool bar, the user will be asked to select the respective surface faces. The annotation editor will be appear allowing the annotation parameters to be entered.

GD&T Datum Annotation Properties

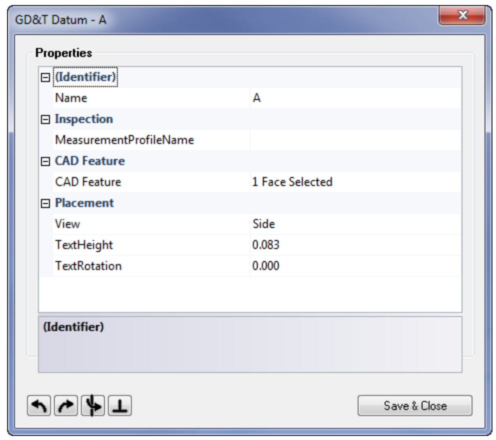


Figure 20-57. Annotation properties.

- **Name.** Datum name used for graphical annotation.
- **Measurement Profile Name.** Sets the measurement mode for the selected instrument when the Datums is created for inspection.
- **CAD Feature.** Prompts user to select CAD faces for the respective feature.
- **View.** Changes orientation of annotation.
- **Text Height.** Defines the height of the annotation text. Expressed in job units.
- **Text Rotation.** Defines rotation in the current view.

GD&T Feature Check Annotation Properties

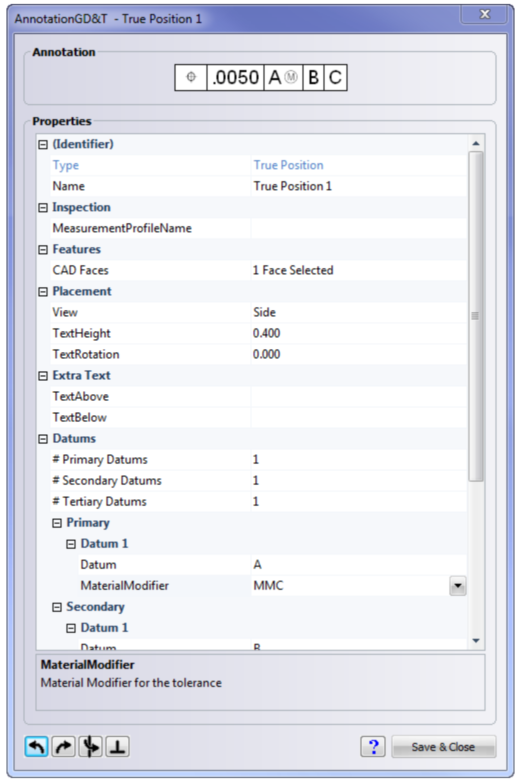


Figure 20-58. Properties for a True Position check.

- **Type.** Feature check type.
- **Name.** The name of the check used to describe the annotation in the SATreebar and Reporting.
- **Measurement Profile Name.** Sets the measurement mode for the selected instrument when the Datums is created for inspection.
- **CAD Feature.** Prompts user to select CAD faces for the respective feature.
- **View.** Changes orientation of annotation.
- **Text Height.** Defines the height of the annotation text. Expressed in job units.
- **Text Rotation.** Defines rotation in the current view.
- **Text Above and Text Below.** Adds additional text above and below the annotation.
- **Number of Primary, Secondary and Tertiary Datums.** Enter the respective Primary, Secondary and Tertiary Datums if applicable.
- **Material Modifiers.** Defines the material modifier for the selected datum. Choose from Maximum Material Condition, Least Material Condition and Regardless of feature size.

- **Zone.** Applicable tolerance zone
- **Tolerance.** Tolerance value for the feature check.
- **Material Modifiers.** Defines the material modifier for the feature check selected datum. Choose from Maximum Material Condition, Least Material Condition and Regardless of feature size.
- **Has Projected Zone.** If checked, the projected zone will be used.
- **Projected Zone Value.** Value used when Projected zone is being used.

Figure 20-59 illustrates how the settings in the property grid relate to the annotation graphic:

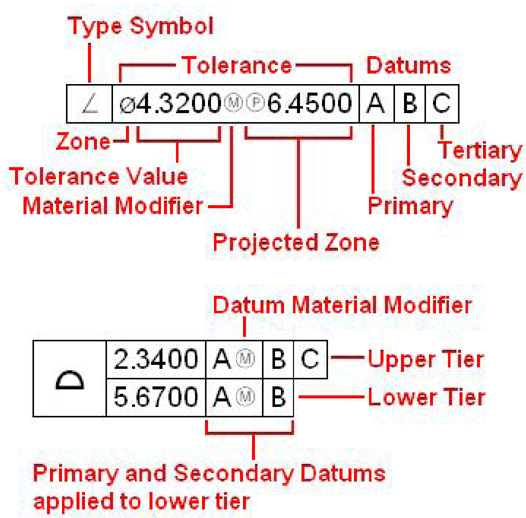


Figure 20-59. Anatomy of a GD&T annotation.

Graphical Placement of Annotations

With the annotation editor open the graphical annotation can be moved and placed in a position that is easy for the user to read. The annotation leader line is defined when the first surface face is selected under the CAD Features parameter. To move the annotation simply select the annotation in the graphics by left-clicking the annotation. While holding the left mouse button the annotation can be moved in the plane of the selected view. The view can be change under the View parameter in the annotation editor. Controls at the bottom of the annotation editor help in fine tuning the position and orientation of the annotation.

- Rotates the annotation box 90 degrees counter clockwise.
- Rotates the annotation box 90 degrees clockwise.
- Mirrors the annotation box.
- Straightens the leader line normal to the selected surface

face.

While moving the annotation, **Alt** can be combined with the left mouse button to move the annotation along the leader line.

Quick Annotation Creation

The GD&T tool bar (Figure 20-60) has several methods that aid in creating annotations quickly. In the tool bar the user can predefine view, tolerance, and datums. So once the surface faces are chosen the annotation properties are populated.

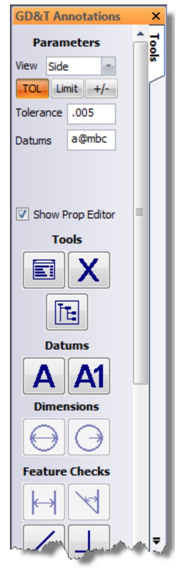


Figure 20-60. The GD&T annotations dialog.

Inspecting a GD&T Feature

To inspect a feature simply right-click and select make feature check (Figure 20-61). The annotation will be decoded and the appropriate datum and feature checks will be created.

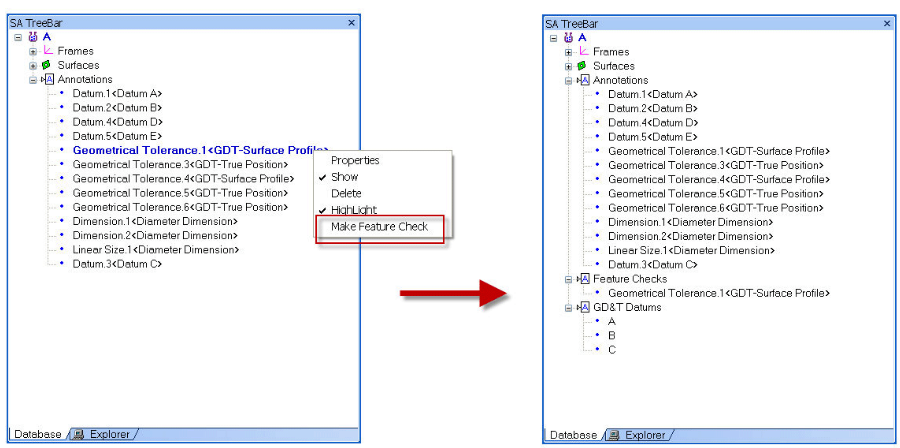


Figure 20-61.

Now measurements need to be associated to the datums and feature checks. There are two ways to associate data, manually associate or trap measurements.

To associate data right click the respective datum or feature check and select **Associate Data** (Figure 20-62). The user will prompted to select the points.

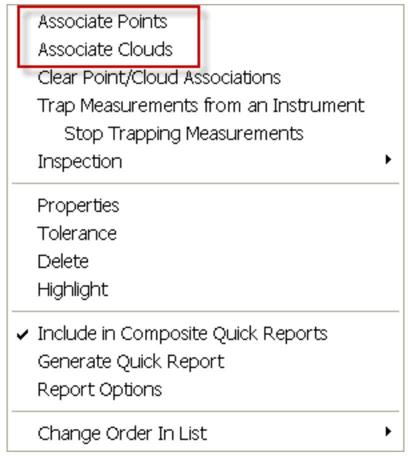


Figure 20-62. Associating data.

Trapping measurements allows any measurement coming from an instrument to be associated to that datum or feature check. Right-click **Trap Measurments from an instrument** to start trapping and **Stop Trapping Measurements** to stop the association (Figure 20-63).

Figure 20-63. Choosing to trap measurements.

To clear point associations right-click and select **Clear Point/Cloud Associations**.

Reporting

Once all measurements are associated the inspection is complete. Use the report bar to view each feature check result (Figure 20-64). To create a complete report, right-click the feature check category and select "Composite Quick Report".

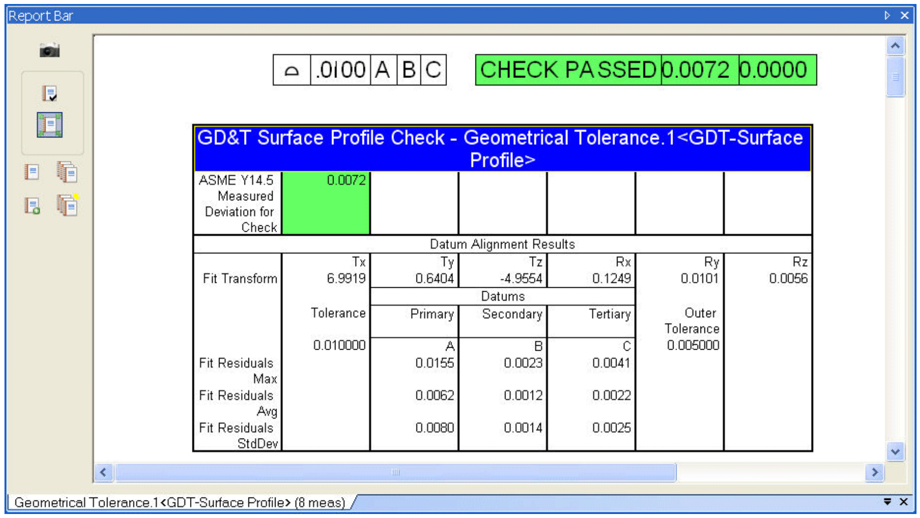


Figure 20-64. An evaluated GD&T check in the Report Bar.

Alignment to Part

When performing a GD&T inspection the graphical alignment to the part is usually not important. During the evaluation of the feature check the data must be aligned to the respective datums. Therefore, the alignment will always be changing. For a "feel good", the data can be aligned to any of the datums. Right-click the datum category and select "align". Here the user can select the datums for alignment (Figure 20-65). This is for visual purposes only, it will not be used during the evaluation of the features.

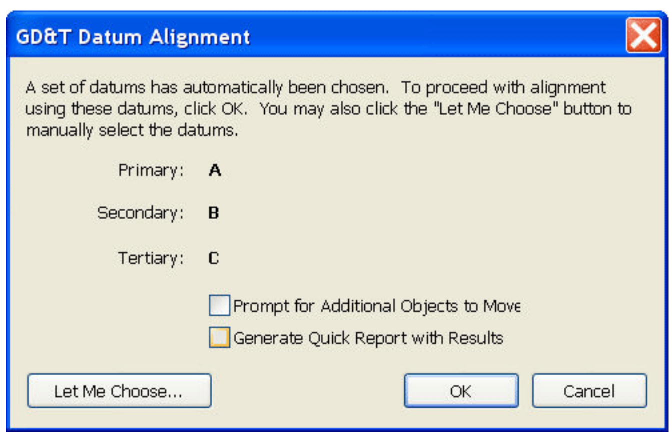


Figure 20-65. Aligning to GD&T datums.

Guided Inspection

A inspection plan can be created for GD&T. The user can specific nominal points and preset views. This will guide the measurement technician through the GD&T inspection. To create a Guide Inspection right-click the Collection, Datum or Feature Check and select **Inspection>Design**. The design dialog will appear. Orient the view as desired and double left click where the user should measure. A red dot will appear marking the area for the user to measure. Press the next button in the bottom right hand corner to advance to the next datum or feature. Continue the process until complete (Figure 20-66) .

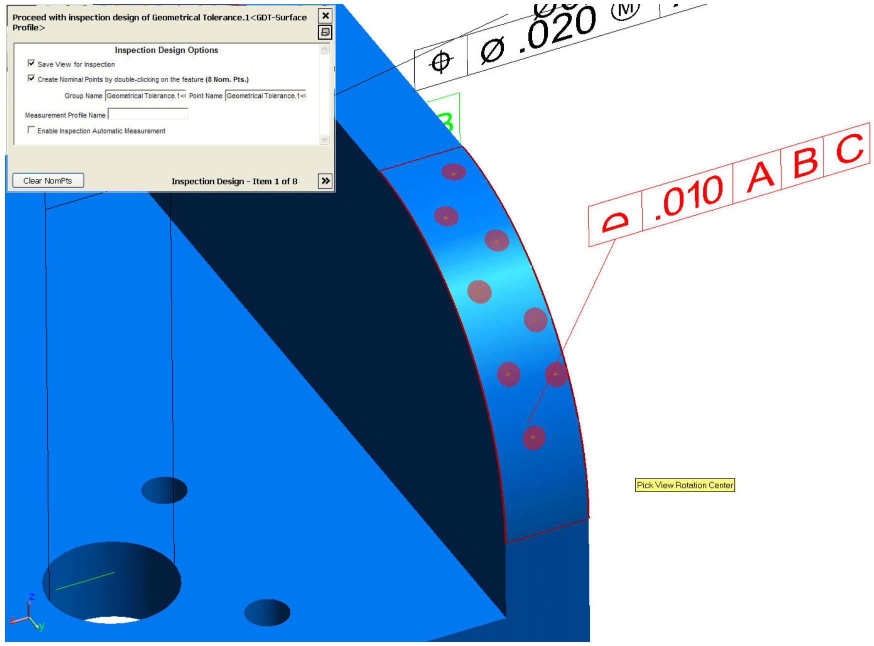


Figure 20-66. Designing a GD&T inspection routine.

When a user is ready to inspect, they right-click the Collection, Datum or Feature Check and select **Inspection>Inspect**. The guided dialog will appear (Figure 20-67), prompting the user through the inspection. If nominal points are present a red dot will appear. As the user measures the red dot will advance to the next measurement area. Once the nominals for one datum or feature are complete the inspection will advance.

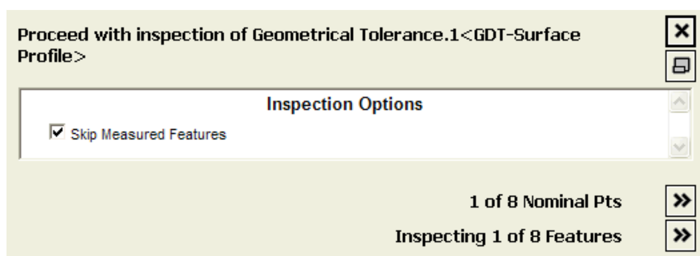


Figure 20-67. The inspection window.

Feature Inspection Auto Filter

If the measurements are roughly aligned to the part, the Feature Inspection Auto Filter can be used to automatically associate measurements to datums and feature checks based on proximity to respective surface faces (Figure 20-68). To perform an Auto Filter inspection, simply make the desired feature checks and use the menu command **GD&T>Feature Inspection Auto Filter**. The user will be prompted to select either points, point groups or point clouds. This command is extremely beneficial when working with point clouds. The below interface allows the user to choose surface proximity and thinning options. The auto filter will filter the points and clouds to the proximity of the respective surfaces faces and automatically associate to the datum or feature check.

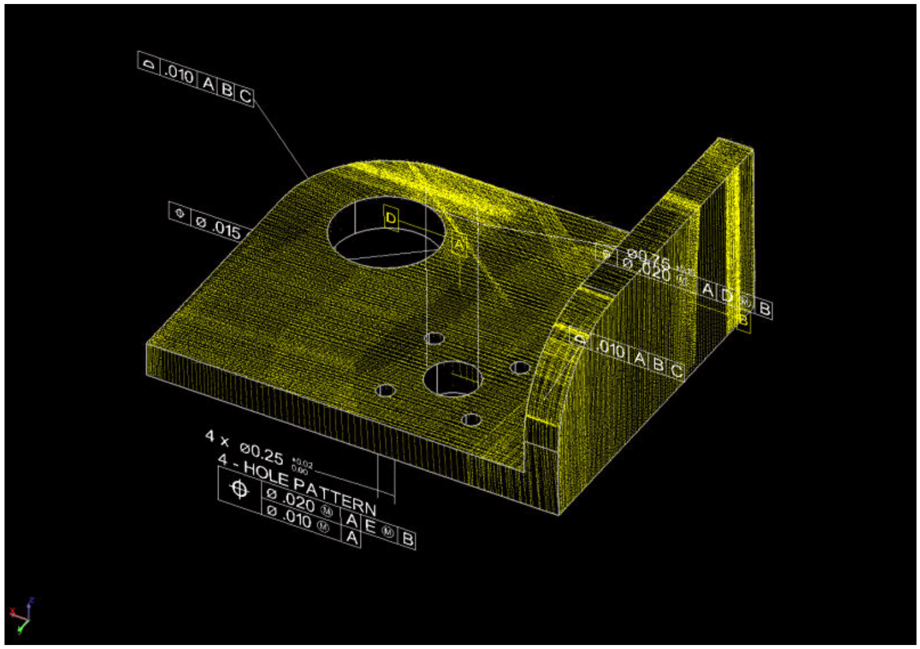


Figure 20-68. Prior to a feature inspection auto-filter.

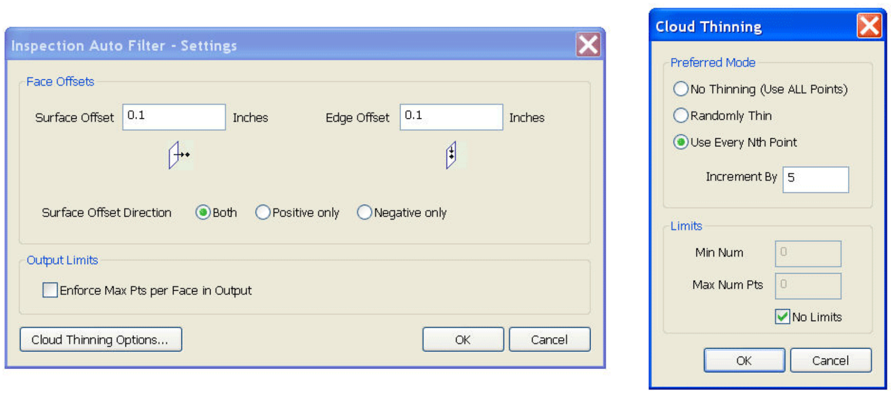


Figure 20-69. Auto-filter settings.

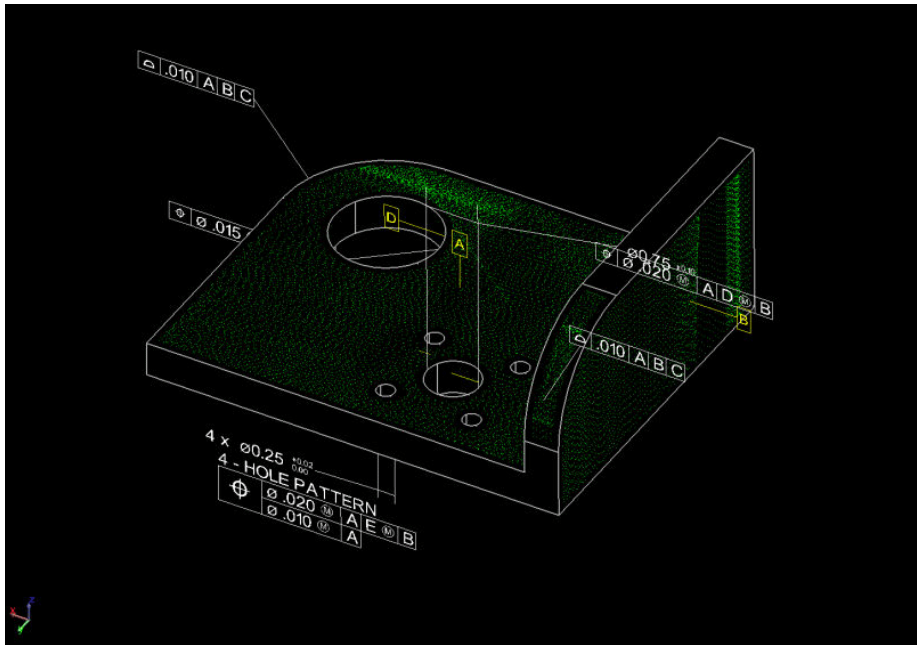


Figure 20-70. After the feature inspection auto-filter command.

GD&T References and Resources

- **ASME Y14.5M-1994.** http://catalog.asme.org/Codes/PrintBook/Y145M_1994_Dimensioning.cfm
- **Tec-Ease.** http://www.tec-ease.com/gdt-tips.htm

Minimizing Relationships

In most applications one of the goals is to orient measured points to critical part features. SpatialAnalyzer is able to help you accomplish that task by letting an optimization routine use relationships to drive a fitting process. The net effect is the distances defined between objects in each relationship are minimized. You can use this functionality to produce a best-fit between the points and the critical features. So by creating relationships between the measured points and the part's critical features you can find orientations that yield the best performing part. Since SA's relationships can be made between any object types you're able to define a comprehensive set of part features and then optimize their orientation.

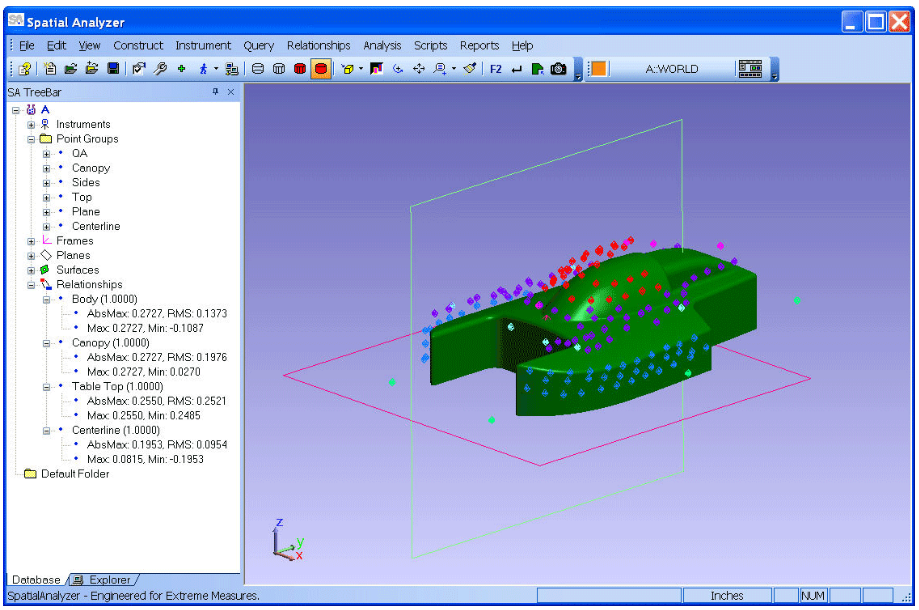


Figure 20-71. Relationships set up in the tree.

Move Objects by Minimizing Relationships

This is a great way to transform objects and instruments into a coordinate system (Figure 20-72). This is often used when traditional alignment methods are not applicable. The relationship minimization gives full control over each individual relationship by using weighting and fit constraints. The user also has control over the translation and rotation degrees of freedom during the minimization. This can allow a minimization to translate and/or rotate about user defined axis.

Run Optimization

This is the default mode of operation. The optimization method performs a version of the steepest-descent algorithm on the system. This means it assembles a matrix that is 6 (unknowns) x N (number of points), inverts it and determines move directions. Though quick for reasonable data sets, it can take a long time when using a large set of points depending on the amount of RAM on the computer. The stopping tolerance is the amount that the objective function (RMS error) must repeat by in order to indicate a minima has been reached.

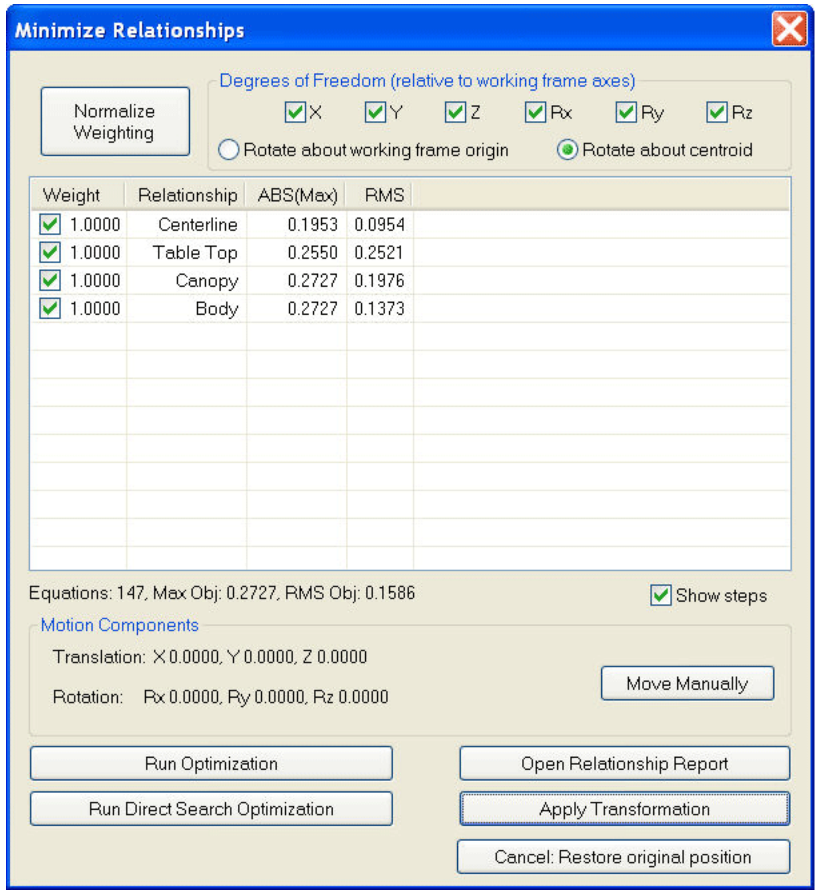


Figure 20-72. The Minimize Relationships dialog.

Run Direct Search Optimization

This optimization provides a powerful method for searching the solution space. It is also required when using the toleranced best-fitting options (in the advanced area) since they introduce non-linearities into the solution space. This method basically steps in a direction, evaluates the fit, then steps in another direction, etc. There is a lot of advanced logic built into the search, but at the core, it is a searching method. The advantage is that it does not have to assemble a large partials matrix and invert it. This means that it can effectively deal with huge point sets. The disadvantage is that since it is trying to exhaustively search, it can sometimes take a long time to complete. At any time during the fit, however, you can cancel the operation and use the current location. To use this mode, you set step sizes. They are set as a ratio of the position and angular perturbation values in the SA units options page. The step size starts at the starting value then decays when it detects no improvement in any direction. Once it decays to the Ending step size, a minimum is reached.

Relationship Fit Settings

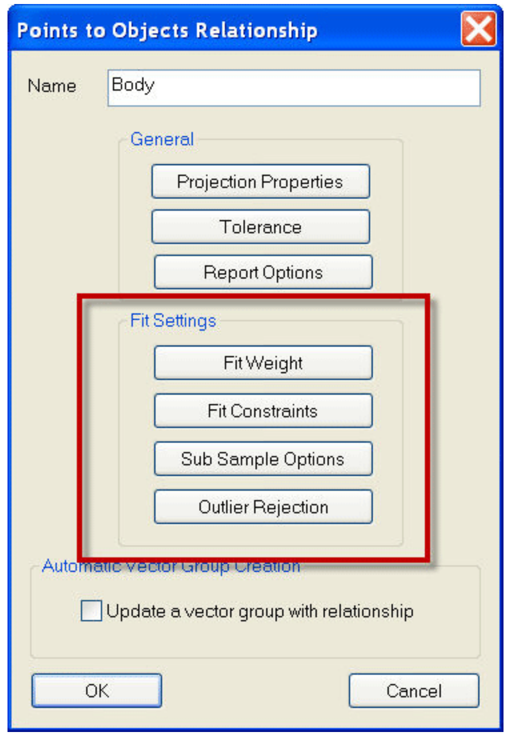


Figure 20-73. Relationship settings.

Fit Weight

Allows more emphasis to placed on the selected relationship. Often times weighting can be tweaked to bring a part in tolerance.

Fit Constraints

A fit region can be described that will represent a "dead" region where the solution can satisfy a zeroed out condition. For example a high limit can be set at +0.1 and low at 0. This will define a region from 0 to +0.1. When the solution reaches this region, it will stop.

Sub Sample Options

Sub sampling is useful for large data sets (Figure 20-74). Often all of the data is not required for the fit.

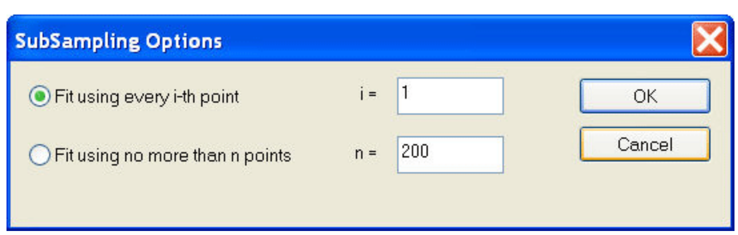


Figure 20-74. Subsampling options.

Outlier Rejection

Outlier rejection is a great way of removing stray data for the fitting solution. Stray points can often impose difficulties for the optimization.

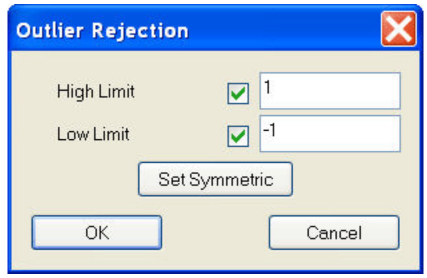


Figure 20-75. Outlier rejection options.

USMN

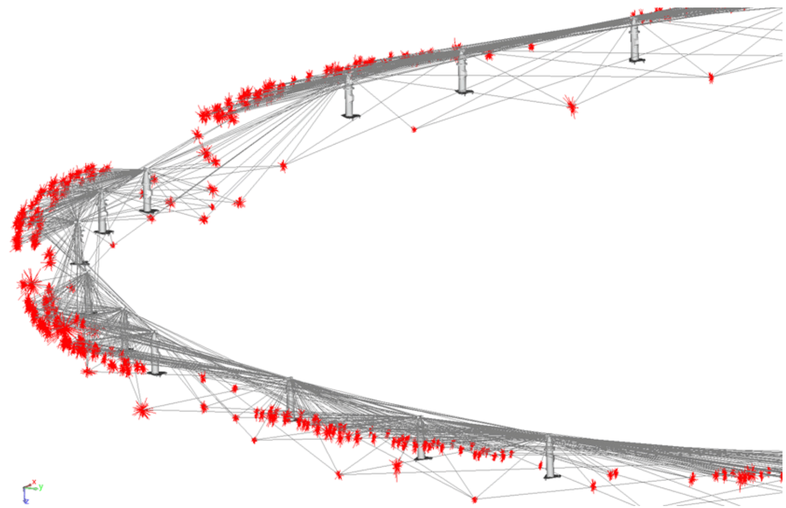


Figure 20-76. Part of a large USMN network.

Many organizations have standards requiring that measurements include not only measured values, but also a statement of the measurement's uncertainty. For instance, the International Organization for Standardization (ISO) "Global Product Specification" requires that part measurements have both measured values and uncertainties, and the National Institute of Standards and Technology (NIST) specifies that "a measurement result is complete only when accompanied by a quantitative statement of its uncertainty." In addition, others recommend that measurement systems provide uncertainty statements in order to be considered "accredited".

A lot of effort has been expended in the metrology industry to quantify the accuracies of different measurement systems, along with the uncertainties of those measurements. While these laboratory specifications generally remove other variables from the process and provide a fairly accurate "apples-to-apples" comparison of one measurement system to the next, they rarely reflect the achieved values in real-world measurement conditions.

In addition, measurement uncertainty statements typically assume a spherical uncertainty and do not address the geometric nature of a measurement in determining its accuracy.

One might have a reasonable estimate of the uncertainty of a measurement in a sterile lab environment--but what is really needed is an estimate of the uncertainty of a real-world measurement. Without this uncertainty, it is somewhat reckless to state that a measurement does or does not conform to a tolerance. For instance, if a measurement has an uncertainty of +/-0.015", and the measurement is required to be 8.125" +/-0.005", then is a measurement of 8.127" truly in-tolerance? In this case, there is a fairly significant chance that the actual value is 8.131" (or worse) and is therefore out-of-tolerance. If it were known to a high degree of confidence that the uncertainty of the measurement were instead +/- 0.0025", then one could say with high confidence that the measurement is in-tolerance.

There are a number of effects experienced in real-world measurement that are either excluded or not provided in typical instrument specifications but which have true impact on real measurement. This may be by choice (for marketing reasons) or out of necessity (e.g. the inability to quantify errors due to poor operator technique). Some of these effects include:

- Measurements are often combined into a network of interrelated measurement systems whose combined characteristics are dramatically different than the individual source measurements.
- Individual instrument characteristics dictate dramatically different uncertainties for a given measurement depending on the relative orientation of the instrument and measurement. (For instance, measurements reasonably close to a laser tracker are inherently more accurate than measurements in the distance).
- Poor operator technique may have adverse effects on accuracies and uncertainties.
- Temperature gradients, fluctuations, wind, and other environmental conditions increase uncertainty.

Based on these factors (and others), it is clear that knowing the uncertainty of a real-world measurement is very valuable.

What is USMN?

USMN (Unified Spatial Metrology Network) is an advanced tool for combining common measurements for two or more instruments into a single instrument network. It has several major benefits over other analysis tools, such as a best-fit:

- It provides a means for calculating a "weighted bundle": the most likely position and orientation for all instruments and targets in a measurement network.
- It provides the capability of calculating realistic uncertainties for measurements in a measurement network.
- It can be used to evaluate the real-world uncertainties of instruments on an individual basis.

Let's consider each of these benefits in order to get a true grasp on the benefits of USMN.

Weighted Bundle

All instruments have different characteristics. Perhaps the simplest example is the comparison of a laser tracker and a total station--both spherical measurement devices (they each measure a distance, azimuth angle, and elevation angle).

Figure 20-77 depicts the uncertainty cloud for a laser tracker. Laser trackers have accurate distance measurements, but the angular measurements are not nearly as accurate. Notice how the uncertainty clouds inherit a flat "pancake" shape--there is less uncertainty in distance than angle, so the uncertainty cloud tends to be thin (along the line of sight) and wide (perpendicular to the line of sight).

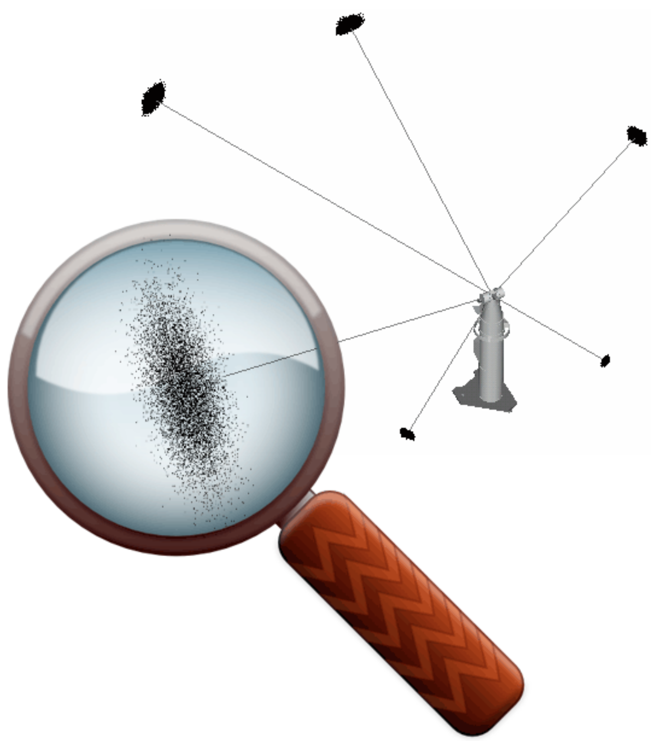


Figure 20-77. A tracker measurement's uncertainty cloud.

Notice also that as measurements get further away from the tracker, the uncertainty becomes larger, manifested as larger uncertainty clouds. (Compare the nearby measurement in the lower-right compared to the more distant measurement in the upper-left).

On the other hand, a total station's azimuth/elevation angle measurements are very accurate compared to its distance measurements. Notice how the uncertainty clouds are elongated and cigar-shaped: there are high uncertainties along the line-of-sight, but low uncertainties perpendicular to the line-of-sight (Figure 20-78).

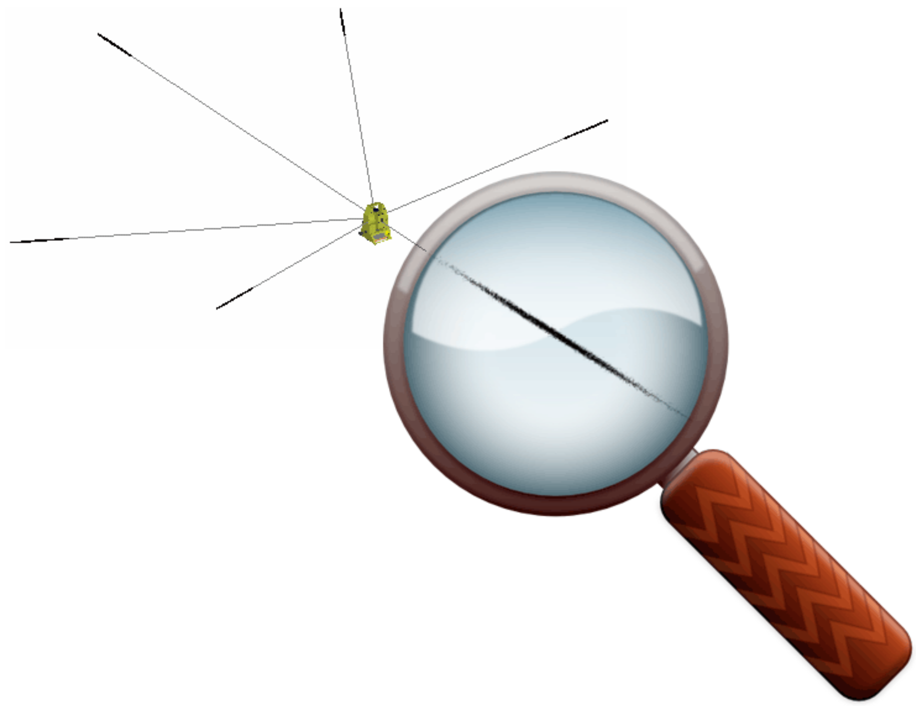


Figure 20-78. The uncertainty cloud for measurements from a total station.

Viewed side-by-side (Figure 20-79), the differing characteristics of each instrument result in drastically different measurement uncertainties (in size, shape, and density):

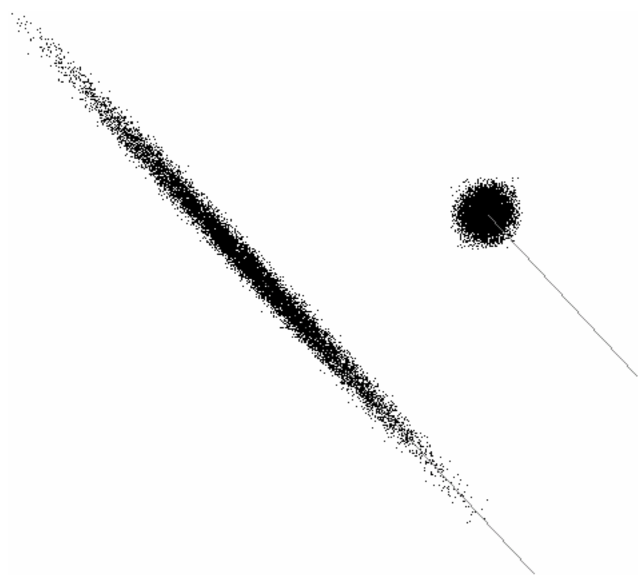


Figure 20-79. Comparing the shapes of the two uncertainty clouds.

The uncertainty characteristics of a portable CMM arm are even more complex (Figure 20-80), depending on the behavior of the encoder in each joint and the configuration of the arm when the measurement is taken.

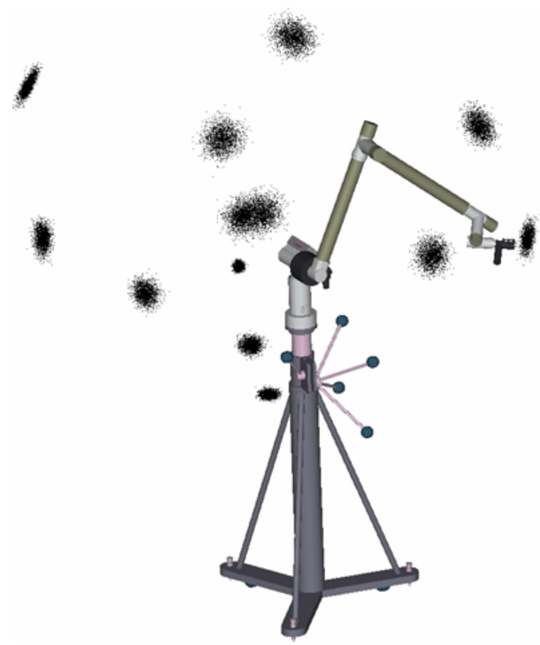


Figure 20-80. The uncertainty cloud shapes for a portable arm.

Now, consider the situation in which a long part is to be measured. Due to its shape, this part might require multiple instrument stations along its length. In order to tie all of the measurements into a single coordinate system, a number of common points are measured between adjacent sets of instruments, and then these points are best-fit together to determine the overall instrument network. For some cases, this method is a perfectly valid approach, and in fact, this is the approach taken by other metrology software packages for tying a network of instruments together. In particular, fairly small instrument networks with low uncertainty requirements can use this approach without problems.

However, using best-fit introduces error stack-up, very similar to the idea of tolerance stack-up in engineering drawings. This error stack-up can be significant. Figure 20-81 depicts an instrument network using a best-fit compared to the true instrument positions (ghosted). A slight error for the first set of common points (between Instruments A & B) will move the position and orientation of Instrument B out of its true position. By fitting Instrument C to Instrument B, not only does Instrument C inherit the error from the first set of common points (which has now been "leveraged" by its distance from those common points), but now errors in the common points between Instruments B & C cause even more error to stack up. The end result is a network that could potentially be quite different from reality.

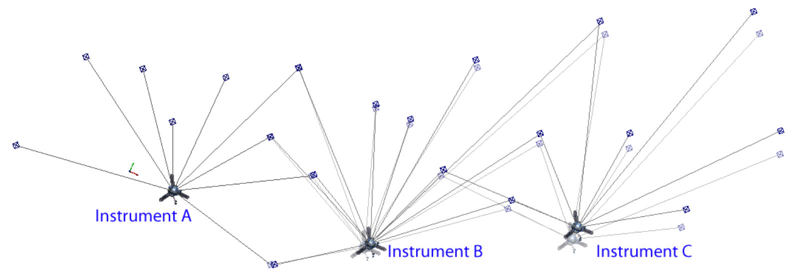


Figure 20-81. Positional errors resulting from leap-frogging an instrument along a length.

When more accuracy is desired--or when the measurement network is elongated, large scale, fairly 2-dimensional, or C-shaped (non-closed, meaning the first instrument is not tied back to the last), then best-fitting is usually inadequate. Some examples of real-world measurements that get significant benefit from USMN include particle accelerator measurements (very large and mostly 2-dimensional), large linear measurements with high accuracy requirements (such as the catapult rail on an aircraft carrier), and large "open" networks (such as measurement of two sides of a building).

Instead of using a best-fit, USMN takes an intelligent weighted bundle approach. USMN examines each common measurement and considers the characteristics of the instruments that measured them and their positions in space. Components of instrument observations which are considered to have low uncertainty are assigned a higher weight than components that are considered to have higher uncertainty. If a total station and laser tracker measure a common point from different angles, then the angular measurement from the total station will be assigned more weight than that of the tracker, and the distance measurement from the laser tracker will be assigned more weight than that of the total station.

Once these weights are assigned, the instrument transforms are adjusted to calculate their most likely positions based on all available information. The net result is an instrument network that is significantly closer to reality than the best-fit network described above.

In summary, USMN considers each instrument's uncertainty variables and perturbs instrument positions in order to get minimal measurement closures on the common points. This is similar to a traditional bundle process, except the optimization process uses the estimates of the instrument uncertainties and the range of each observation to weight the individual contributions for each measurement.

Network Measurement Uncertainties

Instrument measurements (observations) can have uncertainties calculated that indicate the uncertainty of a given measurement relative to the instrument. But when instruments are networked together, this simple uncertainty no longer applies, because the common point position in a networked measurement is really determined by a combination of instruments that measured the common point. In fact, in the real world the computed common point in the real world never lies on top of any of the component observations (because the world is not perfect--unfortunately!). This highlights the second major advantage to USMN: the ability to calculate "net uncertainties" for common points.

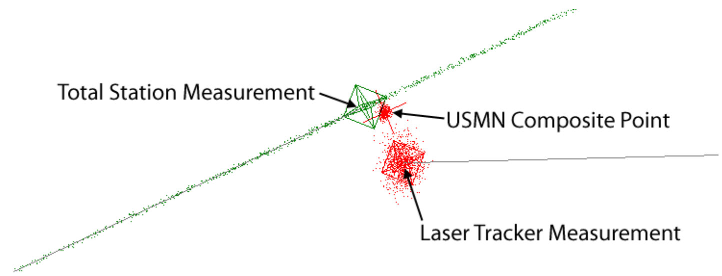


Figure 20-82. The USMN composite point. (Note: The error between the total station and laser tracker measurements are exaggerated for clarity).

Refer to Figure 20-82. A total station and laser tracker have measured a common target in a nest. Due to noise, operator error, and a number of other factors reminding us that the world is not perfect, each instrument "sees" the target in a slightly different position in space. As a result, the measured points end up in slightly different positions.

Not only can USMN calculate the most likely position of the real target based on instrument characteristics (referred to as a USMN Composite Point), but it can also calculate the uncertainty of that composite point. In Figure 20-82, the composite point is closer to the total station's line-of-sight position (due to lower angular uncertainties of the total station), but is also much closer to the laser tracker's distance measurement (due to less uncertainty in the laser tracker's distance measurement). The uncertainty of the composite point itself has less uncertainty than either of the instrument observations, because much of the total station's uncertainty along the line of sight is rendered extremely unlikely based on the laser tracker's measurement.

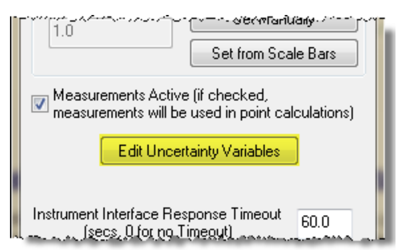


Figure 20-83. Accessing the uncertainty variables for an instrument.

"Real-World" Instrument Uncertainties

As mentioned earlier, instrument uncertainty values as published by the manufacturer are a good way to compare different measurement systems, but they are a poor way to predict real-world measurement uncertainty. A number of factors, such as the calibration/performance of an individual instrument, the technique of the operator, and the measurement environment have significant effects on uncertainty.

Each instrument in SA has associated with it Instrument Uncertainty

Variables (Figure 20-84). These can be found in the Instrument Properties window. Uncertainty variables are used to model an individual instrument's uncertainty characteristics.

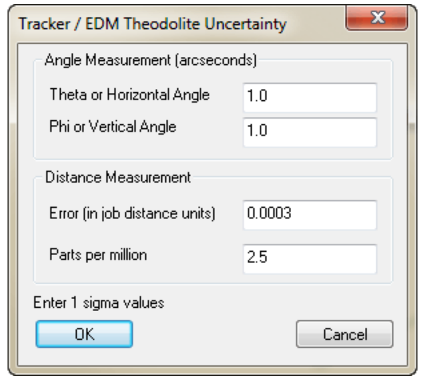


Figure 20-84. Uncertainty variables for a tracker.

If you look at the uncertainty variables for a laser tracker or total station, four values are provided (see image). These values together dictate the uncertainties for the horizontal angle, vertical angle, and distance components for an instrument. They are used in concert with a measurement's relative position from the instrument to determine the uncertainty of the individual measurement.

By default, the values assigned to uncertainty variables are experimentally determined, conservative values for a given class of instrument. In other words, they generally give a result that is "in the ballpark", but are probably a bit worse (higher uncertainty) than can truly be achieved with the instrument. This is done to a) err on the safe side, b) avoid conflict with instrument manufacturers, and c) avoid the need to continually update data as instruments evolve, which could be a time-consuming task.

In order to get the most realistic uncertainties for measurements and USMN, it is necessary to use uncertainty variable values that closely match the individual instrument, operator, and measurement environment characteristics. But how is this done? Manufacturers don't (and can't possibly) publish these values, and they differ from one instrument to the next and (to a lesser extent) from one operator/measurement situation to the next.

USMN itself can be used to determine the most accurate values for these variables. This is done by setting up a set of monuments, and then moving an instrument around through the workspace, measuring each of those monuments in turn.

Consider a situation in which an instrument, its operator, and the environment is perfect (and therefore would have zero uncertainty). A perfect instrument, after having measured all monuments from different positions, would see a given monument in the exact same location in space. When fit together (using a traditional best-fit), there would be zero error. The observations would lie on top of each other perfectly. USMN would see this "perfection" and would report uncertainty variables of zero (no error).

Of course, in reality this never happens. In the real world, an instrument can measure a monument two different times and "see" it in a slightly different place. By analyzing the error between measurements of a common point from the same instrument, in different positions, USMN can perform a complex calculation and determine the real-world uncertainty variables for a given measurement situation.

Observations and Point Group Requirements for USMN

When setting up a network for analysis with USMN, there are several organizational requirements for instruments, measurements, point groups, and point names. These are relatively simple in construction as they follow the process commonly used to locate an instrument (Figure 20-85).

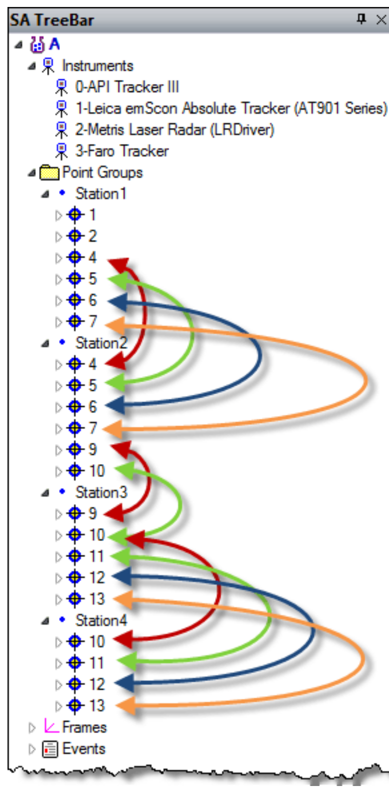


Figure 20-85. Organizational setup for a USMN.

Note: A group name does not need to include a specific reference to the instrument that made measurements to the points contained within it. The designation of S1, S2…SN used above is to illustrate a convenient means of organizing the data for USMN analysis.

The requirements for USMN analysis are as follows:

- As with any measurement network, common targets must be measured by more than one instrument to form a network.
- Individual point names must be common between groups as USMN will associate measurements from different instruments based on point names. For example, MeasStation1::P1, MeasStation2::P1, … MeasStationN::P1 will be combined to compute the composite point P1.

- Points without measurements are ignored, unless the point group is included as a 'Nominal Group.'
- A specific point in a given point group should have observations from only one instrument.
- An instrument can make more than one measurement to a specific point. As an example, an instrument can have a front and back sight to a point or an instrument might have N front sight measurements to a point.
- Observations that are marked as 'ignored' are not included in the USMN network. The measurement details dialog for a point provides an interface to determine whether or not an observation is ignored.

Consider an example tree for a job ready for a USMN network as seen at right.

- Points 1 and 2 (in the Station1 group) have no common measurements with other instruments.
- Points 4, 5, 6, and 7 are shared with Station2.
- Point 9 is shared by Station2 and Station3.
- Point 10 is shared by Station2, Station3, and Station4.
- Points 11-13 are shared by Station3 and Station4.

The points which are measured by more than one instrument station will affect the network solution.

The primary output from USMN is a Composite Point Group. This group contains the optimal point coordinates (the most likely position of the points) after solving the network. Inputs to USMN include both the instruments and their measurements. For a measurement to affect the calculation, it must have been measured from multiple instrument stations. The observations from different stations are made into different point groups, but should have the same target name.

USMN Dialog

The primary purpose of the USMN dialog is to help you optimize the network confidently and get the best Composite Point Group possible from the instrument stations and measurements. The window is composed of 7 main parts (Figure 20-86).

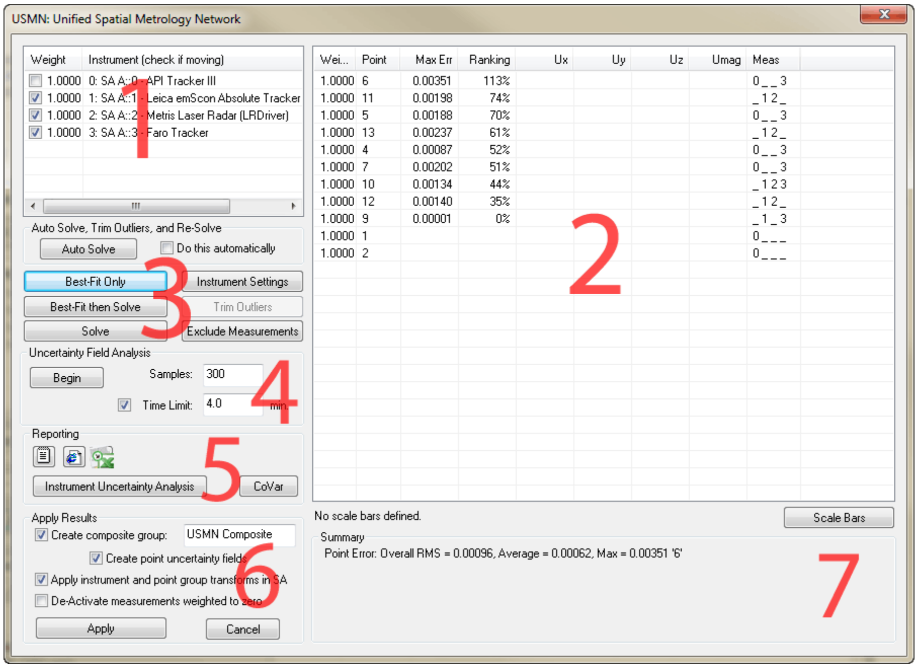


Figure 20-86. The USMN dialog.

1. Instrument Section

This section lists the instruments whose measurements are included in the USMN network solution. Instruments can have different weights applied. So, for example, if one instrument is known to not perform as well as others, its weight might be decreased from the default value of 1.

Each instrument has a checkbox indicating whether that instrument is allowed to move during the calculation. A USMN bundle (or any bundle, or that matter) is a relative solution. That is, the solution can be reached relative to any instrument. As a result, all instruments may be checked (meaning all instruments will be perturbed to find the solution) or one may be unchecked (so that the solution is reached by perturbing all instruments EXCEPT the unchecked instrument). In both cases, the same end result is reached for the network itself, although the networks will end up in different places in space (and the uncertanties will be relative to different baselines). Regardless, allowing all instruments to move will typically take longer to solve. Although the network solution is the same, the calculated uncertainties for the USMN Composite points will be different, because they are relative to different baselines. However, the relative uncertainties (differences between uncertainties) between each pair of points will remain the same.

More than one instrument may be fixed (unchecked) during a solution, although this is typically only done in rare cases and is generally not recommended, as the end result is a solution that is not an ideal spatial representation of an instrument network.

An individual instrument in this list may be double-clicked to access specific instrument properties. For example, you can adjust an instrument's degrees of freedom or change its weight.

2. Measurement Section

This section shows measurement-specific information and has a number of columns which are described below:

- **Weight.** Each measurement can have a weight applied. If a point is considered less important or perhaps more erroneous, its weight can be reduced.
- **Point.** Each point measured by an instrument in the USMN network is listed.
- **Max Err.** The maximum error is listed for each point that has been measured by more than one instrument. This depicts the maximum error between any individual measurement of a point and its optimal value.
- **Ranking.** Displayed as a percentage, this indicates the percentage of the expected 3-sigma envelope consumed by the maximum measurement residual. This is an efficient method of determining which points may have measurement outliers. A point with a ranking of 100% has observations which are consuming the entire 3-sigma envelope. Points with smaller rankings are more tightly clustered around the optimal value. Lower rankings indicate better measurements.
- **Ux/Uy/Uz/Umag.** The component and magnitude uncertainties (in the active coordinate frame) for the optimal point. These values only appear after an uncertainty field analysis has been performed.
- **Meas.** A list of the instruments that have observations for a given point. For instance, point 6 shows 0 _ _ 3, which indicates that instruments 0 and 3 each have a measurement for that point.

Further detail for a point can be obtained by double-clicking the point in the list.

3. Solve Section

This section contains the parameters and controls for actually calculating the USMN solution. They are summarized below:

- **Auto Solve.** This button performs a "Best-Fit then Solve" (see below), trims outliers at 100% (see below), then performs a "Solve" (see below). This automated process is recommended for well-understood networks and in applications where the process has to be automated. It is not recommended when first optimizing an unfamiliar network of measurements. Anytime outliers are eliminated from the network, the causes and details should be characterized and documented.

- **Do This Automatically.** When the USMN dialog is displayed, an Auto Solve is automatically performed once.
- **Best-Fit Only.** Performs a best-fit of the networked points. This is equivalent to performing a traditional best fit. Performing a best-fit prior to initiating the USMN solve drastically speeds up the calculation, because it provides for a much better initial starting point for the calculation.
- **Best-Fit then Solve.** Performs a Best-Fit (see above) and Solve (see below) in one step.
- **Solve.** Initiates a USMN calculation from the instrument's current positions.
- **Instrument Settings.** Allows you to modify properties for all instruments at one time.
- **Trim Outliers.** Removes points or observations exceeding a specified Ranking threshold from the USMN solution. Recall that ranking is an indicator of the quality of a point. Points above the ranking threshold can either be weighted to zero or removed from USMN entirely. Therefore, this is an effective means for preventing poor measurements from tainting the USMN solution.
- **Exclude Measurements.** Any measurements that should not be included in the USMN calculation can be excluded with this button.

4. Uncertainty Field Analysis Section

USMN does not by default calculate the uncertainties of the resulting optimal (USMN Composite) points, because this calculation can take a fair amount of time. As a result, you must initiate this calculation yourself.

Uncertainty field analysis works by using the instrument uncertainty characteristics to slightly perturb measurements following a realistic Gaussian distribution. Each iteration of perturbed measurements is then treated as a separate USMN network. Each iteration is solved in turn, which allows for statistical generation of optimal point uncertainties.

- **Samples.** Specify the number of perturbations, or iterations, to perform. Higher numbers take proportionately longer to solve, but have more accurate uncertainty values. It has been statistically determined that 300 or more samples will provide results within 5% of the "statistically true" value, so 300 is considered the default. About 10,000-15,000 samples are required to get a result within 1% of the statistically true value.

- **Time Limit.** When checked, the uncertainty field analysis will time out at this limit, regardless of the number of samples requested. In this case, the number of completed iterations will be used.
- **Begin.** Initiates the Uncertainty Field Analysis.

5. Reporting Section

This section provides different methods for reporting results.

6. Apply Results Section

This section controls how the calculated results are applied to the job file.

- **Create Composite Group.** When checked, creates the optimal (USMN Composite) point group.
- **Create Point Uncertainty Fields.** When checked, applies uncertainty fields (calculated via the Uncertainty Field Analysis) to the USMN composite points. These fields represent the resulting uncertainty of the network points.
- **Apply instrument and point group transforms in SA.** If unchecked, no instruments or points in SA will be physically moved in the file.
- **De-Activate measurements weighted to zero.** If checked, any measurements that have been weighted to zero in the Measurement Section will be deactivated in SA.

7. Summary

This section lists a general summary of the USMN calculation, including values such as Overall RMS, average point error, uncertainty average, and other statistical information. It also provides access to information about scale bars included in the USMN calculation.

The point list is automatically organized in the interface. Points with the highest ranking are listed first. A high ranking indicates a point with a higher chance of problems. Points with lower ranking suggest the target solution is better.

An auto-solve function automates the solution and outlier trimming process. USMN runs with default options without displaying the main interface dialog. The results are then applied to the instruments in the metrology application automatically.

Basic USMN Solution Process

The following describes the steps for a basic USMN solution process, without calculating uncertainty fields for the USMN composite group.

1. Ensure that the job file matches the requirements in Observations and Point Group Requirements for USMN.
2. Select **Analysis>Coordinate Uncertainty>Unified Spatial Metrology Network**.
3. When prompted, select the instruments to include in the USMN network, then press `Enter`.
4. The ***USMN*** dialog will be presented (see USMN Dialog).
5. Ensure that the instrument settings are correct (moving/non-moving, weights, DOF constraints, etc).
6. Click Best-Fit then Solve.
7. Examine the max error and ranking values.
8. If desired, click the Trim Outliers button and trim any measurements beyond your desired ranking.
9. Click the Solve button and observe the results.
10. Verify that **Create composite group** is checked.
11. Click the Apply button. The instruments/point groups will be located, and a USMN Composite point group will be created representing the ideal position of each meausurement.

For a more in-depth example on basic USMN, see the tutorial Basic USMN.

Advanced Surface Fitting

From Point Groups (Interpolate)

Note: Points in the selected point groups must be in order or the b-spline will be created incorrectly.

Creates a interpolated surface by fitting b-splines to each selected group (Figure 20-87) and then lofting the b-spline to create a surface (Figure 20-88).

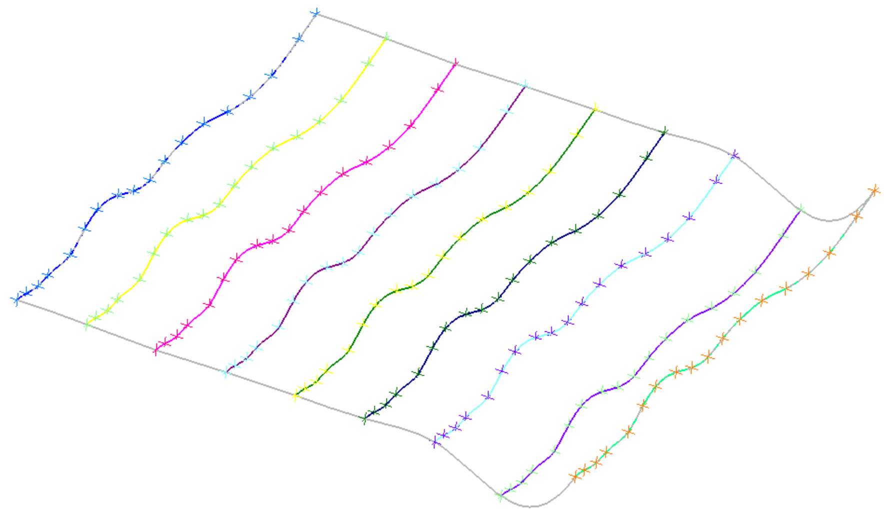


Figure 20-87. Fitting B-Splines to a series of points.

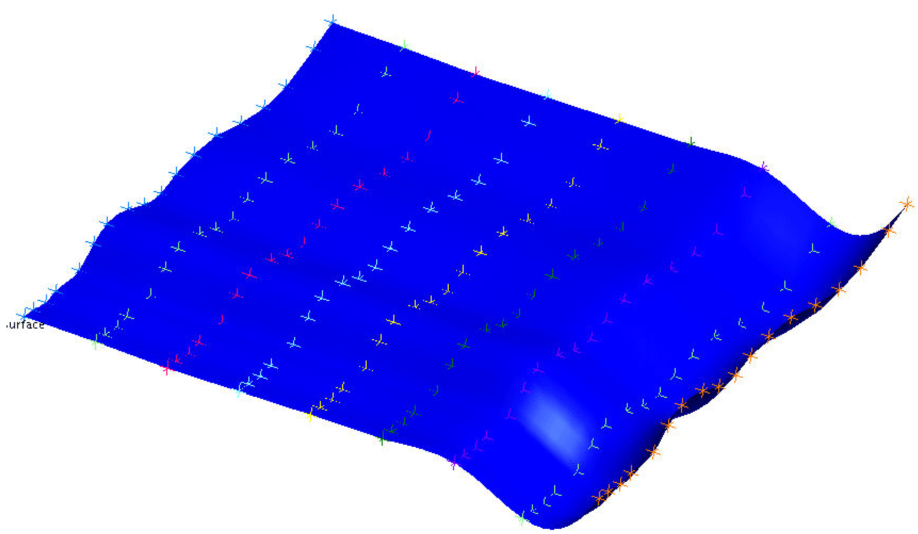


Figure 20-88. A surface has now been fit from the B-Splines.

From B-Splines (Approximate)

Similar to the above command, this time each b-spline is an approximate fit. The user has control of the control points, degree of curve and the skinning tolerance.

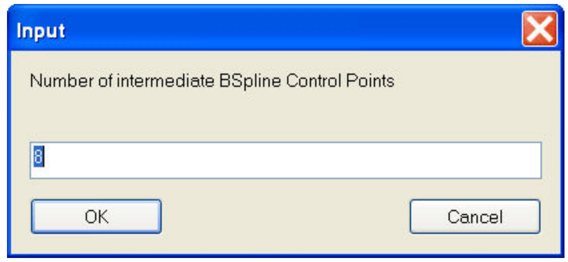


Figure 20-89. Defining the number of intermediate control points.

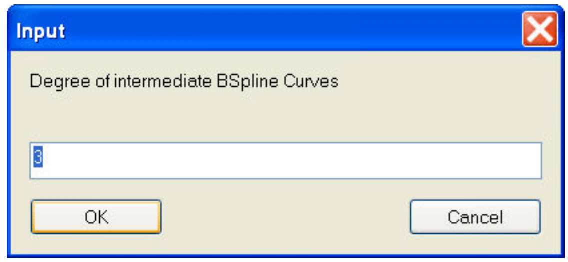


Figure 20-90. Specifying the degree of intermediate curves.

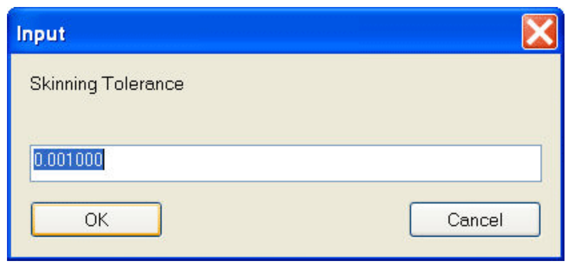


Figure 20-91. The skinning tolerance indicates how far the surface may deviate from the B-Splines.

From Plane and Closed B-Spline

Projects selected closed B-spline to selected plane and creates a surface from the projection (Figure 20-92).

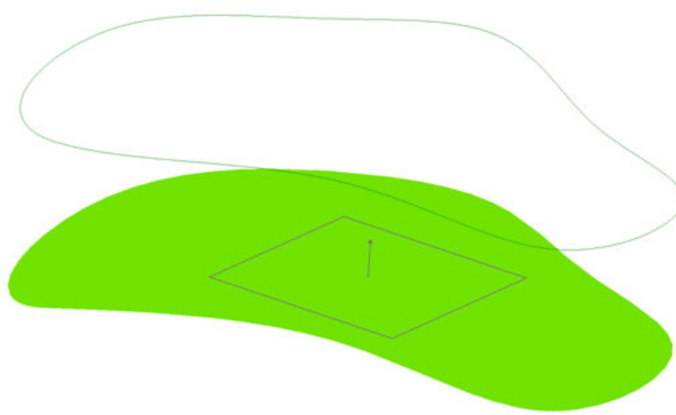


Figure 20-92. A closed B-Spline has been projected to a plane.

Fit From Bounding Curves and internal points

Use for B-splines that bound a group of points (Figure 20-93) to create a surface (Figure 20-94).

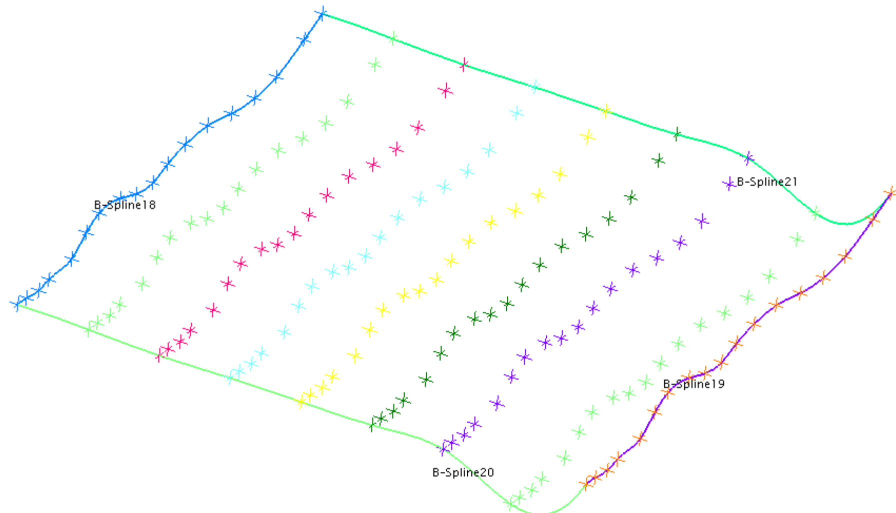


Figure 20-93. Four bounding B-Splines around a surface.

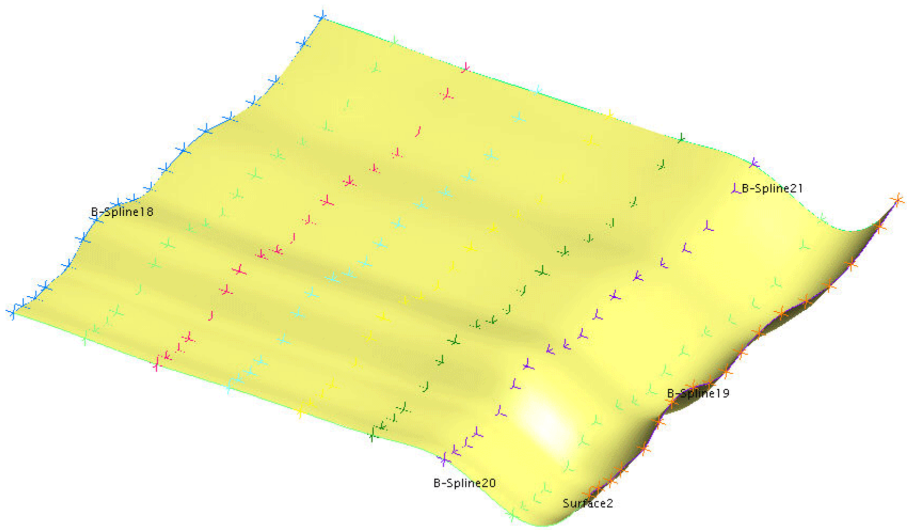


Figure 20-94. The resulting surface.

Fit From Nominal Face and Actual Data

A surface will be created by fitting the selected surface face to the selected data (Figure 20-95).

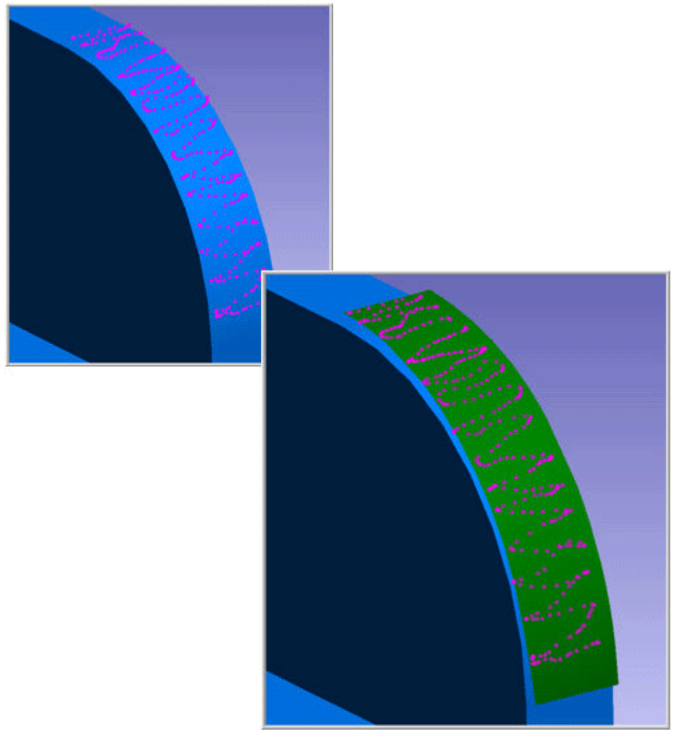


Figure 20-95. Fitting a surface face to selected data.

Frame Wizard

The Frame Wizard (**Construct>Frame>Frame Wizard**) is used to create a coordinate frame based on different combinations of entities, which can then be optionally offset in the new frame's X, Y, and Z directions.

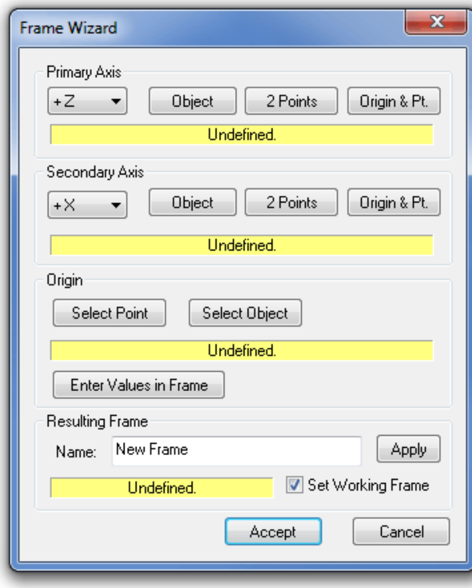


Figure 20-96. The Frame Wizard dialog.

The ***Frame Wizard*** dialog (Figure 20-96) allows you to interact with the graphical view while changing the settings for the frame. Once enough parameters have been defined to construct a coordinate frame, you see the frame in the view and it changes as you modify the settings.

The following describes the specific settings for this command:

- **Primary Axis.** The axis to use as the primary axis for the coordinate frame. Since this axis is primary, it is perfectly parallel to the vector constructed by its defining entities. The primary axis and secondary axis must make sense--for example, you cannot define the +Z axis as primary and the -Z axis as secondary.
- **Secondary Axis.** The axis to use as the secondary axis for the coordinate frame. The plane defined by the primary and secondary axes will be parallel to the vector constructed by the secondary axis's defining entities.
- **Origin.** A single point in space that (assuming no offset) defines the origin of the coordinate frame.
- **Enter Values in Frame.** Click this button to define a new XYZ coordinate. The coordinate frame will be shifted such that the originally defined origin point ends up at the supplied coordinates.
- **Name.** The name for the resulting coordinate frame (in the ac-

tive collection). To change a name, type it in, then click the Apply button.

- **Set Working Frame.** If checked, the new coordinate frame will be set as the active coordinate frame.

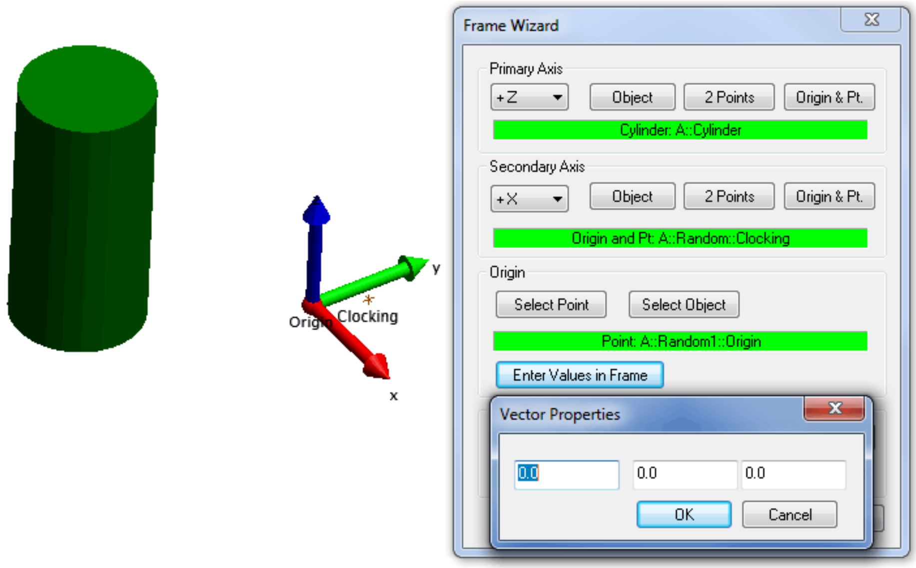


Figure 20-97. A frame defined by an origin at point Origin, a primary axis (+Z) by a cylinder, and a secondary axis (+X) by the origin and a point named Clocking. No offsets have been applied.

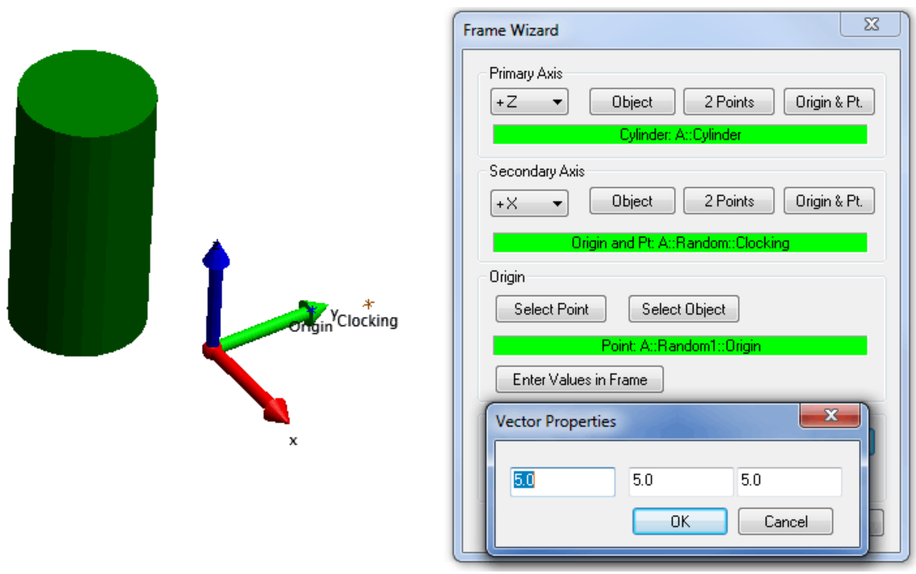


Figure 20-98. The same frame which has been shifted using Enter Values in Frame so that the origin ends up at (5, 5, 5).

The primary and secondary axes can be defined by an object, 2 points, or an origin and point:

- **Object.** The object's internal normal direction defines the direction of interest. For example, if a plane is selected, the normal to the plane is used. Note that some objects (such as spheres) do not define a direction and therefore cannot be used with this option.

- **2 Points.** The vector from the first point to the second defines the direction of interest.

- **Origin & Pt.** The vector from the defined origin to the selected point defines the direction of interest.

The origin can be defined by either an object or a point:

- **Select Point.** A single point will define the position of the origin. Point offsets are not applied.
- **Select Object.** An object's internal origin defines the origin of the frame. For example, if a sphere is selected, the center of the sphere is used.

To Create a Frame with the Frame Wizard:

1. Select the appropriate primary and secondary axes to define.
2. Click the appropriate buttons to define entities for the primary axis, secondary axis, and origin. These entities can be defined in any order.
3. If desired, enter coordinates to shift the frame using the Enter Values in Frame button.
4. Type a desired name for the frame and click Apply.
5. When finished, click the Accept button.

This Page Intentionally Left Blank.

Relationships

CHAPTER 21

Relationships are extremely powerful tools that have a number of uses: Dynamic inspection, alignment, virtual fit-up, real-time assembly, and optimization can all be driven by SA's relationships. Relationships provide optimization results that far surpass traditional methods, and without a complicated setup process.

Pipe Relationships

In some industries, such as nuclear energy and shipbuilding, a common need is to align mating sections of pipe in order to calculate and lay off cut planes so that the pipes can be adjoined with a high-quality weld. In general, the goal is for these adjacent pipe sections to be as coaxial as possible and to share enough material overlap such that a valid cut plane can be calculated that enables a quality weld joint.

Consider the example of replacing a steam generator that has reached the end of its service life at a nuclear power plant. A nuclear steam generator typically has a number of nozzles or pipe openings—one variant design has a "hot leg" pipe for hot water, a "cold leg" pipe for cold water, and a pipe for steam (Figure 21-1).

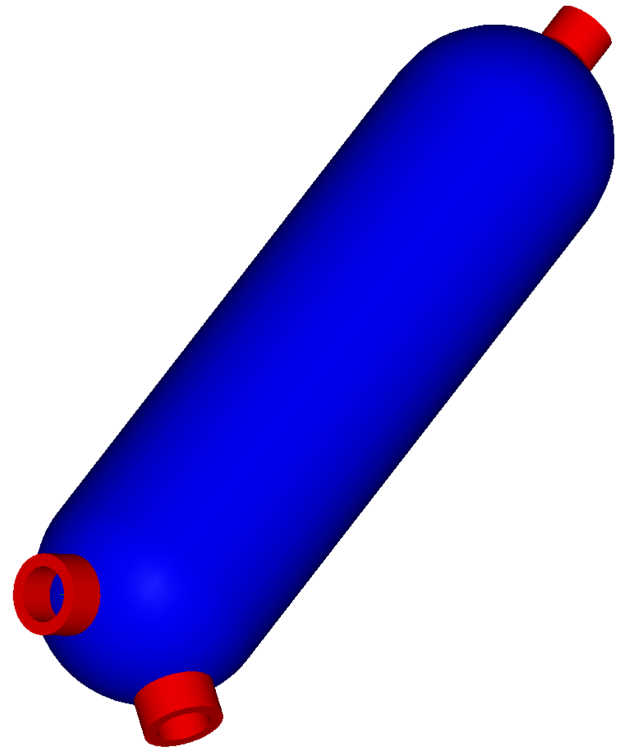


Figure 21-1. A typical steam generator, with hot and cold legs (bottom) and a steam leg (top).

These piping interfaces are critical to the safety and performance of the nuclear power plant, and since they provide a direct barrier between the radioactive and non-radioactive sides of the plant, they must be aligned and cut with optimal precision.

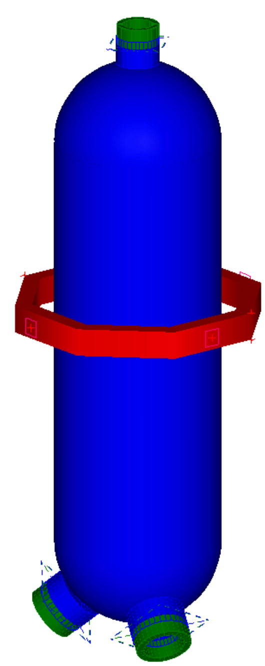


Figure 21-2. The steam generator after virtual assembly.

Since a nuclear reactor must be shut down in order to replace the steam generator—and since downtime at a nuclear power plant is enormously costly—anything that minimizes this downtime is extremely beneficial. Historically, cut planes were calculated manually and refined with the steam generator in its final alignment. However, this proved to be a time-intensive and therefore expensive process. Instead, the steam generator can be digitally assembled (Figure 21-2) and resulting pipe cuts can be calculated before the steam generator is in its final location (Figure 21-3). As a result, the pipes can be cut prior to installation—pipe relationships play a critical role in this process.

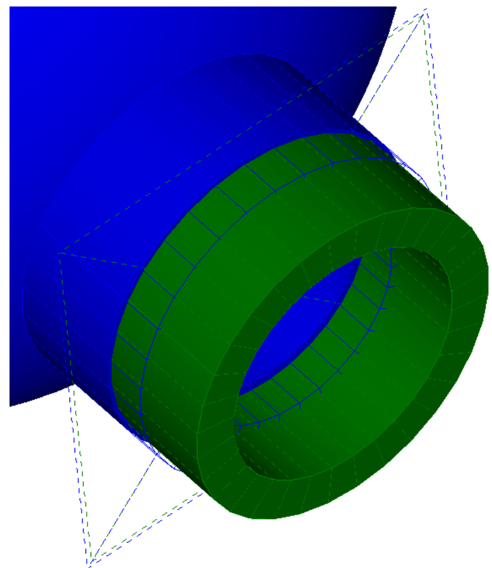


Figure 21-3. A cut plane calculated by a pipe relationship.

Using Pipe Relationships

The general process of using a pipe relationship is as follows:

1. Define the pipe relationship between mating pipe/fittings.
2. Align the pipes using traditional relationships, pipe relationships, or a combination.
3. If the pipe relationship can find a valid cut plane, the proposed cut can be made. A frame can also be constructed on this cut for layoff purposes.
4. After physically cutting one end of the pipe, the actual cut plane can be fed into the solution, re-solving for a tweaked cut plane on the other end.
5. This tweaked cut plane can then be used to cut the mating piece.

Defining Pipe Relationships

Pipe Relationships can be defined between two objects that have positional and directional information. Typically, this means lines, cylinders, or planes. The particular type of object does not matter—as long as the object properly describes the pipe's position and orientation. When a pipe relationship is created, a tube representing the allowable cut region and ID/OD appears in the graphical view, and the ***Pipe to Pipe Relationship*** dialog is displayed (Figure 21-4).

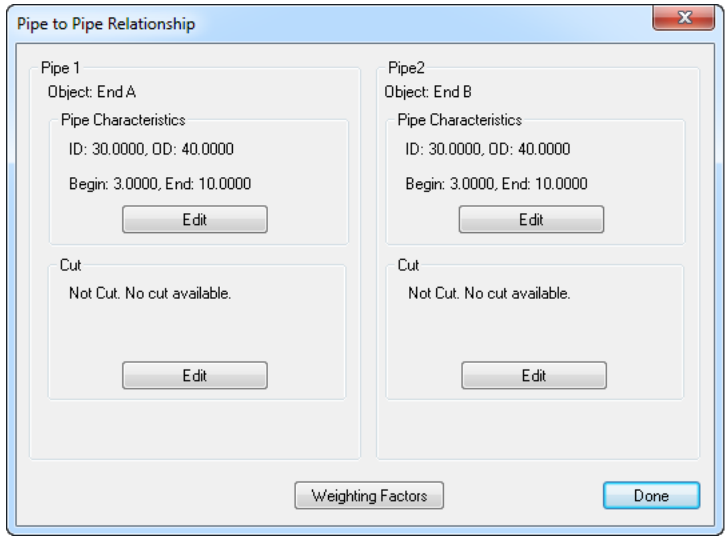


Figure 21-4. The ***Pipe to Pipe Relationship*** dialog.

The first step is to define the properties of each pipe segment (Figure 21-5):

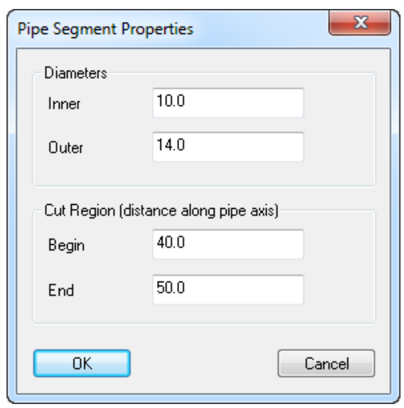


Figure 21-5. Properties of a pipe segment in a pipe relationship.

- **Inner.** The inside diameter of the pipe.
- **Outer.** The outside diameter of the pipe.
- **Begin.** The start of the permissible cut region, as a distance along the pipe axis from its origin. This value should typically be less than the End value.
- **End.** The end of the permissible cut region, as a distance along the pipe axis from its origin. This value should typically be greater than the Begin value.

A pipe relationship will supply several forcing functions to an optimization:

- It will try to make the pipe segments coaxial.

- It will try to make the pipe ends parallel.
- It will try to overlap the cut regions evenly.

After defining the characteristics of the pipe segments, the weighting factors for the relationship may be modified, if desired. Typically, the default values (Figure 21-6) are sufficient:

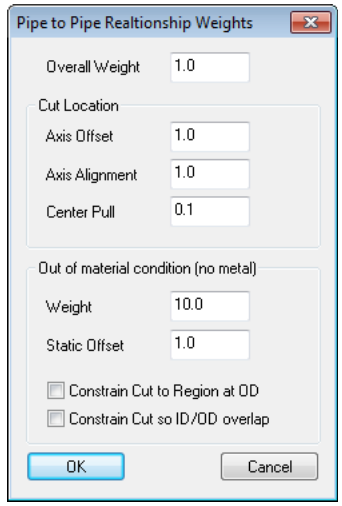


Figure 21-6. Weighting Factors for a pipe relationship.

- **Overall Weight.** The weight of the pipe relationship as a whole, which determines the influence of the pipe relationship in the overall optimization scheme.
- **Axis Offset.** The importance of minimizing the mutual perpendicular midpoint between the two pipe segments (i.e. how "coaxial" the two pipe segments are). Larger values attempt to bring the segments closer together.
- **Axis Alignment.** The importance of minimizing the angle between the two pipe segments. Larger values will force the pipe segments to be more parallel.
- **Center Pull.** The importance of overlapping the center of each pipe segment's cut region (along the axis direction). Larger values assign more importance.
- **Weight.** The importance of having sufficient material for the weld. A value of zero implies that the solution does not care if there is sufficient material for the weld.
- **Static Offset.** A value that controls "how bad" the solution gets when there is insufficient material for the weld. Higher values assign a larger penalty to having insufficient material, effectively ensuring that material requirement is more crucial in the eyes of the solution.
- **Constrain Cut to Region at OD.** If checked, ensures that the cut angle at the outer diameter does not cause the cut plane to exceed the defined region. If unchecked, this restriction is

ignored.

- **Constrain Cut so ID/OD overlap.** If enabled, the inner diameter/outer diameter values for the pipes are used to ensure that there is sufficient material based on the angle of alignment.

To Solve a Pipe Relationship:

1. First take a moment to think about the assembly. If there is more than one pipe segment that will be assembled (that is, if there will be more than one independently moving object in the optimization), then each moving object (pipe segment) should be placed into its own collection. This includes the objects defining the pipe endpoint, the pipe surface/cylinder, and anything else you want to rigidly move with that pipe segment. In this situation, you must use **Relationships > Move Collections by Minimizing Relationships** in step 7 below.

2. Create a primitive SA object with directionality (circles, lines, cylinders, or planes—but circles are preferred) to represent the end of each pipe opening in the assembly. Each object's normal direction *must* face the mating pipe segment as depicted in Figure 21-7.

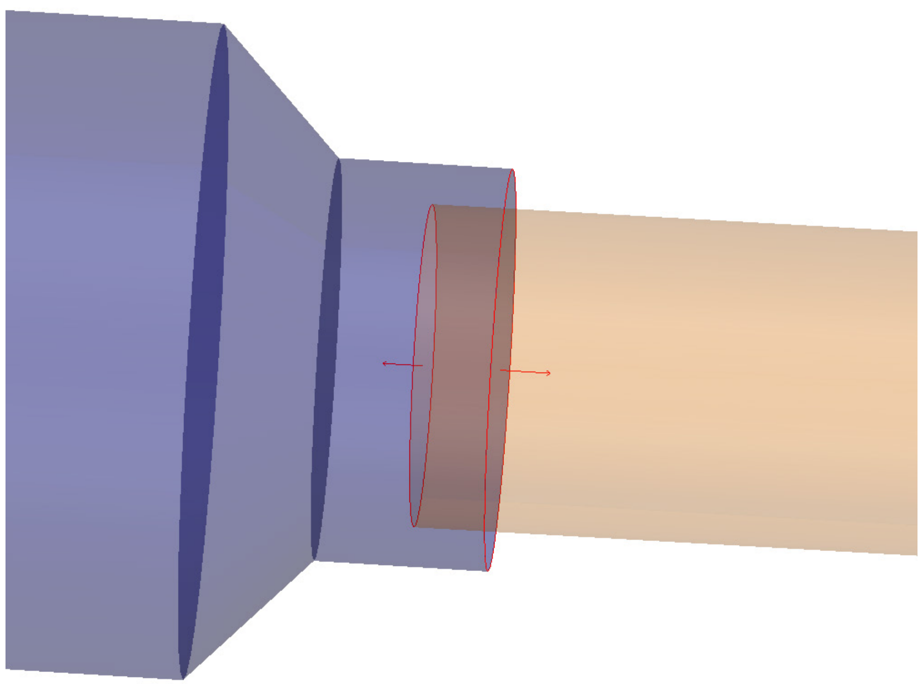


Figure 21-7. The circles defining each pipe opening have their normals pointing toward the mating pipe segment.

3. From the menu, select **Relationships > Pipe Fitting**.

4. Select the object (i.e. circle) describing the first pipe segment, then the second.

5. In the *Pipe to Pipe Relationship* dialog (Figure 21-4), click the Edit button for each pipe segment and define the ID, OD, and begin/end values for the allowable cut region (Figure 21-5).

Warning: If the ID and OD values of mating pipe segments match, a valid cut plane will typically not be found unless you uncheck the *Constrain Cut so ID/OD Overlap* option.

6. If any pipe ends are already cut, create a frame representing that pipe end (usually **Construct > Frame > On an Object** is appropriate) and then click the Force CUT to Frame button (Figure 21-8).

7. Create any other conventional relationships governing the alignment, if applicable.

8. Solve the relationship using **Relationships > Move Objects by Minimizing Relationships** or **> Move Collections by Minimizing Relationships**, as applicable. Enable/disable and adjust the weights on all relationships as desired.

Tip: In general, pipe relationships are best solved by clicking the Direct Search Optimization button instead of the Run Optimization button.

Working With the Cut

If the pipe relationship can find a valid cut plane, it will display a dashed plane indicating the location and orientation of the cut plane (Figure 21-3).

Typically, the next step is to create a frame on the cut, which can then be laid off on the real pipe for cutting. This involves making the proposed cut, then constructing a frame on the cut plane. The XY plane of this frame now defines the perfect cut location on the pipe. Watch windows can be used to lay this cut off on the real pipe.

In some applications, it may be desirable to simply calculate the cuts and lay them all off at once, as described above. This would be a faster but less stringent approach. A more rigorous approach would involve laying off the cuts on one segment, performing the cuts on the actual pipe, then measuring the actual cut plane.

Once the actual cut plane is measured, a coordinate frame can be constructed that represents the location of the as-built cut. This as-built cut location can then be fed back into the pipe relationship, and the as-built cut can be used to tweak the alignment once more. After the alignment is further tweaked, a new tweaked cut plane can be laid off on the other pipe segment.

This process can be iterated, as desired, for each pair of pipe segments.

To Lay Off Cut Planes:

1. Once the relationship has been solved (see “To Solve a Pipe Relationship:” on page 632), double-click the pipe relationship in the tree. In the *Cut* section, ensure that “Valid Cut Available” is indicated.

2. Click the Edit button in the *Cut* section for one of the pipe segments. In the ***Pipe Cut Properties*** dialog, click the Make Proposed Cut button (Figure 21-8).

3. With the cut applied, click the Create Frame on CUT button to create a coordinate frame whose XY plane defines the cut plane (Figure 21-9).

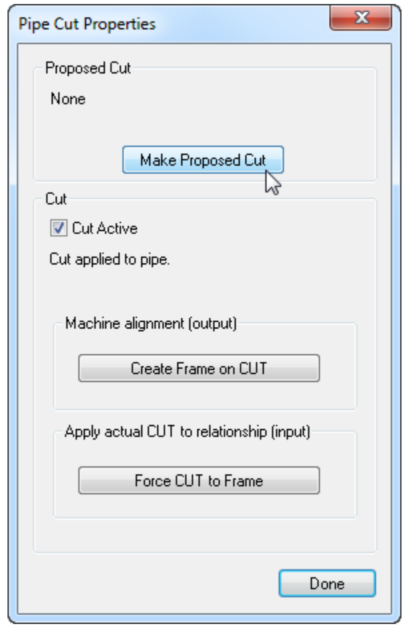


Figure 21-8. The ***Pipe Cut Properties*** dialog.

4. Use a Watch Window to lay the cut plane onto the actual pipe.

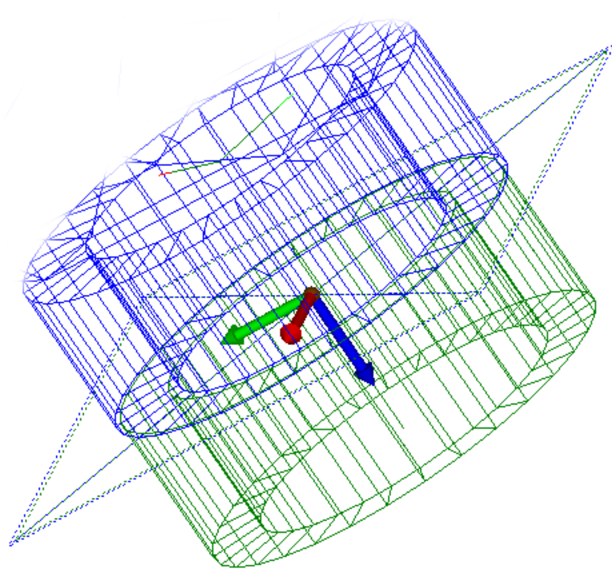


Figure 21-9. A cut frame constructed on the cut plane.

The above steps can be repeated for the other pipe segment, if desired. However, if applying cuts sequentially, the process contains a few more steps.

Applying Cuts Sequentially

1. After cutting the first pipe segment, measure the actual cut and construct a frame on the resulting cut plane. Ensure that this data is aligned appropriately in the SA file.

2. Double-click the pipe relationship, and click the Edit button for the appropriate pipe cut.
3. Click the Force CUT to Frame button, and select the appropriate frame. This will force the cut to sit on the provided frame.
4. Re-solve the optimization. Based on the as-built cut location, you should see the solution get tweaked to account for the changes.
5. Based on the results, make the cut on the other pipe segment using the steps in "To Lay Off Cut Planes:" on page 633.

This Page Intentionally Left Blank.

Reporting

CHAPTER 22

If you cannot communicate your results with your customer, then all of your measurement and analysis work is pointless. Fortunately, SA provides you with many different tools and capabilities for reporting your measurement results.

Callouts

Callouts offer a quick and easy way to annotate the graphical view in SpatialAnalyzer. Callouts can be created for a number of objects, points, vector groups, and other items (Figure 22-1).

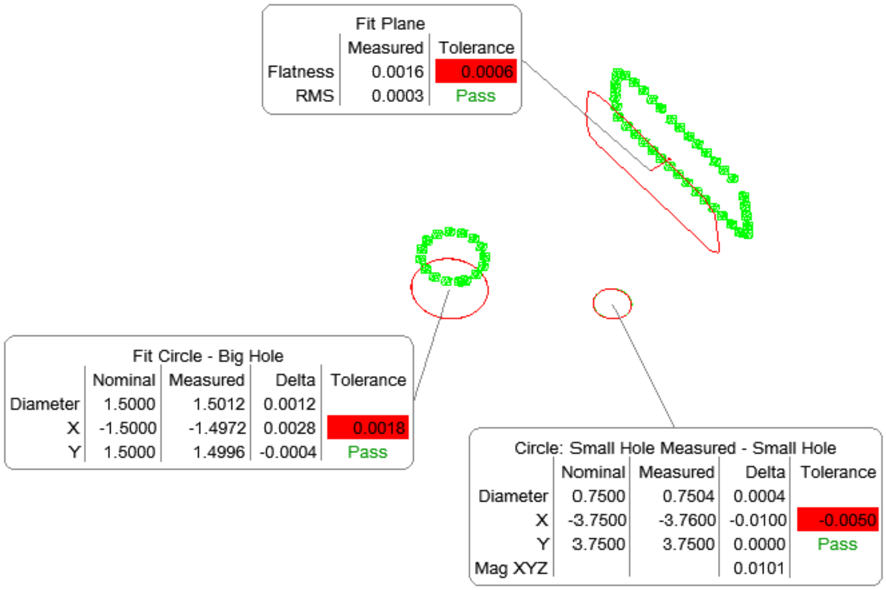


Figure 22-1. Geometry relationship callouts in the graphical view.

Creating a Callout

To create a callout simply press the button. This will create and activate a callout view in the tree. Double-Clicking a callout will activate or deactivate a callout (Figure 22-2).

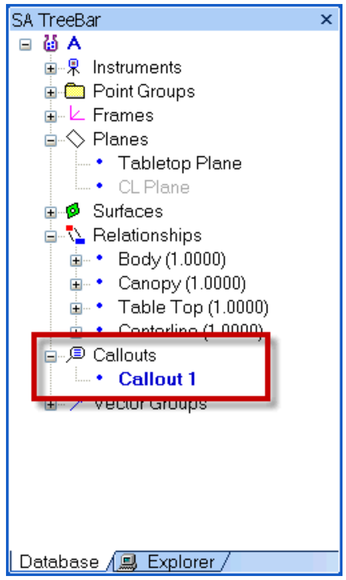


Figure 22-2. An activated callout as it appears in the tree.

Callout Annotations

Text

Allows user to enter text.

Point Coordinates

Annotates selected point coordinates (Figure 22-3).

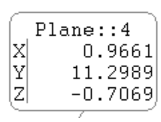


Figure 22-3. A point coordinate callout.

Right-Click the callout to access properties (Figure 22-4).

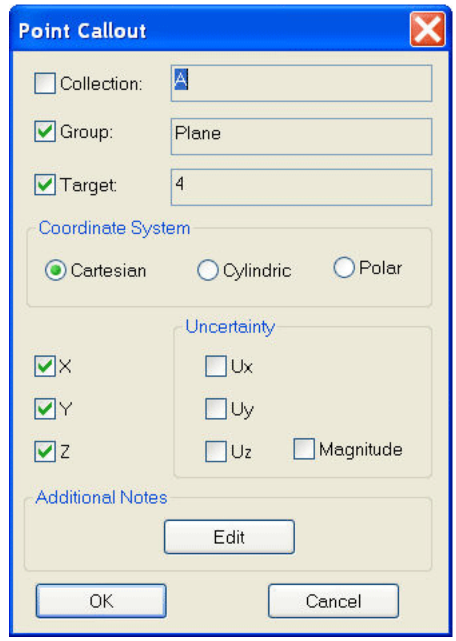


Figure 22-4. Point callout properties.

Point Comparison

Compares the two selected points (Figure 22-5). Their deltas are computed by subtracting the first selected from the second selected. Right-Click the callout to access properties (Figure 22-6).

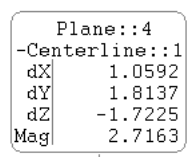


Figure 22-5. A point comparison callout.

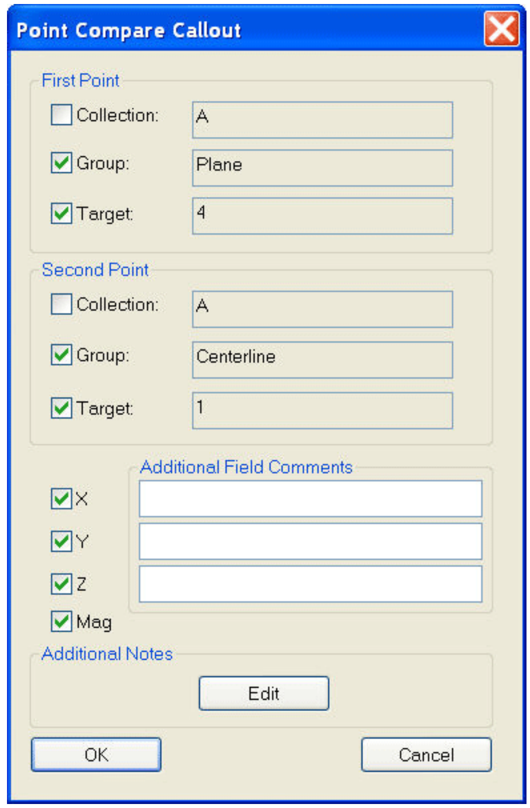


Figure 22-6. Properties for a point comparison callout.

Point Labels

Creates annotations for selected points. User can pick items to include (Figure 22-7).

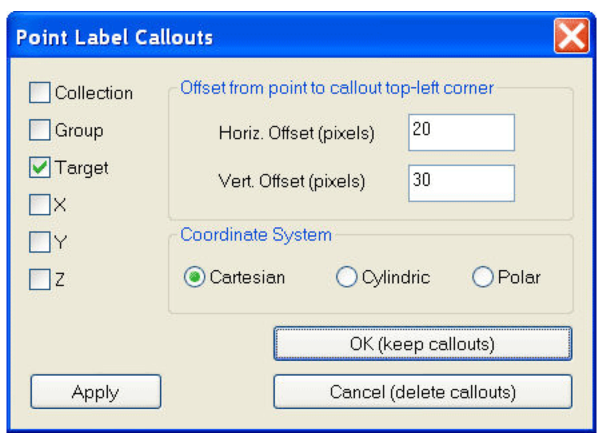


Figure 22-7. Point label callout properties.

Group Comparison

Compares the two selected groups. Their deltas are computed by subtracting the first selected from the second selected. Right-Click the callout to access properties.

Vectors

Annotates selected vector properties and tolerance (Figure 22-8). Right-Click the callout to access properties (Figure 22-9).

	20
dX	-0.0357
dY	0.0183
dZ	0.2697
Mag	0.2727
+	

Figure 22-8. A vector callout.

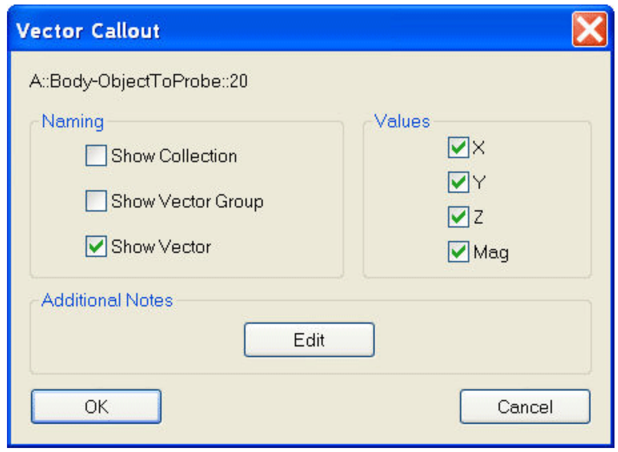


Figure 22-9. Vector callout properties.

Relationship

Annotates selected relationship report (Figure 22-10). Right-Click the callout to access properties.

Fit Circle
Circle3

	Nominal	Measured	Delta	Tolerance
Diameter	1.5000	1.5012	0.0012	Pass
X	-1.5000	-1.4972	0.0028	0.0018
Y	1.5000	1.4996	-0.0004	Pass
Circularity		0.0022		Pass
RMS		0.0008		
Mag XYZ			0.0036	
Mag XY			0.0028	
Angle Between			0.0856	

Figure 22-10. A relationship callout.

Object

Annotates selected object properties (Figure 22-11). Right-Click the callout to access properties.

Plane
A::Tabletop Plane

A	B	C	D
0.0000	0.0000	1.0000	1.0800
Proj. Ang.	Rx from Y	Ry from Z	Rz from X
(deg.)	90.0000	0.0000	0.0000

Figure 22-11. An object callout.

Callout View Properties

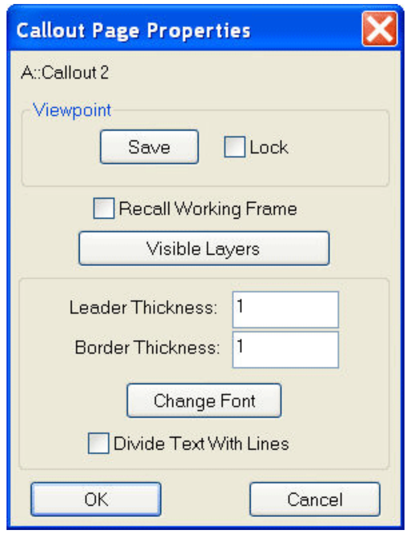


Figure 22-12. Callout view properties.

Viewpoint

Allows a user to save and lock the current view of the callout. When the callout is activated the saved view will be restored.

Recall Working Frame

Allows the current frame to be recalled when the callout is activated.

Visible Layers

Allows for layers to be used when a callout is activated.

Annotation Display Properties

Access to leader/border thickness and font control.

SA Reports

The SA Report allows a user to “drag and drop” items from the tree into the Spatial Analyzer Report Designer Editor where items can be arranged, formatted and edited to convey the user intent. If values/ information change in the tree, the SA report will reflect that change.

The Report Designer

The Report Designer is the canvas on which the user will create their report. The Report Designer supports standard functionality such as click and drag, re-sizing and Left/Right Clicking with context menus. The Report Designer also contains a toolbar (Figure 22-13) with useful reporting tools and options.

Report Toolbar

Figure 22-13. The Report Designer toolbar.

- **Print Current Report.** Displays print dialog for the current SA Report.
- **Save As PDF.** Allows the current SA Report to be saved as a PDF document.
- **Global Reporting Options/Page Setup.** Displays SA Report options such as Page Layout, Default Fonts and other options.
- **Report Refresh.** Refreshes the SA Report with the current values from the tree.
- **Document Zoom In.** Increases the scale of the report.
- **Document Zoom Out.** Decreases the scale of the report.
- **Document Zoom 100%.** Restores the report scale back to 100%.
- **Export to Excel.** Exports the current SA Report to Microsoft Excel.
- **Auto Adjust Report Whitespace.** Removes excess vertical spaces between report items.
- **Lock Report.** When clicked the SA Report becomes static and will no longer receive updates from the tree.
- **Help.** Displays useful information regarding how to navigate and interact with the Report Designer.

Report Header

The contents of the report header can be formatted to include a logo image and a company byline (Figure 22-14). To include this information navigate to **Edit>User Options**. For more information on global report options please refer to the Reporting Tab section.

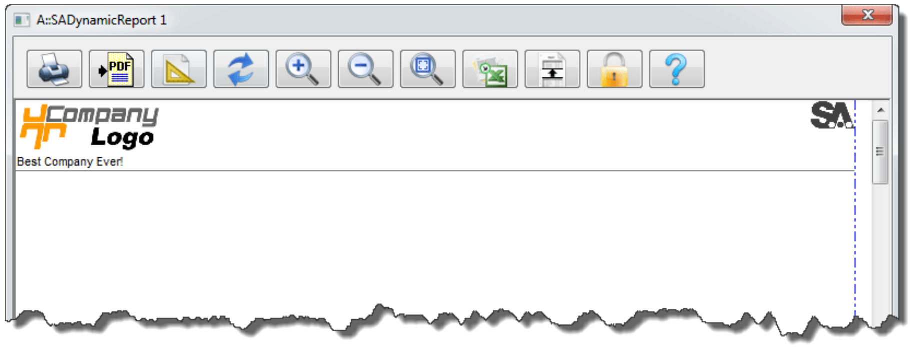


Figure 22-14. A report header in the Report Designer.

Page Layout

Warning: Before creating a report, it is important to select the appropriate page layout. The layout can be changed during report design, however it could change the layout of some existing items.

The report can be formatted in either Portrait or Landscape orientation. Use the page Global Reporting Options/Page Setup button to format the current report.

Creating an SA Report

An SA Report can be created several ways--as the result of a:

- Quick Report
- Composite Report
- Report Template

To create a blank SA Report, use the menu item **Reports>Add SA Report**.

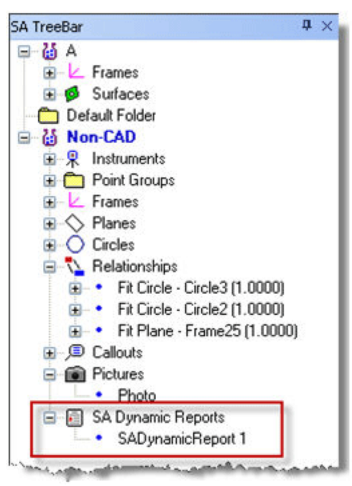


Figure 22-15. An SA report as it appears in the tree.

To open/edit an existing SA Report, expand the SA Reports category in the tree and right-click and select **View**. You can also double-click the respective SA Report to view it (Figure 22-15).

Adding Items to the Dynamic Report

Simply click on an item in the tree and drag into the report window. Once inside the report, items can moved by dragging them.

Adding Snapshots to the Report

Use the Camera Button to take a snapshot of the SA graphic window. The snapshot will be placed into the tree under the *Pictures* category.

The picture can be placed into the report by dragging, as with other items (Figure 22-16).

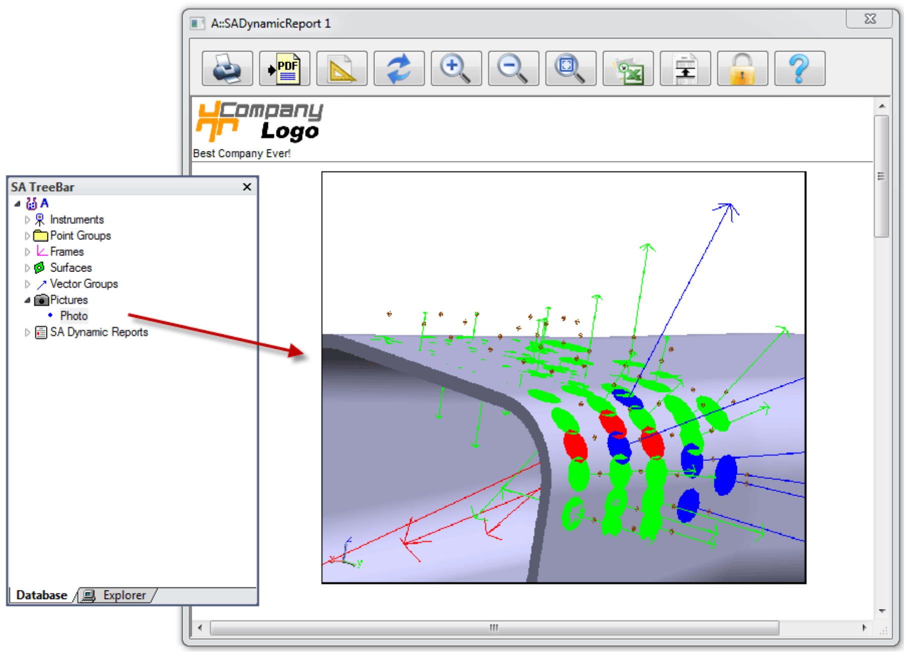
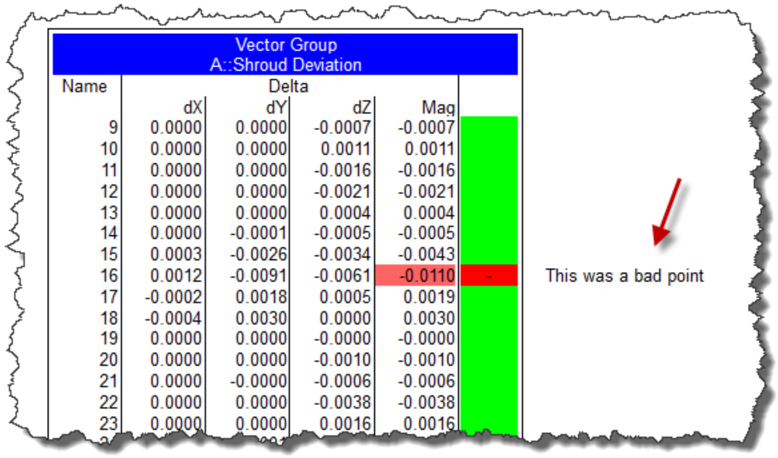


Figure 22-16. Dragging a picture into a report.

Once the picture is in the report it can be re-sized by dragging a corner of the image.

Adding Text to Report

Click any open space and start typing to add text. Or paste text from clipboard into any open space (Figure 22-17).



Vector Group A::Shroud Deviation					
Name	Delta				
	dX	dY	dZ	Mag	
9	0.0000	0.0000	-0.0007	-0.0007	
10	0.0000	0.0000	0.0011	0.0011	
11	0.0000	0.0000	-0.0016	-0.0016	
12	0.0000	0.0000	-0.0021	-0.0021	
13	0.0000	0.0000	0.0004	0.0004	
14	0.0000	-0.0001	-0.0005	-0.0005	
15	0.0003	-0.0026	-0.0034	-0.0043	
16	0.0012	-0.0091	-0.0061	-0.0110	-
17	-0.0002	0.0018	0.0005	0.0019	
18	-0.0004	0.0030	0.0000	0.0030	
19	0.0000	0.0000	-0.0000	-0.0000	
20	0.0000	0.0000	-0.0010	-0.0010	
21	0.0000	-0.0000	-0.0006	-0.0006	
22	0.0000	0.0000	-0.0038	-0.0038	
23	0.0000	0.0000	0.0016	0.0016	

Figure 22-17. A report annotated with some text.

Report Fields

Fields are text boxes that accept input via the keyboard or can be set by a Measurement Plan. There are two types of fields: *Input fields* and *Output fields*.

- **Input Fields.** Fields intended for input from the user. They display a yellow background (red until filled in, if a required field).
- **Output Fields.** Fields intended to display output. These fields have gray backgrounds.

To add a field to a report, right-click on the canvas and select **Add New Field**.

To add an input field, choose the **Input** radio button. If it is a required field, check the **Required** option. A required field will have a red background until it is filled in, and any attempts to print the report or send it to PDF will display a warning dialog.

To add an output field, choose the **Output** radio button. The value displayed in output fields can be set from a specific value, or from a tag (see below).

Tags

Every field (both input and output fields) can be assigned a tag. There are two types of tags: *system tags* and *user-defined tags*.

- System tags include `<< Date/Time >>` and `<< Filename >>` and automatically populate the field with the current date/time or filename (additional system tags will be added in the future). System tags can *only* be assigned to output fields. To create a system tag, select one of the system tags from the dropdown list in the *Tag* field.
- User-defined tags are tags that you have created yourself (such as "`MyTag`" that allow you to reference the field from measure-

ment plans so that a script can fill out the field automatically. To create a user-defined tag, just type a unique tag name into the *Tag* field. You can also assign an existing tag to a field in order to populate multiple fields with the same value.

Field Value

A field value is used to populate the field. Field values are always displayed in the field itself, and are changed when a script or a user modifies the field.

Tables

To add a table to a report, right-click the canvas and select **Add New Table**. Specify a name for the table, and the desired number of rows and columns.

Tables can be resized by dragging their corners, or the left/right/bottom sides. Interior cells can be resized by dragging any internal horizontal or vertical line. Drag the interior of any cell to move the entire table.

To modify the number of rows or columns in a table, right-click the table and select **Table Properties**.

To insert or delete rows or columns or to change the alignment of a cell, right-click a cell and then choose **Cell Options**.

Add an input or output field to a cell by right-clicking a cell and selecting **Cell Options**.

Modifying Items in the report

Items in the report can be modified by right-clicking the report item and selecting **Report Options**. Depending on the item, a context sensitive menu will appear. All items have the following common options: **Report Options**, **Remove from Report**, **Bring to Front**, **Send to Back**, and **Properties**.

Events

SA *Events* provide detailed information for major events that occur in SA. These events can be viewed in the Report Bar or be added to any Report (Figure 22-18). Events are created automatically when certain functions are performed and are stored in the tree.

SA Events are generated for:

- Best-Fit Transformations
- Best-Fit Geometry
- Move Objects by Minimizing Relationships

- Unified Spatial Metrology Networks (USMN)
- Quick Align to CAD
- Queries
- CAD Imports

and more to come!

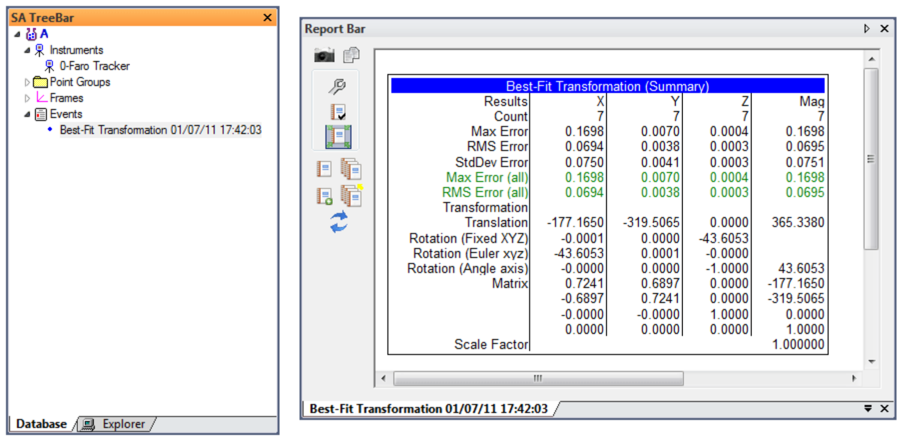


Figure 22-18. A best fit transformation event in the tree.

Object Report Options

Items in the tree have report options that can be accessed and configured per the users needs. You can access by right-clicking the object and selecting **Report Options** (Figure 22-19).

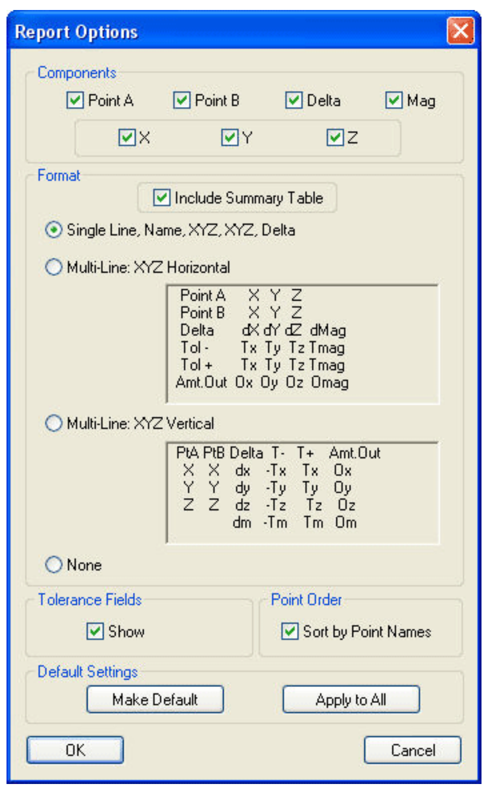


Figure 22-19. Report options.

Quick Report

A *Quick Report* is a fast and easy way to create an SA Report of a single object or multiple objects when using Composite Quick Report.

Creating a Quick Report

To create a Quick Report of a single item, right-click the item of interest in the tree and select **Generate Quick Report**. An SA Report of the object will be created and saved in the tree.

Creating a Composite Quick Report

A Composite Quick Report is a fast way to create an SA Report of several items. A Composite Report can be created by right-clicking a category in the tree and selecting **Composite Quick Report** or from the menu **Reports>Composite Report - All Collections**.

To exclude or include an item from a Composite Quick Report, simply right-click the respective item in the tree and select **Include in Composite Quick Report**.

Report Bar

The Report Bar can serve as quick reporting tool allowing the user to view items quickly and also store the views in a tabular format (Figure 22-20). The report bar can also combine the report tabs into a composite report.

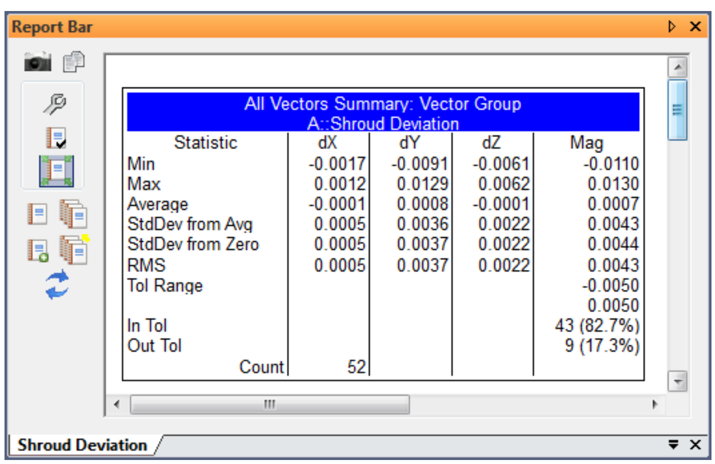


Figure 22-20. A vector group in the Report Bar.

Report Bar Toolbar

- **Capture Current View.** Creates a picture item in the tree of the current graphical view. The picture will display in the report bar under the current tab.
- **Clipboard Help.** Displays help information regarding meth-

ods of copying Report Bar data to the clipboard.

- **Object Properties.** Displays the current tab object properties.
- **Current Object Report Options.** Displays the report options for the current tab.
- **Include Current Object in Composite Report.** Sets the current object to be used in a Composite Report.
- **Quick Report.** Generates a Quick Report from the current tab.
- **Generate Quick Report from Tab Order.** Generates a Quick Report from all Report Bar tabs.
- **New Report Tab.** Creates a new tab in the Report Bar.
- **Close All Tabs except Current.** Closes all Report Bar tabs except for the currently selected tab.
- **Refresh Report.** Refreshes the Report Bar with current information.

Displaying the Report Bar

To display the Report Bar use the menu option **Report>Report Bar Visible**.

Using the Report Bar

The Report Bar is a great way to view object properties and values quickly. Multiple items can be added to the Report Bar for continuous reviewing. Once finished, the Report Bar can be cleared or sent directly to an SA Report for final output.

Adding an Item

To add an item in the Report Bar simply select an item in the tree by single-clicking the item. Once selected it will display in the Report Bar. The currently selected item in the tree will always display in the current Report Bar Tab (Figure 22-21).

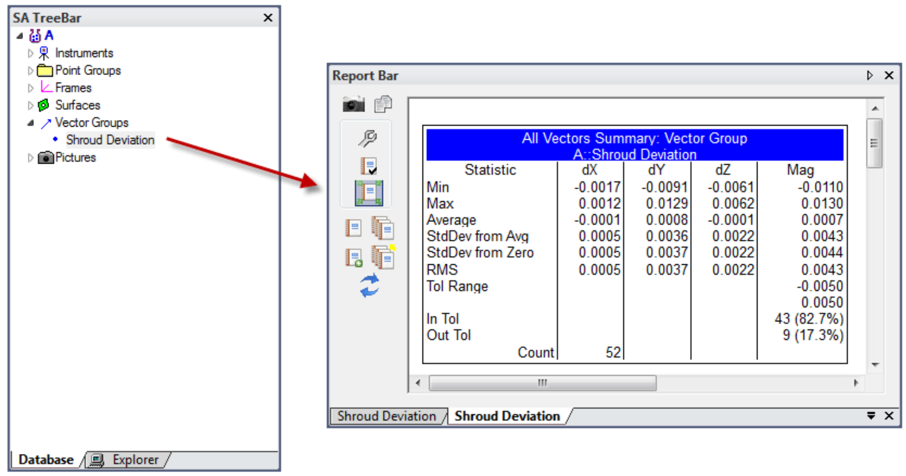


Figure 22-21. The item currently selected in the tree is displayed in the Report Bar.

Adding multiple items to the Report Bar requires multiple tabs. Add a new tab by using the icon. The new tab will appear with the currently selected tree item. Select the next item from the tree to view.

Removing an Item

To remove a Report Bar item, click the "X" icon in the bottom right-hand corner of the Report Bar (Figure 22-22). All tabs, except the current tab, can be cleared by using the icon.

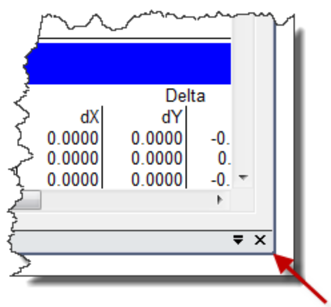


Figure 22-22. Removing a Report Bar item.

Generating a Report

If needed, an SA Report can be generated from one or all of the Report Bar tabs. Use to generate a report of the current tab or to generate a report from al Report Bar tabs.

Report Templates

SA Report Templates offer a repeatable method for creating reports of items from the tree. Once created they can be used repeatedly.

Creating an SA Report Template

Select **Reports>Add New SA Report Template** from the menu and a report template will be created in the tree (Figure 22-23).

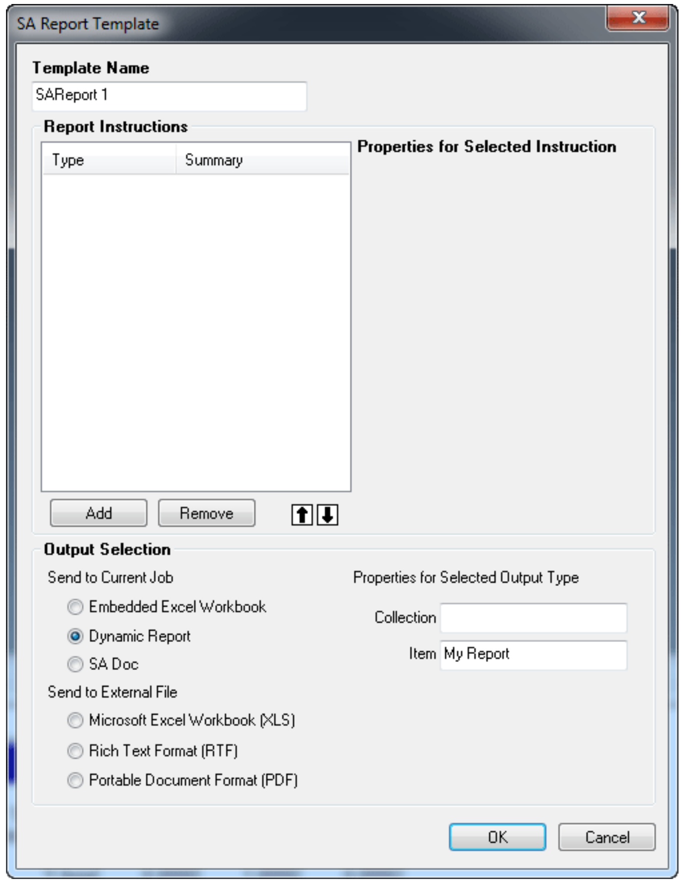


Figure 22-23. The SA Report Template dialog.

Inside the SA Report Template the inputs and outputs are controlled in the *Report Instructions* box (Figure 22-24). Simply use the Add or Remove buttons to control items availability in the report.

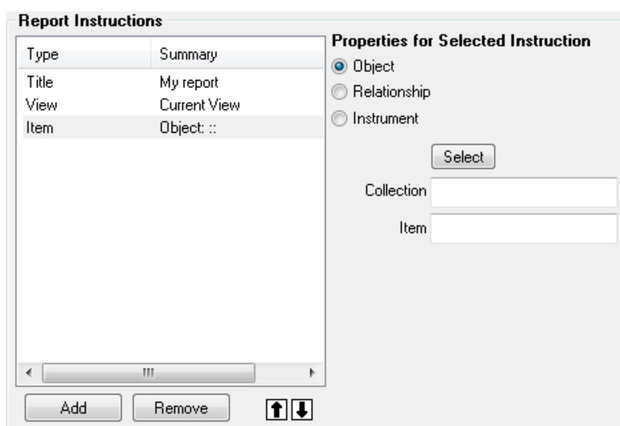


Figure 22-24. The Report Instructions box.

The *Output Selection* defines the output format and location of the report when it is generated (Figure 22-25). Choose from embedded locations or external locations.

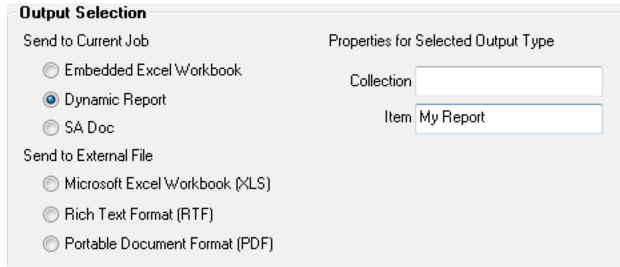


Figure 22-25. The Output Selection box.

Generating the Report

Once the Report Template is designed and saved, a report can be generated. Simply right-click the respective SA Report Template and choose **Generate** (Figure 22-26). The report will be created and saved in the prescribed output format.

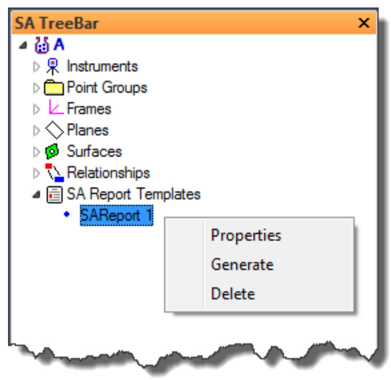


Figure 22-26. Generating a report from a template.

This Page Intentionally Left Blank.

Dimensioning

CHAPTER 23

The ability to depict as-built distances and angles provides a great way to communicate measurement results visually. Dimensioning is a flexible tool that is designed for displaying these measurements in the graphical view, with flexible options to provide you a great deal of control.

Dimensions provide a method to annotate the graphical view with both linear and angular dimensions between any combination of points and objects (Figure 23-1). They are dynamic entities: Both the numeric values and the leader lines update as the points and objects move.

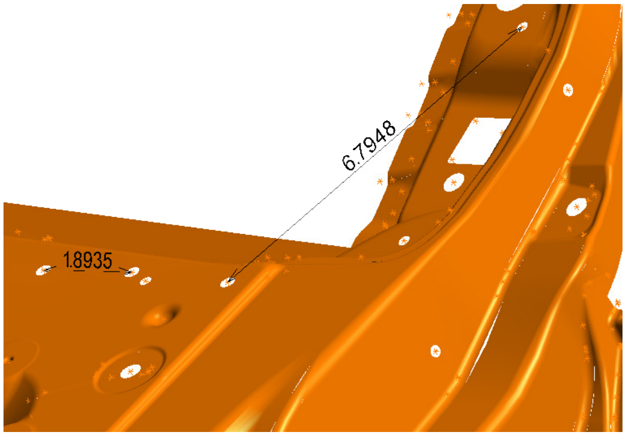


Figure 23-1. Dimensions between hole centers.

Types of Dimensions

There are several different types of dimensions that you can create:

- **Point to Point.** Shows the linear distance between two points.

- **Point to Object.** Shows the linear distance between a point and any object.
- **Object to Object.** Shows the linear distance or angle between any two objects.
- **Diameter/Radius.** Shows the diameter or radius of a circle or cylinder.

Working With Dimensions

To Create a Dimension:

Do one of the following:

- From the **Query>Make Dimension** menu, select the desired dimension type. Select the appropriate points or objects to dimension. A new dimension will appear in the graphical view.
- When the ***Query Results*** dialog is displayed as the result of an appropriate query, check the **Create Dimension** checkbox (Figure 23-2):

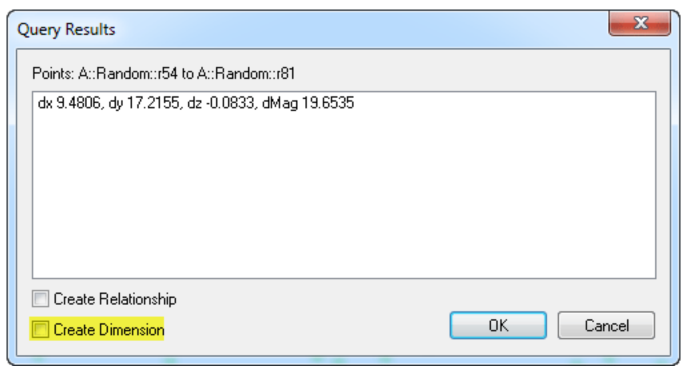


Figure 23-2. The *Create Dimension* checkbox in the ***Query Results*** dialog.

Common Properties

All dimensions in a collection have a set of common properties which can be adjusted for all dimensions in the collection at once. These include the text size for the dimension, and the dimension orientation.

Dimensions can be oriented based on their leader lines, or can be set to always face the screen:

- **Screen.** The dimension always faces the screen.
- **Along.** The dimension is oriented along the leader line.
- **Leave Unchanged.** The dimension orientations are left unchanged.

To Change Settings for All Dimensions in a Collection at Once:

- Right-click the *Dimensions* category in the tree Dimensions and select **Set Common Properties** to open the ***Common Dimen-***

sion Properties dialog.

To Change the Size of All Dimensions in a Collection:

- In the ***Common Dimension Properties*** dialog, enter a new value in the *Text Size* field, then click OK.

To Change the Display Orientation of All Dimensions in a Collection:

- In the ***Common Dimension Properties*** dialog, select the *Screen*, *Along*, or *Leave Unchanged* option.

To Change a Single Dimension's Properties:

- Double-click the dimension in the tree, or
- Right-click the dimension in the tree or graphical view and select **Properties**.

Dimension Properties

Point to Point Dimensions

These dimension types depict the distance between two points (Figure 23-3).

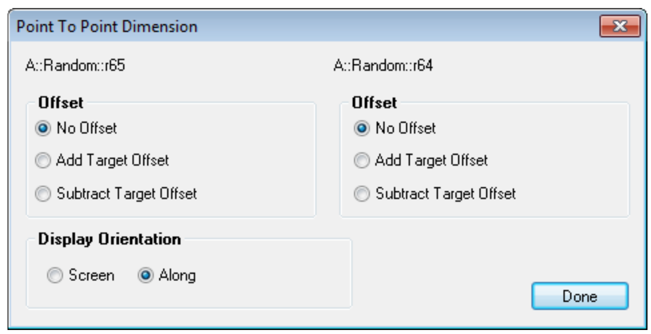


Figure 23-3. The properties of a Point to Point dimension.

Each point has an offset setting:

- **No Offset.** Dimension from the target or point's center.
- **Add Target Offset.** Adds the target's offset to the dimension. This simulates an inside caliper (as if you'd measured the inside of a box).
- **Subtract Target Offset.** Subtracts the target's offset from the dimension. This simulates an outside caliper (as if you'd measured the outside of a box).

You can also control the display orientation for the dimension, as described in "Common Properties" on page 656.

Point to Object Dimensions

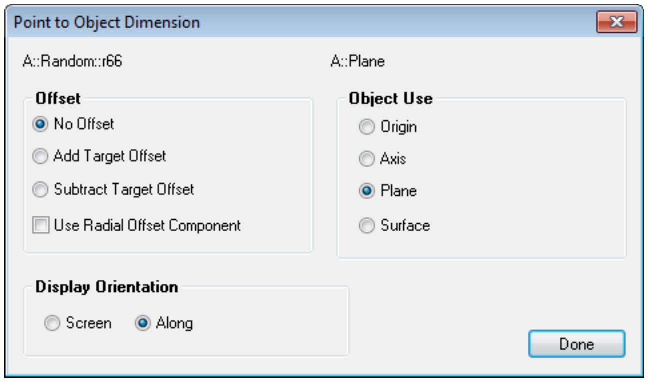


Figure 23-4. Properties of a Point to Object dimension.

The point end of this dimension is the same as described in "Point to Point Dimensions" on page 657. There is an additional option to *Use Radial Offset Component*. If enabled, the radial component of the measurement offset will be used for compensation instead of the planar comonent. This is useful, for instance, when you are measuring the distance from the closest point in a hole to a surface edge plane and are acquiring the hole measurement with a pin nest. On the other end of the dimension, the object can have one of four settings:

- **Origin.** Distance to the object's internal origin (Figure 23-5). For instance, the center of a circle or plane.

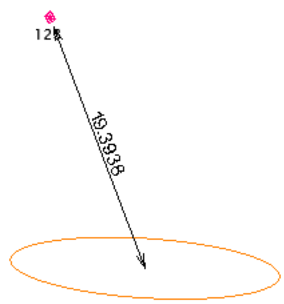


Figure 23-5. Origin.

- **Axis.** Distance to the object's projected (extended) axis (Figure 23-6).

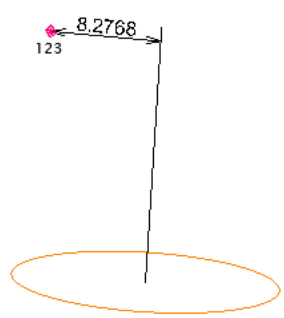


Figure 23-6. Axis.

- **Plane.** Distance to the plane of the object, if applicable (Figure 23-7).

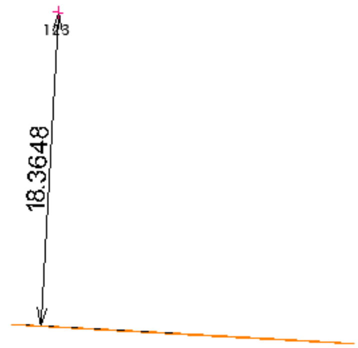


Figure 23-7. Plane.

- **Surface.** Show the distance to the closest point on the object's surface/line/curve (Figure 23-8).

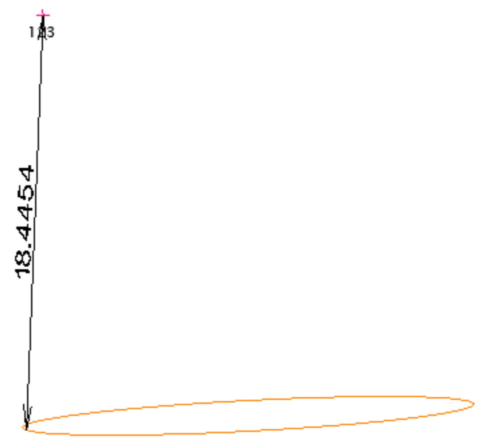


Figure 23-8. Surface.

You can control the display orientation of the dimension as described in "Common Properties" on page 656.

Object to Object Dimensions

In addition to dimension linear distances between objects, object to object dimensions enable angular dimensioning as well (Figure 23-9).

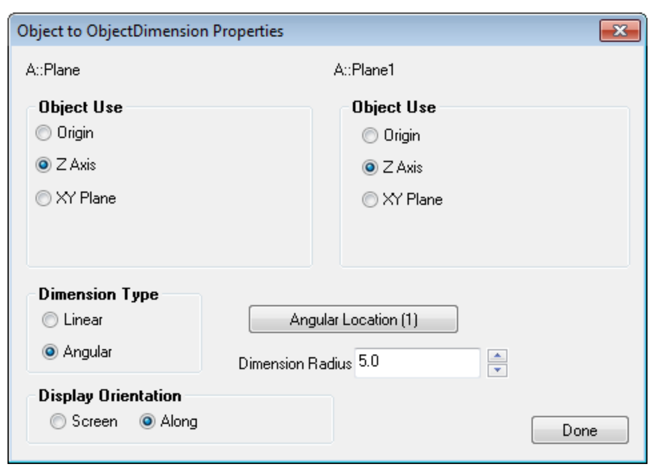


Figure 23-9. Object to Object Dimension properties.

The dimension can be tied to one of three features of each object:

- **Origin.** The distance from the object's internal origin. This feature setting for the dimension does not allow angular dimensioning, therefore the **Angular** option becomes grayed out.

> **Note:** The internal Z axis of an object is typically the object's normal (for planes, circles, etc.) the Z axis (for frames) or the line direction (for lines).

- **Z Axis.** The distance to the object's projected Z axis (if **Linear** is selected), or the angle to the object's internal Z axis (if **Angular** is selected).
- **XY Plane.** The distance to the object's XY plane (if **Linear** is selected), or the angle to the object's XY plane (if **Angular** is selected).

Two settings particular to angular dimensioning dictate where the angular dimension appears:

- **Dimension Radius.** The distance of the angular dimension from the mutual intersection of the two objects.
- **Angular Location.** This button cycles through the four quadrant representations of the given angle, as illustrated in Figure 23-10 through Figure 23-13.

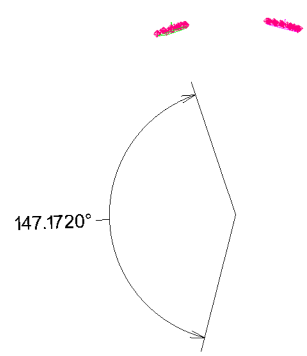


Figure 23-10. Angular quadrant 1.

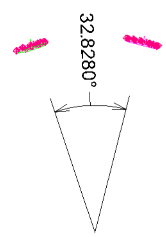


Figure 23-11. Angular quadrant 2.

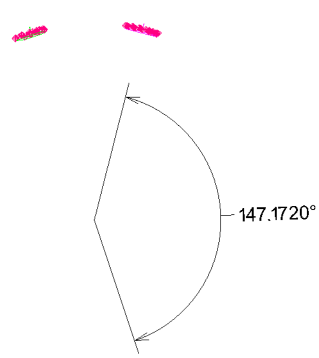


Figure 23-12. Angular quadrant 3.

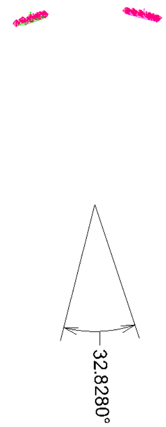


Figure 23-13. Angular quadrant 4.

You can control the display orientation of the dimension as described in “Common Properties” on page 656.

Diameter / Radius Dimensions

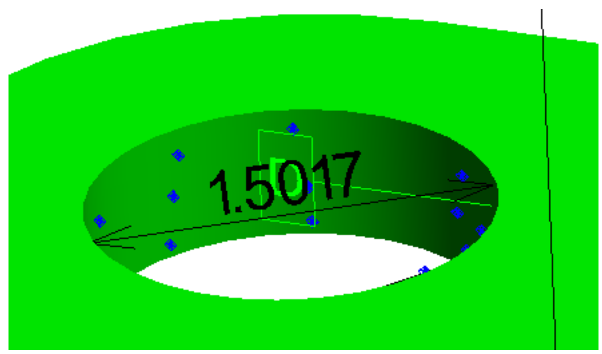


Figure 23-14. A diametrical dimension.

Diameter/Radius dimensions are ideal when the diameter or radius of a circle, cylinder, or sphere needs to be dimensioned (Figure 23-14).

The diameter/radius dimensions are shown in Figure 23-15:

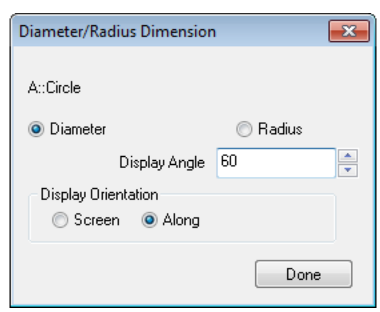


Figure 23-15. Properties of a Diameter/Radius dimension.

- **Diameter/Radius.** Specify whether you’d like to dimension the object’s diameter or radius.
- **Display Angle.** Specifies the angle around the circle to which the dimension is oriented.

You can control the display orientation of the dimension as described in “Common Properties” on page 656.

This Page Intentionally Left Blank.

GD&T

CHAPTER 24

Geometric dimensioning and tolerancing, more commonly known as GD&T, is a system for defining and implementing engineering tolerances for part and assembly geometry. In this chapter, we will discuss how SA supports the engineer's ability to read and communicate GD&T design from CAD drawings.

SpatialAnalyzer's GD&T Inspection tools provide engineers with the ability to perform standardized inspections in accordance with the guidelines of the ASME or ISO standards. It provides the tools to allow the technician to import fully annotated CAD models and utilize the annotations and tolerances provided, or build their own annotations from CAD models, SA geometry, or even directly from measured points that define geometry. It also provides the ability for a designer to build an inspection process that walks an operator through the process of measuring the datums and features of interest following predefined measurement location indicators. Once points are assigned to a feature check it is immediately validated with a pass or fail result. The check can then be added directly to dynamic reports.

SA makes every effort to be true to the ASME standard in the GD&T evaluation process. All calculations are based on the ASME Y14.5M-1994 guidelines but provide users a wide range of flexibility dependent upon their needs. Introduced in the 2012.12.06 version are GD&T pre-check validations which ensure full compatibility with the desired standard. The supported standards include:

- ASME (1994)
- ASME (2009)
- ISO (1983)

- ISO (2004)

You may select the standard you wish to use by navigating to the Analysis tab in the ***User Options*** dialog. Select GD&T Options and then choose one of the options listed in Figure 24-1.

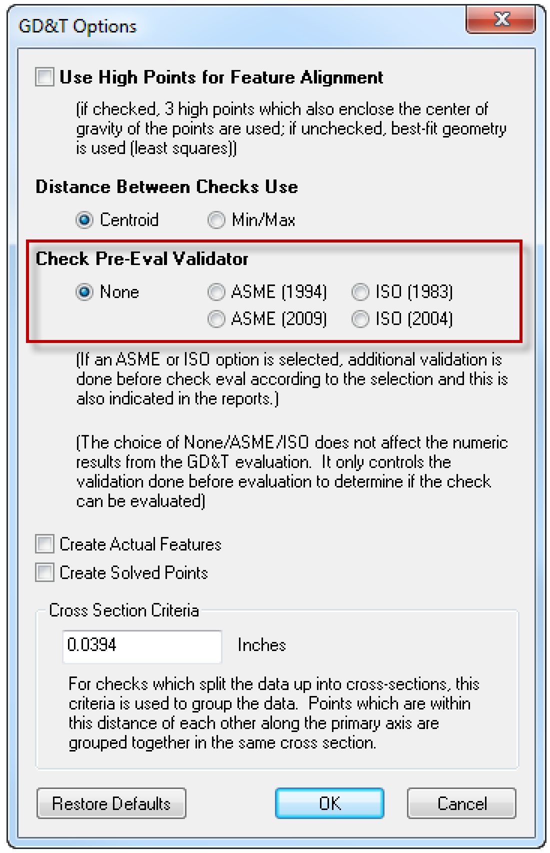


Figure 24-1. GD&T Options dialog.

Choosing a pre-check evaluation does not affect the numeric results but rather allows the user to validate the check in advance to ensure that the check is in accordance with the specific standard. As a result some checks will fail due to a noncompliance with a particular standard that would pass without pre-checks in place. However, this allows the user to be fully-compliant with the desired version of either the ASME or ISO standard.

Importing CAD with GD&T Annotations

To import a CAD model with annotations select **File > Import > Direct CAD Access**. If you would like to change your import preferences, navigate to **File > Import > Direct CAD Access Settings**.

In the ***Direct CAD Access Settings*** dialog, check the *Annotations* option. Once imported, the annotations will appear graphically and in the tree in the Annotations category (Figure 24-2) .

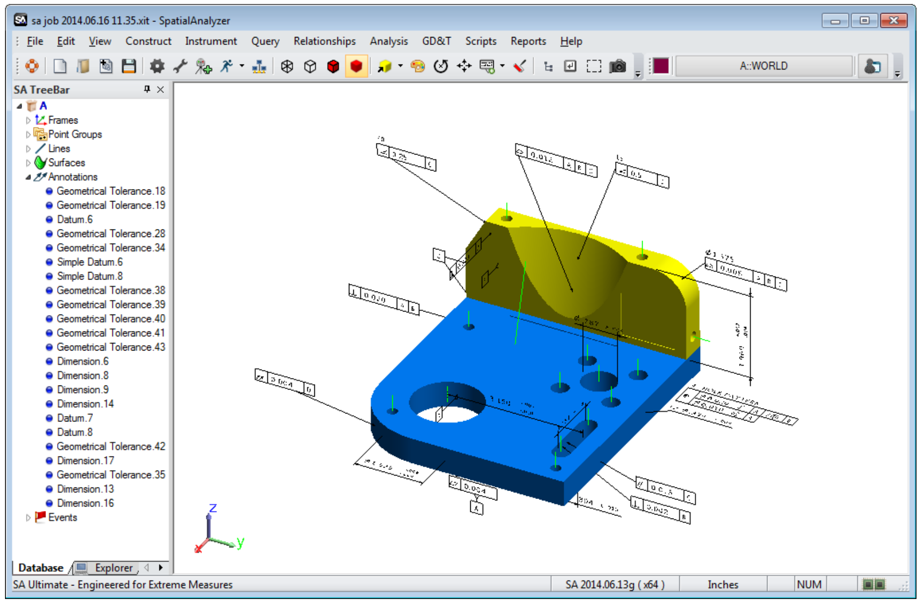


Figure 24-2. Imported CAD Annotations.

Creating GD&T Annotations

In many cases the GD&T annotations will be imported with the CAD model. However, SA provides the ability to add GD&T annotations to a CAD model that does not have them and provides the tools necessary to conduct a GD&T analysis on constructed SA geometry and also points within SA that define geometry. To build your own annotations in SA use the GD&T tab in the SA toolkit (Figure 24-3).

Creation Options

The GD&T Annotation tab is located within the SA Toolkit. If it is not visible, select **View > Show Toolkit Bar** from the menu. This toolbar is the starting point for building annotations and setting up GD&T evaluations when not importing annotations with a CAD model. With a logical workflow starting from the top and working toward the bottom, the toolbar begins with the *Creation Options* section.

View

This sets the orientation of the text for the annotation to be added. The view orientations available are drawn from the Preset Views list which can be accessed and edited by going to **View > View Control > Preset Views**. The view selected, as well as the text size, can be edited within the annotation properties once the annotation is created.

Figure 24-3. GD&T tab in Toolkit.

Feature Type

The Feature Type defines and limits the type of objects to which an annotation may be added. You can choose from *CAD Faces*, *SA Objects* or *SA Objects (fit to Points)*. In all cases the GD&T annotations must be added to the *nominal (design)* features. If these nominal features do not exist in your job they must be created.

- When you select *CAD Faces*, the annotation will be added to and include the specific selected CAD faces you chose from a particular model regardless of whether or not a model has been dissected into individual faces.
- Selecting *SA Objects* allows selection of any primitive SA geometry type as the nominal feature.
- *SA Objects (fit to points)* allows you to add an annotation directly to measured points. When you select this option you will be limited in your choices to building Datums, Form, and Orientation Checks. When adding annotations you will be prompted to define the geometry type you wish to fit to the points. This nominal geometry is then created for you from the points se-

lected and the type of feature check used.

Datum Annotations

Once the *Creation Options* are set, Datums and Datum targets can be added to the model by simply clicking on a CAD face, SA object or selecting points. The datum letter is incremented automatically. To place a specific datum letter on a model, hold down the letter you wish to add on the keyboard when you click the datum button. You can then click on a CAD face or SA object and that particular letter will be used. Datum targets are added with the datum name and a numeric indicator in the same way that datums are added. Simply click on the datum target button and on the point on the datum surface that you wish to add the target. That click point will then be used to anchor the leader line for the annotation. Once a datum/target is selected, the ***GD&T Datum*** dialog will appear (Figure 24-4).

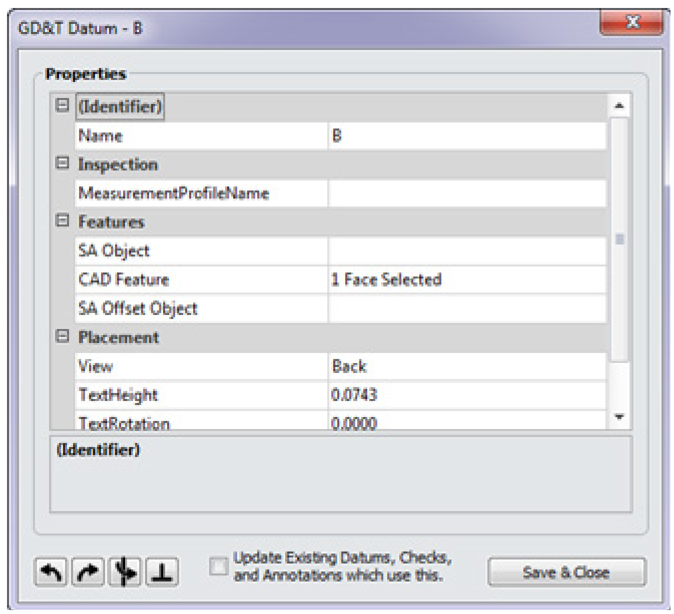


Figure 24-4. GD&T Datum dialog.

The ***GD&T Datum*** annotation properties dialog allows you to edit the following settings:

- **Datum Name (Identifier).** The datum letter used as an identifier in the annotation (currently support single and double letter annotations).
- **Inspection.** *MeasurementProfileName* sets the measurement mode for the selected instrument when the Datum is created for inspection through the guided inspection feature.
- **Feature Type.** Provides a selection prompt and resulting selection list of the SA features for reference in the check. You must select either CAD faces or SA objects. The selected SA objects define the annotation placement and the features used during datum alignment. The selected CAD faces define the annota-

tion placement and the features used during datum alignment. Where applicable, SA Offset Objects provides directional reference information or alignment restrictions for surface datums.

- **Placement.** This will define the text size and orientation in the job. The View defines the saved orientation of annotation text. The Text Height defines the height of the annotation text. This is expressed in job units. The Text Rotation defines the rotation in the current view.
- **Cylinder Mode.** Allows for selection between hole and pin cylinder evaluations. It is used to properly define probing offsets.
- **CAD Feature.** Provides a radio button which forces the selected feature to be interpreted as a general surface.

GD&T Feature Check Annotation Properties

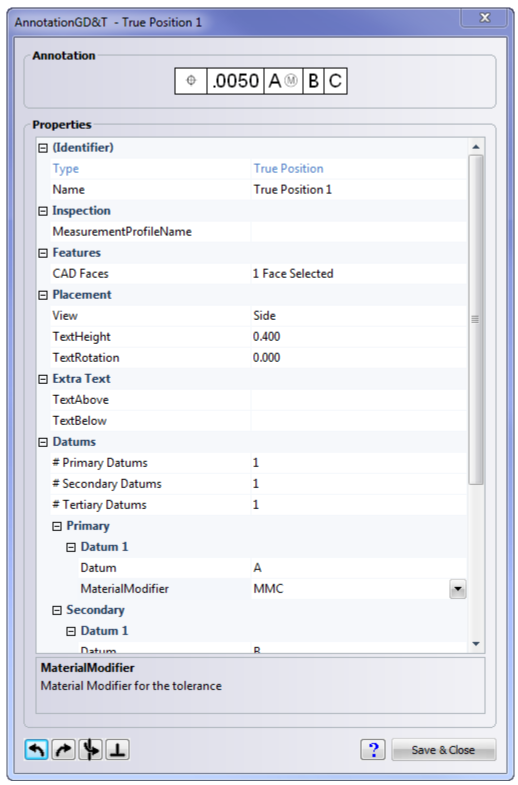


Figure 24-5. Properties for a True Position check.

- **Type.** Feature check type.
- **Name.** The name of the check used to describe the annotation in the tree as well as reporting.
- **Measurement Profile Name.** Sets the measurement mode for the selected instrument when the Datum is created for inspection.
- **CAD Feature.** Prompts user to select CAD faces for the respective feature.

- **View.** Changes orientation of annotation.
- **Text Height.** Defines the height of the annotation text. Expressed in job units.
- **Text Rotation.** Defines rotation in the current view.
- **Text Above and Text Below.** Adds additional text above and below the annotation.
- **Number of Primary, Secondary and Tertiary Datums.** Enter the respective Primary, Secondary and Tertiary Datums if applicable.
- **Zone.** Applicable tolerance zone
- **Tolerance.** Tolerance value for the feature check.
- **Material Modifiers.** Defines the material modifier for the feature check's selected datum. Choose from *Maximum Material Condition, Least Material Condition* or *Regardless of Feature Size.*
- **Has Projected Zone.** If checked, the projected zone will be used.
- **Projected Zone Value.** Value used when projected zone is being used.
- **Tolerance Lower Tier.** Tolerance used in Lower Tier evaluation.
- **Outer Tolerance Lower Tier.** This designates the amount of the tolerance outside the surface of the part for the lower tier check. It allows for asymmetrical tolerance zones to be set. For symmetric tolerances this value should be set to one-half the total tolerance.
- **Material Modifier Lower Tier.** Optional material modifier designation for the lower tier tolerance.

GD&T Dimension Tolerances

To add dimensional annotations you must have first set the desired View and Feature Type followed by the Tolerance type, whether that be Limit (which provides the option to enter the low value limit and high value limit) or Nominal +/- (which provides the option to enter the nominal value and the low and high deviation values from that nominal). These settings are saved with the annotation and used in the evaluation. The tolerance definition can then be edited in the annotation properties later dialog if needed. Figure 24-6 illustrates how the settings in the property grid relate to the annotation graphic:

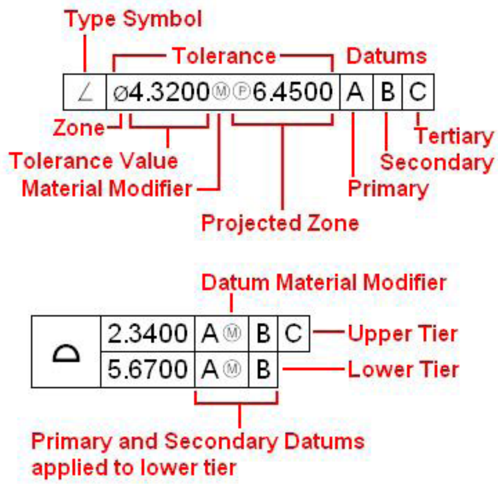


Figure 24-6. Anatomy of a GD&T annotation.

Graphical Placement of Annotations

With the annotation editor open the graphical annotation can be moved and placed in a position that is easy for the user to read. The annotation leader line is defined when the first surface face is selected under the CAD Features parameter. To move the annotation simply select the annotation in the graphics by left-clicking the annotation. While holding the left mouse button the annotation can be moved in the plane of the selected view. The view can be change under the View parameter in the annotation editor. Controls at the bottom of the annotation editor help in fine tuning the position and orientation of the annotation.

- Rotates the annotation box 90 degrees counter clockwise.
- Rotates the annotation box 90 degrees clockwise.
- Mirrors the annotation box.
- Straightens the leader line normal to the selected surface face.

While moving the annotation, **Alt** can be combined with the left mouse button to move the annotation along the leader line.

GD&T Entities

If you look in the TreeBar in SA you will find that all imported and constructed annotations are contained within an *Annotations* category. Often it is useful to contain all the design CAD and Annotations within a single collection. You can then create a new collection for a particular inspection. To inspect a feature simply right-click on the Annotations category and select *Make Feature Checks.* The annotation will be decoded and two additional categories will be built: *Feature Checks* and *Datums*. Feature Checks and Datums are much like relationships in that they maintain the association between measured points and the features specified in the annotation. (For more information on associating points to with datums and feature checks refer to the GD&T Workflow section).

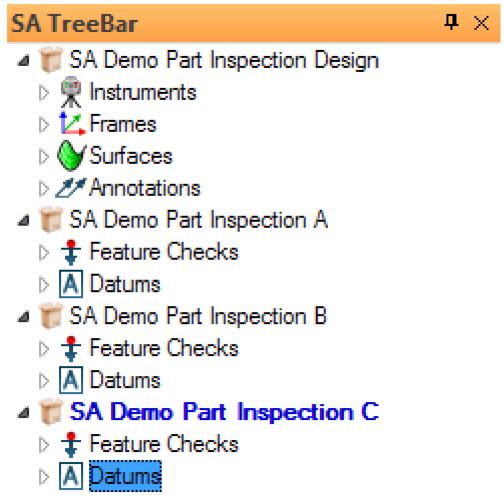


Figure 24-7. GD&T Entities in the Treebar.

Datums and Feature Checks

The ***Datums*** dialog (Figure 24-8) as well as the ***Feature Checks*** dialog (Figure 24-9) have identical settings with a few minor exceptions:

Figure 24-8. Datum properties dialog.

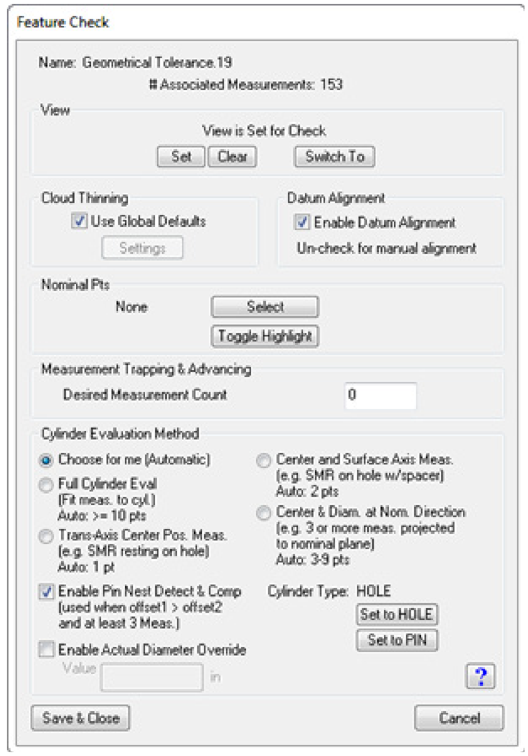


Figure 24-9. Feature Check properties dialog.

- **Indirect Measurements (Datums only).** At times, it can become necessary to measure a feature indirectly. Meaning for example, that you measure the table top to represent the back surface of a part clamped to the table. Normally the probing direction of a surface defines the way in which we handle probing offsets, however in this case, you are both measuring off the true surface and from the opposite direction. To account for this we have an option in the ***Feature Check*** properties dialog that allows you to specify that the measurements were taken indirectly.
- **View.** Provides controls for the views saved in the GD&T Inspection Design, Inspection Inspect workflow process (see section on workflow). The views saved from the Inspection Design interface are saved here and can be recalled or edited by going directly to this section in the specific datum or feature check.
- **Cloud Thinning.** Controls the extent of thinning performed on point clouds associated with this check during evaluation and defaults to the global settings.
- **Datum Alignment (Feature Checks only).** When doing a GD&T inspection, alignment is performed relative to the specified datums for each individual check. The current instrument alignment *is not* used for analysis unless specified. To do so, disable datum alignment for a particular feature check by going to this feature in the feature check properties dialog. When you

do so, the datum annotation will be crossed out in the evaluation (Figure 24-10).

Figure 24-10. Disabled datum alignment.

- **Nominal Points.** Provides control of the selected nominal points used in the GD&T Inspection Inspect routine. These points are used to lay out the bulls eyes on the part used by the inspector as a guild to measure the part in the correct location. Nominal points can be added to the job through the Inspection Design interface or can be selected manually for a particular feature through this dialog. The highlight button can be used to graphically identify the nominal points.
- **Measurement Trapping & Advancing.** This parameter is used to control the number of points captured from a live instrument during trapping. When the specified number of points are associated with the feature trapping is stopped automatically (refer to GD&T Workflow section).
- **Cylinder Evaluation Methods.** (Figure 24-11) The way in which cylinders are fit to the data depends on the number of measurements. A full cylinder fit is only attempted if you have 10 or more measurements of the feature. This is essential to understanding how the data is used in the check.

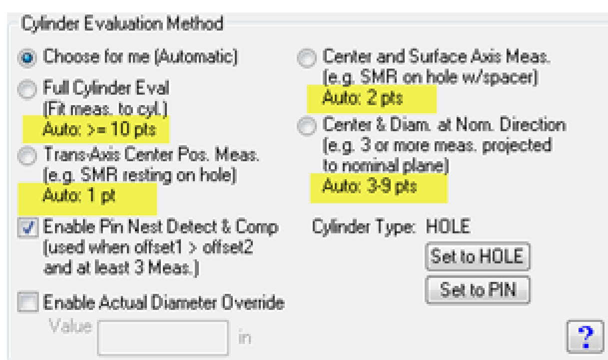


Figure 24-11. Cylinder Evaluation Method section.

Cylinder Evaluation Methods

- **Choose for me (Automatic).** This is the default used when creating datums and checks with cylindrical features. In this mode, the evaluation method is determined automatically based on the number of measurements. When set to this option the following designations will be made: if one point is associated, *Trans-Axis Center Pos. Meas.* will be chosen; if two points are associated, then *Center and Surface Axis Meas.* will be selected; if three to nine points are associated, then *Center & Diam at Nom. Direction* will be selected; and if ten or more points are associated then *Full Cylinder Eval* will be chosen.

- **Full Cylinder Evaluation.** This method does a cylinder fit from the provided measurements. The points are assumed to be measurements of the cylinder walls (pin or hole), and the length of the cylinder is determined by the extent of the measurements along the cylinder axis. In this case the measurements are used to determine the central axis and radius/diameter of the cylindrical feature.
- **Trans-Axis Center Position Measurement.** This method takes either a single point or averages a set of points and compares the resulting point to the nominal cylinder axis while using the nominal direction and radius for the evaluation. An example of this type of check is resting a probe tip of a larger diameter on a hole.
- **Center and Surface Axis Measurement.** This method assumes that multiple measurements of the cylinder axis were taken at varying depths. A line is fit to the measurements and that line determines the center and direction of the cylinder while the nominal cylinder is used for the radius. For example, using a probe resting on a hole with and without a spacer could be used to define a cylinder center axis.
- **Center & Diameter at Nominal Direction.** This method projects the associated measurements to the end cap of the nominal cylinder. The results of the projection are then used to fit a circle. The circle then determines the center point and the diameter while the nominal cylinder is used for the direction. This is an excellent choice in situations where you have very little coverage of the cylinder on either side of a cross section plane to define the cylinder axis.

GD&T Alignment

When performing a GD&T inspection, alignment is performed relative to the specified datums for each individual check. The current instrument alignment is not used for analysis unless specifically specified for the specific feature check within its properties dialog (refer to the feature check properties section for more details).

To get an idea of the alignment used by the GD&T evaluation with specific datums, right-click on the Datum Category and select "Align". You will be prompted to specify the datum scheme, picking primary, secondary and tertiary where applicable. This will then move the instruments and points associated with the datums to the correct orientation based on this datum scheme. Because each feature check is evaluated based upon its own datums it will be necessary to include or exclude only the datums you wish to use in the alignment to simulate the alignment for the particular check. This alignment tool is for visual purposes only, it will not be exactly duplicated during the evaluation of the features.

To accelerate and the calculation process and to ensure that an ideal alignment solution is found for a particular check it is highly recommended that an initial instrument alignment, using one of SA's other alignment tools such as Quick Align to CAD or Relationship Fitting, is performed first prior to performing the check, particularly if using surface datums.

Surface Alignment

When using datums that reference complex (non-planar / cylindrical) surface features some additional requirements are involved:

- The data should be pre-aligned to the part using other non-GD&T techniques before attempting GD&T analysis, such as Quick Align to CAD or relationship fitting.
- It is necessary to specify an offset plane or line with the datum annotation which will be used to constrain the degrees of freedom for that datum during alignment.

This offset geometry is what will "maintain contact" during alignment to subsequent datums and/or the main tolerance feature and so it needs to be chosen in such a way that it represents the control the datum is intended to exert over the solution.

Normally GD&T evaluation begins with an initial alignment step internally and also uses an iterative solution. When surface datums are involved both are disabled. This was necessary to ensure stable results with surface datums and since the data is required to be close before starting they aren't needed anyway.

GD&T Evaluation

The GD&T Analysis within SA follows the ASME standard evaluation. Therefore you will find that the results of a geometry fit or relationship fit which are based on an RMS fit will not return exactly the same results. Take the example of a flatness check. When we build a Geometry relationship fit to the points we report the flatness. This value is based on comparing the point deviations to the best fit plane. In GD&T, however, flatness is evaluated as two parallel planes bounding the data. These planes are then free to rotate in space to find the closest position they can reach to each other while still containing all the measured points between them. This process will almost always return a slightly better result than an RMS fit because of this additional degree of freedom.

Decimal Precision

It is important to understand that the displayed decimal precision imported with an annotation or set in the Users Options does not limit the underlying GD&T Analysis. So, for example, if you have the decimal precision set to 2 digits and have a flatness check with a tolerance of 0.02, the check can still fail if the measured deviation is 0.02001. This makes sense considering that the standard is based upon hard limits. The measurement is compared directly to the specified tolerance value and any deviation outside the limit signifies nonconformance to those limits. If you can't get the pin in the hole then it fails.

Evaluation Zone/Length

The evaluation zone/length is set by the extents of the measurements associated with the feature check. Therefore if you have a tolerance specification for a particular length part but can only measure a portion of that part. It may be necessary to adjust the tolerance to account for the shortened evaluation zone.

Cylinder Coverage

When measuring cylindrical features and using the "full eval" mode, it is important to provide enough measurements so that the intended cylinder orientation can be determined from the measurements.

Consider the following cylinder with 8 measurements in Figure 24-12. It is possible to interpret the cylinder in 3 different basic orientations.

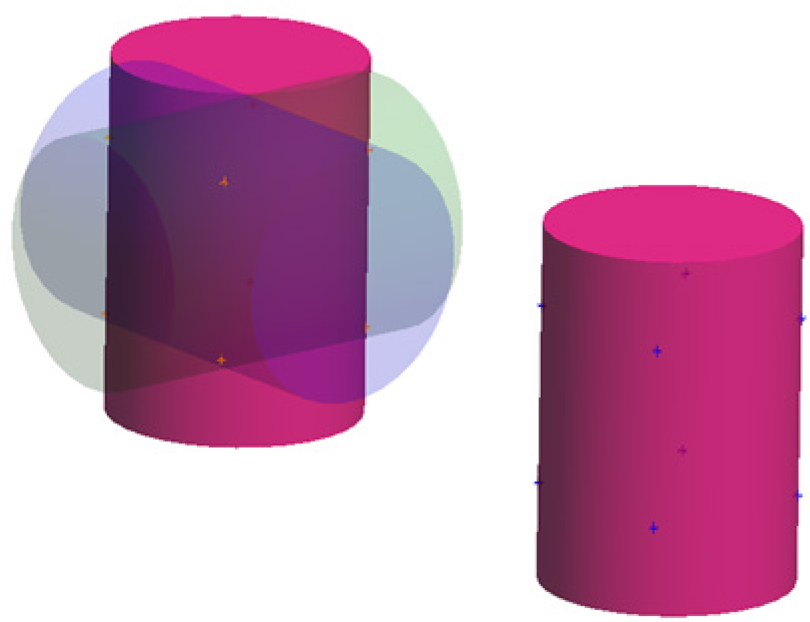


Figure 24-12. Possible cylinder fit orientations with only eight points defining a cylinder location.

To avoid this problem, cover the cylinder more completely with measurements so that it is clear which of those 3 orientations is intended.

One question which may arise is why the nominal feature is not sufficient to lock in the correct orientation. When SpatialAnalyzer evaluates a GD&T check it first performs an initial alignment on the data representing the datums. This initial fit is done without knowledge of the nominal orientation because data is often far out of alignment initially as is common with portable metrology. Once a fit is performed the fit cylinder is compared with the nominal cylinder. Using the above as an example, if the wrong cylinder fit result was chosen (not having the nominal information) then the alignment would attempt to rotate the data incorrectly. Providing more complete measurement coverage avoids this problem.

GD&T User Options

There are several important global evaluation settings to select from found in the GD&T Options within the Users Options Dialog (Figure 24-13):

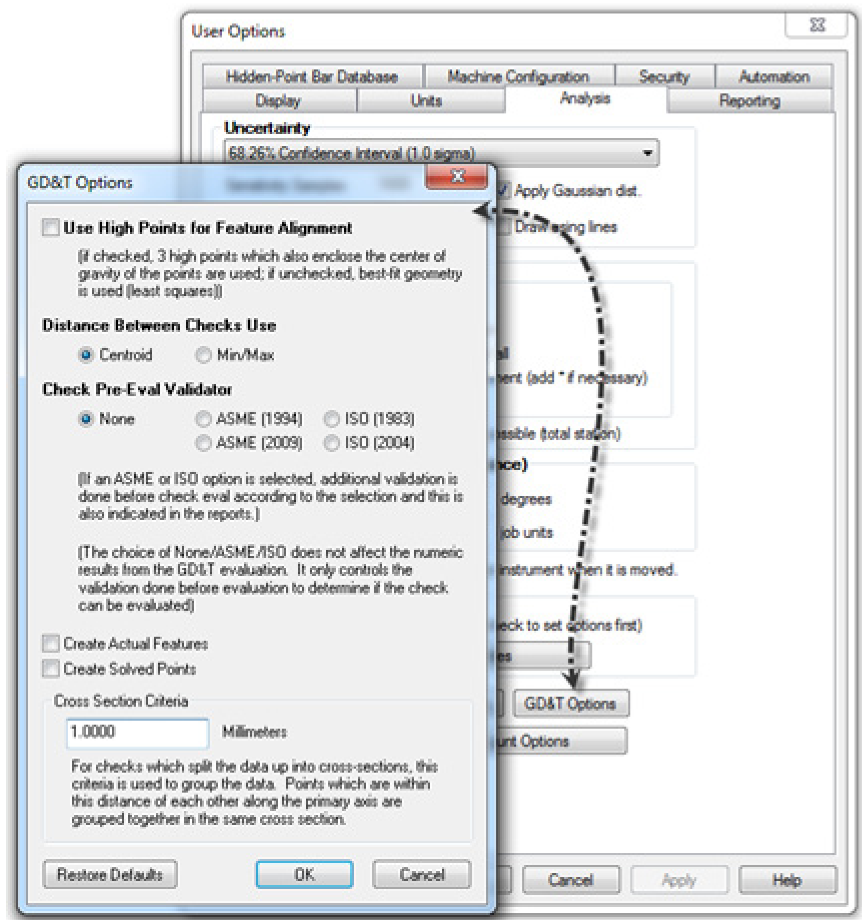


Figure 24-13. GD&T Options dialog within the User Options dialog.

- **High Point Utilization for Feature Alignment.** If checked, three high points which also enclose the center of gravity of the points are used while if unchecked, the best-fit geometry is used (Least Squares).

- **Distance Between Checks Used.** You can choose between using the Centroid or the Min/Max when reporting the distance between checks.
- **Check Pre-Evaluation Validator.** SA follows the ASME Y14.5M-1994 standard when performing GD&T analysis, however, GD&T pre-check validations can be used to ensure full compatibility with the desired standard. The supported standards currently supported are: ASME (1994); ASME (2009); ISO (1983); ISO (2004).

Choosing to use a pre-check evaluation does not affect the numeric results but rather allows the user to validate the check in advance to ensure that the check is in accordance with the specific standard. As a result some checks will fail do to a noncompliance with a particular standard that would pass without pre-checks in place. However, this allows the user to be fully compliant with the desired version of either the ASME or ISO standard. To use a Pre-Evaluation Check simply select the desired standard in the users options prior to evaluating the feature check.

- **Feature Creation Check Boxes.** Both the calculated points and geometry used for the evaluation can be obtained for some feature checks by turning on these check boxes in the Users Options. This can be a very helpful diagnostic tool, however, it creates a lot of new points and geometry. Figure 24-14 shows the resulting geometry created by enabling the Feature Creation check boxes on a Composite True Position check.

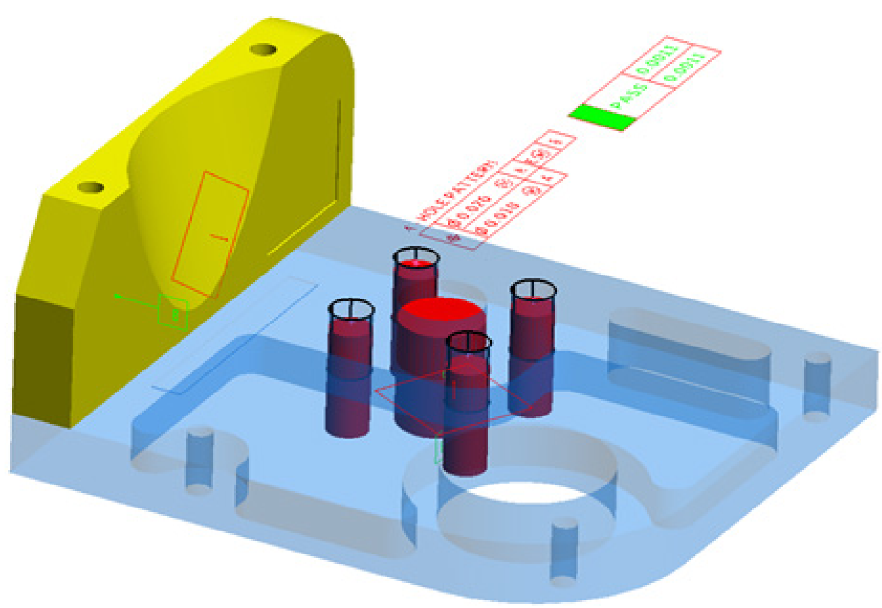


Figure 24-14. Resulting geometry from enabling the Feature Creation check boxes on a Composite True Position check.

- **Cross-Section Criteria.** In order for a group of measurements to be considered a cross section, they must be within the default 1mm of each other in the cross section direction. In other words if you fit a plane to the data none of the points would be greater than .5mm from the plane. In order to take measurements in this format a geometry trigger measurement mode would be easiest. This cross section criteria can be adjusted by selecting GD&T Options under the *Analysis* tab in the ***User Options*** dialog. The number of cross sections used in the analysis is reported in the summary table of the check.

GD&T Workflow

SA provides the option for a GD&T inspection designer to layout exactly how and where to measure a part. It also provides a rehearse function that allows the designer to walk through the script in simulation to check the workflow. Once complete, the resulting script can then graphically walk a measurement technician, with little to no metrology knowledge, through the measurement process. This guided inspection is broken into 3 pieces: Design, Rehearse and Inspect.

Guided Inspection - Design

To build a guided GD&T Inspection an inspection designer should begin with a fully annotated model. This annotated model can either be imported from a CAD package or built within SA. SA objects should be converted to bounded surfaces before beginning the inspection design to allow nominal guild points to be placed on the nominal features. The design process should be conducted as follows:

1. Begin by right-clicking on the annotations in the tree and building feature checks. The majority of the inspection design including view orientations and nominal points are saved with the datums and feature checks.
2. Right-clicking in the tree and select **Inspection > Design**. There are three location where you can find this option. You can right-click on the collection containing the GD&T datums and feature checks to work through the design of all datum and feature checks at once, or you select the datums or feature checks only if you only want to work on designing the portion of the GD&T inspection.
3. Once started the GD&T inspection designer walks through designing the inspection for each GD&T feature in order just as a user would measure them. For each feature the following designations can be made:

- **Save View for Inspection.** If checked the current view will be saved with the datum or feature check and the view will be set to this view when the **Inspection > Rehearse** or **Inspection > Inspect** routines are run.

- **Create Nominal Points.** Nominal point measurement locations can be layed out on the model by graphically clicking on the model in the graphic view. The click selection order will be duplicated in the guided inspection. These point locations are saved as a point group and are saved with the Group Name and Point Name specified in the dialog. If you wish to remove the associated point locations you can use the "Clear NomPts" button at the bottom of the dialog.
- **Measurement Profile Name.** The measurement profile for the instrument being used for the inspection can be set through the inspection designer as well. When the inspector is prompted to measure this feature the measurement profile specified here will be sent to the active instrument.
- **Enable Inspection Automatic Measurement.** When enabled this option will automatically begin the specified measurement mode on the active instrument.
- **Inspection Design - Item # of #.** This set of toggle buttons provides a convenient way of progression the included inspection features. The total number of features included in the item list depend on the selection method, whether you selected to design only the datums, feature checks, or included the entire inspection in the design by selecting **Inspection > Design** for the entire collection. The settings for each inspection feature are saved automatically when you exit the current feature.

Guided Inspection - Rehearse

Once a guided GD&T inspection has been built the designer can then use the Inspection>Rehearse routine to simulate the measurement process. This option allows the designer to walk through the inspection process in simulation using mouse clicks for measurements to simulate the complete GD&T inspection routine built through the Inspection>Design process.

1. To begin the inspection right-click on either the collection, the datums or the feature check categories and select **Inspection > Rehearse** depending on the number of features you wish to include in the rehearsal. When you do so the following dialog will be displayed with the following options (Figure 24-15):

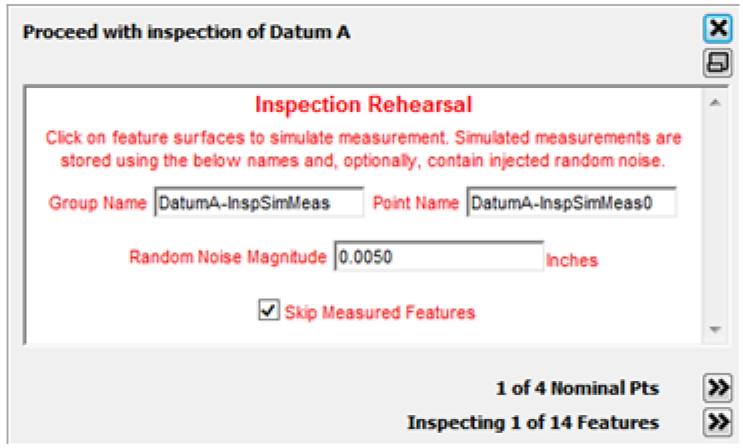


Figure 24-15. Inspection > Rehearse dialog.

- **Group Name/Point Name.** Specifies the group and target names for the click points (simulated measurements) to be saved.
- **Random Noise Magnitude.** This entry allows you to add random noise to the data measurements to simulate a more realistic result. The recorded point location will be randomly generated within the specified radius of the point click point on the surface.
- **Skip Measured Features.** When checked this option allows the feature progression buttons to skip features that already have measurements associated and progress to features that do not. This is helpful when all but a few skipped features have been measured and you want to automatically progress to those particular features.
- **Number of Nominal Points.** These progress buttons provide the number of design measurements expected for the features and allows the user to skip through the measurement points to record measurements for only the points desired. It also allows you to backtrack and add additional measurements at a particular guide location.
- **Inspection Design - Item # of #.** This set of toggle buttons provides a convenient way of progression the included inspection features. The total number of features included in the item list depend on the selection method, whether you selected to design only the datums, feature checks, or included the entire inspection in the design by selection Inspection>Design for the entire collection. The settings for each inspection feature are saved automatically when you exit the current feature.

2. Simply click on the surface where the bulls eyes is displayed to simulate a measurement of that location. The bulls eyes will automatically increment through the saved measurement locations. When all measurements are taken for a feature the

dialog will automatically increment to the next feature in the list just as the inspection was designed.

Guided Inspection - Inspect

Once a guided GD&T inspection has been built through the Inspection>Design process, a measurement technician can then right-click on the collection and select Inspection>Inspect to conduct the inspection of a part. The guided inspection will walk the operator through the inspection laid out by the designer and allow them to measure the part as specified. As measurements are taken they will be associated with the appropriate feature is the tree and an analysis will be carried out automatically. For an operator it can be as easy as measuring the point on the part designated by the bulls-eyes laid out on the nominal CAD part. The process flow is as follows:

1. To begin the inspection right-click on either the collection, the datums or the feature check categories and select Inspection> Inspect depending on the number of features you wish to include in the inspection. When you do so the inspection will begin and the following will occur: the view will be set to the saved view for the feature; the measurement profile will be set on the instrument specified; or the first inspection point will be displayed on the nominal part.
2. A dialog box (Figure 24-16) will be displayed with the following options and active trapping will begin for that feature:

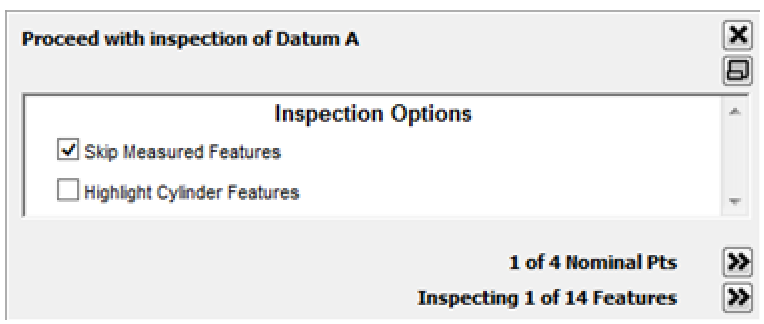


Figure 24-16. Inspection > Inspect dialog.

- **Skip Measured Features.** When checked, this option allows the feature progression buttons to skip features that already have measurements associated and progress to features that do not. This is helpful when all but a few skipped features have been measured and you want to automatically progress to those particular features.
- **Highlight Cylinder Features.** This check box triggers an additional cylinder to be graphically drawn on the CAD model to clearly depict which cylindrical surface to measure in addition to highlighting the specific CAD faces.
- **# of # Nominal Points.** These progress buttons are displayed

only when saved measurement locations are included in the inspection. They provide the number of design measurements expected for the features and allows the user to skip through the measurement points to record measurements for only the points desired. It also allows you to backtrack and add additional measurements at a particular guide location.

- **Inspecting # of # Features.** This set of toggle buttons provides a convenient way of progression the included inspection features. The total number of features included in the item list depend on the selection method, whether you selected to design only the datums, feature checks, or included the entire inspection in the design by selection inspection>design for the entire collection. The settings for each inspection feature are saved automatically when you exit the current feature.

3. Simply measure the actual part where indicated by the bulls eye is displayed on the nominal part within SA. The bulls eyes will automatically increment through the saved measurement locations. When no nominal measurement locations are save with the feature it is assumed that the measurement technician will measure the appropriate number of on the designated surfaces.
4. When all measurements are taken for a feature the dialog will automatically increment to the next feature in the list just as the inspection was designed. If nominal measurement locations were not included in the inspection design then the feature should be incremented manually when sufficient measurements are take.

Direct Data Association and Trapping

In addition to classical Design and Inspection routines intended to separate the GD&T inspection design phase from the inspection or operation phase, SA provides a number of tools available to more advanced GD&T users performing the inspection of a part.

Trapping Measurements from a Live Instrument

Measurements can be captured or “Trapped” directly from a live instrument while measuring without either building or utilizing the guided inspection process. For an advanced user, directly measuring points on a part and having those measurements automatically associated to a particular feature can be the fast way to conduct an analysis. Trapping can be started for any datum or feature check in either of two ways:

Trapping through the Instrument

1. Connect to your instrument.

2. Right-click on the particular Datum or Feature Check you wish to trap.

3. When finished measuring, right-click on the feature a second time and select **Stop Trapping Measurements**.

Trapping through the Inspection Tab in the Toolkit

1. Connect to your instrument.

2. Double-click on the particular Datum or Feature Check you wish to trap measurements to in the *Inspection* tab.

3. When finished measuring, press the stop button (INSERT ICON) or press the progress button (INSERT ICON) to move to the next or previous item in the *Inspection* tab list.

Either of these operations will begin trapping to the feature. If you have more than one instrument you will be prompted to choose which instrument you wish to trap measurements from. While trapping is active the feature will become highlighted in both the Inspection tab and in the tree as well as being displayed with the point count in the Heads Up Display and will remain highlighted until trapping is stopped.

Directly Associating Data with a Feature

As soon as Annotations are available, Feature Checks and Datums can be built and data can be associated with those features. A user could potentially measure all the features on the part independently of the GD&T analysis process and then come back after the fact and associate the appropriate measurements with the corresponding Datums and Feature Checks. To associate data with a datum or a feature check just right-click on the feature in the tree and select Associate Points or Associate Clouds. This will clear the list of currently associated points and or point clouds associated with the feature, if any are currently associated, and associate the selected points or clouds with the feature. Once features have associated data they can be evaluated all at once by right-clicking on the feature checks and selecting Evaluate all Checks.

Feature Inspection Auto Filter

If you have measured your part independently of any type of Guided Inspection or Measurement Trapping and you have pre-aligned to a CAD model it is quite simple to perform a GD&T analysis using the annotations contained in the model. To do so simply do the following:

1. Navigate to **GD&T > Feature Inspection Auto Filter**. This filter goes through all the measurements in the job within a specified distance from the part, whether they are individual points, point groups, or even point clouds, filters them into individual groups associated with the CAD surface, and assigns them to

the appropriate feature check or datum in one operation.

2. Once this is done then navigate to **GD&T > Evaluate all Feature Checks** to complete the GD&T analysis.

This is a particularly useful approach to GD&T analysis when using a scanner. If you initially locate a scanner and then fully scan the part you are evaluating you can then filter the scan data to the CAD Faces associate the individual scans to the appropriate Features and in two clicks of a mouse button go from a single unfiltered cloud to a completed GD&T analysis ready for reporting.

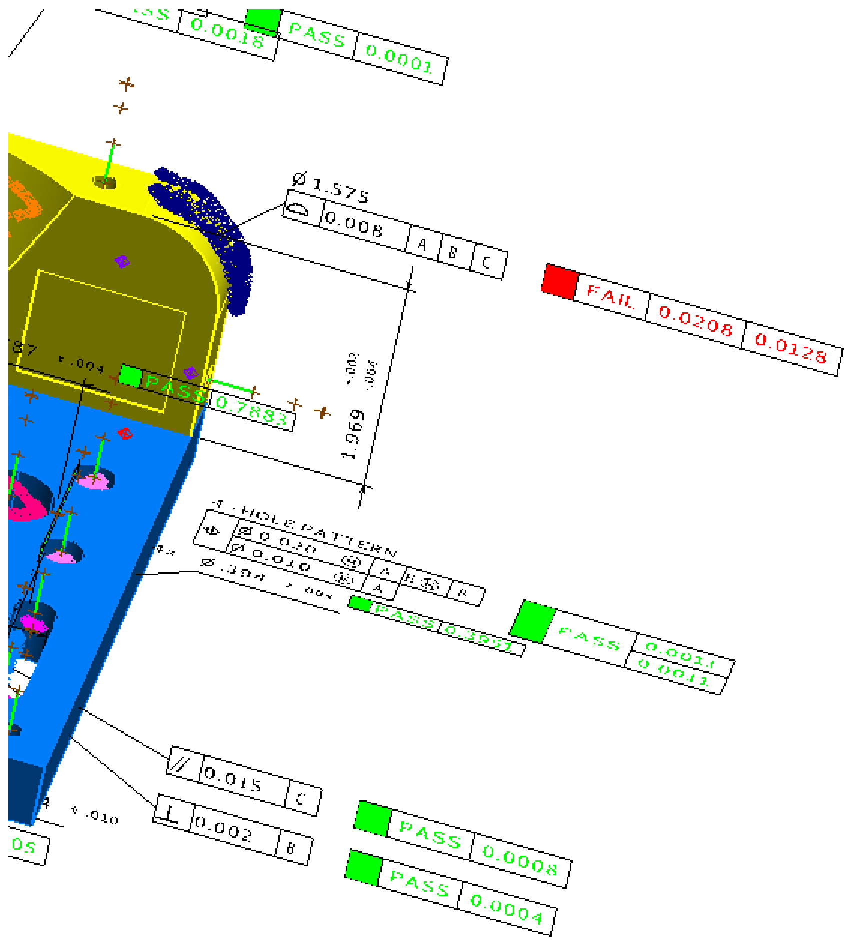


Figure 24-17. Evaluated feature checks.

GD&T Reporting

Once feature checks have been evaluated the number of points used and the PASS/FAIL status is displayed in the tree. The report bar will provide the complete results of the analysis. Datum point associations, locations, and offsets can also be viewed as you select individual datums. Feature checks display only a single line pass or fail results just as displayed in the graphic view by default but more detailed information can be obtained by turning on additional reporting features by right-clicking on the feature of interest and selecting *Reporting Options* (Figure 24-18).

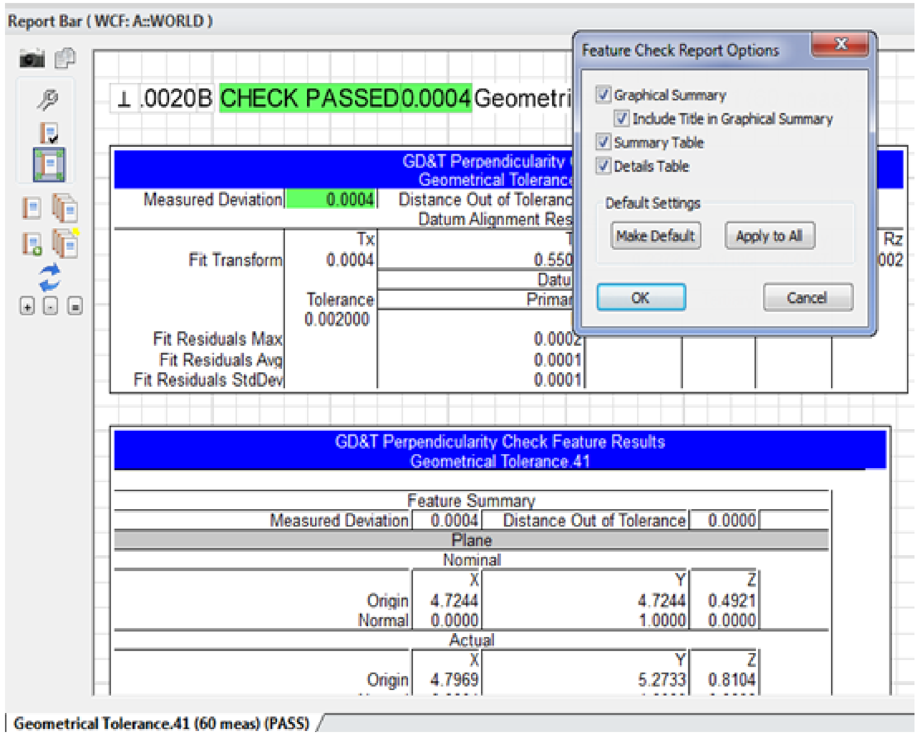


Figure 24-18. Full Feature Check reporting options in the Report Bar.

These feature checks can easily be added to a report through a drag and drop process or a composite report can be generated for all the feature checks at once by right-clicking on the Feature Check category and selecting **Composite Quick Report > Generate** this will automatically generate a report containing a snapshot of the part in the saved view used for the measurement and the resulting success or failure of the check. By right-clicking on the check you can select to add either a summary table, showing more detail on the overall residuals, or a Details view displaying the point coordinates and offsets for each point used in the analysis.

Measurement Plans

CHAPTER 25

SpatialAnalyzer contains a powerful scripting environment that gives you the ability to automate tasks ranging from measurement, to analysis, and reporting. Scripts can introduce efficiencies into your metrology processes that cannot be overstated.

About Measurement Plans

Measurement Plans (MPs) is the name given to the scripting capability built into SpatialAnalyzer. MPs are similar to macros in that they can repeat a series of steps for you—but the similarities end there. MPs actually have significantly more capability—scripts make calculations, branch to different behavior based on decisions, gather user input, read and write to files, and execute custom code. In that sense, scripts share more traits with a programming language than a macro.

However, true programming languages have a complex syntax, often take many years to master, and have a high learning curve. MPs are designed to share as much capability as possible with programming languages while still dramatically lowering the learning curve compared to learning a programming language.

Scripts can vary from simple utilities constructed in just a few minutes to comprehensive inspection plans that provide go/no-go automation and drive other machinery such as robots or CNC machines.

MPs are available for editing and executing on SA Ultimate or higher. They cannot be created, edited, nor executed on SA Professional.

Running a Measurement Plan

Scripts can be created in two formats:

- **Embedded.** An embedded MP is stored in the SA job file and listed in the tree (Figure 25-1).
- **External.** An external MP is stored as a separate .MP file on the file system. It is completely independent of any job file.

Both embedded and external scripts have essentially the same functionality and behavior (although there are some advanced reasons why you might choose one over the other).

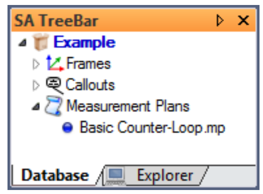


Figure 25-1. An embedded MP.

If you would like some example scripts to view or run, they are included with the SA install in the `Samples/Scripts` folder.

To Run an External MP:

1. From the SA menu, select **Scripts > Run Measurement Plan**.
2. Navigate to the `.mp` file and select it.
3. The script will load into the *MP Bar* (Figure 25-2). If the *Auto-Run MPs* option is enabled (see "To Enable/Disable Auto-Running of Scripts:" on page 689), the script will begin executing immediately. If this option is disabled, you must click the run button in the *MP Bar* to run the script.

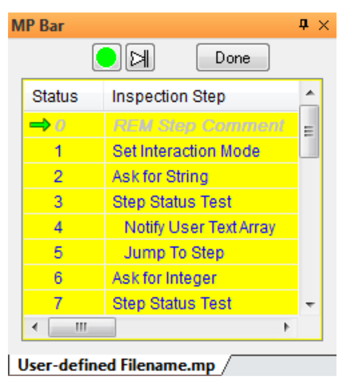


Figure 25-2. A script ready to run in the *MP Bar*.

To Run an Embedded MP:

1. In the tree, locate the script in the `Measurement Plans` category (see Figure 25-1) and double-click it, or right-click it and select **Run** from the context menu.
2. The script will load into the *MP Bar* (Figure 25-2). If the *Auto-Run MPs* option is enabled (see "To Enable/Disable Auto-Run-

ning of Scripts:" on page 689), the script will begin executing immediately. If this option is disabled, you must click the run button in the *MP Bar* to run the script.

To Run a Script from the MP Editor:

1. Load the script into the editor.
2. Do one of the following:

- Click the run button at the top of the editor,
- From the editor's menu, select **File > Run**, or
- Press **Ctrl + R**.

3. The script will load into the *MP Bar* (Figure 25-2). If the *Auto-Run MPs* option is enabled (see "To Enable/Disable Auto-Running of Scripts:" below), the script will begin executing immediately. If this option is disabled, you must click the run button in the *MP Bar* to run the script.

To Enable/Disable Auto-Running of Scripts:

1. Click the User Options icon or press **Ctrl+U** (default keyboard shortcut.
2. Navigate to the *Automation* tab. Toggle the *Auto-run MPs (applies for double-click in tree and Scripts>>Run Measurement Plan)* option.

Quick MPs

Warning: Making changes to Quick MP slots modifies your User Interface Profile. Changes must be saved to your UI profile in order to retain the settings the next time SA is run.

You can assign Measurement Plan scripts to the menus (Figure 25-3), so that any script can be executed by selecting it specifically from the SA menu (or from a toolbar, if you've customized the SA user interface). You can assign up to 30 scripts to Quick MP slots for easy execution.

This immediate execution of Quick MPs is excellent for listing a library of commonly-used scripts or for testing a script during development.

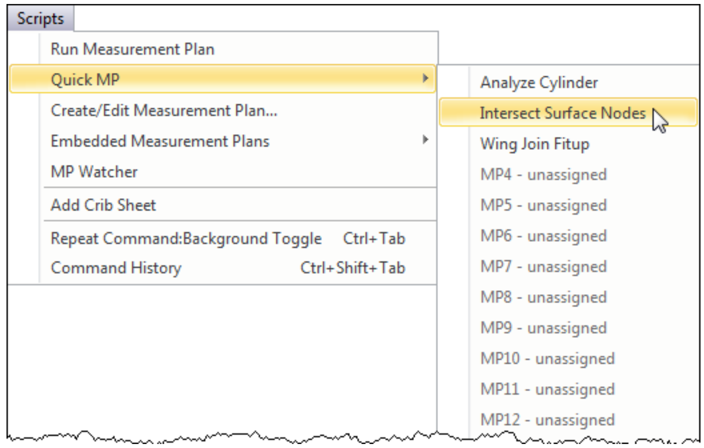


Figure 25-3. Scripts have been assign to Quick MP slots 1, 2, and 3.

To Assign a Script to a Quick MP Slot:

1. In the menus, select **Scripts > Quick MP > Configure Quick MPs**.
2. In the ***QuickMP Assignments*** dialog, click the Add button.
3. In the ***Quick MP Assignment*** window, specify the parameters for the script:

- **ID.** Which of the 30 slots (from one through 30) should be used for the script.
- **Display Name.** The name for the script that you'd like to use in menus or toolbars.
- **Location.** Select *Embedded MP* if the script is embedded, or *External MP File* if the script is an external file.
- **Collection/Name/File.** If the script is embedded, specify the collection containing the script and the name of the script itself as it appears in the tree. If the script is external, the path to the script file to run.

Tip: In most cases, you should use external scripts with Quick MPs.

4. Click OK to accept the settings, then OK again to close the ***QuickMP Assignments*** window.
5. Save the changes to the UI profile as specified in "Custom UI Profiles" on page 144.

If you now select **Scripts > Quick MP**, you should see your script listed.

To Edit a Script's Quick MP Slot:

1. In the menus, select **Scripts > Quick MP > Configure Quick MPs**.
2. In the ***QuickMP Assignments*** dialog, select the Quick MP to modify and click the Edit button.
3. Specify the desired settings as indicated in "To Assign a Script to a Quick MP Slot:" on page 690.
4. Click OK to accept the settings, then OK again to close the ***QuickMP Assignments*** window.

5. Save the changes to the UI profile as specified in "Custom UI Profiles" on page 144.

To Remove a Script from a Quick MP Slot:

1. In the menus, select **Scripts > Quick MP > Configure Quick MPs**.
2. Select the Quick MP script to delete, then click the Delete button.
3. Save the changes to the UI profile as specified "Custom UI Profiles" on page 144.

To Run a Script from a Quick MP Slot:

1. In the menus, select **Scripts > Quick MP**.
2. Choose the script you'd like to execute.
3. The script will execute immediately, even if the *Auto-Run MPs* option is disabled (see "To Run an External MP:" on page 688)

To Assign a Script to Any Menu or Toolbar:

1. Assign the script as a Quick MP as described in "To Assign a Script to a Quick MP Slot:" on page 690.
2. Assign the Quick MP slot's menu item to a toolbar or menu as described in "Customizing Toolbars & Menus" on page 145.

Working With Measurement Plans

Measurement Plan Scripts are not text-based. Instead, SA uses a graphical editor for writing scripts, which is not as basic as a macro recorder but also not as complicated as a text-based scripting environment.

Tip: In general, we recommend using external script files unless your script is fairly short and simple.

As described in "Running a Measurement Plan" on page 688, you can create your script as an embedded script or as an external script. There are advantages and disadvantages to each.

Embedded Measurement Plans:

- Are stored as part of the SA file, therefore are easier to find and bundle with a script.
- Are easier to execute and edit, since you can access them directly from the tree.
- May not work if you need to create a new SA file or load an SA file as part of the script, since such actions will clear the embedded scripts from the tree.
- Make version maintenance, reuse, and distribution of scripts more difficult, since you need to export the script to a separate file in order to use it with other SA files.

Warning: You cannot use embedded scripts that have more than one script (such as a script with a subroutine) with an "Open SA File" or "New SA File" command. An external script is necessary in such a scenario.

External Measurement Plans:

- Are stored as a separate `.mp` file, therefore you must navigate to the file to execute it.
- Are able to work with any scripting commands—creating a new SA file or loading an existing file does not create problems.
- Are much better suited to script maintenance, reuse, versioning, and distribution.

You can start writing an embedded script or start writing a script in an external file. Likewise, you can export an embedded script to an external file or import an external script file as an embedded script.

To Start a New Embedded Measurement Plan Script:

1. In the menus, select **Scripts > Embedded Measurement Plans > Create New**.
2. Specify a name for the new script. This is how the embedded script will be named in the tree.
3. The MP Editor will appear, ready for input.

To Start a New Measurement Plan Script in an External File:

1. In the menus, select **Scripts > Create/Edit Measurement Plan**.
2. The MP Editor will appear, ready for input.

To Import an External Script File as an Embedded Script:

1. Select **Scripts > Embedded Measurement Plans > Embed existing .MP file**.
2. Navigate to the script file to embed and select it.

To Export an Embedded Script to an External File:

1. Right-click the script in the tree and select **Export** from the context menu.
2. Specify a file name for the new external file, then click Save.

To Edit an Existing External Script File:

1. From the menu, select **Scripts > Create/Edit Measurement Plan...**
2. The ***MP Editor*** will appear.
3. Do one of the following:
   - In the editor's menu, select **File > Load**, or
   - Press `Ctrl + L`.
4. Select the desired file.

To Edit an Existing Embedded Script:

1. Right-click the script in the SA tree and select **Edit** from the context menu.

To Start a New Script in an Existing Editor Window:

- From the editor's menu, select **File > New**, or
- Press **`Ctrl + N`**.

To Save Your Script:

- From the editor's menu, select **File > Save** or **File > Save As...**, as applicable, or
- Press **`Ctrl + S`** (Save) or **`Ctrl + Shift + S`** (Save As...).

Protecting Scripts

Once you have finalized a script, you will often want to protect it from accidental (or intentional) modification. You can password-protect a script so that it cannot be modified. Once protected, the script will no longer load into the editor. However, protecting a script does not prevent it from being executed, or from viewing the commands used to create the script.

To Password-Protect a Script:

1. Load the script into the editor.
2. From the editor's menu, select **Protect > Set Password**.
3. In the ***Set Password*** dialog, enter the desired password twice, then press OK.
4. Save the script.

To Open A Protected Script:

1. Load the script into the editor as usual.
2. When prompted, enter the password for the script.

To Remove Protection from a Script:

1. Load the script into the editor. You will need the password for this operation.
2. From the menu, select **Protect > Remove Password**.
3. Confirm the operation.

Generating SDK Code

The MP Editor has the capability of automatically generating the SDK code needed to repeat a script's functionality in both the C++ and Visual Basic languages. This code can then be copied, pasted into an SDK project, and (with limited changes) be executed directly in compiled code. For more on the SDK, refer to the next chapter (SA SDK).

To Generate SDK Code for a Script:

1. Load the desired script into the editor.

2. From the editor's menu, select **SDK > View SDK Code**.

3. In the ***SDK Code Viewer*** dialog, select the desired language, *Visual C++* or *Visual Basic*.

4. The code will automatically be copied to the Windows clipboard and can be pasted directly into any text editor or Integrated Development Environment.

The MP Editor

If you will be writing Measurement Plans, you'll be doing it in the MP Editor (Figure 25-4). Some companies even have employees whose primary (or only) job responsibility is to write Measurement Plans—these people may spend most of their working hours in this editor!

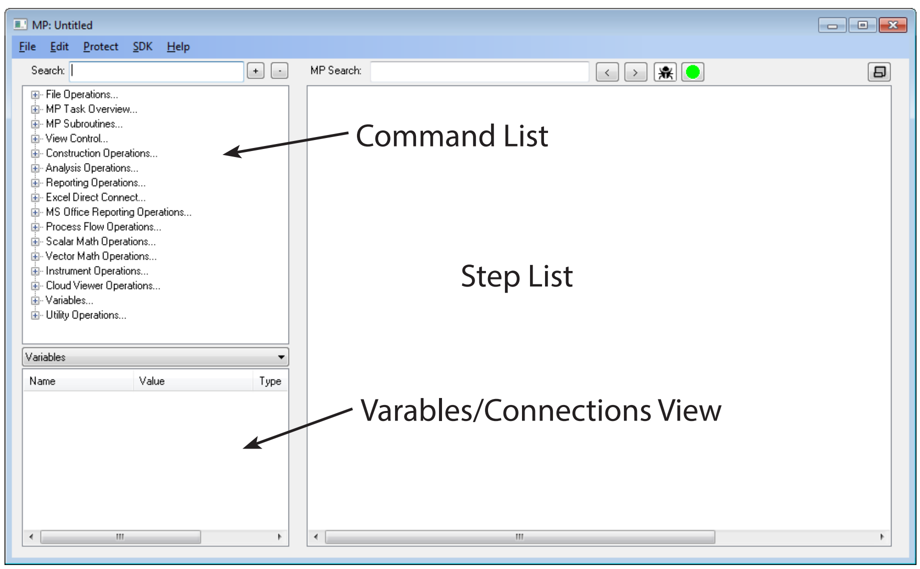


Figure 25-4. The MP Editor.

For those used to the legacy MP Editor in SA 2014.04.15 and earlier, this new editor has an enormous number of improvements. The most notable are:

- Command list and script are now integrated into one window. More screen real estate is devoted to your script.
- Multiple commands can be displayed at once for easy reference.
- Every command is a collapsable tree item, allowing easy expand/collapse behavior. This is particularly useful for "keeping track" of commands that you haven't finished editing.
- Commands can be rearranged through easy drag and drop.
- Commands can be locked to prevent changes once they're finalized.
- Changing entry methods (such as switching from "Enter Meth-

od" to "Reference" is dramatically faster than before with quick-select buttons.

- Most references can be selected directly in the step editor (instead of the large table that was displayed in older versions), causing less disorientation.
- Long comments can be seen completely, whereas they would be obscured in the old editor.
- A variable list shows any defined variables and their types for reference.
- A debug mode allows you to get "MP Watcher" functionality directly in the editor. You can see argument values directly in the editor as the script executes.
- Current variable values are displayed while in debug mode.
- A Connections viewer allows you to easily view commands that reference or are referenced by the current command, and jump to those steps quickly.
- Lots of keyboard shortcuts for loading, saving, running, expanding and collapsing, selecting, etc. are available—making for a fast workflow.
- Automatic expansion and collapse of applicable commands (an option) makes for quick selection of references.
- An "expert mode" suppresses warning messages about deletion of commands.
- Configurable alternating row colors for less eyestrain.
- Automatic refresh of subroutine arguments.
- Faster setting of breakpoints/comments.
- More obvious indentation.
- For new commands, if a step reference becomes invalid, it will switch to -1, forcing a command to fail instead of jumping to the start of the script. This makes it easier to find incomplete arguments in a command.

Overview

The MP Editor provides a graphical interface for writing scripts. It carries a few traits from a traditional IDE (Integrated Development Environment) that experienced programmers might recognize. But most importantly, it is not a text editor. Scripts are written in a point-and-click, drag-and-drop fashion instead of writing text-based commands. This makes the learning curve more acceptable for most users.

There are three main sections in the editor (Figure 25-4):

- **Command List.** This is a hierarchical tree showing every command that could possibly be added to a script, organized by type.

- **Variables/Connections View.** This view shows either a list of the variables, variable types, and (if debugging) variable values defined in the current script, or shows the connections for the currently selected command. (For now, think of connections as the commands that relate to the selected command in some way).
- **Step List.** The majority of the window is devoted to the *Step List*, which displays the individual steps of the script itself and allows for editing of the script's arguments.

A Measurement Plan is a procedural script that executes one line at a time. Normally, the script executes in a linear fashion, proceeding from one command to the next. However, there are commands that cause execution to jump to another location in the script. There are also commands that branch to different parts of the script (or to different scripts altogether) based on some value.

Everything in Measurement Plan scripting is zero-based. That is, all counting begins with zero, whereas in the regular world we typically start counting from one. The first step in a script is step zero, the first argument in a command is argument zero, the first item in a list (array) is item zero, and so on. If you have any programming experience, you will be used to this convention.

About Commands

When a command is first added to a script, it will be in expanded form so that it can easily be modified (Figure 25-5).

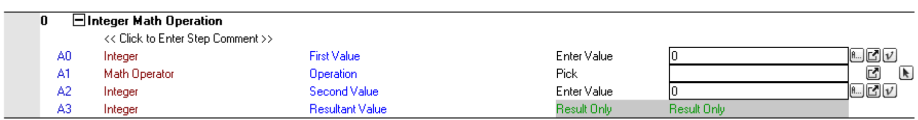


Figure 25-5. A newly-added command, in expanded form.

Commands can also be collapsed to one line at any time to save space or simplify your view (Figure 25-6).

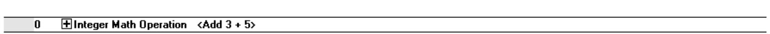


Figure 25-6. A collapsed command.

To understand the command's parameters, let's look at this command in more detail. Each item in the command's detail view is numbered in Figure 25-7 and discussed below.

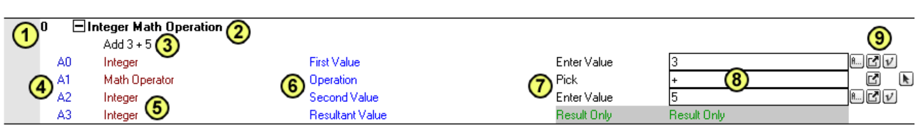


Figure 25-7. A command's parameters.

1. Step Number

Every command in a script has a specific step number, used to order and identify steps in the script. Step numbers start from zero, so the first step in a script is step zero. You can rearrange single steps or entire groups of steps.

2. Command Name

The name of the command is listed at the top of the entry. This is a read-only field that allows you to view the name of the command being examined. The name of the command in Figure 25-7 is **`Integer Math Operation`**.

3. Comment

Each step can have a user-defined comment (**`Add 3+5`** in this case). This comment is simply a single line into which you can type anything you'd like. Comments are typically used to provide "human-readable" text to accompany the commands, so that someone reading a script (often times, that's you!) can understand the intent of the command.

Comments can only be one line in length. When you collapse a command, the comment appears to the right of the command name. When expanded, the comment appears below the command name in a field that can be edited.

A few tips regarding comments:

- **Keep them brief.** Long comments clutter up the screen, take time to enter, and take longer to read.
- **Comment Only When Necessary.** New script writers have a tendency to add a comment to every line. Don't. Only add comments if you feel like they're needed.
- **Don't be redundant.** If the command itself is clear, there's no need to add a comment. For instance, in Figure 25-7, it's clear that we're adding 3+5, so the comment is redundant.
- **The above three rules are not set in stone.** If you feel like a comment is needed, add one. For example, it may be clear what a command is doing when you're looking at its details, but when it's collapsed, you can't see what it's doing. So it may be acceptable to add a comment in that case. Use your own discretion.

4. Arguments

Every command added to a script has zero or more *arguments*. Arguments are the vehicle by which you can communicate with a command, and by which a command can communicate with you as the script's author. There are two types of arguments:

- **Input Arguments.** These are values that can be supplied to a command as input. They help the command determine what it is supposed to do. Most input arguments are required, but some are optional.

- **Return Arguments.** Return arguments are used to return information or results back from a command after it executes.

In Figure 25-7, the command has three input arguments: zero through two (A0 through A2). It has one return argument: A3 (notice the grayed-out "Result only" fields).

In this example, we want to perform a math operation on two integers (whole numbers). We obviously need to provide both of those numbers, in addition to the math operation that we want to perform. Those are the input arguments.

Once the command completes, we need a place to store the result of the operation so that we can refer to it later. The result of the addition in this case is stored in A3.

5. Data Type

Every argument has a data type (this will not be new for those with programming experience). A data type indicates the type of information that an argument is expecting. You must feed an argument data of a compatible data type—otherwise, SA will prevent the operation or the script will fail at runtime. For instance, you cannot provide a letter to an argument expecting a number. After all, what is the numeric value of that letter?

There are just four primary data types:

- **Integer.** Whole numbers, such as 10, 10000, 0, and -502. These values are represented exactly, but there is a limit to the range of this number. In general, you need not worry about this limit for scripting applications.
- **Double.** Decimal numbers, such as 3.1415926535, 2.781, 0.0, or -1.125. These numbers are in general not represented exactly, so expect numerical errors to creep into your calculations. These errors are typically well beyond the precision of portable metrology work, so they're not usually a worry, but you should take care when trying to compare one double value to another. You may expect them to be the same but they may actually be very slightly different. Double values also have limits on the range of values they can represent, but in general you need not worry about this limit for scripting applications.
- **Boolean.** A value that represents one of two possible values, such as true/false, on/off, left/right, or yes/no.
- **String.** A sequence of one or more alphanumeric characters, such as a sentence or prompt.

Note: Some data types are simply more descriptive names for other data types. These are called *Proxy data types*. **`Collection Name`** is an example of a proxy data type.

There are many more data types in scripts in addition to these four, but virtually all other data types in scripts are derivations of these four main types. For example, there is a **`Collection Name`** data type, but it is really a string. This means that any argument that expects a **`Collection Name`** can also be fed a string, and vice versa.

6. Description

Every argument has a description field. This is more of a human-readable description of what the argument is for.

7. Entry Method

A dropdown which appears when clicking in this column allows you to specify *how* you want to supply the information for the argument. For example, do you want to type it in, or feed it from some other command?

There are many entry methods. The most common are covered here:

- **Enter Value.** The argument's value is typed directly into the command—known as *hard coding* the value.
- **Reference.** The value is fed into the argument from some other command. For example, the result of a math operation might be fed into another command for further processing.
- **Variable.** The value for the argument is retrieved from a variable by specifying the variable's name. The variable must be of a compatible data type for the given argument.
- **Pick.** The value is selected from a limited dropdown list of values.
- **Dialog.** The value is specified graphically in a dialog.

8. Argument Value

The argument's value is specified in this field.

- Hard coded values appear as typed.
- References appear as a combination of step number and argument number. For instance, **`Ref {S3 A4}`** indicates that you are feeding the value stored in step 3, argument 4 into the current argument.
- Variables are specified by entering the variable name.

9. Entry Method Quick-Select

A series of buttons for the most common entry methods are listed to the right of the argument value. By clicking these buttons, you can immediately change the entry method for the argument and directly enter the value. Users familiar with SA's legacy script editor will find this new method of argument entry dramatically faster than the old method, while also requiring less clicking.

> **Note:** If you cannot find a button for the appropriate entry method, look for it in the entry method dropdown (7).

Only buttons applicable to a given argument are displayed, although often times there are more entry methods than buttons. (In other words, only the most common entry methods have buttons).

Entering Commands

Before you add a command to a script, you must find it in the *Command List* first. When you are just starting out, you will probably find commands by browsing the list. However, once you have more experience under your belt, you will begin memorizing key phrases of common commands and find them by simply typing a phrase to search for in the *Search* field above the *Command List*.

To Add a Command to a Script:

1. Select the step after which you'd like the new command to be added. (If no step is selected, the new command will be added to the end of the script).
2. Find the desired command in the command list, or search for it by typing into the *Search* field. Use the **+** and **-** buttons to expand or collapse categories in the command list.
3. Double-click the desired command, or select it and press **`Enter`**.

To Delete a Command from a Script:

1. Select one or more commands by clicking on their command name. (Standard Windows contiguous and non-contiguous selection is supported).
2. Press the **`Delete`** key.

Editing Commands

Once you have commands in your script, various editing operations can be performed.

To Select a Single Command:

- Click in the row containing the command's name, or
- Use the up/down arrow keys on the keyboard while the step list has focus.

To Select Several Contiguous Commands:

1. Click the row containing the first command's name.
2. Hold down **`Shift`** and click the row containing the last command's name.

To Select Several Non-Contiguous Commands:

1. Click the row containing the first command's name.

2. Hold down **Ctrl** and click additional rows containing subsequent command's names.

All clicked commands will now be selected.

To Select All Commands:

- Use contiguous selection above, or
- Choose **Edit > Select All** from the editor's menu, or
- Press **Ctrl + A**.

To Clear Your Selection:

- Click away from any commands in the step list, or
- Choose **Edit > Deselect All** from the editor's menu, or
- Press **Ctrl + Shift + A**.

To Rearrange Commands:

- To move a single command, drag it by clicking on its command name and drop it anywhere on the *command you'd like the command to follow*.
- To drop a command BEFORE another command, hold down the **Shift** key while dropping.
- To rearrange multiple steps, select them (standard Windows contiguous and non-contiguous selection is supported), then drag and drop them onto the step after which you'd like the group of steps to appear. Hold down **Shift** when dropping to place the group of steps before the command.

To Expand or Collapse Commands:

1. Select one or more commands (standard Windows contiguous and non-contiguous selection is supported).
2. Do one of the following:

- Click the ⊞ or ⊟ button,
- Double-click the row containing the command name, or
- Press the right arrow (expand) or left arrow (collapse) keys on the keyboard.

To Copy Commands:

Note: Cut is not currently supported.

1. Select one or more commands.
2. Do one of the following:

- From the editor's menu, select **Edit > Copy**, or
- Press **Ctrl + C**.

To Paste Commands:

1. First copy one or more commands as described above.
2. Select the command after which you'd like to paste.

3. Do one of the following:

- From the editor's menu, select **Edit > Paste**, or
- Press **Ctrl + V**.

Resizing Columns

Although it may not be obvious, the MP Editor has a number of resizable columns. You can adjust these to better fit the column's contents, or if you have made the editor window large.

To Resize Columns:

1. Expand any command so that its arguments are visible.
2. Place the cursor just to the left of the Description, Entry Method, Value, or Entry Method Quick-Select Button columns. The cursor will change to indicate that you can drag left and right.
3. Drag the column to the desired width.

Hard Coding Argument Values

Hard coding involves manually entering values into an argument. This is done when the value for an argument is known at development time and is not expected to change.

To Hard Code an Argument Value:

1. Ensure the entry method dropdown for the desired argument is set to *Enter Value*, or click the [A...] button.
2. Enter the value into the argument's value field, as specified below (the most common cases are described):

- **Simple Data types.** Doubles, strings, integers, and other simple data types can be entered by typing the values directly into the field.
- **Boolean types.** For arguments of type **Boolean**, you can either type **TRUE** or **FALSE** (case insensitive) or, as a shortcut, type a nonzero value to represent **TRUE** and **0** to represent false. (It will automatically convert this value to **TRUE** or **FALSE** when you press **Enter**.
- **Collection Object Names.** If you're entering the name of an object, type the collection name, followed by two colons, followed by the object name. For example, **Measured::AxisLine**. If you leave off the collection name and simply type **AxisLine**, the command will try to find the object named **AxisLine** in whichever collection is active at the time the command is executed.
- **Point Names.** To specify a point name, type the collection,

group name, and point name separated by a pair of colons. For instance, **`Measured::Tie-Ins::1`**. You can leave off the collection name to indicate that you want to find the group and target in the active collection at the time the command is executed. For example, **`Tie-Ins::1`** will look for point **`1`** in a **`Tie-Ins`** group for the active collection.

- **Reference Lists.** Some commands, such as **`Concatenate Strings`**, expect reference lists, which are simply lists of items. To enter these, click the entry field, which will open the ***List Editor*** (Figure 25-8). Use this to add, rearrange, or remove items from the list.

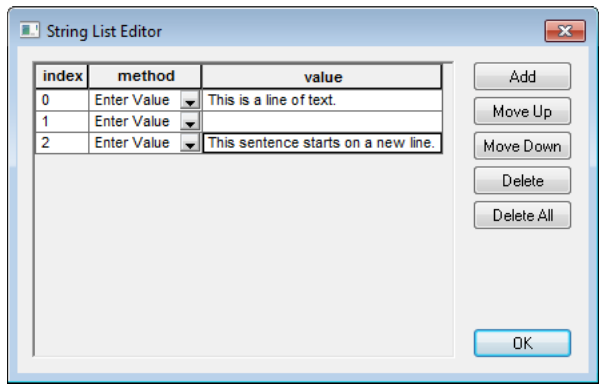


Figure 25-8. The ***String List Editor***.

Referencing Arguments

The true power of scripting comes not from a script that repeats a sequence of steps exactly as before, but in making decisions and behaving differently based on parameters that may be different from the last time the script was executed.

> **Tip:** If multiple arguments from different commands are going to refer to the same value, it is generally best for all of them to reference the same argument. Doing so ensures that future changes only require changing one argument.

The most common way of supporting this "non-hard coded" behavior in scripts is through the use of references. A reference is essentially a link or a connection to another value in your script. It is a way to feed a value from some other command in your script into the current argument.

For example, one step may multiply two values together. You can then feed the result of that calculation into an input argument in another command to further operate on it.

To Reference an Argument:

> **Note:** An argument cannot reference itself. Such circular

> **Warning:** Do not attempt to reference an argument that will not contain a value at the time the command is executed. This results in undefined (and generally incorrect) behavior.

1. Ensure the entry method dropdown for the desired argument is set to *Reference*, or click the button.

The editor will highlight yellow all arguments that are of a compatible type for the given argument. If an argument is already being referenced, it will be highlighted blue. If the *Auto Expand/Collapse Steps* option is enabled in the MP Editor options, any commands containing compatible arguments will be automatically expanded to show them. These commands will collapse again when the reference is selected.

2. Click on the highlighted argument to reference, or **Esc** to cancel.

Using Variables

Most if not all programming languages support the concept of variables. Variables allow you to define a name that is associated with a value. This allows you to refer to that value by name. The advantage of this is that while a script is running, you may change or overwrite the value of that variable, but anywhere you refer to it by name, you'll always retrieve its current value. In many scripting situations, this can lead to dramatically simpler scripts.

Variables must be of the appropriate data type to store a given value. For instance, an integer variable can store a whole number, but it cannot store a double. Likewise, a point name variable can store a point name, but it cannot store a vector.

You can define a variable by using the **Set [____] Variable** commands. (There is a different command for each variable type). These commands create a variable with the specified name that stores the specified value. You can then refer to that value by its variable name.

To overwrite or change the value stored under a variable, simply use another **Set [_____] Variable** command using the same variable name.

To Enter a Variable Argument:

1. Ensure that the entry method dropdown for the desired argument is set to *Variable*, or click the button.
2. Type the name of the variable into the argument's value field.

The value of the variable will be retrieved at the time the command is executed and used as the argument's value.

Variables View

In the bottom-left-hand side of the ***MP Editor*** is a dropdown that allows you to select the *Variables* view (Figure 25-9).

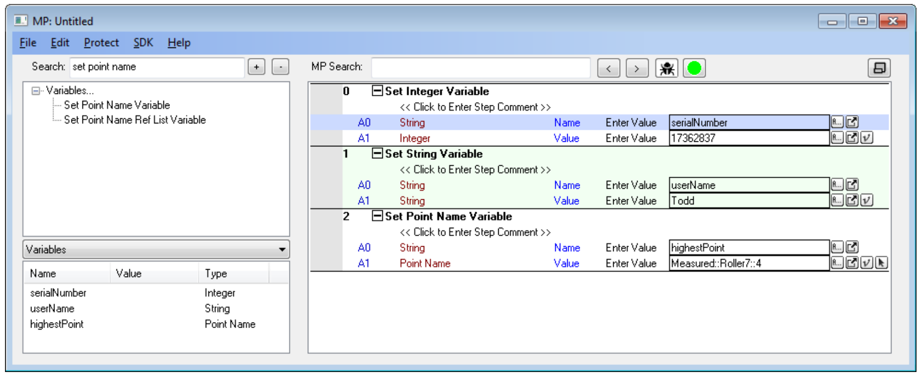


Figure 25-9. A list of the defined variables.

> **Note:** For simplicity, variables defined in other scripts are not displayed. Variables defined in subroutines are not visible to a calling script, so listing a subroutine's variables would be pointless. While editing a subroutine, the editor doesn't know which script is considered its parent, so a parent script's variables are not displayed either.

The Variables view allows you to see all of the defined variables in the current script, as well as their data type. Additionally, while debugging, the value of the variable at the current point of execution is displayed in the *Value* column.

In addition to debugging uses, having a list of defined variables available allows you to easily reference the name when typing a variable in by hand.

Connections View

Occasionally, you may wish to see or jump to commands which are somehow related to the currently selected step. For example, if you delete a command, it would be nice to see which commands reference that step. After all, by deleting that step, any commands that reference it will need to be corrected.

As another example, you may have a command that jumps to a given step. If you want to insert a new command before that step and have the command jump to it instead, you will need to adjust the step reference. These types of situations are where the Connections View comes in handy.

Commenting Out Commands

Sometimes, you may wish to "comment out" commands. Commenting out commands will temporarily disable them—they remain in your script, but the script will skip them during execution. As a result, you can easily remove a portion of your script and then restore it into the script with a simple click. This is extremely useful while testing your script—you may wish to try something without a section of commands, or you may wish to remove a block of your script simply to shorten the length of time it takes for a script to execute. Deleting these commands instead would not be a great approach, since adding them back in would require a lot of work.

Although commented commands are not executed, they are still a part of your script—references to those commands remain. If other commands reference a commented command, they will most likely fail at runtime, because the commented command is not executed, so its values may not be calculated.

Steps which have been commented out will have a speech bubble icon next to them (Figure 25-10).

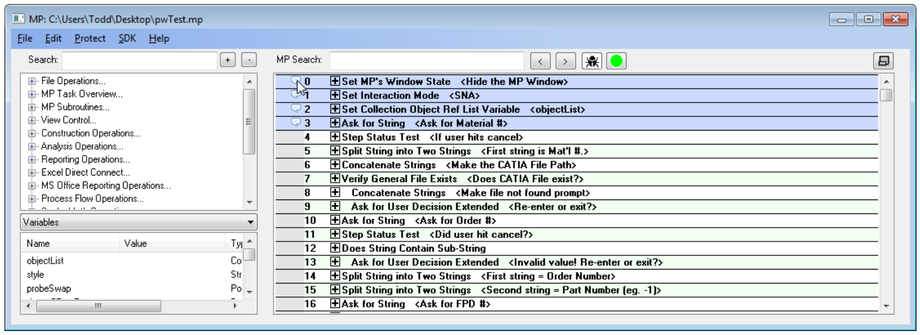


Figure 25-10. Steps 0 through 3 are commented out.

To Comment Out Commands:

1. Select the commands you'd like to comment out.
2. Do one of the following:

- Click in the right half of the gray column to the left of the selected step numbers until a small comment bubble icon appears , (Figure 25-10), or
- Right-click the selection and select **Comment Line (REM)** from the context menu.

To Uncomment Commands:

1. Select the commands to uncomment.
2. Do one of the following:

- Click in the right half of the gray column to the left of the selected step numbers until all icons are cleared, or
- Right-click the selection and deselect **Comment Line (REM)** in the context menu.

To Clear All Comments:

- Right-click any step and select **Clear All Comments** from the context menu.

Setting Breakpoints

When testing, you often want to run a script at "full speed", then have the script pause at a certain point so that you can slowly step through a certain area of your script. For example, if there is a tight loop at the start of a script that executes thousands of times, it would be exhausting to step through the loop manually one line at a time. Instead, what is needed is a way to run the loop at full speed, then pause afterward.

Breakpoints enable this capability. Any line can be set as a breakpoint, and you can set as many breakpoints in your script as you'd like (Figure 25-11). When set as a breakpoint, an executing script will pause prior to executing the line on which the breakpoint is set. You can then choose to step forward one line at a time, or continue running "at full speed".

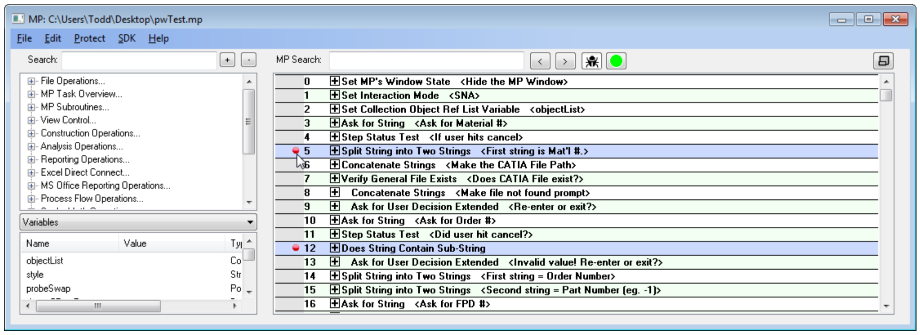


Figure 25-11. Steps 5 and 12 have been set as breakpoints.

To Set Breakpoints:

1. Select the commands you'd like to set as breakpoints.
2. Do one of the following:

- Click in the right half of the gray column to the left of the selected steps until a breakpoint icon appears (Figure 25-11), or
- Right-click one of the selected steps and choose **Breakpoint (BRK)** from the context menu.

To Clear Breakpoints:

1. Select the steps whose breakpoints you'd like to clear.
2. Do one of the following:

- Right-click one of the selected steps and uncheck **Breakpoint (BRK)** in the context menu, or
- Click in the right half of the gray column to the left of one of the selected steps until no icon is displayed.

To Clear All Breakpoints:

- Right-click any command and select **Clear All Breakpoints** from the context menu.

Locking Commands

When you are satisfied one or more command's parameters, you can lock those commands to prevent accidental modification. Locking a command provides a number of advantages:

- They cannot be reordered in the script, so you cannot accidentally move them around.
- They cannot be deleted.

- Changes cannot be made to their arguments.

To Lock or Unlock Commands:

1. Select the commands you'd like to lock/unlock.
2. Do one of the following:

- Right-click one of the selected commands and toggle the **Lock** entry in the context menu, or
- Click in the left half of the gray column to the left of one of the selected steps until the lock icon appears 🔒 (to lock) or disappears (to unlock).

To Unlock All Commands:

- Right-click any command and select **Clear All Locks** from the context menu.

Entering Comments

Note: Multi-line comments are not currently supported.

When a command is expanded, a line directly beneath the command's name says `<< Click to Enter Step Comment >>`. In this field, you can type any comment to help document your script. Simply click in the field and start typing.

When you collapse a command, its comment will be placed next to the command name.

Indenting Commands

You can indent commands to help set them off as being part of a loop or branch of code, similar to how programmers indent their code to improve readability of the source code.

To Indent Commands:

1. Select the commands you' d like to indent.
2. Press **Tab**. The command will shift over by one tab space.

To Remove Indention:

3. Select the commands you'd like to unindent.
4. Press **Shift + Tab**.

Setting Command Colors

You can change the background color of one or more commands in the editor to help differentiate a section of your script.

To Change Command Colors:

1. Select the commands you'd like to change.
2. Right-click one of the selected commands and choose **Set**

Background Color from the context menu.

3. Choose a new color, then click OK. To reset the color back to its default, choose white.

Writing Your First Script

Let's go through the process of writing a short simple script just to see how it's done. This script will ask for a number between 1 and 10, and it will count up to that number starting from 1. For instance, if the user enters **2**, our script will display **1**, **2**, then exit.

1. Start a new script in the editor.

We first need to ask the user for the number to count to. We'll do this using the **`Ask for Integer`** command, which allows the user to enter a whole number and returns the value entered.

2. In the search field above the command list, type **`Ask For I`**.

The search field will filter the available commands down—the **`Ask For Integer`** command should be the only one remaining.

3. Double-click the command in the command list, or select it and press **`Enter`**. It will get added as the first line in our script.

4. In the *Question to Ask* argument, click in the box and type **`Please Enter a number from 1 to 10`**, then press **`Enter`**.

> **Tip:** For any boolean argument, you can type **`0`** to represent **`FALSE`** or any nonzero number to represent **`TRUE`**. The value will be converted automatically when you hit **`Enter`**.

5. In the *Enforce Min/Max Values?* argument, click in the box and type **`1`**, then press **`Enter`**. The value becomes **`TRUE`**. This allows us to ensure that the user can only supply a value within a given range.

6. Set the *Min Value* to **`1`** and the *Max Value* to **`10`**.

7. In the search field/command list, find the command **`Exit Measurement Plan`** (again, starting to type this name in the search field is the fastest way to find the command). Add it to the script (it will appear at the end).

8. Back in the **`Ask for Integer`** command, set the *Step to jump to if Canceled* argument to **`1`**. If the user clicks the cancel button, the script will jump to the **`Exit Measurement Plan`** command in step 1 and exit.

9. Make sure a line in step 0 is selected, then add the command **`Create Counter`**. In the *Counter Value* argument, set the value to **`1`**. This will be the initial value of our counter.

10. Next, add an **`Integer Comparison`** command. This will allow us to check the current counter value against the value

the user entered so we can determine when to stop counting.

11. Click below the command name where it says **`<< Click to Enter Step Comment >>`**. Type **`Keep Counting?`** or a similar comment.

12. In the *Integer A* argument, click the reference quick-select button, then click the yellow box in Step 1, argument 0. The argument should now say **`Ref {S1 A0}`**.

We have now set this argument to reference the current counter's value in step 1, argument 0. This will feed the current counter value into this argument at runtime.

13. In the *Comparison Type* argument, click the box and select **`<=`** from the dropdown list.

14. For *Integer B*, again click the reference button and select step 0, argument 7.

Essentially, this command is now comparing the "current" counter value against the number that the user entered. If the current counter is less than or equal to the supplied number, the command will jump to the *Step if TRUE* step. Otherwise, the command will jump to the *Step if FALSE* step. However, we don't yet have these steps written, so we can't yet fill them in.

Let's consider what happens if the current counter is less than or equal to the supplied value. In this case, we want to display the current counter value.

15. After the **`Integer Comparison`** command, add a **`Notify User Integer`** command. Click the command name to select it and press **Tab** to indent the line over once (we're now inside the body of a loop).

Your script should now appear as depicted in Figure 25-12.

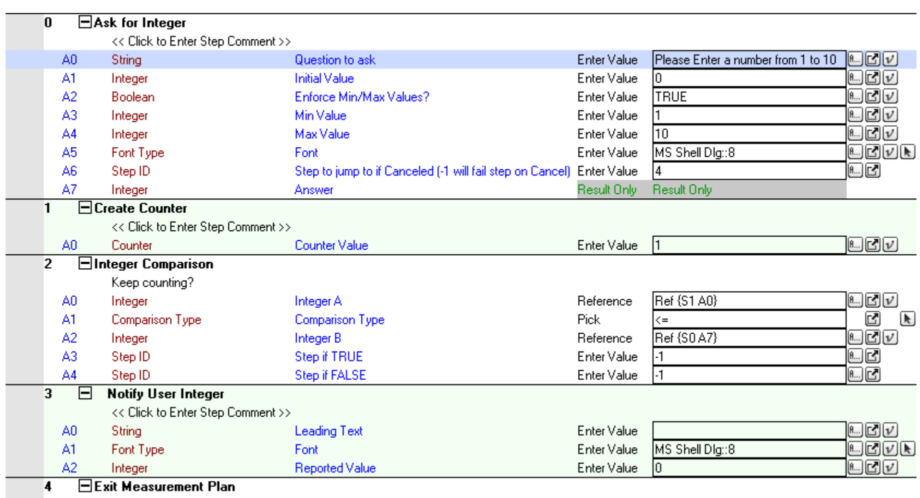


Figure 25-12. Our first script, so far.

16. In the *Reported Value* argument, click the reference button and select step 1, argument 0 (the counter value). This will display the current counter value to the user at runtime.

We need to increment the counter, or else we'll be stuck in an infinite loop.

17. Add an **`Increment Counter`** command after the **`Notify User Integer`** command. Click the reference button and select the counter in step 1, argument 0. Press **`Tab`** to indent the command name.

When this command is reached, the current counter value will automatically be incremented by one.

Almost done. Now that we've incremented the counter, we need to jump back to our test—the comparison—to see if we should continue the loop.

18. Add a **`Jump to Step`** command. In the *Step ID* argument, enter **2** to jump back to the **`Integer Comparison`** command.

The last thing we need to do is fix the step references in the **`Integer Comparison`** command.

19. In the **`Integer Comparison`** command, set the *Step if TRUE* value to **3**. This will enter the body of the loop if we're still counting.

20. Set the *Step if FALSE* value to **6**. When we're done counting and the counter becomes greater than the entered value, we'll exit the script.

21. Select all commands by pressing **`Ctrl + A`**. Then press the left arrow key to collapse all commands at once.

Let's lock the commands so we don't accidentally change them.

22. Ensure all commands are still selected, then click the left side of the column to the left of the step numbers. A padlock should appear on every line, indicating the lines are now "read-only" (Figure 25-13).

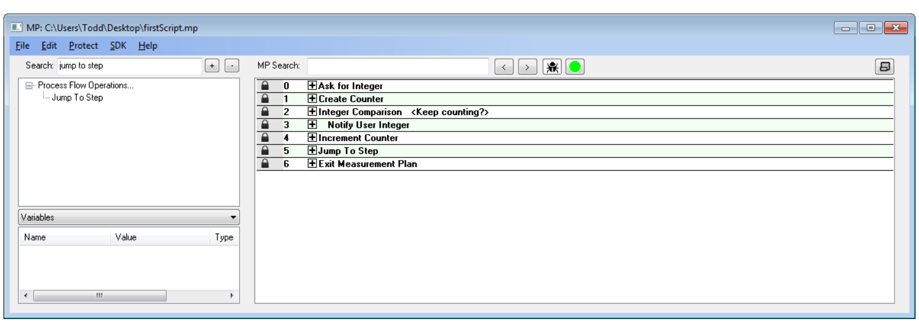


Figure 25-13. All steps are now locked.

That's it! Our script is done. Let's test it.

23. Click the run button in the editor. If auto-run is disabled (see “To Enable/Disable Auto-Running of Scripts:” on page 689), click the same button in the *MP Bar*.

Did your script run as you expected? If not, try to find any errors and correct them.

Debugging

A large portion of the time spent writing a script is often spent on debugging—that is, finding any bugs and errors in the script and correcting them so that the script runs flawlessly in an execution environment.

There are several ways to test and evaluate your script’s behavior.

Stepping Through a Script

One of the most obvious but still useful techniques for testing a script is to simply step through its evaluation one line at a time. This gives you the opportunity to view the effect of each command as it’s executed—typically, scripts execute too quickly to observe what’s happening.

When a script is loaded into the MP Bar, you can step forward one line at a time by clicking the Next Step button.

In the ***MP Editor***, you can enable debugging mode by clicking the Debug button . When you do this, the Next Step button will appear in the editor window, allowing you to step through the script.

Debug Mode

The ***MP Editor*** has a debug mode that you can enable. Debug mode (Figure 25-14) provides several benefits for testing:

- You can step through a script one line at a time.
- Each command’s arguments will display their runtime values, so that as you step through the script you can see the actual runtime values stored with each argument.
- The Variables view shows the current value of each variable as you step through the script.

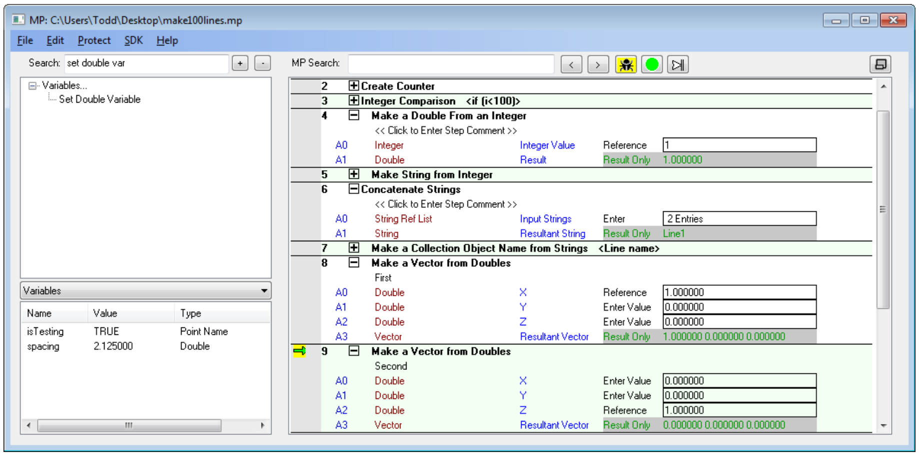


Figure 25-14. Debug mode shows variable values (left column) and current argument values.

These three items provide most of the tools you need to discover errors in your script.

Note: Scripts will run noticably slower in debug mode. This is typically fine, since you're testing—but it is something to be aware of.

Debug mode only works with a single script. This means that if a subroutine is called, you will not see any changes in the ***MP Editor*** window until the subroutine returns and execution continues in the current script. To view values in parent scripts *and* subroutines as a script executes, use the ***MP Watcher***.

To Enable Debug Mode:

1. In the editor, click the Debug button . In debug mode, all arguments will become read-only and they will display their current values.
2. Run the script or step through it using the controls at the top of the window.

MP Watcher

The ***MP Watcher*** is very similar to debug mode in the editor. It is a separate window that allows you to view the current values of all command arguments as a script is executing. However, the MP Watcher adds each line to a list as it is executed. This means that unlike debug mode in the editor, if a script calls a subroutine, the subroutine's values can be seen while that subroutine is executing. Moreover, in a loop each iteration through the loop will add additional lines to the list, whereas in debug mode the previous iteration is "overwritten" with the current iteration's values.

The MP Watcher is shown in Figure 25-15. It will only show commands that are executed starting from the time that the window was opened. In other words, it is possible to run partway through a script, then open the MP Watcher. Only commands from that point on will be displayed.

Scripts will run noticeably slower with the MP Watcher open, so make sure that this window is not open when execution speed is important. (Typically, this is fine, since the window is only used during the testing phase of script development).

To Print the Contents of the MP Watcher Window:

- Click the Print button.

To Export the Contents of the MP Watcher to an Excel Spreadsheet:

- Click the Excel Export button.

To Export the Contents of the MP Watcher to a PDF File:

- Click the PDF Export button.

To Clear the Contents of the MP Watcher Window:

- Click the Clear button.

To Search for Content in the MP Watcher Window:

- Click the Search button. You can search for text in the title, the comment field, for a specific value, or search among a specific command.

To Limit the Number of Steps Displayed in the MP Watcher:

- Click the Options button. Set the *Max Displayed Steps* option, then click OK.

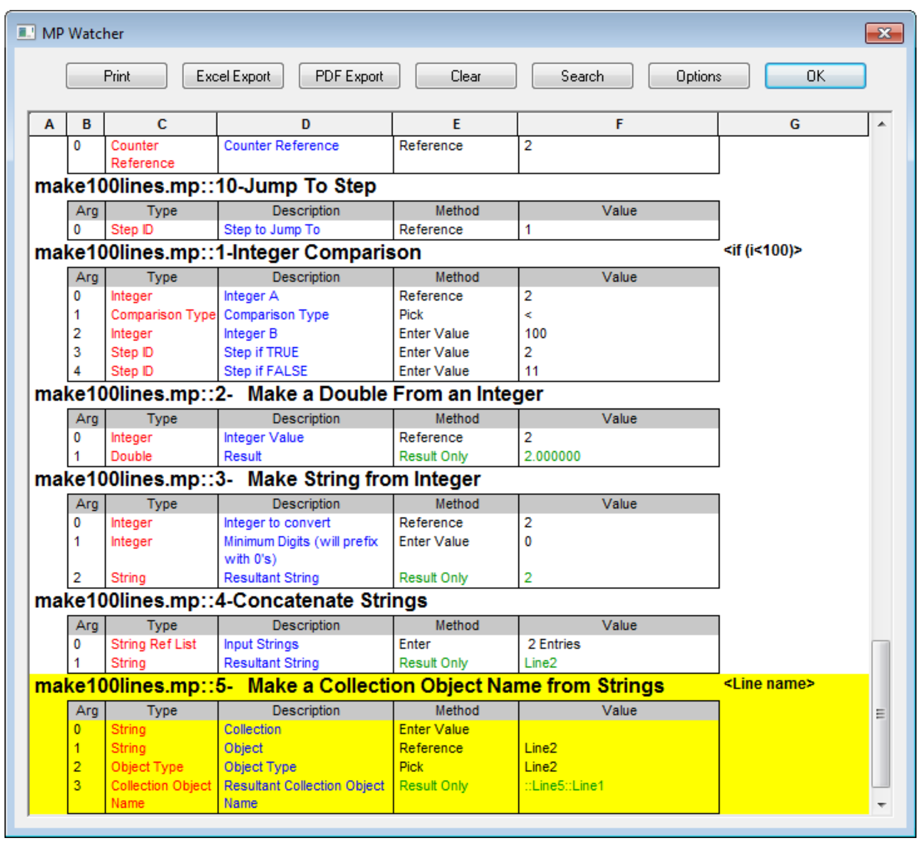


Figure 25-15. The MP Watcher in mid-execution.

SA SDK

CHAPTER 26

The ability to automate measurement, analysis, and reporting tasks in SA can prove to save an enormous amount of money, time, and effort. Automation through Measurement Plans or the SA SDK can introduce efficiencies in your metrology processes unmatched by anything else.

SA SDK

The SpatialAnalyzer SDK provides a means by which to write custom applications that utilize Measurement Plan functionality within Visual C++, VB.NET, or other development environments that support ActiveX controls. Once the SDK engine has been added to the development environment, Measurement Plan steps can be executed from within the programming language.

The advantages of the SDK over Measurement Plans include the following:

- Since the code is written in a full-featured programming language (such as Visual C++ or VB.NET), it offers the full flexibility and power of those programming languages.
- Complex math operations and data manipulation is often easier than when written in MPs.
- There is no need to write routines to transfer data back and forth between MPs and custom external applications--all data is handled within the SDK application itself.
- SDK code is compiled and can be distributed as a true application.
- SDK code is more difficult to reverse engineer than MPs, so pro-

prietary processes can be kept more secure.

The disadvantages of the SDK when compared to MPs include the following:

- It is more difficult to understand for a user not familiar with programming. Creating automated routines is less intuitive.
- It requires at least basic knowledge of a real programming language and development environment.

SDK Engine

Whenever a client application that you have developed is executing, the SpatialAnalyzer SDK engine will automatically start and appear minimized on the Taskbar. Figure 26-1 shows the engine in its maximized state:

Figure 26-1. The window displayed when the SDK Engine is running.

Automatic Code Generation

SpatialAnalyzer supports the automatic generation of SA SDK code for Visual Basic and Visual C++ languages through the MP Editor. This is an excellent way to prototype SDK applications, or convert MPs into full SA SDK applications. To access this sample code, set up an MP with your desired arguments. Then, press the SDK Code Generation button in the MP Editor (Figure 26-2).

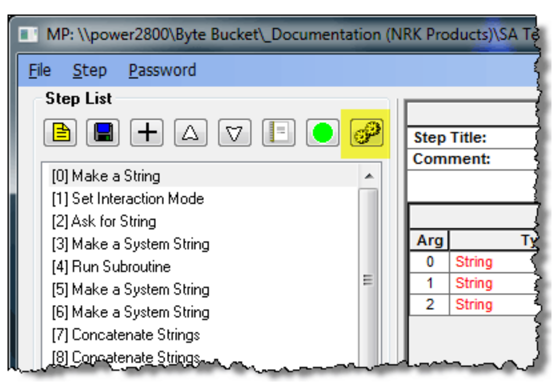


Figure 26-2. The SDK Code Generation button.

The necessary code will be displayed in a window and automatically copied to the Windows clipboard (Figure 26-3).

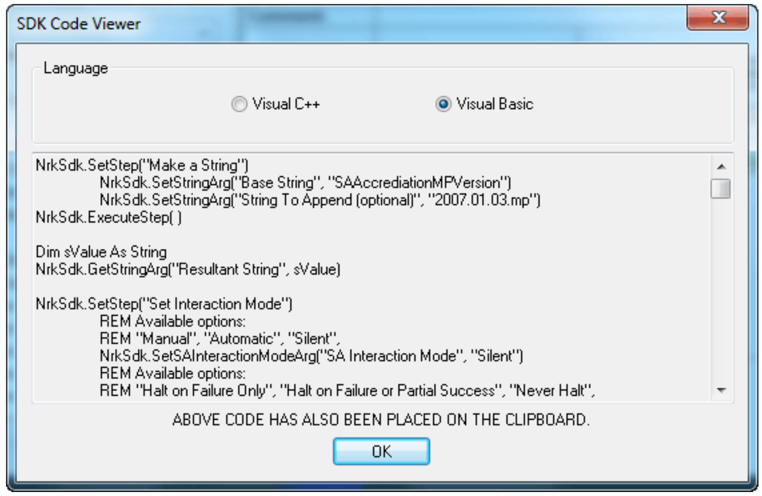


Figure 26-3. The SDK Code Viewer.

You can then paste this code into your SDK application.

Setting Up an SDK Project

VB.NET

1. Create a new Visual Basic Windows Forms Application project within Visual Studio (Figure 26-4). Select **File>New Project**.

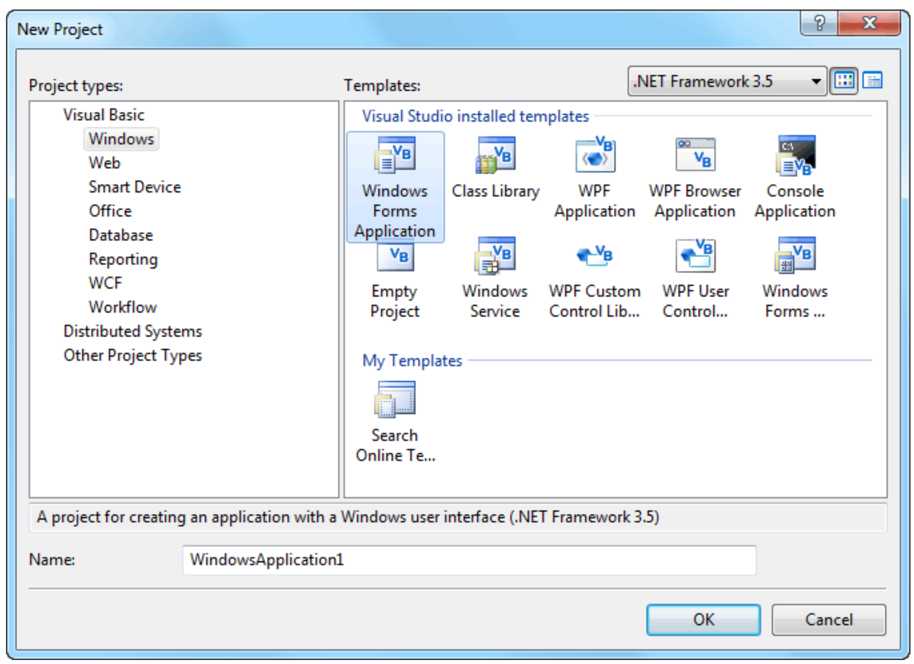


Figure 26-4. Creating a new VB.NET project.

2. Type a name for your application, then press OK.
3. Select **Project>Add Reference...**

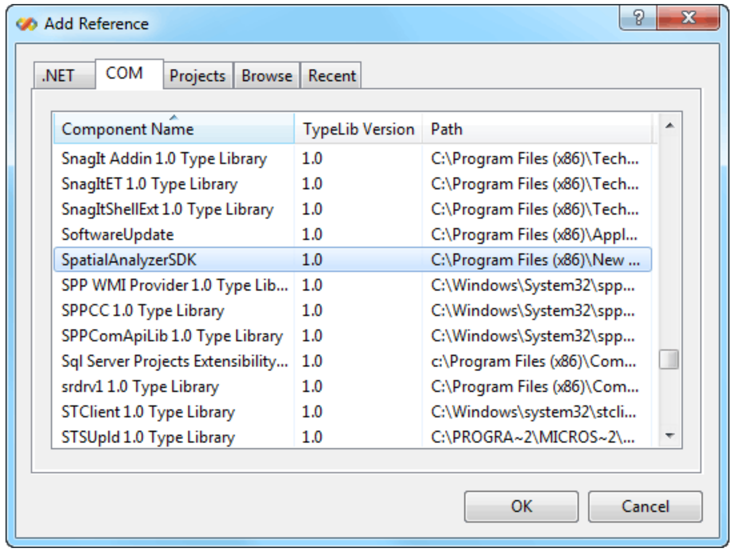


Figure 26-5. Adding the SpatialAnalyzerSDK COM object.

4. In the ***Add Reference*** dialog, select the *COM* tab, then scroll down in the list and choose **SpatialAnalyzerSDK**. Click OK (Figure 26-5).

5. Double-click the form in your project. You should be presented with the code for the `Form1_Load` method of your form (Figure 26-6).

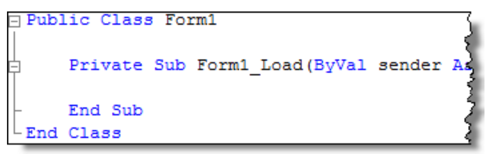


Figure 26-6. The default form.

6. Replace this class code with the following:

```
Public Class Form1
'Declare an object for the SA SDK Interface
Private NRKSdk As SpatialAnalyzerSDK.SpatialAnalyzerSDK

'Declare an enumerated type that defines the various return codes from the SA
'SDK Interface.

Enum MPStatus
        SDKERROR = -1
        UNDONE = 0
        INPROGRESS = 1
        DONESUCCESS = 2
        DONEFATALERROR = 3
        DONEMINORERROR = 4
        CURRENTTASK = 5
End Enum


'Allocate the SDK Interface object
Private Sub Form1_Load(ByVal sender As System.Object, ByVal e As System.
EventArgs)
Handles MyBase.Load
        NRKSdk = New SpatialAnalyzerSDK.SpatialAnalyzerSDK
End Sub

End Class
```

7. Ensure SA is running, then compile and run the application.

Visual C++

1. Create a new Visual C++ MFC project within Visual Studio (Figure 26-7). Select **File>New Project...**

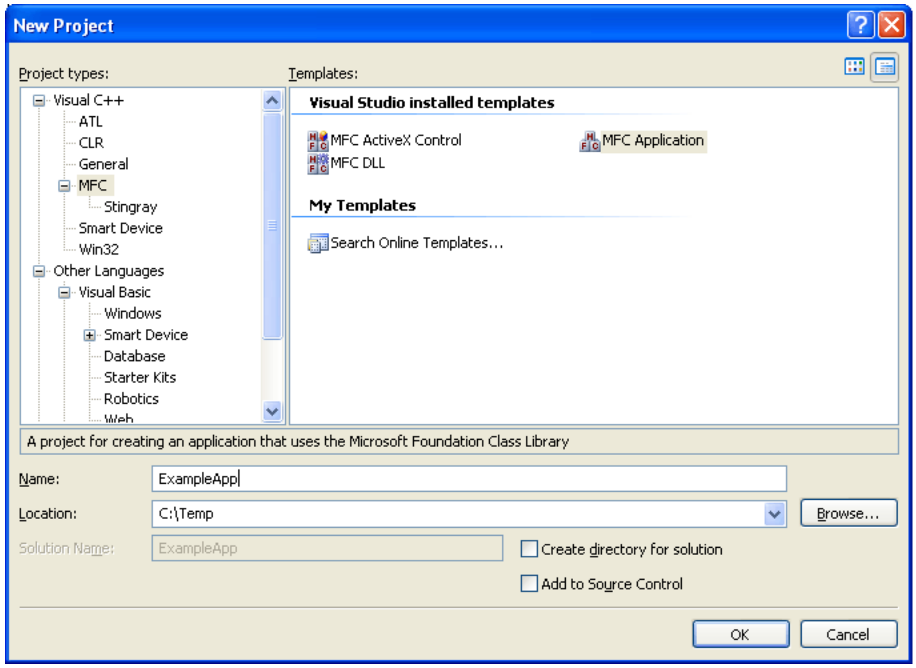


Figure 26-7. Creating a new MFC C++ project.

2. Type a name and location for your project, then click OK.

3. For purposes of illustration, let's assume a dialog application is created (Figure 26-8). Note that the **Dialog based** radio button is selected:

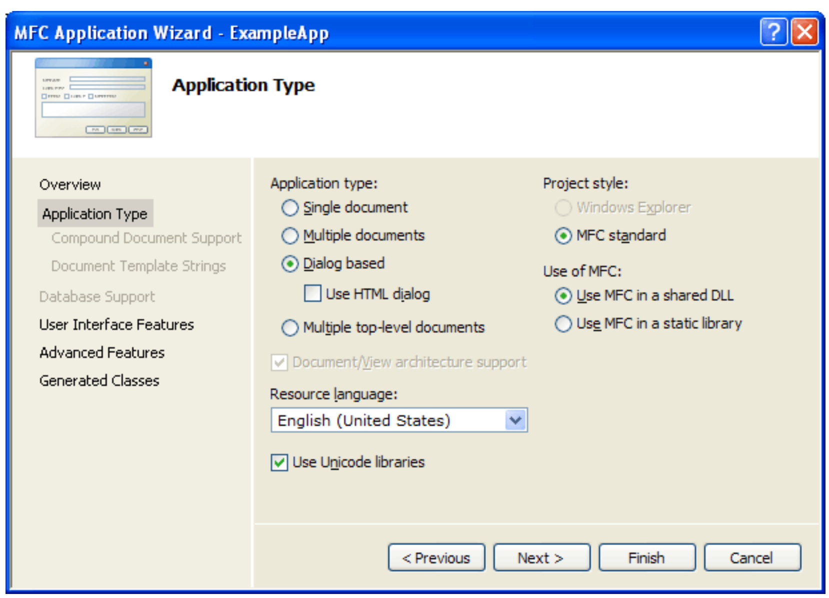


Figure 26-8. The MFC Application Wizard.

4. In the next dialog, ensure that **Automation** is selected (Figure 26-9).

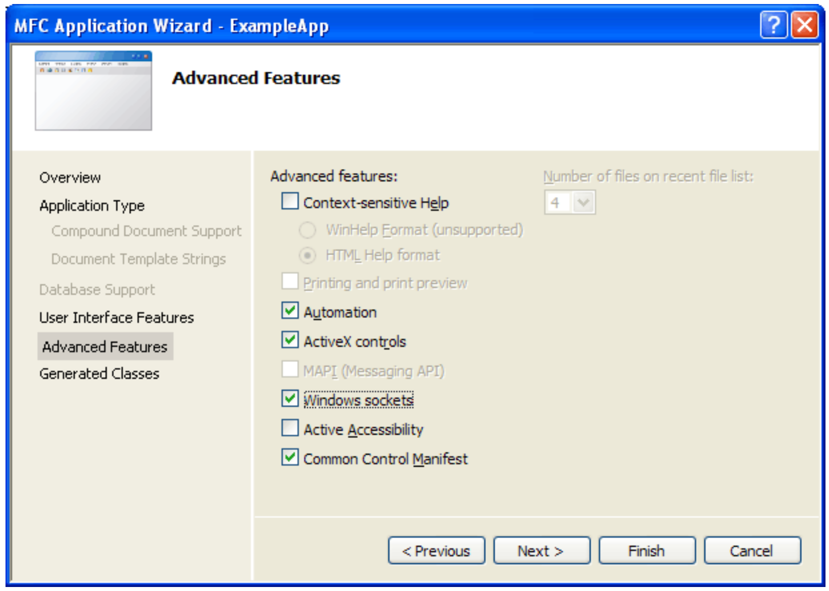


Figure 26-9. Ensuring that the Automation checkbox is selected.

5. Now add the SDK to the application. From the Solution Explorer, right-click the application in the tree and select **Add>Class** (Figure 26-10).

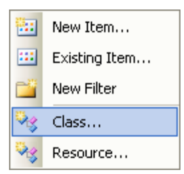


Figure 26-10. Adding a new class.

6. Select **MFC Class From TypeLib**, then click Add (Figure 26-11).

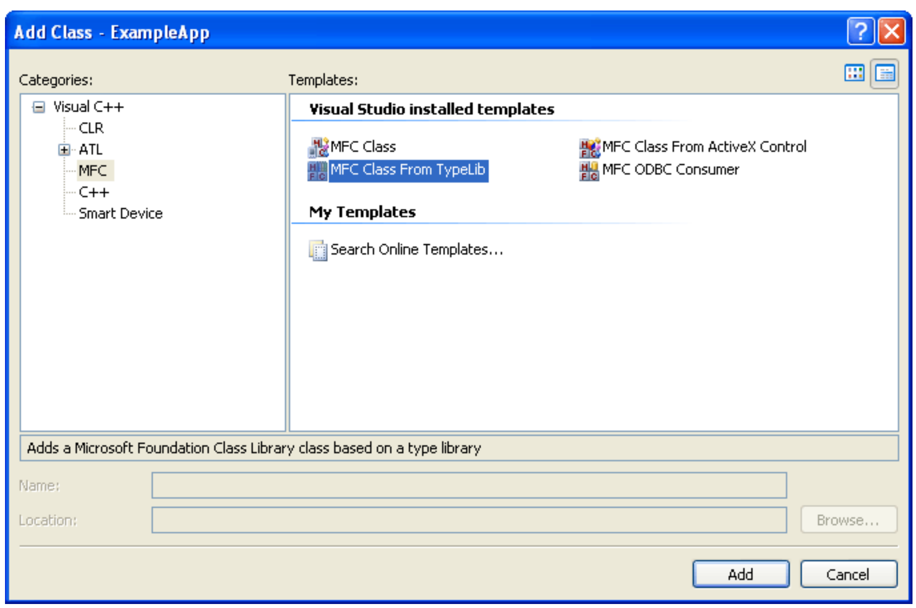


Figure 26-11. Adding an MFC class.

7. Select **SpatialAnalyzerSDK<1.0>** from the *Available type libraries* drop-down. Then click the >> button to add `ISpatialAnalyzerSDK` to the *Generated Classes* section. Click Finish.

8. `CSpatialAnalyzerSDK.h` will automatically be added to

your project and contains all of the available SDK declarations.

Sending Commands to SA

Establishing a Connection

The SDK consists of a set of function calls that allow the specification and execution of MP Steps. Before executing any commands in SA, your SDK code must first connect to SpatialAnalyzer. You can connect to SA using one of two methods: `Connect` (deprecated) and `ConnectEx`. The deprecated `Connect()` method returns a simple boolean `YES` or `NO` answer, indicating whether the connection was successful:

```
'Connect to SA. If unable to connect, display a failure message.
If Not(NRKSdk.Connect("localhost")) Then
        MsgBox("Failed to connect with automation server!")
End If
```

The `ConnectEx()` method allows you to pass a long integer variable (by reference) and obtain a status code from the command which indicates whether a connection was successful or not, and why:

```
'Connect to SA. If unable to connect, display a failure message.
Dim statusCode As Long
NRKSdk.ConnectEx("localhost", statusCode)
If (statusCode)
        Msgbox("Connection error, code: " & statusCode)
End If

'Possible values for statusCode include:
'CONNECTED = 0,
'CONNECTION IN USE = -1     An  SDK Client is already connected to SA
'INVALID_LICENSE   = -2     Invalid SA SDK license
'COULD_NOT_CONNECT = -3     Failed to connect to specified host
'SDK_SOCKET_ERROR  = -4     Other Socket error
```

Executing an MP Step

Once you've successfully connected, you can now execute any MP command by adhering to the following process:

Declaring the Command

The first step in executing a command is to tell the SDK which MP command you want to execute by using the `setStep()` method:

```
'Tell SA which MP step we're defining.
NRKSdk.setStep("Construct a Point in Working Coordinates")
```

This example would tell SA that you are in the process of defining the `Construct a Point in Working Coordinates` command. Note that the string provided as an argument to the `setStep()` method must match the name of the MP command exactly.

Setting the Arguments

Once you've declared the command, you need to provide input arguments using the argument setting methods such as `setVectorArg()` and `setPointNameArg()`:

```
'Set the "Working Coordinates" argument of the command.
NRKSdk.SetVectorArg("Working Coordinates", 10.1234, 20.2345, 30.5678)

'Set the Point Name argument of the command.
'We want to put this point in the active collection, in the TestGrp point
'group, and name it "TestPt".
NRKSdk.SetPointNameArg("Point Name", "", "TestGrp", "TestPt")
```

When doing this, the first string provided to the methods is from the *Description* field of the MP command. This string must match what is seen in the MP, in English. The remaining arguments to the method depend on the type of argument.

Executing the Command

Now that the MP command is fully defined, you need to tell SA to execute the step. This is performed using the `executeStep()` method:

```
'Create a boolean variable to hold the result of the execution.
Dim bSendStatus As Boolean

'Execute the step, and assign the result to our boolean variable.
bSendStatus = NRKSdk.ExecuteStep()
```

Checking the Result of a Command

If the SDK fails to execute the command (for example, because you have not properly defined the arguments), then bSendStatus (above) will be `False`. If the command succeeds, the value will be `True` and you can further obtain the MP step result using the `getMPStepResult()` method. That is, you can determine whether the MP step passed, partially failed, or completely failed:

```
'Only do this if the MP command executed properly.
If (bSendStatus) Then
        'Declare a long integer to hold the result code.
        Dim result As Long
        'Get the MP step result, and assign it to result.
        NRKSdk.GetMPStepResult(result)

        'Check the result of the MP command.
        Select Case result
                Case MPStatus.DONESUCCESS
                        Msgbox("Success!")
                Case MPStatus.DONEMINORERROR
                        Msgbox("Partial Success!")
                Case MPStatus.DONEFATALERROR
                        Msgbox("Failure!")
                Case MPStatus.SDKERROR
                        Msgbox("There was an SDK Error!")

                        'Create an object to hold the error message
                        Dim msgArray As Object

                        'Display the text returned from execution attempt to
                        'user...
                        If (NRKSdk.GetMPStepMessages(msgArray)) Then
                                'Declare a variable to hold the array index.
                                Dim i

                                'Declare a string to hold the message
                                Dim msg As String = ""

                                'Loop through the array to assemble the
                                'message.
                                For idx = LBound(msgArray) To
                                           UBound(msgArray)
                                    msg += msgArray(idx) + Chr(10) +
                                           Chr(13)
                                Next

                                'Now display the message.
                                MsgBox(msg)
                        End If
                Case MPStatus.UNDONE
                  msgbox("Command hasn't been executed!")
                Case MPStatus.INPROGRESS
                  msgbox("The command is currently being executed...")
                Case MPStatus.CURRENTTASK
                  msgbox("The step is the current task.")
        End Select
End If
```

Getting Return Arguments

If an MP contains a "Result Only" argument, then the result argument can be retrieved by a calling the appropriate argument get methods, such as `getVectorArg()` and `getPointNameArg()`. For example, if the MP step successfully executes and a vector result argument is returned, the following code could be used to retrieve the result:

```
If (bSendStatus) Then
        'Declare a long int to hold the return code.
        Dim result As Long
        NRKSdk.GetMPStepResult(result)
        If (result = MPStatus.DONESUCCESS) Then
          'Get the return values and store in variables.
          Dim xVal, yVal, zVal As Double
          NRKSdk.GetVectorArg("Vector Representation", xVal, yVal, zVal)
        End If
End If
```

This Page Intentionally Left Blank.

Machine Calibration

CHAPTER 27

SpatialAnalyzer has the unique ability to integrate portable metrology measurements for the calculation of true metrology-compensated robot and CNC machine calibrations.

SA Machine

Note: While SA Ultimate allows robot calibration, only SA Machine can connect with robots to enable true real-time compensation and control.

SA Machine contains all of the features of SA Ultimate + Native CAD, but also allows interfacing with robots and CNC machines. This permits calibration and compensation of robots and large volume CNC machines to high degrees of accuracy. It also enables robotic scripting, teach pendant behavior, and linear or joint space robotic control.

Usually, robots are delivered by the manufacturer with default kinematic models that are accurate to on the order of about 1/2 - 1/4". This depends on the manufacturer, product, calibration level, and other factors. Many manufacturers offer calibration packages that might improve this accuracy to perhaps 0.1". CNC machines (especially large-volume machines) can also encounter the same sorts of issues (although they are usually more accurate). Regardless, this default accuracy is nowhere near what's necessary for precision applications. There are several reasons for this. One might be that the manufacturer's kinematic model is not quite right or doesn't incorporate things like the stiffness of the linkages or the weights of tools on the machine very well (if at all).

SA's Robot and Machine calibration functions have a few purposes. The first benefit is that they can be used to calculate a significantly more accurate kinematic model than is provided by the manufacturer by default. The following discussion will refer to robots, but it applies equally well to CNC machines.

To calculate a new kinematic model, a robot is commanded to go to a set of poses throughout the desired volume of use and whose positions in the robot controller's coordinates are known. For each pose, measurements are taken on the robot's end effector to determine the actual position of the tool compared to where it was "asked" to go. These measurements can be taken with a single measurement from a 6-DOF measurement device, or they can be taken by measuring 3 targets with a 3-DOF measurement device. (Any measurement method that results in 6-DOF information can be used, including photogrammetry).

Once a set of commanded positions and measured "actual" positions are determined, this data is run through a series of kinematic algorithms to determine an accurate kinematic model for the specific robot (machine) of interest. Using this kinematic model, any position that you'd like to send the robot to is first backed through this kinematic model to give the controller a new set of positions. In other words, based on the calculated kinematic model, SA Machine tells the controller where the robot ought to go (based on its kinematics) to end up where you want it to be. This results in a drastically improved result, (to accuracies of about 0.010", for example). Although this is a big improvement, it can be taken a step further.

Now that the robot has gone where we think it should go, the end-effector can be measured again, which tells us where the robot actually went compared to where we expected it to be. We can then iterate through the kinematic model to give a small correction to the robot, getting it even closer to the desired target. We have demonstrated accuracies to about 0.001"-0.002" using this method, which is quite accurate for a robot. Given the accuracy of the encoders on the robot's servos, this can't be beat without more accurate hardware encoders. This process is a great way to position tools and other items with very high accuracy/repeatability.

Robots in SA

Adding a Robot to SA

There are two ways to add a robot to SA.

- To load an existing SA Robot use **Analysis>SA Machine - Robots and Machines>Add Machine>Load .SAMachine File (kinematics and graphics)**.
- To create a new Robot use **Analysis>SA Machine - Robots and Machines>Add Machine> Add Serial Robot>Using Kinematic Definition File**.

Once a robot is added to the tree it is displayed under the *Robots and*

Machines category.

Robot Category

This category contains the specific Robot or Machine with the respective components and calibrations (Figure 27-1).

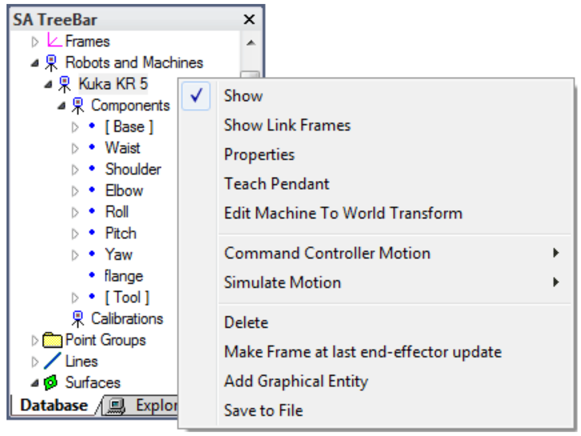


Figure 27-1. The properties of a Kuka KR 5 robot.

- **Show.** Show/Hide all components
- **Show Link Frames.** Show/Hide Link Frames
- **Properties.** Displays properties
- **Teach Pendant.** Displays teach pendant dialog in which user can control robot via several different methods.
- **Edit Machine to World Transform.** Displays machine base transform dialog allowing for the machine to be transformed with respect to the current frame.
- **Command Controller Motion.** Sends move commands to machine based on controller kinematics or SA kinematics.
- **Simulate Motion.** Simulates move commands to machine based on SA kinematics.
- **Delete.** Delete machine from SA TreeBar.
- **Make Frame at last end-effector update.** Creates a SA coordinate frame at the last end-effector position.
- **Add Graphical Entity.** Adds a new component.
- **Save to File.** Saves current machine to .SAMachine file.

Components

The Components sub-category contains each link of the robot with its respective graphical representations (Figure 27-2).

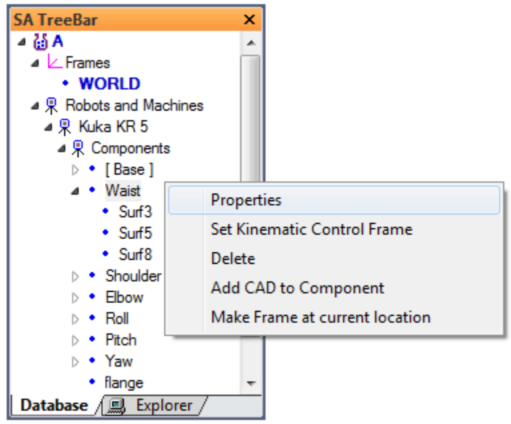


Figure 27-2. The properties of the robot's waist joint.

- **Properties.** Displays the Manipulator Segment Properties - Denavit-Hartenberg Dialog. Here the Segment Transform and Joint Variables can be modified.
- **Set Kinematic Control Frame.** Select the Link Frame for the component.
- **Delete.** Deletes the selected component.
- **Add CAD to Component.** Select surfaces to attach to component at current location.
- **Make Frame at current location.** Creates a SA coordinate frame at the current component location.

Calibrations

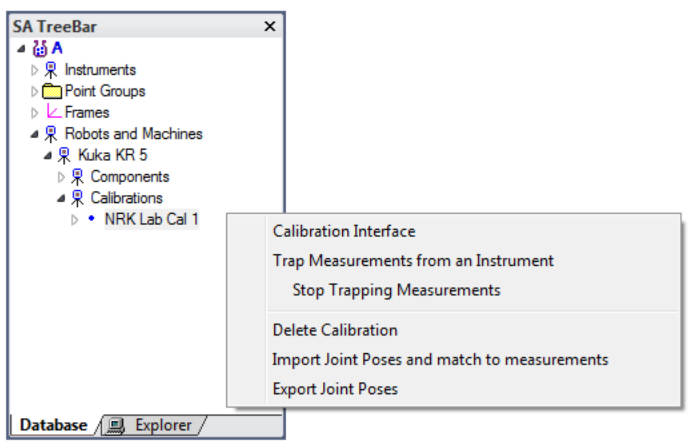


Figure 27-3. A calibration in the tree.

- **Calibration Interface.** Displays the Calibration Interface for the respective calibration set.
- **Trap Measurements from an Instrument.** Allows incoming measurements to be paired with current joint sets.
- **Stop Trapping Measurements.** Stops the trapping of incoming measurements.
- **Delete Calibration.** Deletes selected calibration.

- **Import Joint Poses and match to measurements.** Imports Joint Poses and pairs with selected measurements. Often used when a live interface to a Robot is unavailable.
- **Export Joint Poses.** Exports the Joint Poses from the selected calibration set.

Controlling a Robot from SA

There are several ways to control a Robot from SA. The following sections will cover the various methods.

SA Teach Pendant

The SA Teach Pendant (Figure 27-4) allows for Robot control from SA instead of the physical teach pendant. The SA Teach Pendant allows for individual actuator inputs, Cartesian moves and graphical input.

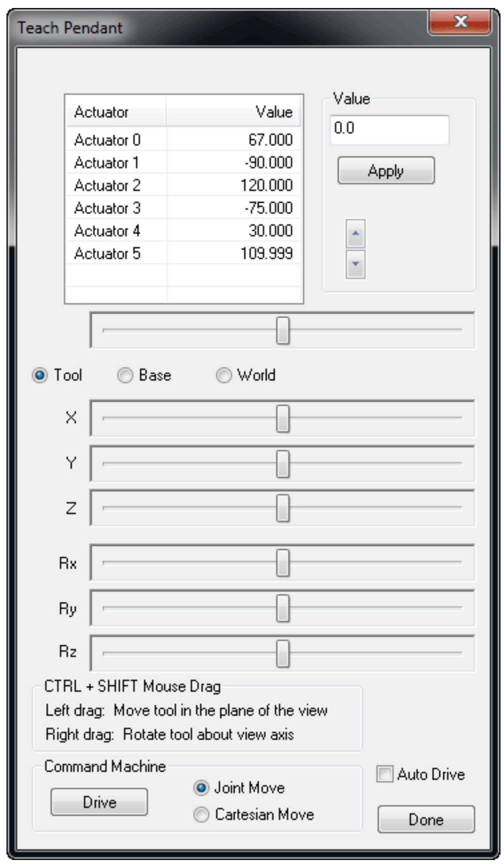


Figure 27-4. The Teach Pendant dialog.

- **Actuator Movement.** Select the actuator to move and either enter the value or use the slider to increase/decrease the value. Select the Drive button to apply the movement to the Robot Controller.
- **Cartesian Movement.** Here the TCP can be moved in Cartesian space about either the Tool, Base or World frame. Simply slide the respective sliders and then click the Drive button to apply the movement to the Robot Controller.

- **Graphical Movement.** Allows the user to graphically move the Robot and then apply the movement to Robot Controller by clicking the Drive button.
- **Command Machine.** The Drive button sends over the specified movement in either Joint space or Cartesian space.
- **Auto Drive.** Automatically sends movement to the Robot Controller when values change.

Move to Frame

The Move to Frame commands (Figure 27-5) allow robot movements to be calculated from the selection of Coordinate Frames in SA. When used, the user will be prompted to select a frame. From here, the robot will be moved to the selected position either in controller kinematics or SA Kinematics.

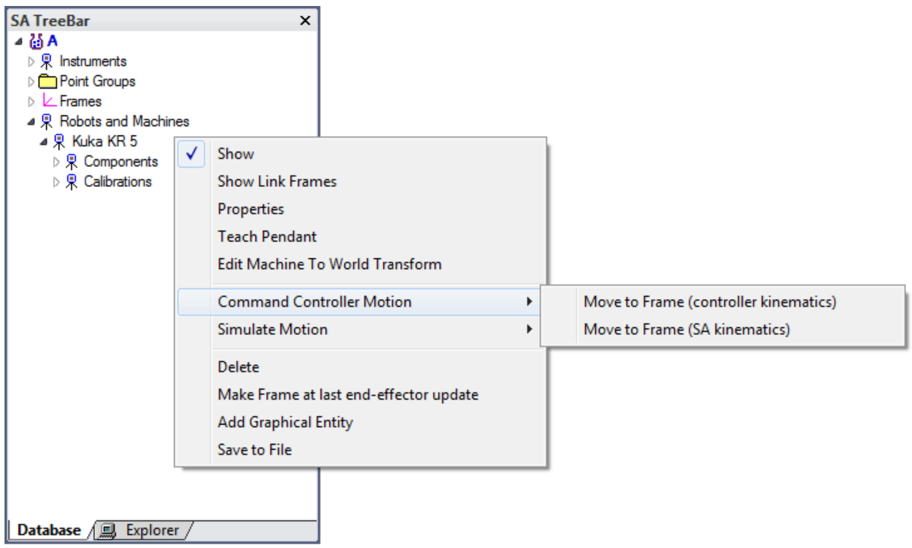


Figure 27-5. The Move to Frame commands.

Measurement Plan

The SA Measurement Plan scripting language also includes several commands for controlling Robots. As the above section describes Move to Frame, similar commands are available in MP. MP is a great tool for automating data acquisition and robot movement.

- Move Robot/Machine to Frame
- Move Robot/Machine to Named Destination
- Set Robot/Machine Parameter
- Compute Robot/Machine Adjusted Goal Frame

Collecting Calibration Measurements

Collecting data for a Robot calibration only takes a few steps. Two scenarios exist: Collecting joint sets from a live robot or importing joint sets.

To Collect Measurements from a Live Robot

1. Ensure that SA and the Robot are connected via the SA Robot Driver.
2. Add a new calibration by right-clicking Calibrations and select **Add Calibration**.
3. Right-click the newly added calibration and select **Trap Measurements from an Instrument**.
4. Now the calibration set is ready to receive measurements and joint sets. As a measurement is received a corresponding joint set is recorded. Simply move the Robot to various poses and record measurements (Figure 27-6).

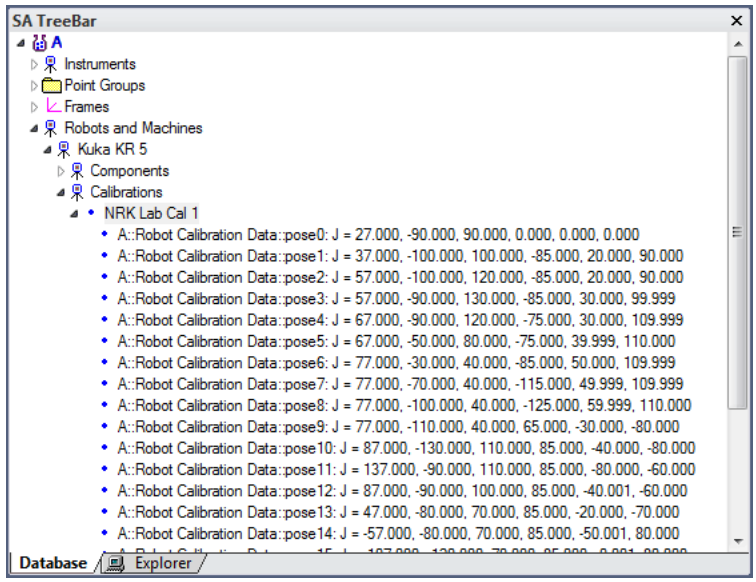


Figure 27-6. Pose data for a calibration set.

To Import Measurements and Robot Poses

1. Ensure the proper Robot poses are formatted in a `.txt` or `.csv` file:

```
<name>,<j1>,<j2>,etc.
```

2. Collect measurements for each pose and ensure the point names match the respective pose names.
3. Add a new calibration by right-clicking Calibrations and selecting **Add Calibration**.
4. Right-click the newly-added calibration and select **Import Joint Poses** and match to measurements. You will be prompted to select the points and pose file.

Calibration Interface

The calibration interface is used to compute the new kinematic model for the robot based on the recorded poses and measurements (Figure 27-7).

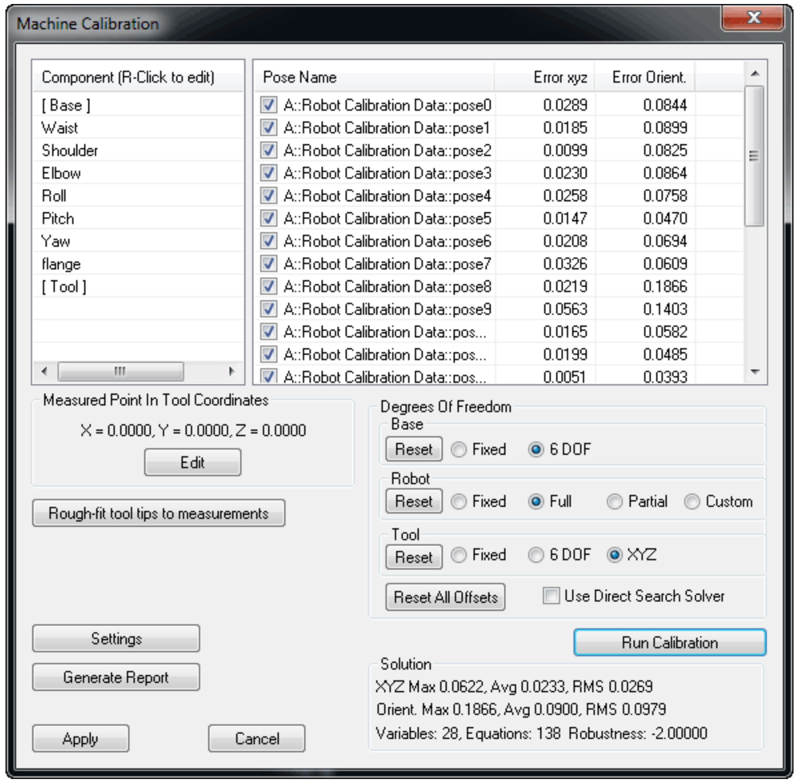


Figure 27-7. The Machine Calibration dialog.

- **Components.** Displays the current components for the current Machine. Right-click a component to access properties.
- **Pose Name and Errors.** Displays each pose and corresponding positional and rotational errors. Uncheck a pose to remove from solution.
- **Measured Point in Tool Coordinates.** Used to define the measured target in the TCP frame when using single point measurements. Not needed when using a 6 degree of freedom target.
- **Rough-fit tool tips to measurements.** Performs a best-fit of the tool tip locations to the measurements to ensure a good initial condition for the optimization.
- **Settings.** Displays the tool orientation weighting factor.
- **Generate Report.** Generates a final report to Excel with solution results and kinematic properties.
- **Degrees of Freedom.** Degrees of freedom for the base, linkages and tool of Robot.
- **Run Calibration.** Executes the optimization.
- **Solution.** Displays current position and orientation errors and RMS values.

Best Practice for Running a Calibration

1. With measurements and poses captured, open the calibration interface by right-clicking the calibration set of interest and choosing **Calibration Interface**.
2. Ensure a good initial starting condition by pressing Rough-fit tool tips to measurements. The overall errors will reduce greatly as the measurements and tool tip positions are best-fit to one another.
3. The initial Degrees of Freedom (DOF) are to allow the base and tool to move freely while maintaining the nominal kinematics (Figure 27-8). The tool DOF should only be set to 6 DOF when 6 DOF measurements are present. Press Run Calibration to start the optimization.

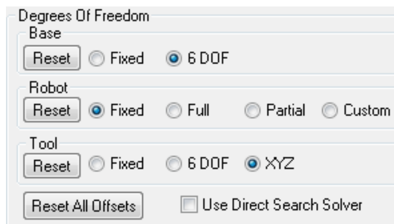


Figure 27-8. The Degrees of Freedom box.

4. Now, if desired, allow the Robot kinematics to vary by choosing **Partial** or **Full** (Figure 27-9). Press Run Calibration to start the optimization.

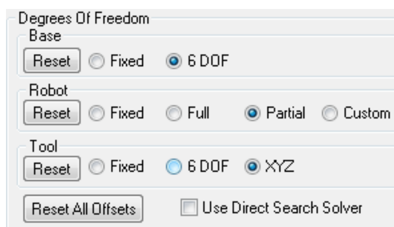


Figure 27-9. Choosing a partial robot DOF solution.

5. At any time, make sure to validate any potential outliers and remove by un-checking the pose and re-running the optimization by pressing Run Calibration.
6. Once satisfied with optimization, generate a report for external sources and/or apply the new calibration to SA by pressing Apply.
7. This calibration will now be used when moving the robot with SA Kinematics.

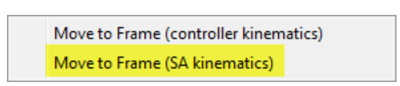


Figure 27-10. The Move to Frame option.

Kuka Robot Deployment

Below are the necessary components to operate SA and SA Robot Driver with Kuka Robots.

XML Channel configuration and Data format files (C:\KRC\Roboter\INIT)

- RAC.xml
- RAC+.xml
- XmlApiConfig.xml (set tcp/ip address and port of remote access computer – secondary fixed ip on remote computer)

KRL Client Communication (C:\KRC\Roboter\KRC\R1\Program)

- SAKRL.dat
- SAKRL.src

KRL Client motion programs (called by SAKRL.SRC C:\KRC\Roboter\KRC\R1\Program)

- MoveToFrame.SRC and .DAT
- MoveToJointSet.SRC and .DAT
- HitEStop.SRC

VxWorks Communication Configuration (C:\Windows)

- vxWin.ini (robot controller VxWorks tcp/ip address modified under e={……})

Server application and data format files on remote computer

- SARobotDriver.exe
- EKX Server.exe
- RACData.xml
- Global Data (c:\KRC\Roboter\KRC\R1\System)
- $config.dat

SA Robot Driver – Kuka Controller Requirements

- KR C2 Controller
- Kuka System Software
- KSS v.5.5
- Kuka Ethernet KRL XML
- v. 1.2
- 3COM Ethernet Card Option

Denavit-Hartenberg Notation[1]

The first step in calibrating a manipulator is to understand its nominal kinematic parameters, and the controller's ability to accept new, calibrated values for these parameters. This section will describe how to

1 For a more detailed description of Denavit-Hartenberg notation see Craig, J. J., 1989, Introduction to Robotics: Mechanisms and Control, Addison Wesley, New York.

represent a robot's kinematics. Most manipulators are serial devices, meaning they are composed of a series of links and joints connected in an open chain. Denavit-Hartenberg notation is a standard means for specifying the kinematics of a manipulator. See the example table below, then a brief description of the meaning of the parameters.

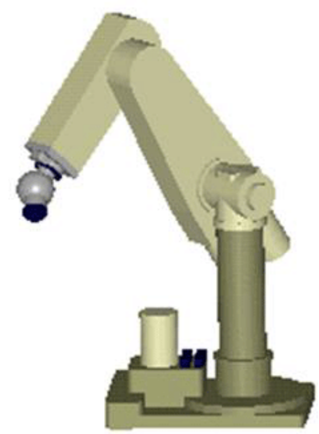


Figure 27-11. A robot and its Denavit-Hartenberg (D&H) table.

Link	α	a	d	θ	Type
1	0.0	0.0	0.0	0	Revolute
2	-90.0	0.0	0.0	0	Revolute
3	0	8.0	-4.8	0	Revolute
4	-90.0	0.0	8.0	0	Revolute
5	-90.0	0.0	0.0	0	Revolute
6	-90.0	0.0	0.0	0	Revolute

Links are rigid members; joints allow relative motion between the links. The chain starts at the base, or ground link, and ends at the working tool. Except for the tool link and the base, each individual link in the chain begins and ends with a joint. The tool may be a gripper, paint sprayer, welding tip, electro-magnet or any of a wide variety of devices. Common manipulators use only revolute (turning) or a prismatic (sliding) joints to connect links. More complicated forms of joints can always be modeled as a combination of these simple elements. The function of a link is merely to fix the relationship between the joints at either end of the link. As long as the two joints are held at the same relative position and orientation with respect to one another, the link serves only as a rigid connector. The actual physical shape of the link can be changed without altering the kinematic relationship between the two interconnected joints.

A kinematic description of the manipulator is generated by SpatialAnalyzer using the standard Denavit-Hartenberg notation. This remarkably simple representation uses only two values to describe each link, and two additional values to describe the joint connection between adjacent links. Links are described in terms of their length (a) and the twist they maintain between adjacent joints (α), as explained below.

- **Twist Angle (α).** The angle between the first and second joint axes on a link. This angle is measured in a right-hand sense when viewed directly along the common normal.
- **Link Length (a).** The distance between the two joint axes of a link. This distance must be measured along a line that is normal (i.e. perpendicular) to both joint axes. This line is referred to as the common normal between the joint axes. If the joint axes intersect (a common arrangement in industrial manipulators), the link will have zero length. The joint connection between adjacent links is described in terms of the offset (d) and the joint angle (θ), as explained below.
- **Offset (d).** The distance between the common normal of one link and the common normal of the next, measured along the axis of the joint connecting the two links.
- **Joint Angle (θ).** The angle between the common normal of one link and the common normal of the next, measured about the axis of the joint connecting the two links.

The four parameters α, a, d, and θ, when taken for each link and joint in the chain, provide a complete kinematic description of the manipulator. When taken together with a description of the physical shape of each link and limits on joint motion, an accurate simulation of robot motion and operating workspace can be developed.

In addition, there is usually a *Joint Offset* value computed during the calibration. This value represents the difference between the "kinematic zero" of a joint, and the mechanical zero based on how the encoder was set in the joint mechanism.

For a more detailed description of Denavit-Hartenberg notation see:

Craig, J. J., 1989, Introduction to Robotics: Mechanisms and Control, Addison Wesley, New York

Instrument Quick-Starts

CHAPTER 28

These instrument Quick-Start guides are designed to supplement the manufacturer's instrument device manuals by providing straightforward, easy to follow steps to quickly get instruments up and running in SpatialAnalyzer. You may wish to reference them when setting up your instrument with SA for the first time.

Connection Configurations

Instruments can be categorized by the connection type used to interact with the computer. These fall into several basic categories:

- **Serial Devices.** Serial devices (such as RS-232) require a functional serial port on the user's computer and the use of a serial cable. USB to serial adapters are available should your computer not have a serial port (We have had success with DIGI Edgeport adapters, which although are more expensive, have been quite reliable). Refer to the individual following sections for more specific configurations.
- **Bluetooth devices.** Bluetooth devices require a Bluetooth receiver on the user's computer. If your computer does not have a built-in Bluetooth connection, a relatively inexpensive module, such as a Rocketfish™ Micro Bluetooth Adapter, can be purchased from most office supply stores.
- **Universal Serial Bus (USB).** USB connected devices typically require the installation of a USB driver for the instrument. The latest drivers for supported instruments can be found on NRK's FTP site at: ftp://ftp.kinematics.com/pub/SA/Install/Driver Downloads/.

- **Network-connected devices (TCP/IP).** The majority of instruments used in SpatialAnalyzer are network-connected devices. This connection type provides extremely fast and reliable connections either directly to the host computer or over a network.

Laser Trackers

API Radian

Hardware Setup

Set up the unit following the manufacturer's directions. API Radian trackers are networked TCP/IP connected devices and should be connected either to a network or directly to a computer with an Ethernet crossover cable.

Connect the temperature probe and ensure that it is well clear of any external heat sources (such as the heat fan on the back of the power supply). Ensure that the instrument is powered on and that an SMR is in the home position.

Software Setup

Each version of SA is built with the most current version of the API software development kit (SDK). This driver set provides the tools to interface with the instrument. Check the Readme file to see which SDK version is being used. No additional software installation is necessary.

Set the IP address on your computer to be compatible with the address set on the tracker. The default Radian IP address is: 192.168.0.168 with the subnet mask 255.255.255.0.

Starting the Interface

1. Select **Instrument>Add** and choose API Radian from the ***Add Instrument to SA*** dialog (Figure 28-1).

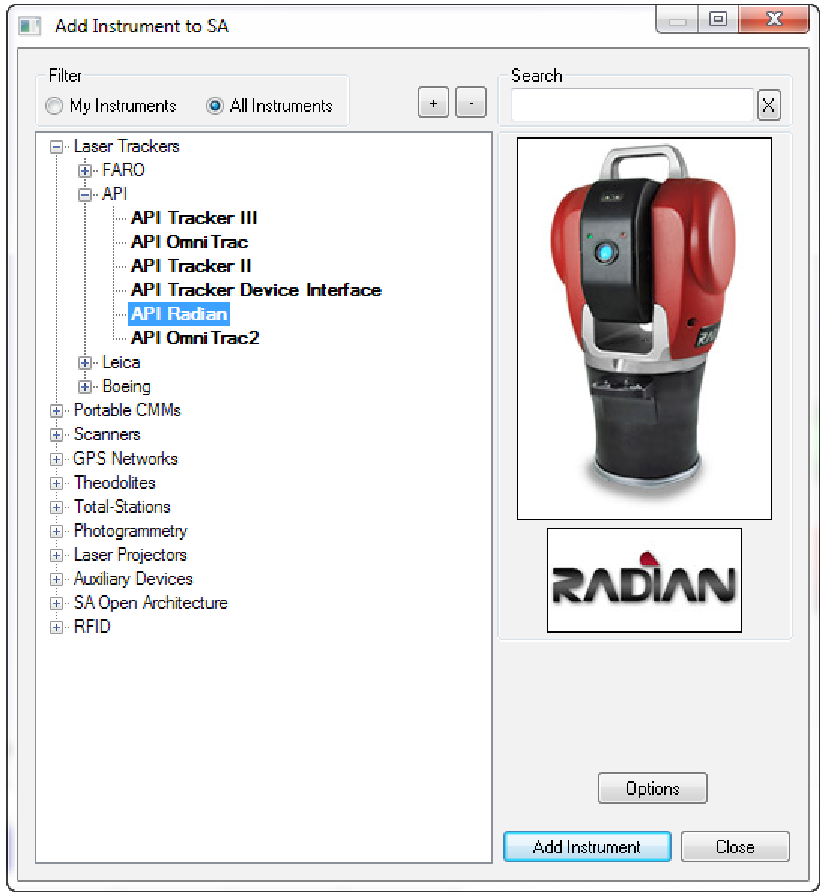


Figure 28-1. Adding a Radian tracker.

2. Run Interface Module without connecting (**Instrument>Run Interface Mode**) and choose *API Device Interface* (not Laser Trackers).
3. Within the ***Connect to SpatialAnalyzer*** dialog, select the instrument station (computer name, job name, Collection::Instrument Name: Serial Number) you wish to connect your instrument to from the network list.
4. In the ***API_DI Connection*** dialog, enter the tracker's IP address (if different than the default) and use the Ping button to test the connection if needed. Once satisfied, click OK. The next time you connect this instrument to the instrument, you can just select *Run Interface and Connect*. This will utilize the last saved settings and automatically connect the instrument.

Intelliprobe 360

Please refer to the current API Intelliprobe 360 user's manual for setup and calibration instructions. A version is available on our webpage: http://www.kinematics.com/ftp/SA/Install/Driver%20Downloads/Laser%20Trackers/API/Radian/.

1. Ensure the I-360 has been fully unpacked, configured to communicate with the controller, and calibrated within API's soft-

ware.

2. Set the reflector in SA to the appropriate I-360.
3. Place a 1.5" inch reflector in the nest.
4. Run a virtual level measurement (**Check/Cal> Virtual Level**). This will align the instrument with gravity, allowing the I-360 pendulum measurements to be correctly interpreted.
5. Specific measurement profiles have been added for the I-360 and must be used to take data from the device (Figure 28-2).

Note: Conducting the Virtual Level measurement will transform subsequent tracker data. If you have taken measurements from this instrument station previously, a new instrument station must be added.

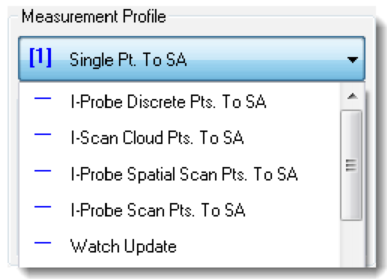


Figure 28-2. I-360 Measurement profiles.

API T2+

The parameter file, `[tkr serial#].prm` contains the kinematic correction info, home location, etc. for the tracker. For example, if your tracker serial number is 3827, the file name will be `3827.prm`. This file must be located successfully by the tracker initialization for the tracker to run. By default the parameter file should be located in `C:\Analyzer Data\Persistence`. To change this location, just press the Browse button in the tracker connection dialog, as pictured in Figure 28-3.

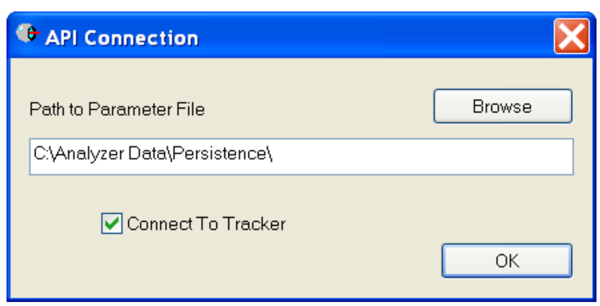


Figure 28-3. The legacy API Connection window.

The parameter file is edited when an API calibration is run. Ensure that the file attributes are NOT set to *read only*.

You must have a working serial port for the connection to the tracker. Use the lowest available serial port number for the tracker connection.

API T3

The parameter file is stored on the controller for the API T3 tracker. Like the API T2+, you will see a default location for a parameter file. In the case of the T3, this is only the location where a back-up copy of the parameter file will be written from the controller. To change this location, just press the Browse button in the tracker connection dialog.

Faro

This guide applies to the setup of the SMX 4000, 4500, Faro X, Xi, Ion and the Vantage.

Hardware Setup

Set up the unit following the manufacturer's directions. Faro trackers are networked TCP/IP connected devices and should be connected either to a network or directly to a computer with an Ethernet cross-over cable.

Connect the temperature probe and ensure that it is well clear of any external heat sources (such as the heat fan on the back of the power supply). Ensure that the instrument is powered on and that an SMR is in the home position.

The Faro Vantage requires SA version 2012.09.14 or later and a minimum of JRE version 4.0.2 or later.

Software Setup

Set your computer's Local Network Connection to be compatible with that of the Faro tracker. Faro trackers are shipped with a standard IP address of 128.128.128.100.

Download the latest java drive from: **`http://www.kinematics.com/ftp/SA/Install/Driver Downloads/Laser Trackers/Faro/`**. Extract the files to the C:\ drive. This should create a directory structure with the Faro Java files contained in C:\FaroJRE.

If you plan to use the video (overview) camera, you must get 'Faro Tracker v.4.0.2 (or later) Camera files.zip' (search the SA readme for "java" to ensure correct version for your install) from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/Laser%20Trackers/Faro/ . Unzip the file and follow the instructions contained in `ReadMe Faro Camera.txt`. Note that cameras are wireless. If your camera has the default IP address, you can set your PC's wireless connection's address to 129.129.0.1.

Starting the Interface

1. Select **Instrument > Add** and choose the appropriate Faro tracker from the ***Add Instrument to SA*** dialog (Figure 28-4).

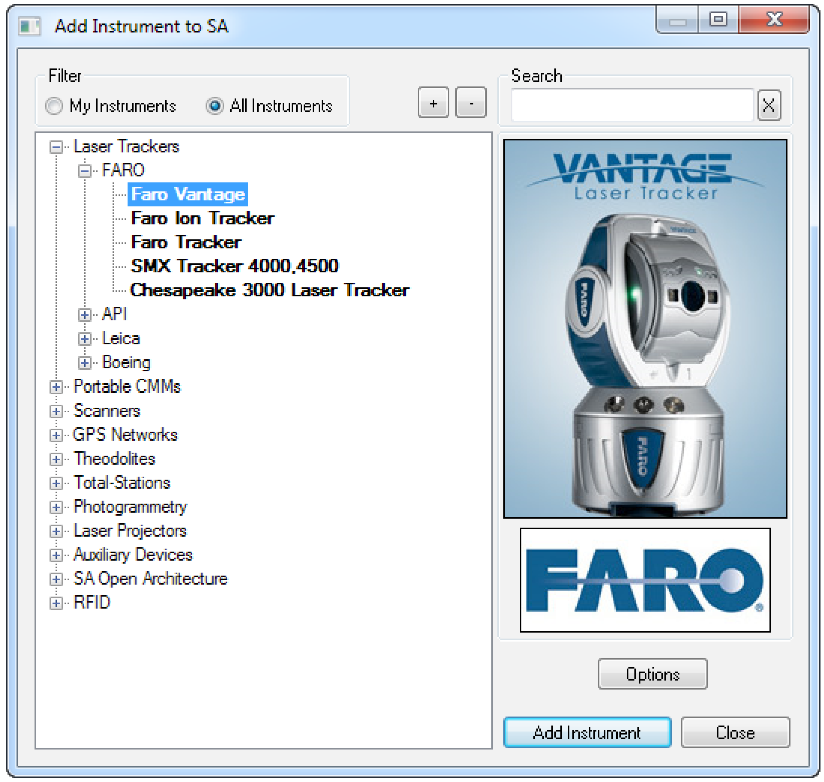


Figure 28-4. Adding a Faro tracker

2. Run Interface Module without connecting (**Instrument > Run Interface Mode**) and choose *Laser Trackers*. This will bring up the ***Faro Connection*** dialog (Figure 28-5). Within it you will see all the available instruments on the network.

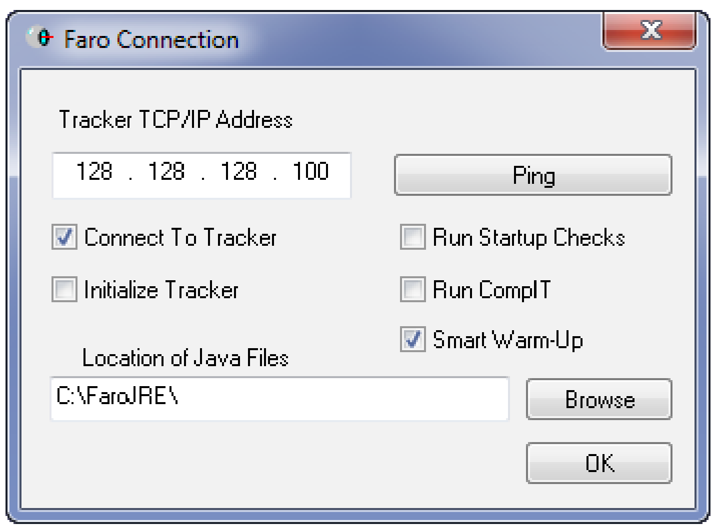


Figure 28-5. The Faro Connection dialog

3. Select the instrument station (computer name, job name, Collection::Instrument Name: Serial Number) you wish to connect your instrument to from the network list.

4. Enter the tracker's IP address (if different than the default) and use the Ping button to test the connection if needed. Once

satisfied, click OK. The next time you connect this instrument to the instrument, you can just select *Run Interface and Connect*. This will utilize the last saved settings and automatically connect the instrument.

5. To use the camera, click the ADM Drive button in the SA interface. The tracker pad will pop showing the drive options. Select *Camera Drive* to use the Faro camera interface.

The interface is now connected and ready for use. Please refer to the Measurements chapter of the manual for more details on the tracker interface and instrument settings options.

Leica EmScon

Prerequisites

- Newer emScon controllers require a Leica dongle. Ensure that the dongle is installed on the controller's parallel port.
- All emScon controllers are shipped with 192.168.0.1 as the IP address. To learn more about configuring IP addresses, see the IP Address Basics section.

Compensation

The Leica emScon trackers and accessories can be compensated per Leica through emScon (Figure 28-6). EmScon can be accessed via the shortcut on the desktop or use the following address http://192.168.0.1:8099/emscon/default.aspx. If your tracker address is not the standard 192.168.0.01, change that component of the address in your browser.

Figure 28-6. The main EmScon window.

- Compensation Password: Expert (Full and Intermediate, ADM, Reflector Definition, Camera Compensation, etc.)
- Server Settings Password: Administrator (TCP/IP address, Time/Date, etc.)

Starting the Interface

1. Select **Instrument > Add** and choose the respective Leica emScon Tracker from the ***Add Instrument to SA*** dialog.

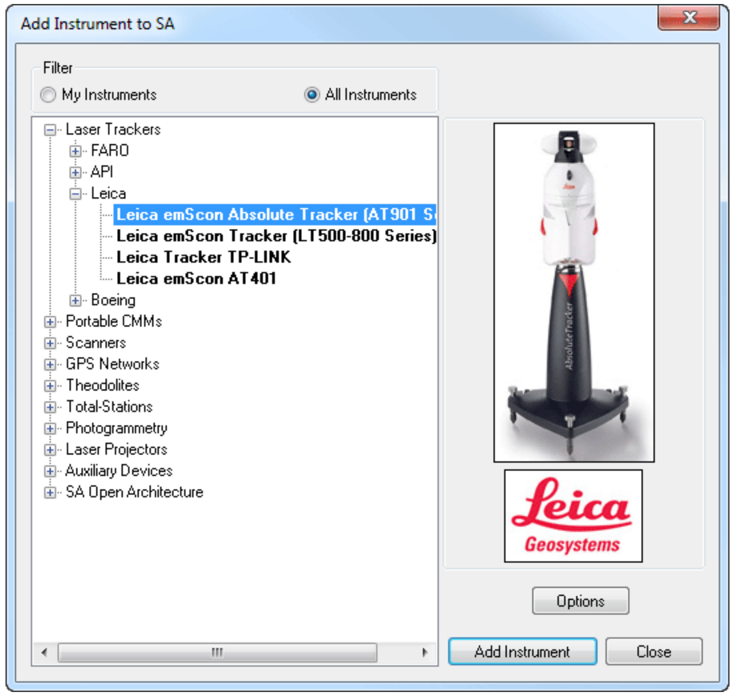


Figure 28-7. Adding an emScon tracker.

2. Now run the instrument interface module under **Instrument > Run Interface Module** and choose **Laser Trackers**.

3. Enter the tracker's IP address and use the Ping button to test the connection if needed. Once satisfied, press OK. The next time the interface is started, you can simply click the Run Interface and Connect icon. This will use the last saved settings and automatically connect the instrument.

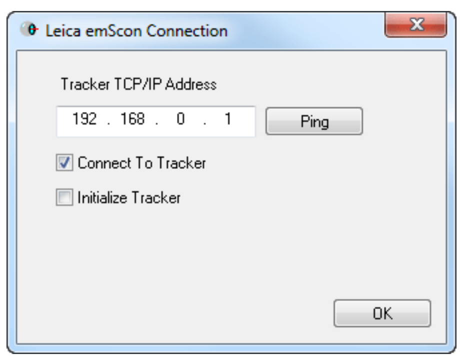


Figure 28-8. The Leica emScon connection window.

4. The interface is now connected and ready for use. Please refer to the Laser Tracker section for details on the laser tracker interface.

EmScon Settings

To access the emScon settings, use **Settings > Tracker > General Settings** or press the button. Then press the tracker specific button at the bottom. This will expose the emScon-specific options (Figure 28-9). These options are the same ones that exist in emScon. This dialog gives you access without having to enter emScon.

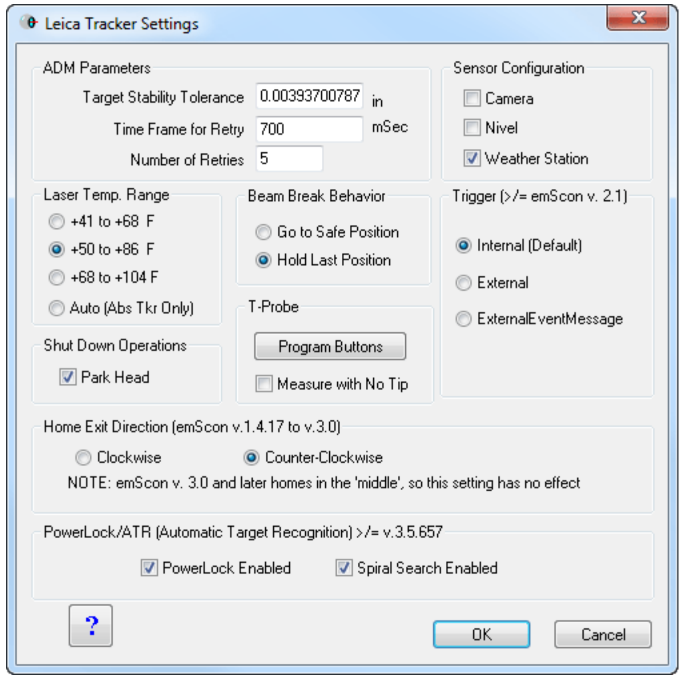


Figure 28-9. EmScon-specific tracker settings.

ADM Parameters

Parameters for recapturing the beam in ADM.

Sensor Configuration

Choose which peripheral devices are being used. Overview camera, Nivel Level and Weather Station.

Laser Temperature Range

Select the temperature range in which the tracker will be operating.

Beam Break Behavior

- **Go to Safe Position.** The head will point directly up toward the overview camera when the beam is broken.
- **Hold Last Position.** The beam will sit still when the beam is broken. This is needed for ADM beam break reset mode to work. If not, each time the beam is broken the tracker will return to its safe position.

Trigger

- Select the type of trigger, if applicable, for triggering measurements.

T-Probe

- Allows for the programming of buttons and the option to measure without a tip.

Home Exit Direction

- Choose which direction for the tracker head to unwrap when leaving the birdbath.

PowerLock/ATR settings

Ability to turn on/off PowerLock and Spiral Search.

Tracker Status

To view the tracker status (Figure 28-10) go to **Utility>Tracker Status**.

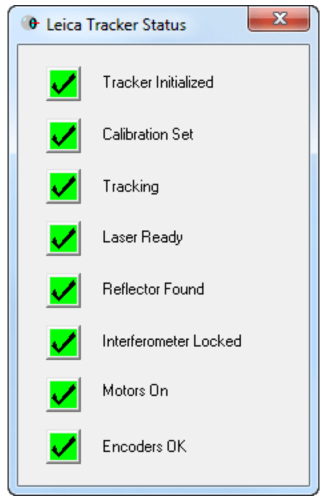


Figure 28-10. The Leica Tracker Status dialog.

T-Probe

If you get a 'usage conflict' error when trying to use the T-Probe, run the emScon BUI. In it, there is a setting for stylus usage:

- Stylus required
- Stylus not required
- Compensated shank

For standard T-Probe usage, you MUST select 'stylus required'.

Leica AT401

Prerequisites

The Leica AT401 connects via a wired Ethernet connection or a wireless connection. In either case, the IP configuration must be correct before the PC and instrument can communicate. The default Leica AT401 IP address is 192.168.0.1. To learn more about configuring IP address see the IP Address Basics section.

Compensation

The Leica AT401 compensation is performed in the Leica Tracker Pilot application. This is supplied by Leica and should be performed before running the instrument in SA.

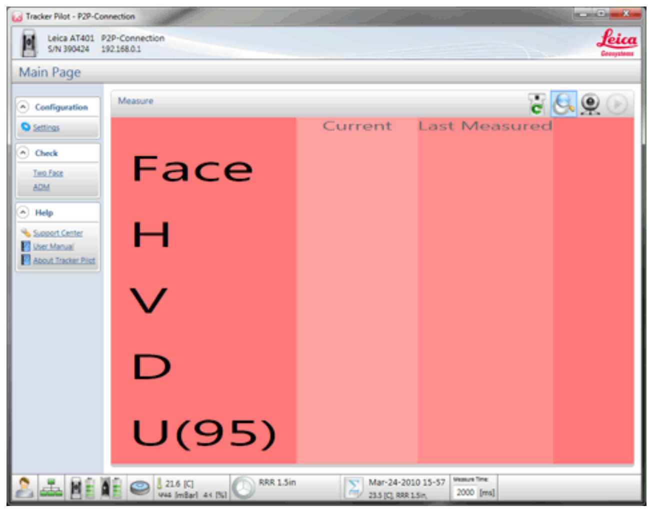


Figure 28-11. Running a compensation routine in TrackerPilot.

Starting the Interface

1. Select **Instrument>Add** and choose the Leica emScon AT401 from the Instrument List (see Figure 28-12 on page 754).

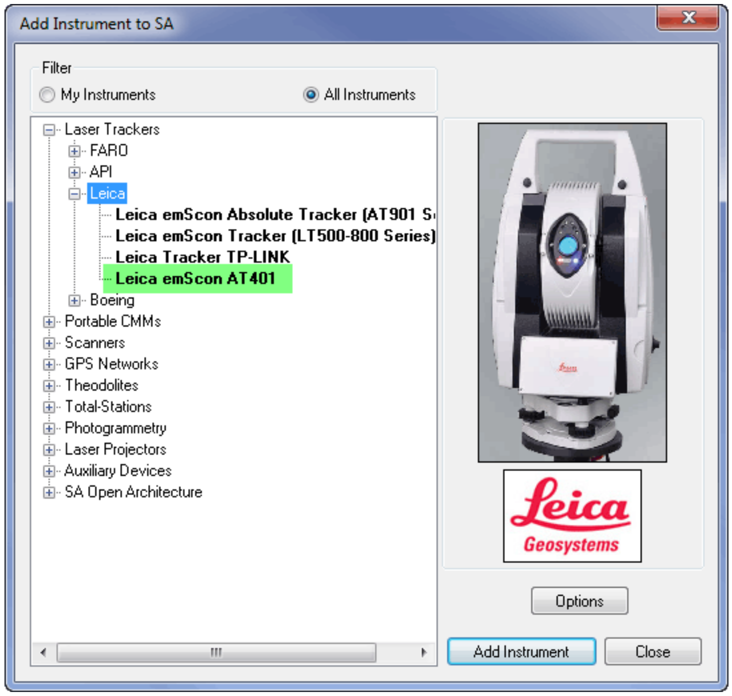


Figure 28-12. Adding an AT401 tracker.

2. Now run the instrument interface module under **Instrument > Run Interface Module** and choose Laser Trackers.

Note: If the Initialize Tracker option is chosen, be sure a reflector is in the tracker's view and at a distance of 1.5 meters or greater.

3. Enter the tracker's IP address and use the Ping button to test the connection if needed (Figure 28-13). Once satisfied, click OK. The next time the interface is started, you can simply click the Run Interface and Connect icon. This will use the last saved settings and automatically connect the instrument.

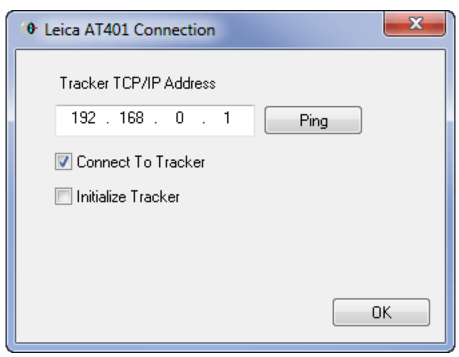


Figure 28-13. The Leica AT401 Connection dialog.

4. The interface is now connected and ready for use. Please refer to the "Measuring With Laser Trackers" chapter for details on the laser tracker interface.

AT401 Settings

To access the AT401's settings, use **Settings > Tracker > General Settings** or press the button. Then press the Tracker Specific button at the bottom. This will expose the AT401's specific options (Figure 28-14).

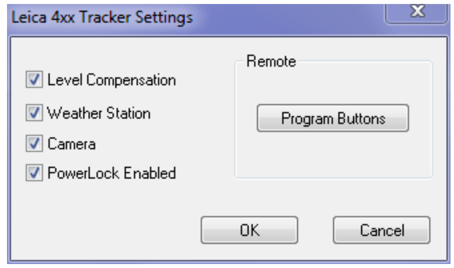


Figure 28-14. The Leica AT401 tracker settings.

The AT401 utilizes a programmable remote (Figure 28-15).

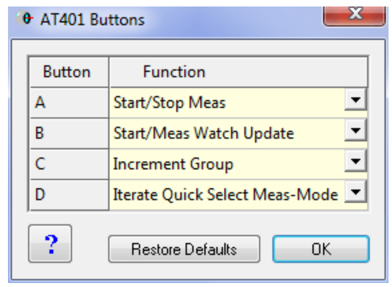


Figure 28-15. Setting the programmable buttons for the AT401.

Battery Status

To view the battery status go to **Utility > Tracker Status** (Figure 28-16).

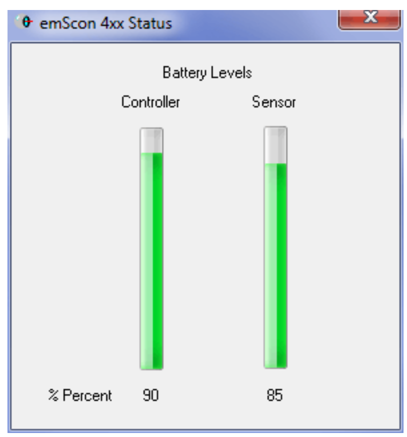


Figure 28-16. The AT401 battery status dialog.

Leica Automation Interface

Hardware Setup

Before starting the AIC driver, it's necessary to ensure that all hardware connections are complete:

- The AT-901 tracker controllers should be connected to the AIC hardware.
- The auxiliary device (T-Mac, T-Probe, external trigger, etc.) should be connected to the AIC hardware.
- The AIC hardware should be directly connected to the computer on which the interface will be run.

Starting the AIC Driver Manually

1. First, add up to four Leica AT-901 trackers to the SA job.

2. From the appropriate SA folder in the Windows Start Menu, navigate to Interfaces and select Leica Automation Interface (Figure 28-17).

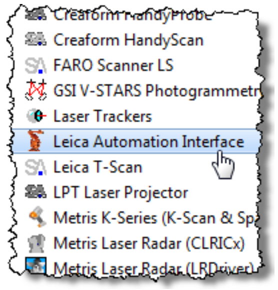


Figure 28-17. Starting the Leica Automation Interface.

3. In the ***Connect*** dialog, select the tracker you'd like to connect to (Figure 28-18).

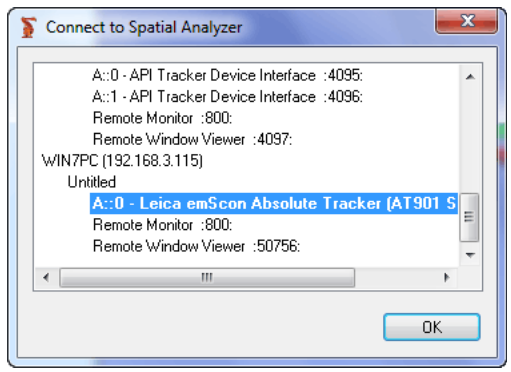


Figure 28-18. Connecting to a tracker.

4. The Leica AIC Driver interface will appear, automatically connect to the AIC, and immediately be ready to use. Note that the AIC Driver will detect the current AI Controller's connection to hardware at the Multiplexer--it is the T-Scan in this case (Figure 28-19).

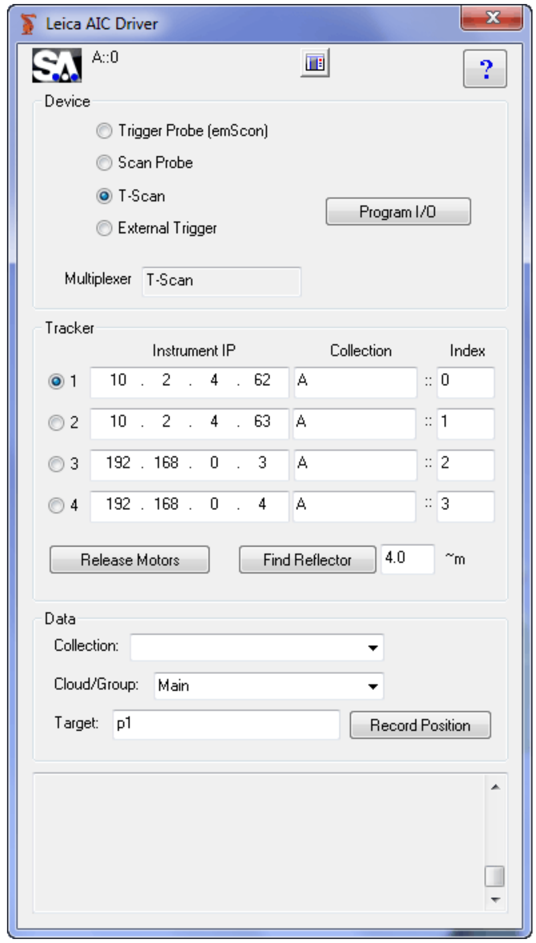


Figure 28-19. The AIC Driver window.

Note: If we'd had a T-Mac connected, the MUX (multiplexer) would show T-Mac and the Device would be Trigger Probe.

Using the Interface

Each tracker in use requires an assigned IP address and collection/index, which indicates which instrument in the SA file is associated with the corresponding hardware. Use the radio button to switch between different trackers.

All settings appropriate to the current device will be automatically set. Measurement parameters can be set via Measurement Plans.

- The Record Position button is used to teach positions for automatically locking on the T-Scan via an MP command. The Collection::Group::Target name is used for the storage of the auto-lock position in SA.
- The Release Motors button will release the motors on the active tracker so that it can be pointed by hand.
- The Find Reflector button will initiate a search for a reflector in order to lock onto the selected device. The distance field next to this button is used to provide the controller with an idea on how far to search for the reflector based on its distance from the tracker.

Running the AIC Driver In Automation Mode

The Program I/O button is used to program the digital I/O signals for Automation Mode. In this mode, the AIC Interface expects to receive signals from the robot, and will send signals to the robot, for handshaking. The Program I/O button allows communication between the AIC interface and the device with which it is working--typically a robot. You can define the meaning for up to 6 input channels coming in from the robot, and up to 3 channels going out to the robot (Figure 28-20).

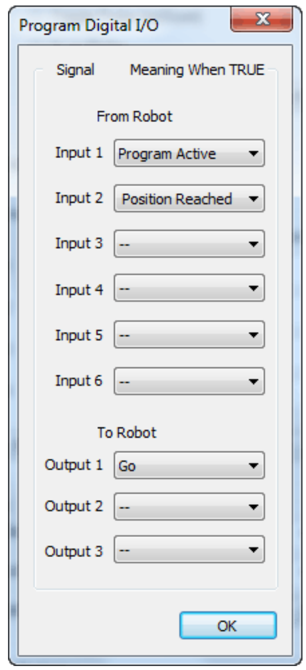


Figure 28-20. Programming the I/O.

A series of **`Instrument Operational Check`** Measurement Plan command strings are available for interacting with the AIC in automation mode. Refer to the "MP Command Reference" document for details.

Portable Arms

API Axxis Arm (6 DOF)

Note: You must have the proper calibration file for the arm (manufacturer-supplied). The file name will contain the arm serial number and will end with `...Arm.tab`. You will be prompted to browse for the path to this file when the interface starts.

1. Download the Baces3D Install zip from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/API/Axxis/.
2. Unzip the file to your hard drive, and follow the instructions in the `Baces3D USB Installation` PDF file.

Running the Arm

1. Select the API Axxis 6DOF arm (Figure 28-21).

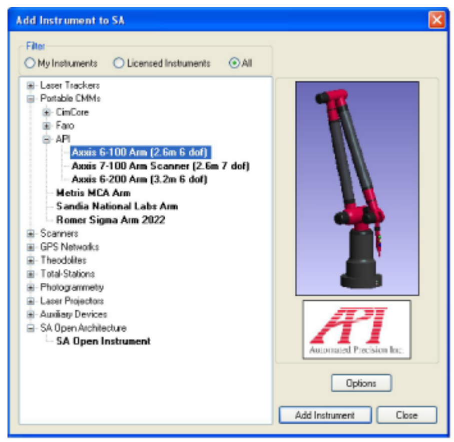


Figure 28-21. Adding an Axxis 6-DOF arm.

2. With the new instrument added, run the arm interface module.
3. You will be asked to choose a communication method and a parameter file (Figure 28-22). This parameter file should be supplied with the arm.

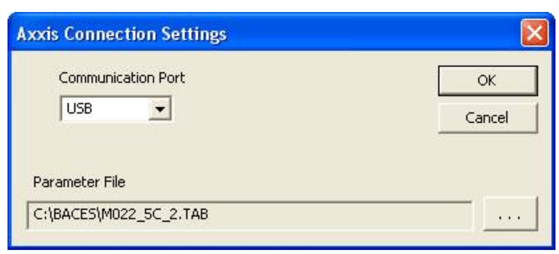


Figure 28-22. The Axxis Connection Settings dialog.

4. Once connection is established the arm must be initialized. Simply exercise the arm until all encoders have a green status (Figure 28-23).

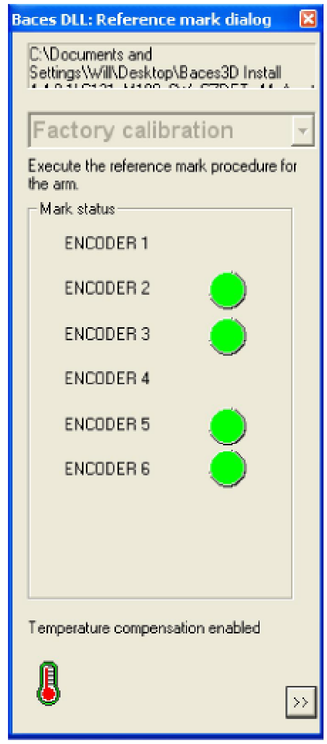


Figure 28-23. Exercising the encoders on the arm.

5. The SA arm interface will now appear. At this point you are ready to perform a point probe calibration. To access the point probe calibration, simply right-click on the Baces3D icon in the system tray (Figure 28-24).

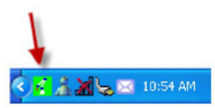


Figure 28-24. The Baces3D icon.

6. Follow the Baces3D calibration dialog (Figure 28-25).

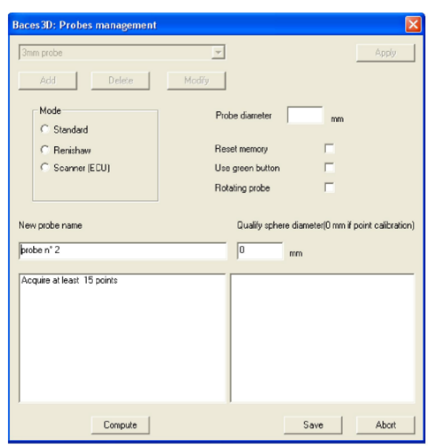


Figure 28-25. The Baces3D calibration dialog.

7. With a satisfactory calibration the arm is ready to measure. Simply use the buttons on the arm to trigger measurements, change measurements, etc. The button configuration is available through the arm User Options. For example, a short press of the blue button will toggle through the measurement

modes. A long press of the blue button will activate a measurement mode.

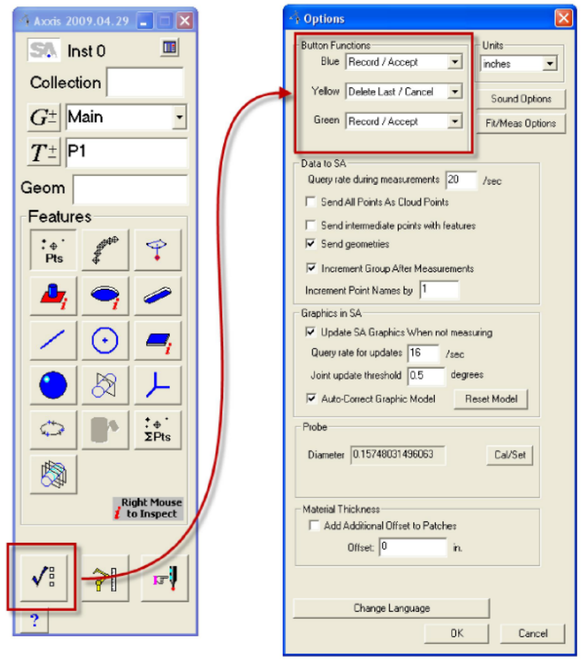


Figure 28-26. Button settings for the arm.

API Axxis Scanner Arm (7 DOF)

Prerequisites

> **Note:** You must have the proper calibration file for the arm (manufacturer-supplied). The file name will contain the arm serial number and will end with ...Arm.tab. You will be prompted to browse for the path to this file when the interface starts.

> **Note:** You must have the proper calibration file for the scanner. The Kreon install will prompt you for the file. It will be named [scanner serial number].cal.

> **Note:** You must plug in the USB license key for the scanner before use.

1. Download `PluginSpatialAnalyzerXXXX.zip` from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/Scanners/Kreon%20Scanner%20Plugin/.
2. Unzip the file to your hard drive, and run `Setup.exe`. The installer will guide you through the process of installing the scanner and the HASP dongle driver.

Network Settings for Running the Scanner

- The scanner's ip address is set to 192.168.244.43 and 255.255.255.0 (subnet mask).
- Your PC's network card MUST have the following address: 192.168.244.x, where x is any number between 1 and 255, except for 43. 255.255.255.0 (subnet mask).
- The above MUST be the ONLY address set on your network card.
- Your network card MUST be configured for 'Speed & Duplex' = '10Mbps/Full Duplex'
- You can find this setting in (e.g. for Windows XP) in Device Manager: **Control Panel>System>Hardware>Device Manager**. Select Network adapter (the one you are configuring), and go to the Advanced tab.
- If you are unable to make this setting change, you will need to consult IT, or someone who can make the change.

> **Note:** The consequence of the above requirements is of course that you will be unable to run any other device which uses network communications on the computer configured as required, UNLESS the computer has more than one network card.

Running the Instrument in SA

1. Add an instrument using the menu item **Instrument>Add Instrument** or use the icon.
2. Select the API Axxis 7DOF arm (Figure 28-27).

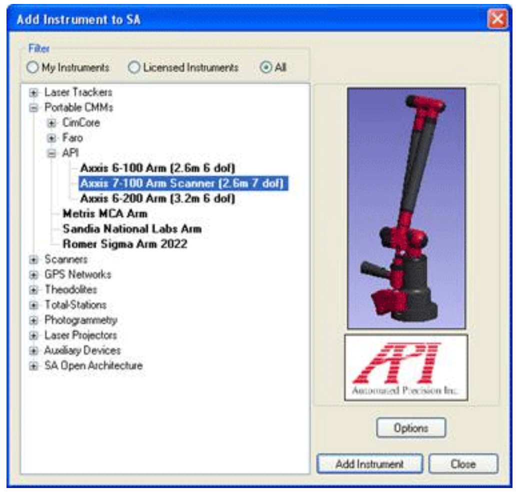


Figure 28-27. Adding a 7-DOF Axxis arm.

3. With the new instrument added run the arm interface module. Simply click the icon.

4. You will be asked to choose a communication method and a parameter file. This parameter file should be supplied with the arm. Be sure to select the scanner parameter file (Figure 28-28).

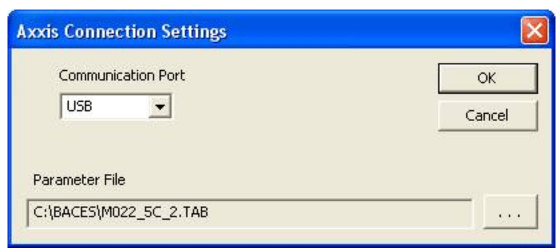


Figure 28-28. The Axxis connection settings dialog.

5. Once the connection is established the arm must be initialized. Simply exercise the arm until all encoders have a green status (Figure 28-29).

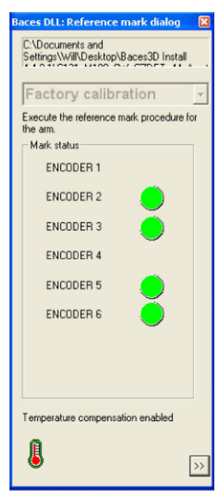


Figure 28-29. Exercising the encoders on the arm.

6. The SA arm interface will now appear. At this point you are ready to perform a point probe and scanner calibration. To ac-

cess the calibration routine press the calibration button in the arm interface. Perform a scanner calibration and accept the results (Figure 28-30).

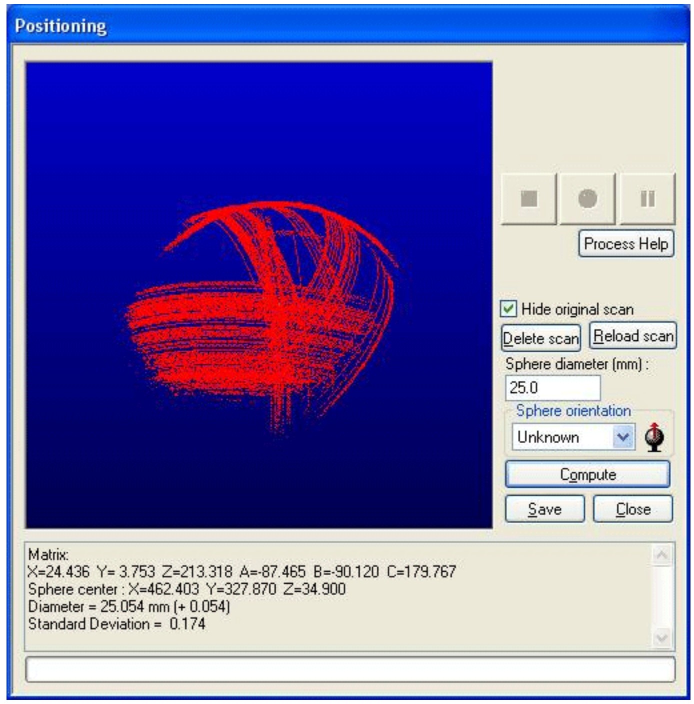


Figure 28-30. Calibrating the arm.

7. If hard probing will be used in conjunction with the scanner be sure to have a valid 6-DOF calibration as the 7-DOF calibration is dependent on the results of the 6-DOF calibration. To define the 7-DOF hard probe press the options button in the arm interface. Now press the Cal/Set button to access the calibration dialog.

8. Follow the instructions for defining the point probe (Figure 28-31).

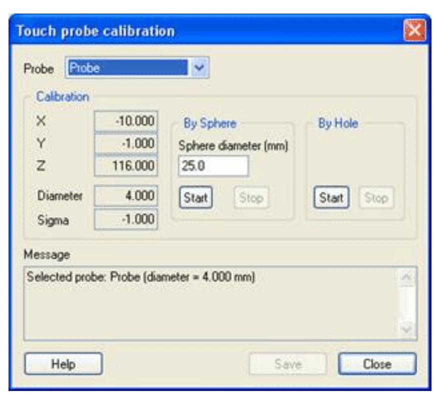


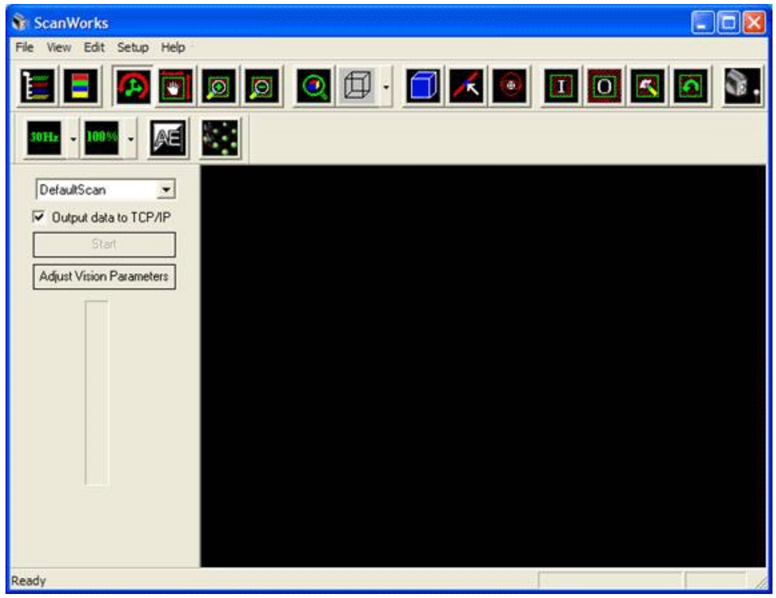
Figure 28-31. Calibrating the touch probe.

9. With a satisfactory calibration the arm is ready to measure. Simply use the buttons on the arm to trigger measurements, change measurements, etc.

Perceptron Contour Scanner (ScanWorks v4)

The Perceptron interface requires that you install and run Perceptron's ScanWorks application. Make certain the scanner is connected to the arm, and the power to the scanner is on. Run ScanWorks. ScanWorks version 4.4a.082 is shown in Figure 28-32.

Figure 28-32.



Note the **Output data to TCP/IP** setting is checked. This MUST be checked in order for the interface to receive data. In older versions of ScanWorks, you must go to the Setup menu, and select User Options. There, you must check **Output data to TCP/IP**, and select **Inches** as the units. In ScanWorks v. 4.4 and later, the SA interface sets the units in ScanWorks automatically.

1. Minimize ScanWorks to preserve screen space. In the SA Interface, select to start the scanner measure mode. As with any measure mode, this is normally done by pressing the Record/Accept button for a short duration to iterate through the measure modes (buttons will appear pressed as you iterate through.) Or you can press the Delete Last/Cancel button to iterate back the other way through the measure modes. Then, a long button press on the Record/Accept button will start the selected measure mode. The scanner laser line will turn on, and the ***Scanner Measurement*** dialog will appear (Figure 28-33).

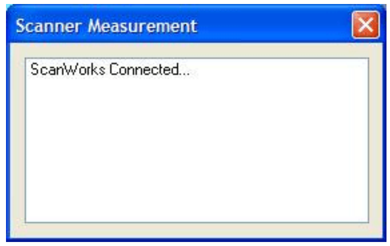


Figure 28-33.

2. If the ScanWorks window is visible, you'll notice most controls are greyed out in ScanWorks, denoting that control has been taken by the SA CMM Arm interface (Figure 28-34).

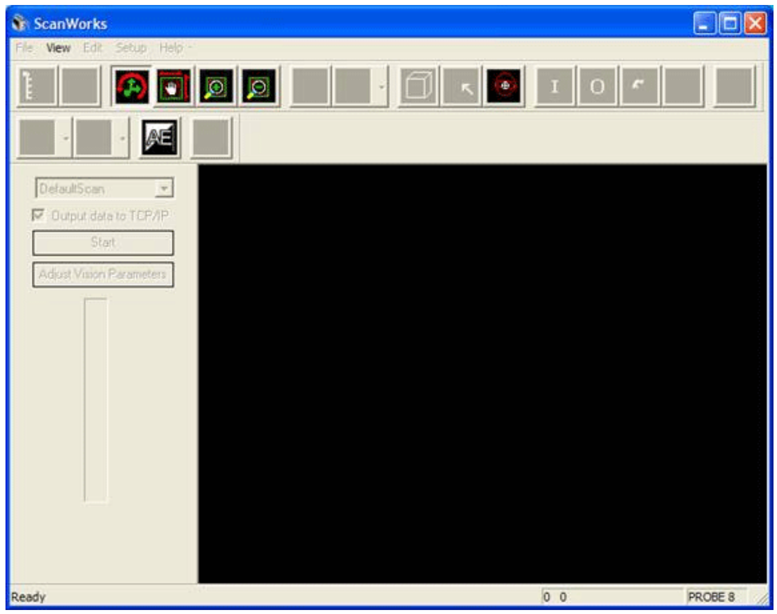


Figure 28-34. Grayed-out controls in the ScanWorks interface.

3. You'll also notice that ScanWorks will pop a scanner distance indicator bar (see Figure 28-35 on page 767) to help determine whether you are a good distance from the surface to be scanned:

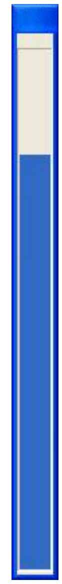


Figure 28-35. Scanner distance guide.

4. You'll also hear a variable frequency beep giving an audible scanner-to-part distance indication. Both these features can be toggled in the ScanWorks Setup menu under User Options. When you reach a good starting position, simply press and release the scan start/stop button on the arm (this is determined by ScanWorks. For example, on a 7-DOF arm with a trigger style handle, the start scan button will be the center, or trigger, button.).

5. During the scan, Scanner Measurement dialog will indicate that a scan has started (Figure 28-36), and will display progress indicating number of cloud data points and Cloud Name being sent to SA.

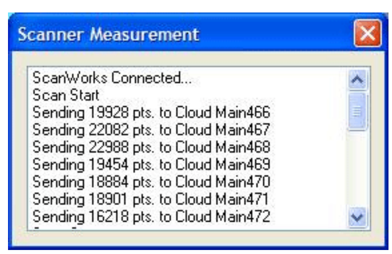


Figure 28-36. Scanning progress is displayed in the Scanner Measurement window.

6. End a scan by again pressing and releasing the arm scan button. When this happens, the Scanner Measurement dialog will indicate the scan stop, and will send and report any left over buffered cloud points.

7. Perform as many scans as are desired in this way. To end the Scanner Measurement Mode, you must press the 'end scan session' button on the arm. Like the start/stop scan button, this is determined by ScanWorks. For example, on a 7-DOF arm with a trigger-style handle, the end scan session button will be either of the outside buttons on the trigger handle.

Note that if you try to simply close the ***Scanner Measurement*** dialog with the "X" button, it will not close, and the dialog will instruct you to end the scan session from the arm (Figure 28-37).

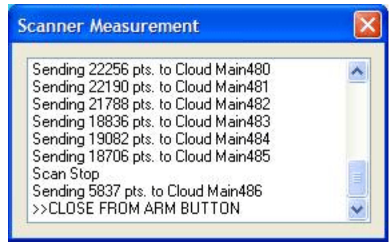


Figure 28-37. The scan session must be closed via the arm button.

8. Once you do press the appropriate arm button to end the ScanWorks scan session, the Scanner Measurement will report that the ScanWorks connection is closing, and will close once ScanWorks is disconnected. If the ScanWorks window is visible, you'll notice that its controls are re-enabled, indicating that scanner control has been passed back to it.

Scanner Settings

You can edit the scan data handling and naming in the ***Arm Settings*** dialog. Press to edit arm settings. Note that if you still have the Perceptron scanner connected, you will briefly see an error message from CimCore (Figure 28-38).

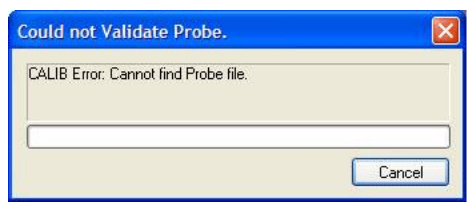


Figure 28-38. Error message.

This is expected, since the CimCore Validate Probe function is called, and CimCore has no knowledge of the Contour Scanner as a valid probe definition. Indeed, this error will be generated any time a measurement other than scanning is attempted while the scanner is connected to the arm. The arm ***Options*** dialog will appear (Figure 28-39).

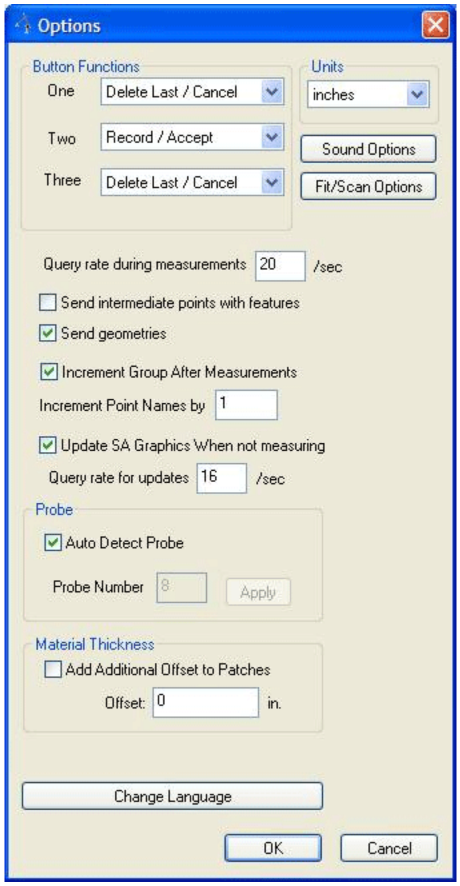


Figure 28-39. Arm options.

Press the Fit/Scan Options button to show the ***Fit/Scan Options*** dialog (Figure 28-40).

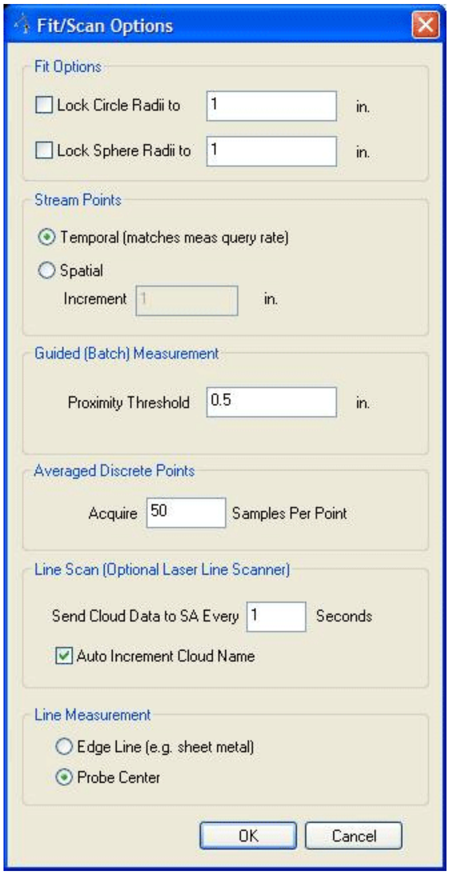


Figure 28-40. The Fit/Scan options.

Under *Line Scan (Optional Laser Line Scanner)*, you can control the interval at which scan clouds are sent to SA (Perceptron Contour can acquire up to ~23,000 Pts/sec), by editing the field labeled *Send Cloud Data to SA Every ___ Seconds*. The Cloud Name is taken from the current Group Name in the main dialog. If **Auto Increment Cloud Name** is checked, then the name will automatically increment after each cloud data packet is sent to SA. The Scanner Measurement dialog is modeless, meaning the Group Name (and therefore Cloud Name) in the main dialog can be edited any time, even during a scan session. The scanner measurement progress shown earlier in this section was performed with the settings as shown here. To apply settings, simply press OK.

Scanner Calibration

The ScanWorks User Guide includes detailed instructions for calibrating the scanner to the arm. Read the instructions through before attempting to calibrate the scanner. If you need to change computers, but will be using the same scanner and arm, you can transfer the existing calibration to the new computer. The calibration file is stored in the ScanWorks install on the application PC. For example, if you used the default location for the ScanWorks install, the calibration file will be stored in `C:\Program Files\Perceptron\ScanWorks\CimCore\data\`.

The name of the file is `xform_[scanner serial number].dat` where *[scanner serial number]* will match the number printed directly on the scanner. Simply copy this file to the same location in the ScanWorks install folder on the new PC, and you are ready to run using the existing scanner calibration.

Perceptron Contour Scanner (ScanWorks v5)

Running the Perceptron Scanner requires Scanworks (version 5.21.116 or later) to be installed and running (Figure 28-41). Please refer to Scanworks documentation for details and settings. The Scanworks interface is controlled by a console application. The console can be set to start on system startup or manually started when needed.

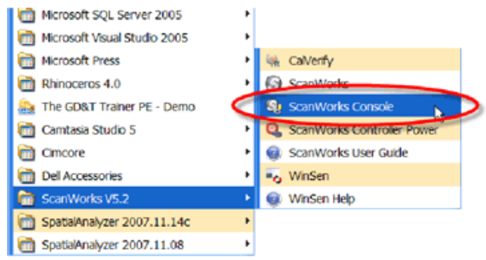


Figure 28-41. Starting Scanworks.

The below options in the console (Figure 28-42) must be set accordingly to allow SA to communicate with the arm and scanner. If settings are not set correctly communication errors will occur.

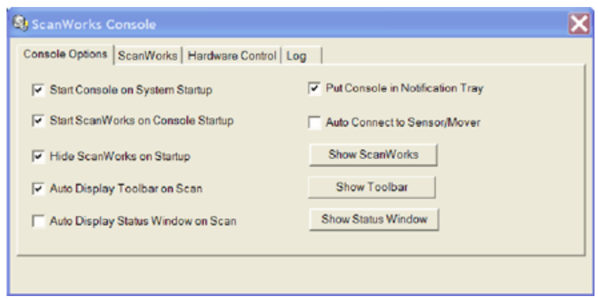


Figure 28-42. ScanWorks Console.

Once the console options have been set, now operation of the scanner can begin. Open SA and add your instrument. Start the instrument interface. Best practice is to have the console running before you start the instrument interface but it is not required. The console must be running before the scanning operation is started.

In the instrument interface there are several settings that can affect the outputs of the contour scanner. These settings can be optimized to best fit your application and the capacity of your computer.

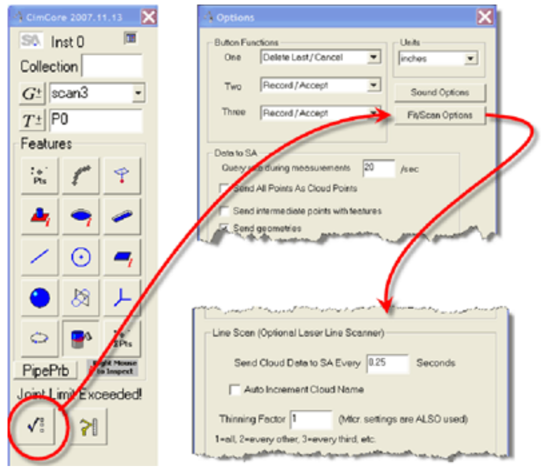


Figure 28-43. Setting the Laser Line Scan options.

Once Line Scan settings are configured (Figure 28-43), simply press to start scanning.

You will see a status window notifying the connection between SA and Scanworks (Figure 28-44).

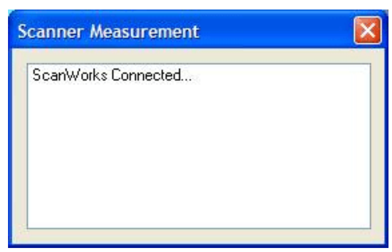


Figure 28-44. The connection is established.

You will notice the below toolbar display when connected to the scanner (Figure 28-45). This will allow access to settings such as scan rate, scan density and exposure settings.

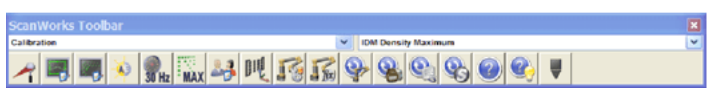


Figure 28-45. The ScanWorks toolbar.

Now press the Red button to start collecting data. The status window will show data being sent to SA (Figure 28-46).

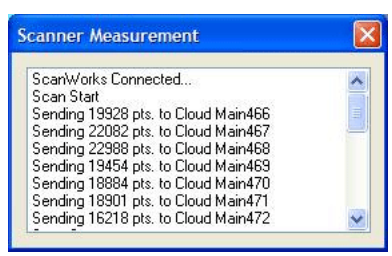


Figure 28-46. Data collection.

To stop data collection, press the Red Button again. To exit the scanning operation press one of the White Buttons and SA will disconnect from Scanworks.

Troubleshooting

If the error in Figure 28-47 occurs try the following:

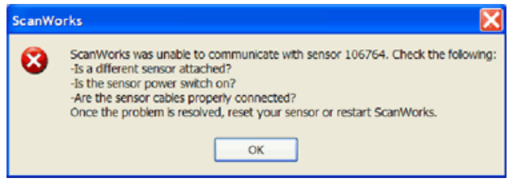


Figure 28-47. ScanWorks communication error.

1. Check to be sure sensor is attached and all connections are correct.
2. Go to the Scanworks Console and close Scanworks (Figure 28-48).

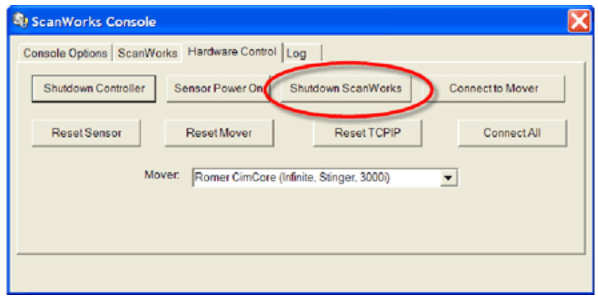


Figure 28-48. Shutting down ScanWorks.

3. Now restart Scanworks and try the scanning operation again (Figure 28-49).

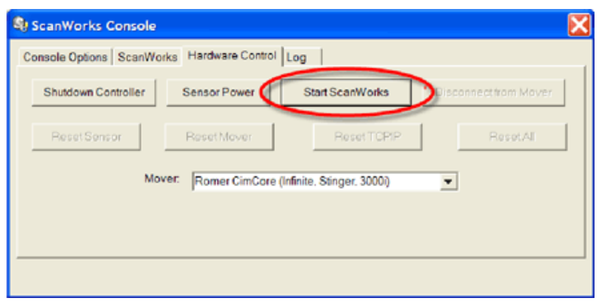


Figure 28-49. Restarting ScanWorks.

Romer/Cimcore Arms

Note: If you are not running the latest version of SA, please check the readme.txt file for the corresponding driver version.

Before adding and running a Cimcore arm in SA, the appropriate instrument driver must be installed. Locate and install the latest WinRDS driver from the SA Driver Download area: ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/Cimcore/

Once downloaded, with the instrument disconnected, install the instrument driver as instructed. Once installed, reboot the PC. Now connect the arm and turn the power on. If the arm uses a USB connection, follow the below steps. If not, proceed to Running the Cimcore Arm in SpatialAnalyzer below.

When connecting via USB for the first time Windows will display the dialog in Figure 28-50.

Figure 28-50. The Hardware Update wizard.

Select **Yes, this time only** and press Next. Follow the prompts until the Hardware Update is complete.

If this dialog did not appear and the drivers have not been associated to the hardware, right click on "My Computer" and select "Properties". Under the properties dialog select the "Hardware" tab and then select "Device Manager".

Under "Device Manager" (Figure 28-51) locate the Cimcore USB Arm, then right click and select "Update Drivers". This will display the "Hardware Update Wizard" as discussed above. Follow the same directions as above.

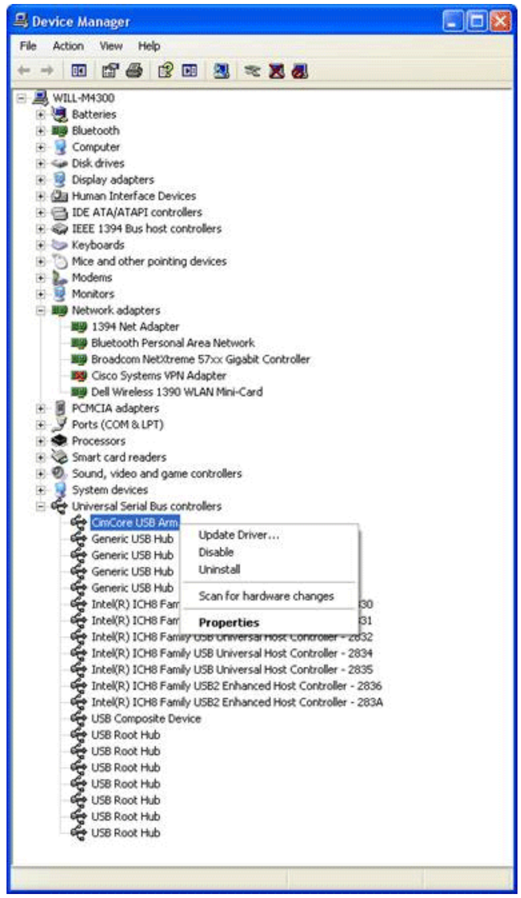


Figure 28-51. Device Manager.

Now that drivers are installed and the hardware is updated, test the connection by running WinRDS. **Start>Programs>Cimcore>WinRDS>Cimcore Arm Utilities**.

If the connection is working correctly, WinRDS will initialize the arm. Move the arm to until the below dialog is satisfied. Now that the connection has been tested close WinRDS.

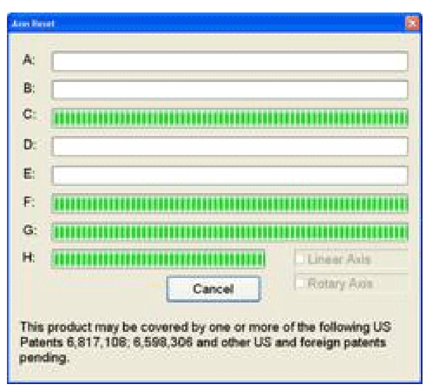


Figure 28-52. Exercising the encoders on the arm.

Running the Cimcore Arm in SpatialAnalyzer

Connecting the arm for the first time in SA requires one extra step than usual. First add the desired Cimcore Arm by selecting **Instrument>Add**. Then select **Instrument>Run Interface Module**. Select the Portable CMM interface (Figure 28-53).

Figure 28-53. Starting the interface.

Select the instrument (Figure 28-54).

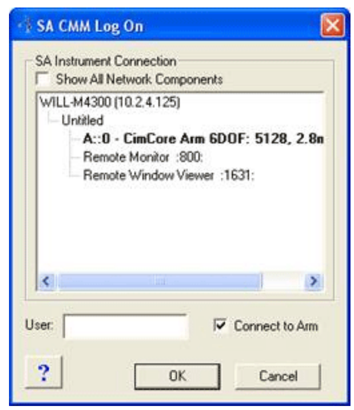


Figure 28-54. Connecting to the instrument.

Once selected, a prompt will display asking for communication method (Figure 28-55). This step will persist the next time the interface is connected.

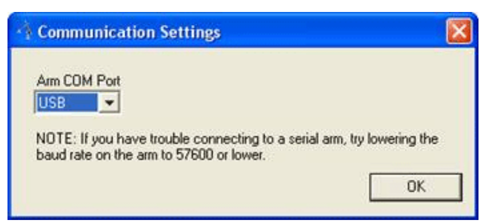


Figure 28-55. Choosing the communication method.

The next time connection is required, simply use the icon to run and connect the interface module. The arm interface will display once connection is complete.

Romer MultiGage

Prerequisites

Before adding and running a Romer Multi-Gage in SA, the RDS drivers must first be installed.

Download the RDS `InstallerV3.1.1.exe` (or later) from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/Hexagon/ or install from the Instrument Drivers Combined Installer at http://kinematics.com/products/spatialanalyzer/spatialanalyzer-downloads.html

1. Run the RDS Installer and follow the prompts. Once installed, plug in the arm via the USB cable and verify Windows has properly recognized the device and installed the drivers properly.
2. RDS will always be active in the system tray (Figure 28-56) and is a great tool for checking arm status and connectivity (Figure 28-57).

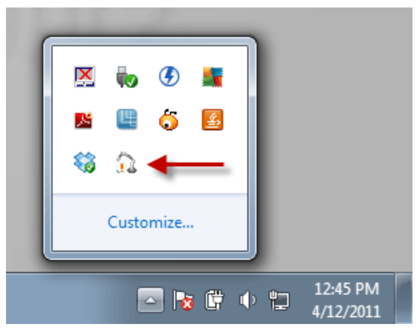


Figure 28-56. RDS in the System Tray.

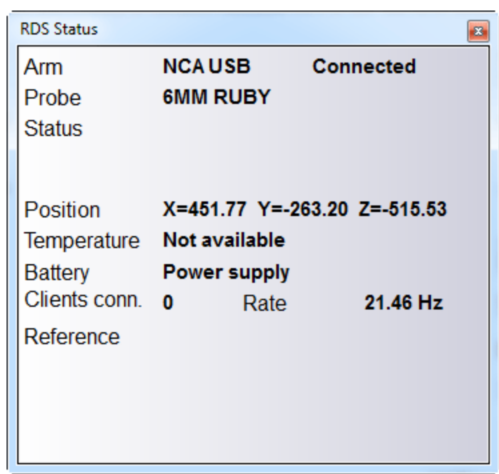


Figure 28-57. The RDS status.

Running the MultiGage

1. Add a MultiGage to SA via **Instrument>Add** or using the icon. Select the Romer MultiGage and click Add Instrument (Figure 28-58).

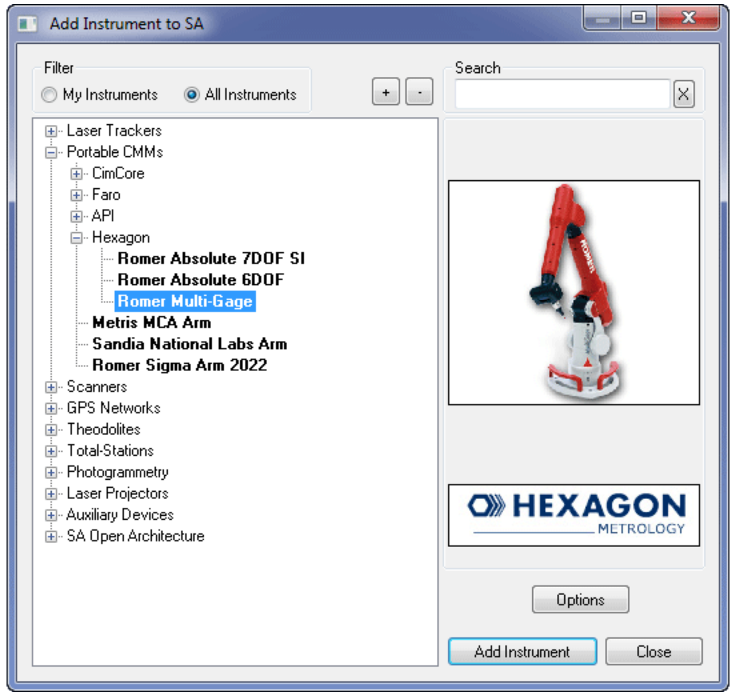


Figure 28-58. Selecting the instrument to add.

2. Run the instrument interface via **Instrument>Run Interface Module and Connect** or using the icon.

Troubleshooting

If arm connection problems occur, be sure to use the RDS application in the system tray for current status (Figure 28-59).

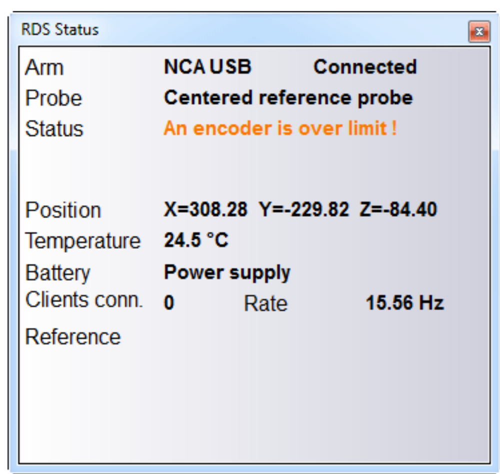


Figure 28-59. The RDS status displaying an error.

If further information or settings are needed access the RDS Control Panel by right clicking the RDS icon in the system tray and select RDS Control Panel.

Romer Absolute

Prerequisites

Before adding and running a Romer Absolute Arm in SA, the RDS drivers must first be installed.

Download the `RDS InstallerV3.1.1.exe` (or later) from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/Hexagon/ or install from the Instrument Drivers combined installer at http://kinematics.com/products/spatialanalyzer/spatialanalyzer-downloads.html

1. Run the RDS Installer and follow the prompts. Once installed, plug in the arm via the USB cable and verify Windows has properly recognized the device and installed the drivers properly.
2. RDS will always be active in the system tray (Figure 28-60) and is a great tool for checking arm status and connectivity (Figure 28-61).

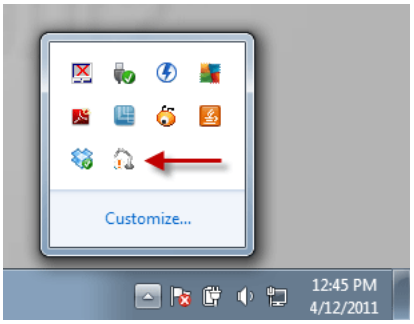


Figure 28-60. RDS in the System Tray.

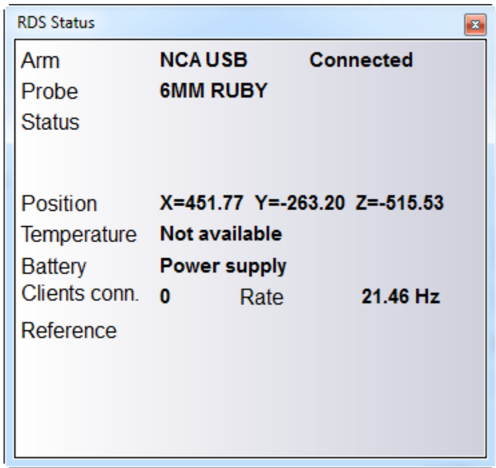


Figure 28-61. The RDS status window.

3. If the arm is an Absolute Arm 7DOF SI (integrated scanner) you will need to ensure the computer's IP address is configured correctly to communicate with the Scanner IP address of 192.168.167.200. Make sure the computer IP address is 192.168.167.XXX with the last digits being between 0-255 but not 200. Example, 192.168.167.100 for the computer. For

more information on changing IP addresses please consult the IP Address Basics section.

Running the Absolute Arm

1. Add an AbsoluteArm to SA via **Instrument>Add** or using the icon (Figure 28-62). Select the respective Absolute Arm and click Add Instrument.

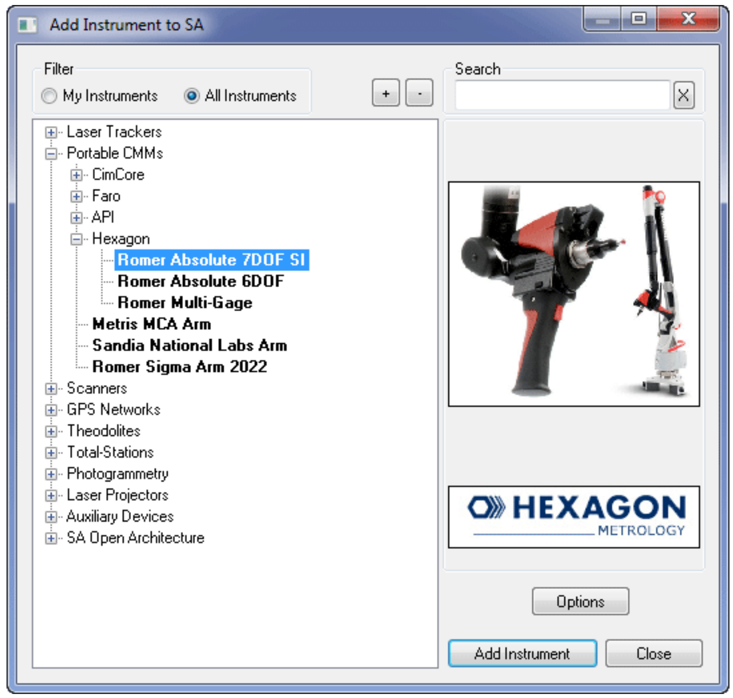


Figure 28-62. Adding the Absolute arm to SA.

2. Run the instrument interface via **Instrument>Run Interface Module and Connect** or using the icon.

Scanning with the Absolute Arm (SI)

Note: When using the scanner make sure the probe is removed. The scanner will not connect with the probe attached.

Note: If the probe is removed and the RDS Scanning window will not appear, verify the Ethernet connection and the IP address configuration.

Scanning with a Absolute Arm 7-DOF SI is as simple as sliding a switch. Slide the switch on the back of the grip to the ON (upper) position and the scanner will be activated. There is no need to use the icon as it will notify you to slide the switch to the ON position. The interface in Figure 28-63 will appear when the scanner is successfully connected.

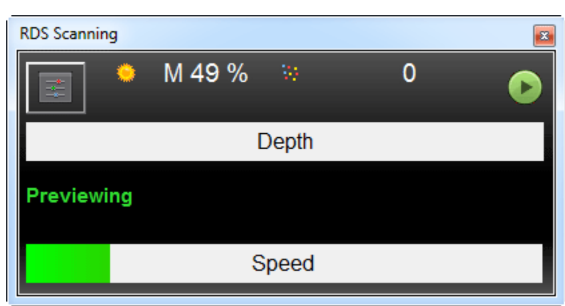


Figure 28-63. The RDS Scanning interface.

Troubleshooting

If arm connection problems occur, be sure to use the RDS application in the system tray for current status.

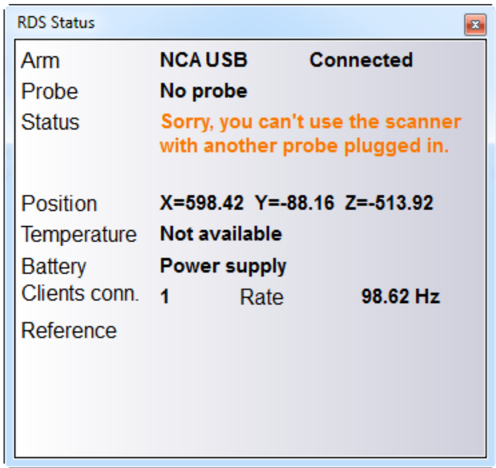


Figure 28-64. The RDS Status window displaying an error.

If further information or settings are needed access the RDS Control Panel by right clicking the RDS icon in the system tray and select RDS Control Panel.

Romer Absolute (SE) with Perceptron ScanWorks v5 Shark

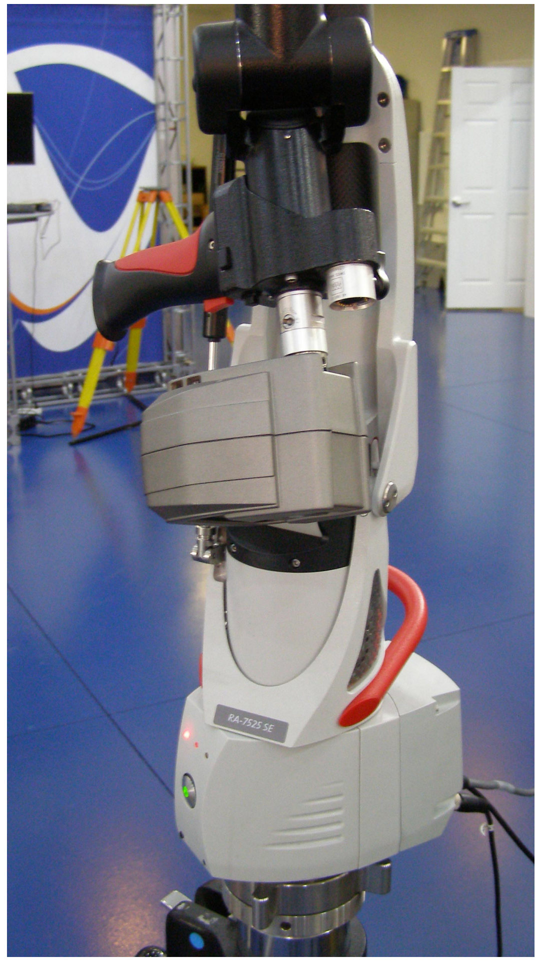


Figure 28-65. The Romer Absolute arm with Perceptron ScanWorks v5 Shark.

Requirements

- A version of SA newer than SA 2011.10.20 is required.
- RDS v3.2 is required and must be installed.
- ScanWorks v5.50.19 or later is required. Refer to the ScanWorks documentation for installation instructions. If running on a 64-bit operating system, be sure to use the 64-bit version of ScanWorks.

Installing RDS v3.2

1. Obtain the installer for RDS v3.2 from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/Hexagon/ or use the combined instrument drivers installer found at ftp://ftp.kinematics.com/pub/SA/Install/SA%20Instrument%20Mfg%20Drivers.exe.
2. Double-click the executable file to begin the installer, and follow the on-screen instructions.

Setting Up the Scanner

1. Ensure that the correct versions of RDS and ScanWorks are installed on the computer as indicated above.

2. Verify that all connections have been made and that the scanner controller (and its sensor power switch) are powered off.

3. On the Absolute arm's handle, ensure the scanner switch is in the **Off** position.

4. Start ScanWorks Console (Figure 28-66), and set it up as depicted in the following image. While these settings are not all required, if settings differ you may experience communication problems.

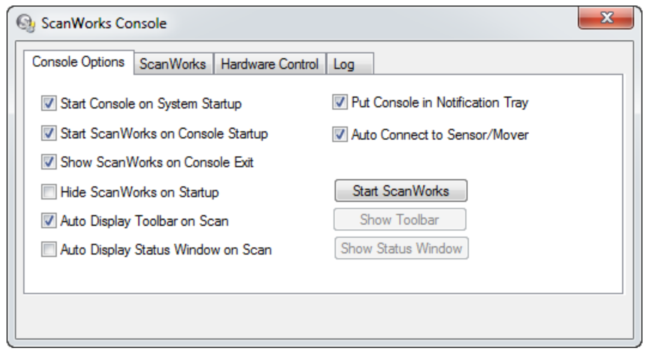


Figure 28-66. ScanWorks console.

5. Open the ***RDS Control Panel*** (Figure 28-67) and go to the ***Probe*** tab. If there are any entries of type **Scanner** in the list, delete them by selecting them and pressing **Delete**. (This is required if you ever change from one scanner to another).

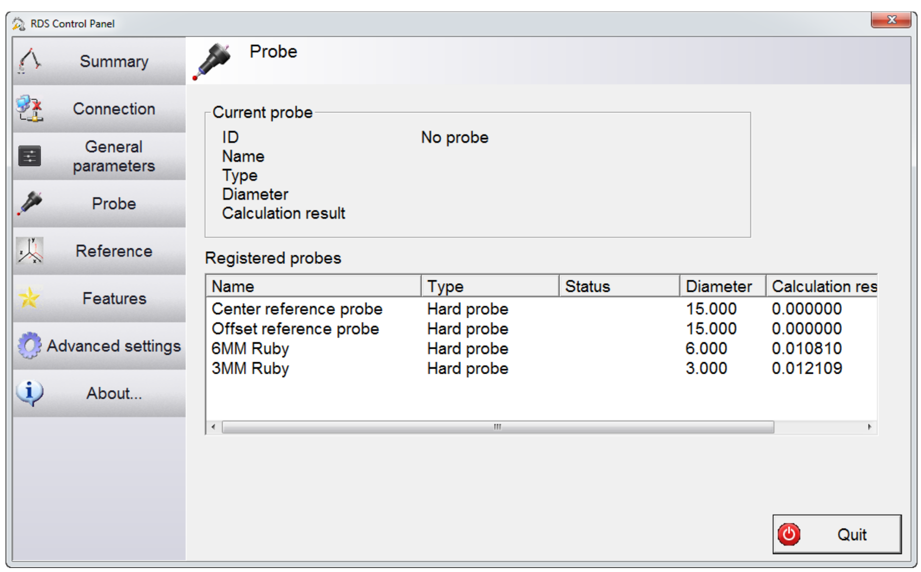


Figure 28-67. ScanWorks Control Panel.

6. If the computer is on a network with a DHCP server, the scanner controller will get an IP address from the server, and you can leave your network settings as they are. If not, set the computer's IP address to `192.168.19.xx`, where xx is any

Note: If your scanner controller has been set to any IP address other than the default, you may need to use a different IP address. Contact Perceptron or Hexagon Metrology for details.

number other than 13.

7. Power on the scanner controller. On the Absolute arm's handle, turn the scanner switch to the **On** position.

8. RDS should automatically detect and identify the scanner. If not, you will be presented with a dialog asking you to select the probe type (Figure 28-68). Choose **External scanner**, then click OK.

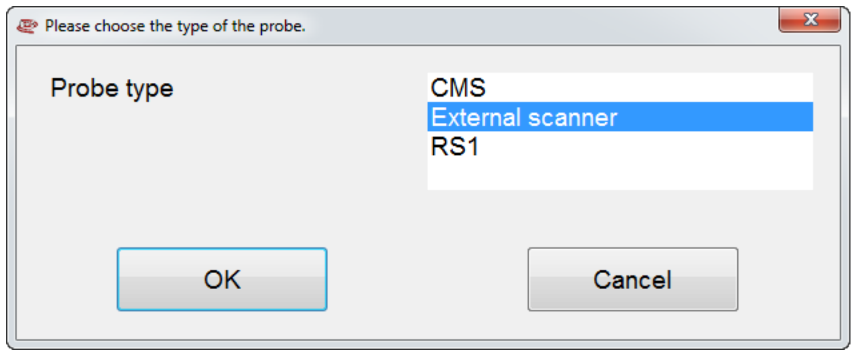


Figure 28-68. Selecting the probe type.

9. Calibrate the scanner in accordance with the manufacturer's guidance.

10. In SA, add the appropriate Absolute SE arm, and start up the instrument interface (Figure 28-69).

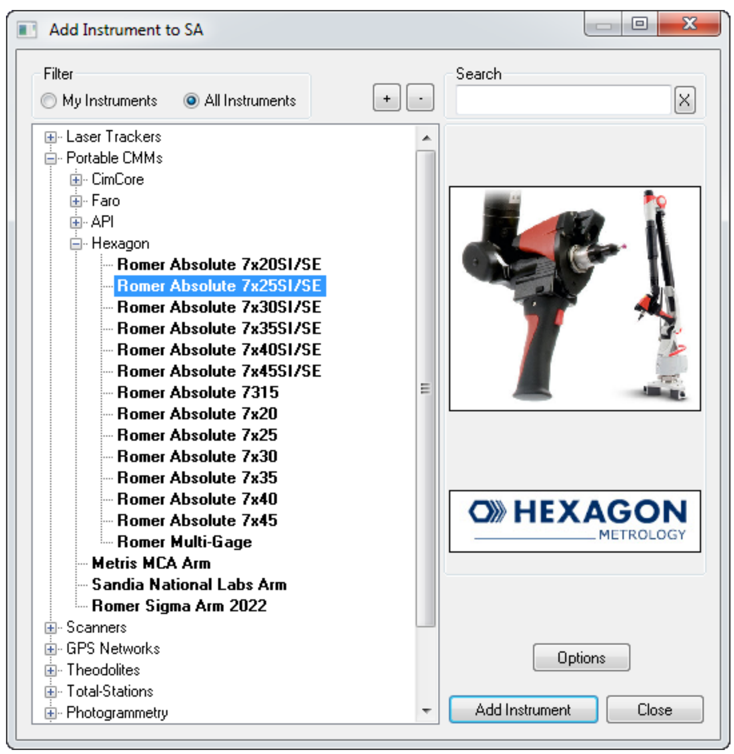


Figure 28-69. Adding the appropriate arm.

11. Any time the scan switch on the Absolute arm's handle is in the **On** position, pressing the trigger button will initiate scanning. Press the side button to stop scanning. When finished scanning, move the scanner selector switch on the Absolute

arm's handle to the **Off** position.

12. To change to a probe tip, turn off the Perceptron controller *Sensor Power* switch, remove the scanner, and attach the probe. To change back to the scanner, remove the probe, attach the scanner, and turn scanner controller's *Sensor Power* switch back on. The current active probe is displayed on the SA Arm Interface main window.

Romer Absolute (SE) with Hexagon CMS Scanner

Figure 28-70. Romer Absolute arm with Hexagon CMS Scanner.

Requirements

- A version of SA newer than SA 2011.10.20 is required.
- RDS v3.2 is required and must be installed.

Installing RDS v3.2

1. Obtain the installer for RDS v3.2 from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/PCMM%20Arms/Hexagon/ or use the combined instrument drivers installer found at ftp://ftp.kinematics.com/pub/SA/Install/SA%20Instrument%20Mfg%20Drivers.exe.
2. Double-click the executable file to begin the installer, and follow the on-screen instructions.

Setting Up the Scanner

1. Ensure that the correct versions of RDS is installed on the computer as indicated above.
2. Verify that all connections have been made and that the scanner controller is powered off.
3. On the Absolute arm's handle, ensure the scanner switch is in the **Off** position.

4. Open the ***RDS Control Panel*** (Figure 28-71) and go to the ***Probe*** tab. If there are any entries of type **Scanner** in the list, delete them by selecting them and pressing **Delete**. (This is required if you ever change from one scanner to another).

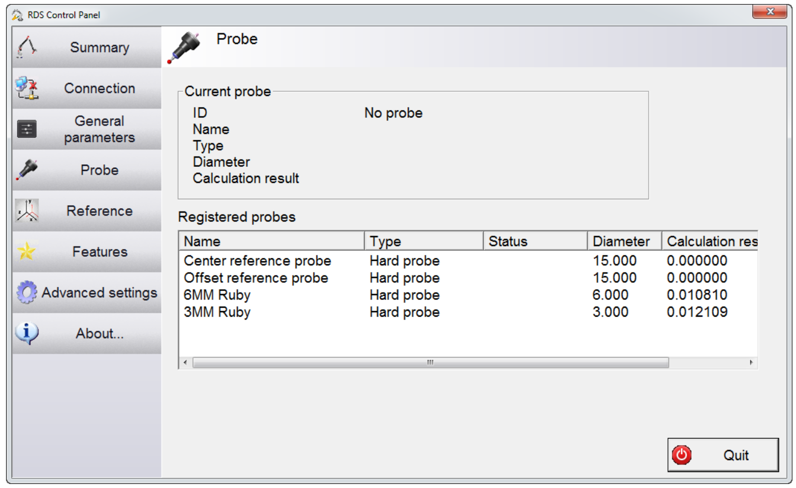


Figure 28-71. The RDS Control Panel.

Note: If your scanner controller has been set to any IP address other than the default, you may need to use a different IP address. Contact Hexagon Metrology for details.

5. Set the computer's IP address to `192.168.150.xx`, where xx is any number other than `100`.

6. Power on the scanner controller. On the Absolute arm's handle, turn the scanner switch to the **On** position.

7. RDS should automatically detect and identify the scanner. If not, you will be presented with a dialog asking you to select the probe type (Figure 28-72). Choose **CMS**, then click OK.

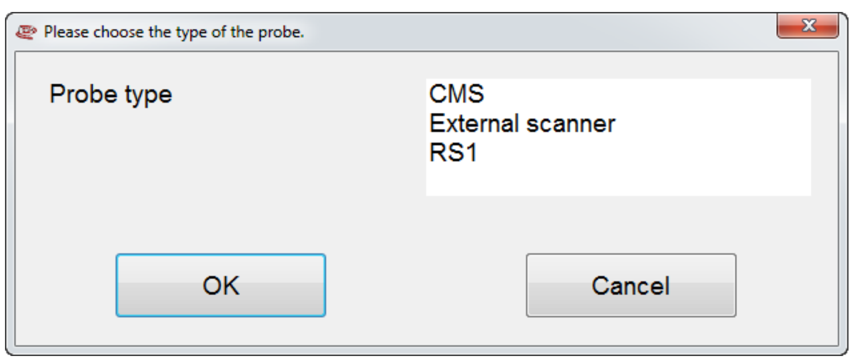


Figure 28-72. Selecting the probe type.

Note: During calibration, the trigger button on the Absolute arm's handle toggles scanning, and the side button confirms the scan for each of the 15 calibration scans.

8. In the ***RDS Control Panel***, click the Calibrate button and calibrate the scanner in accordance with the manufacturer's guidance.

9. Once calibration is complete, return to SA. Add the appropriate Absolute SE arm (Figure 28-73), and start up the instrument interface, as depicted below.

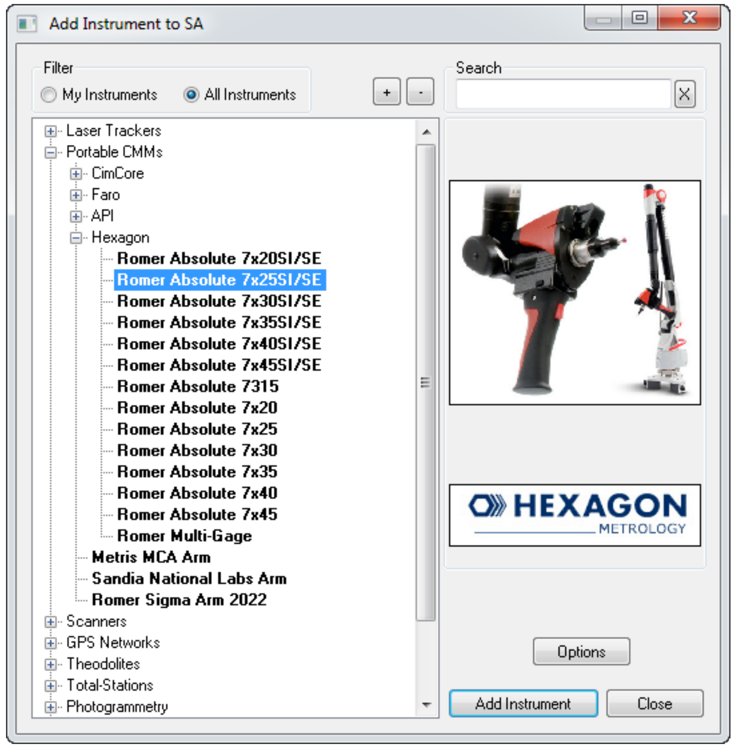


Figure 28-73. Adding the appropriate arm.

10. Any time the scan switch on the Absolute arm's handle is in the **On** position, *press and hold* the trigger button to scan. When finished scanning, move the scanner selector switch on the Absolute arm's handle to the **Off** position.

Total Stations

Leica TCRP, TDRA, and TS-Series

Establishing Bluetooth Connection on PC

Use the Bluetooth wizard installed on the PC to create a new connection (Figure 28-74). Be sure the Leica 1203+ is powered on.

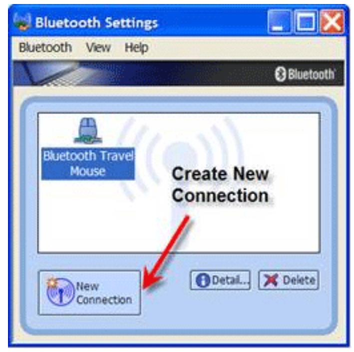


Figure 28-74. The Bluetooth Settings window.

The Bluetooth wizard will search for the device (Figure 28-75).

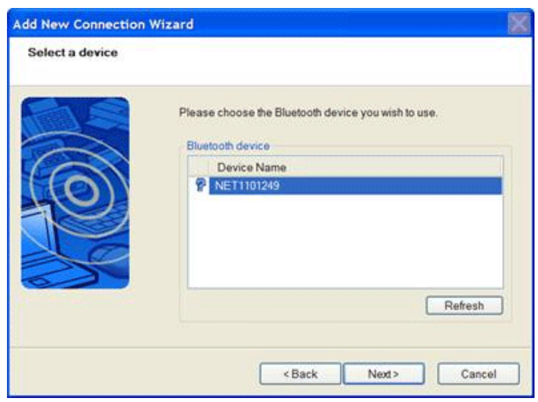


Figure 28-75. Adding the new connection.

The Bluetooth manager will complete the connection and display the COM number being used. Remember this number as it is needed inside of SA to establish connection. If prompted for a password just use "0000".

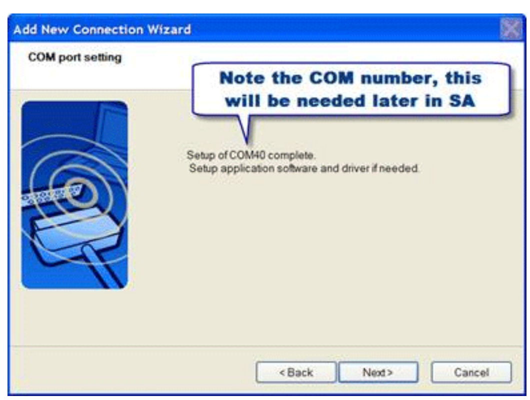


Figure 28-76. This instrument is set up on COM4.

Now the Leica 1203+ is a recognizable connection for Bluetooth. Each time the Leica 1203+ is powered the user must enter the Bluetooth manager and activate the connection (Figure 28-77).

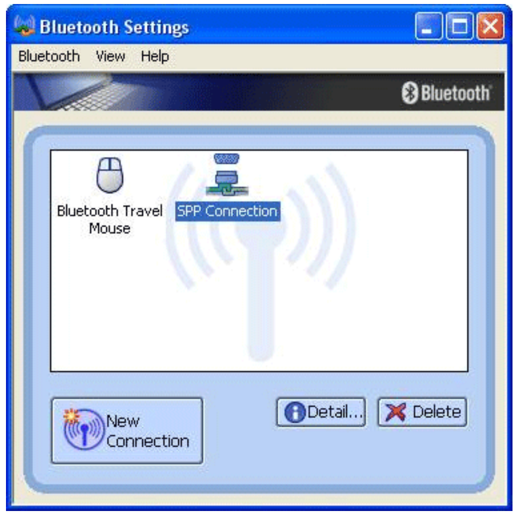


Figure 28-77. Activating the connection.

Starting the Interface

1. Select **Instrument>Add** and choose the Leica Total Station (TDRA6000, TS3X, T120x) from the Instrument List.
2. Now run the instrument interface module under **Instrument>Run Interface Module** and choose **Theodolite Manager**.
3. Select New Setup and then Add (Figure 28-78).

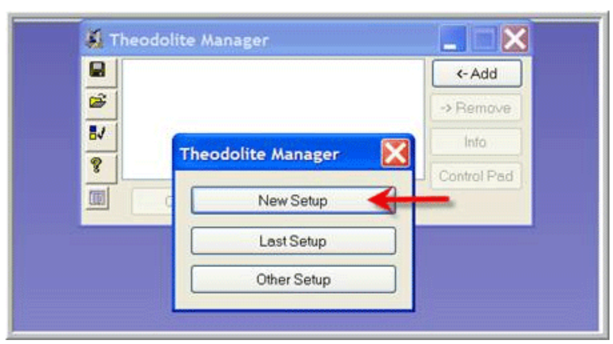


Figure 28-78. Creating a new setup.

4. Select the instrument type and the COM Port. Then select the instrument in the available SpatialAnalyzer list and click Connect.

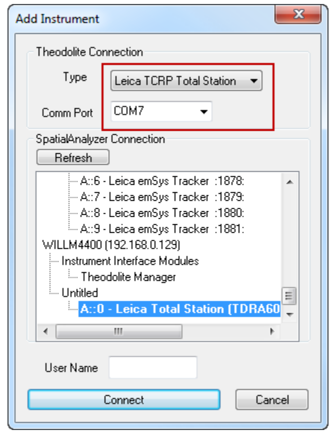


Figure 28-79. Connecting to the instrument.

The Interface

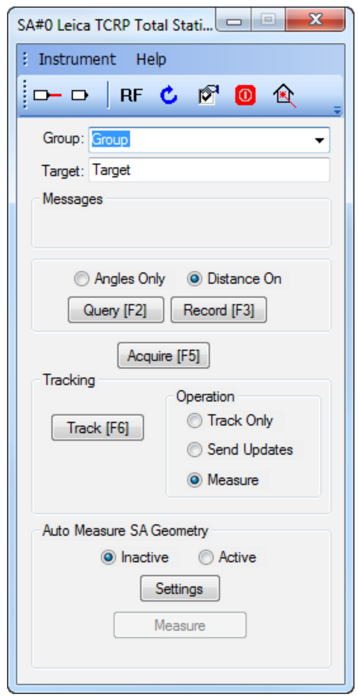


Figure 28-80. The total station interface.

Toolbar

- Laser beam on
- Laser beam off
- Reverses the face of the instrument
- Displays rotation dialog for manual entry of rotation values
- Displays the user options
- Shuts down the instrument
- Opens the reflector database

Group and Target

Defines the Group name and target name for the measured point. If the measured point is recorded from the instrument it will do one of two things:

- If in Freeform Mode it will decode the instrument target based on the parameters in the Theodolite Manager Options (Figure 28-81). For example, if the Target name is 32 on the instrument, when recorded in SA it will be Group3::2 per the below settings.

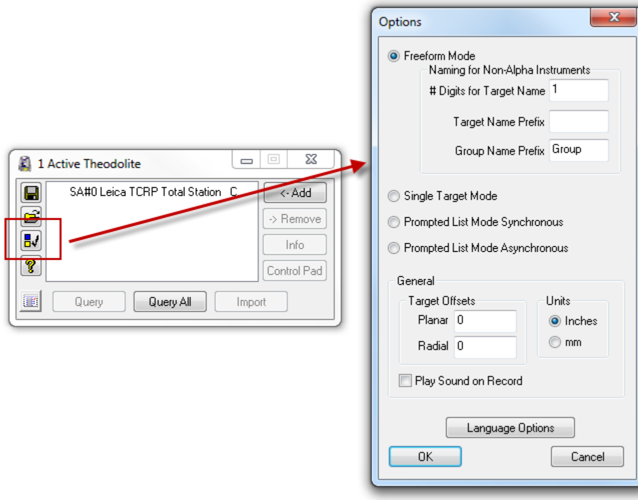


Figure 28-81. Freeform mode.

- If the Theodolite Manger options are set to Single Target Mode, the prescribed point name in the SA interface will be used.

Messages

The message dialog will notify the user if a measurement or process encounters an issue. The message will clear automatically or right-click the message.

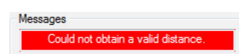


Figure 28-82. An error message.

Query and Record

Query will send updates only to SA. If you wanted to update a watch window, you use Query. Record will send a measured point to SA (Figure 28-83). See above section regarding point naming if recorded from the instrument.

Figure 28-83. Query vs. Record.

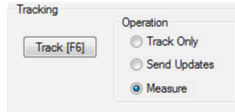
Acquire

Acquire [Acquire [F5]] will use the Leica ATR functionality to lock onto a target. This is especially useful for Prisms and Tape targets. Point the instrument close to the target and press acquire. ATR will find the best center for the target.

Tracking

With the instrument locked onto a prism, tracking can be used. Select the desired options and press Track (Figure 28-84).

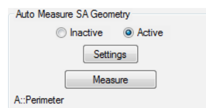
Figure 28-84. Tracking.



Track only will do exactly what it says, it will only track the prism and not send updates or measurements. Send updates is like query, but used for sending updates to SA while tracking. Measure will send measurements to SA while tracking.

Auto Measure SA Geometry

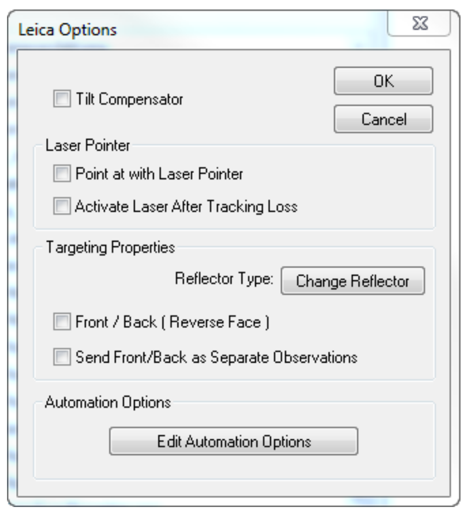
Figure 28-85. Auto-Measure settings.



User Options

The user options (Figure 28-86) can be accessed by using or by **Instrument>Options**.

Figure 28-86. The instrument options.



- **Tilt Compensator.** Turns the tilt compensator on and off. If on, the instrument must be level in order to measure.

Laser Pointer

- **Point at with laser pointer.** When the SA command "point at" is executed the laser will turn on.
- **Activate Laser After Tracking Loss.** When the tracking stops because the line of sight as been broken the laser beam will turn on.

Targeting Properties

- **Change Reflector.** Choose the desired reflector type. For prism, the user has the option of adding Planar and Radial offsets in addition to the Prism Constant. For prism and tape targets, the ATR can be used if desired when a point is recorded.

- **Front/Back (Reverse Face).** When checked, a back sight measurement will be taken with each front sight measurement
- **Send Front/Back as Separate Observations.** When a front/back sight measurement is taken, if checked, they will be stored as separate observations.

Perimeter Scanning

1. Measure points defining the area to scan (Figure 28-87).

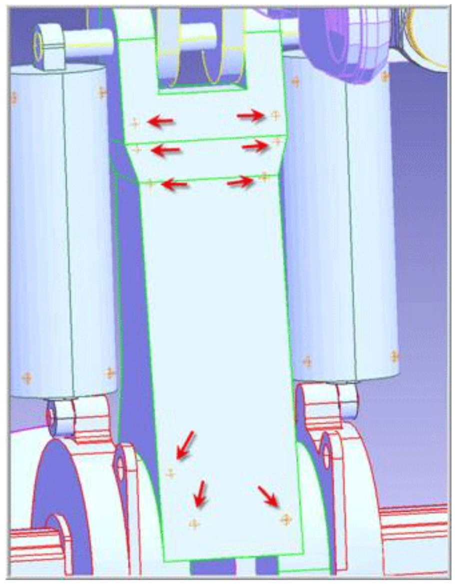


Figure 28-87. Perimeter points.

2. Construct a perimeter from the measured point in SA under **Construct>Perimeter** (Figure 28-88).

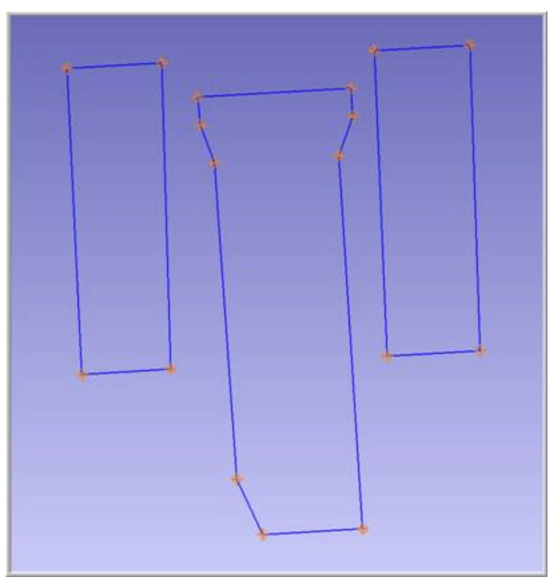


Figure 28-88. Defined perimeters.

3. Two types of perimeters exist in SA, open and closed (Figure

28-89). A single perimeter can be changed from open to closed in the perimeter properties dialog.

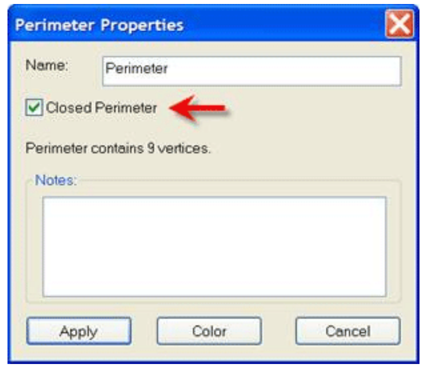


Figure 28-89. Marking a perimeter as closed.

4. Set the scan properties for the appropriate perimeter. If the perimeter is closed the scan will be bounded by the perimeter. If it is open, the scan lines will be perpendicular to the perimeter.

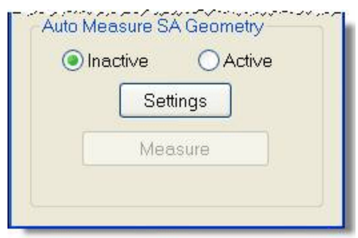


Figure 28-90. Auto Measure SA Geometry settings.

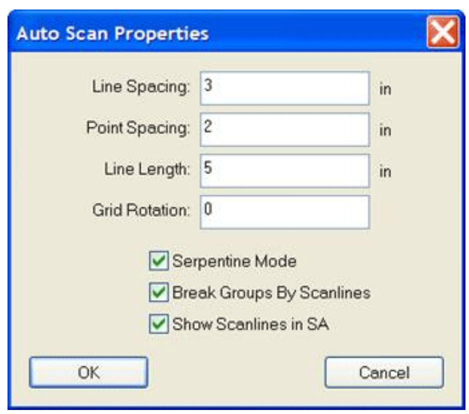


Figure 28-91. Auto Scan properties.

The user can specify the distance between scan lines and points per scan line (Figure 28-91). In addition the user can also specify line length for open perimeters. The orientation of the lines to the perimeter can be controlled by the Grid Rotation field.

To measure a Perimeter, first select Active in the user interface (Figure 28-92) and then select a perimeter from the Treeview in SA. Once the perimeter is selected, select the measure button.

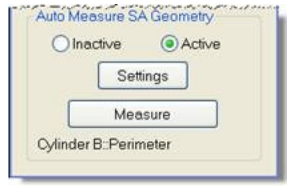


Figure 28-92. Selecting *Active* in the interface.

The scan will start (Figure 28-93) and lines will be constructed (if option was checked). Points will be spaced per user setting. A point will always be measured at the beginning and end of each line.

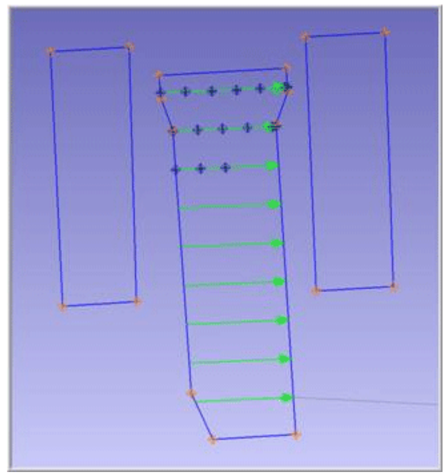


Figure 28-93. Scanning.

To stop a scan, press the Cancel button in the progress dialog.

Auto Measure

The Leica 1203+ can also be used in the SA command Auto Measure. This command will drive the instrument to points in a group and allow a measurement to be recorded.

To perform the Auto Measure, select **Instrument>Automatic Measurement>Auto Measure**.

Select the group to measure and the dialog in Figure 28-94 will appear.

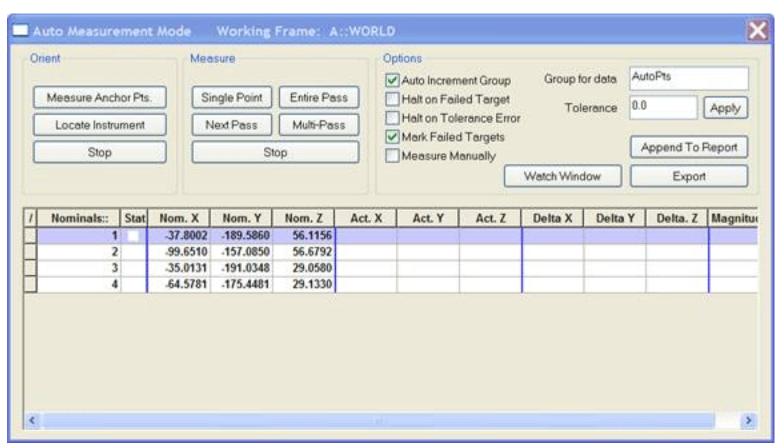


Figure 28-94. The Auto Measure dialog.

This will allow for a single point to be measured or an entire pass be performed. Choose Entire Pass and the instrument will drive to the first point.

Sight the target and press Record (**F3**) in the instrument interface. Once the measurement is recorded the instrument will drive to the next point.

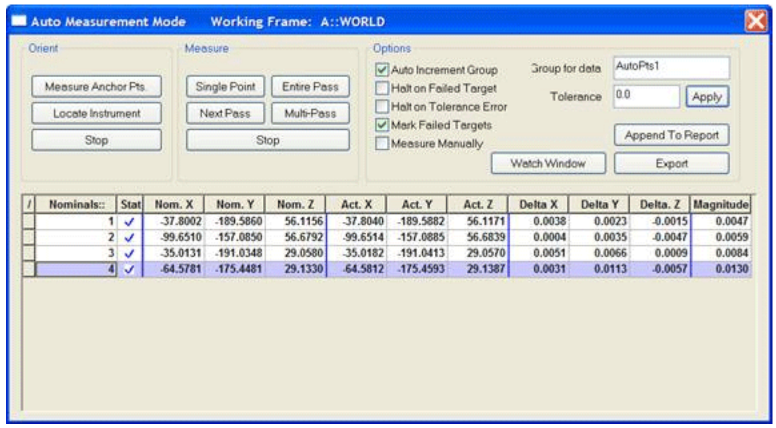


Figure 28-95. Completion of the Auto Measure procedure.

Sokkia Net1

SpatialAnalyzer interfaces with the Sokkia Net1 using the Theodolite Manager via a Bluetooth or RS232C (Serial Port) connection. This quick start guide will cover required Sokkia Net1 settings along with step by step instruction for using the Sokkia Net1 in SA.

Connections

Before attempting to connect to SA, first choose your preferred connection on the Net1.

1. Enter the configuration menu (Figure 28-96).

Figure 28-96. The configuration menu.

2. Select Communications (Figure 28-97).

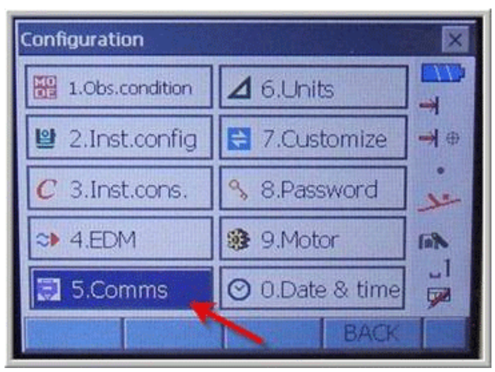


Figure 28-97. Selecting Comms.

3. Set the communication mode (Figure 28-98).

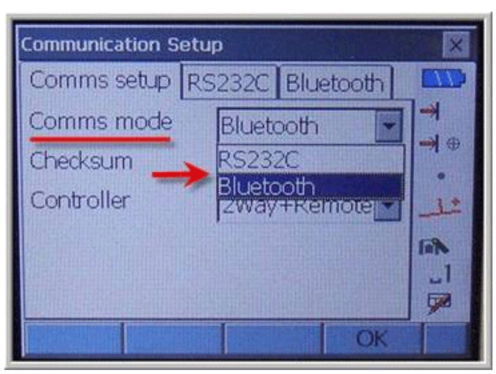


Figure 28-98. Choosing the Comm mode.

4. RS232C Settings (Figure 28-99).

Figure 28-99. The correct RS232 settings for connection.

5. Bluetooth Settings (Figure 28-100).

Figure 28-100. The proper Bluetooth settings.

Establishing Bluetooth Connection on PC

1. Use Bluetooth wizard installed on PC to create a new connection (Figure 28-101). Be sure the Sokkia Net1 power is on.

Figure 28-101. Creating a new connection.

2. The Bluetooth wizard will search for the device (Figure 28-102).

Figure 28-102. The Add New Connection Wizard.

3. The Bluetooth manager will complete the connection and display the COM number being used (Figure 28-103). Remember this number as it is needed inside of SA to establish connection.

Figure 28-103. The assigned COM port.

4. Now the Sokkia Net1 is a recognizable connection for Bluetooth. Each time the Sokkia Net1 is powered the user must enter the Bluetooth manager and activate the connection (Figure 28-104).

Figure 28-104. The connection has been established.

5. Add the Sokkia Net1--select the Sokkia Net1 and pick instrument options (Figure 28-105).

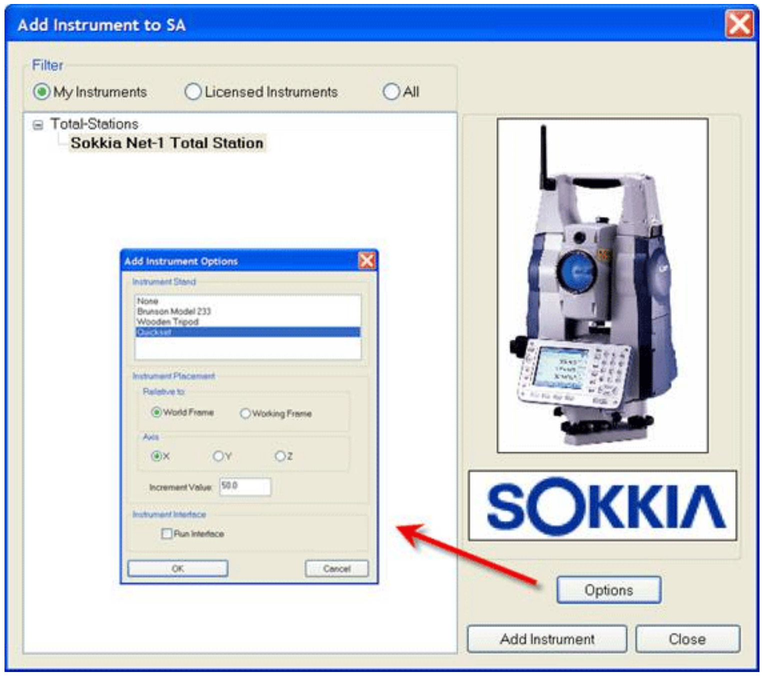


Figure 28-105. Adding the total station to the job.

6. Run the Theodolite Manager and select New Setup, then Add (Figure 28-106).

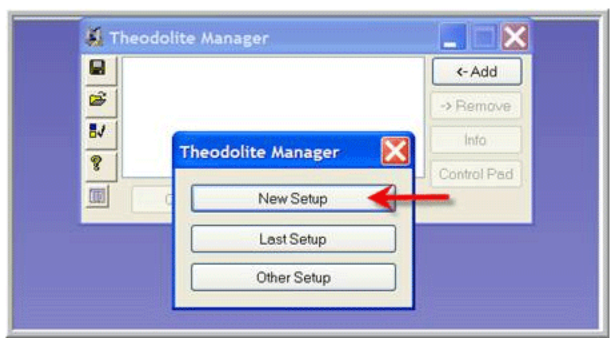


Figure 28-106. Creating a New Setup.

7. Select the instrument type and the Comm Port. This port was displayed in the Bluetooth manager during the creation of the connection. Then select the instrument in the available Spatial Analyzer list. Once connected the Sokkia Net1 interface will display (Figure 28-107).

Figure 28-107. The Sokkia interface.

The following topics will be covered:

- Selecting a Reflector
- Single Point Measurement
- Perimeter Scanning
- Target Tracking
- Auto Measure Points

Selecting Target Reflector

1. Select the options button in the interface (Figure 28-108).

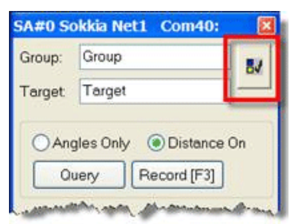


Figure 28-108. The options button.

2. Select the Change Reflector button (Figure 28-109).

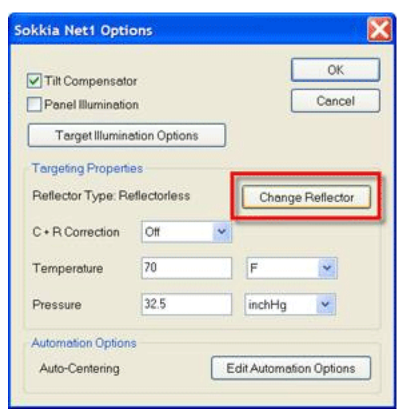


Figure 28-109. The Sokkia options window.

3. Select desired reflector type (Figure 28-110) .

Figure 28-110. Selecting the desired reflector.

Single Point Measurement

1. Specify Group and Target Name.
2. Sight Target.
3. Select Query or Record (**F3**) in the interface. Query will force the instrument to calculate a measurement. Record (F3) will send the measurement to SA (Figure 28-111).

Figure 28-111. Recording a measurement.

Perimeter Scanning

1. Measure points defining the area to scan (Figure 28-112).

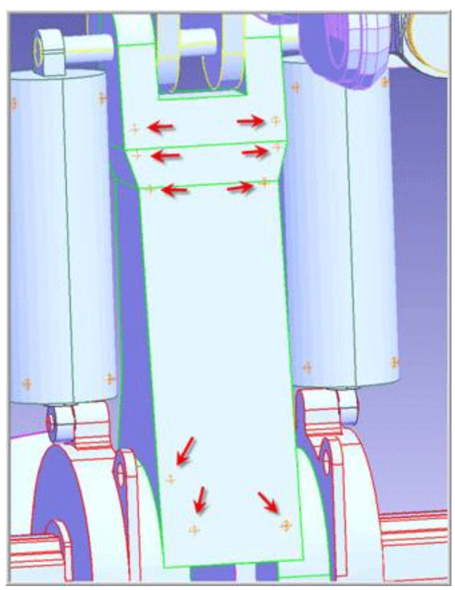


Figure 28-112. Measurements of the area to scan.

2. Construct a perimeter from the measured point in SA under **Construct>Perimeter** (Figure 28-113).

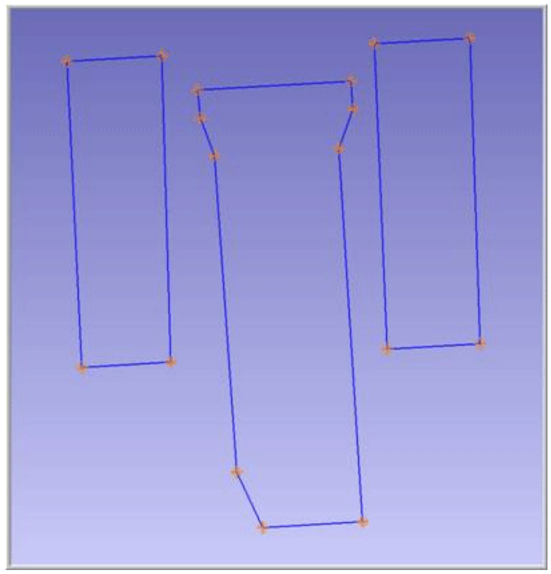


Figure 28-113. The defined perimeters.

3. Two types of perimeters exist in SA, open and closed. A single perimeter can be changed from open to closed in the perimeter properties dialog (Figure 28-114).

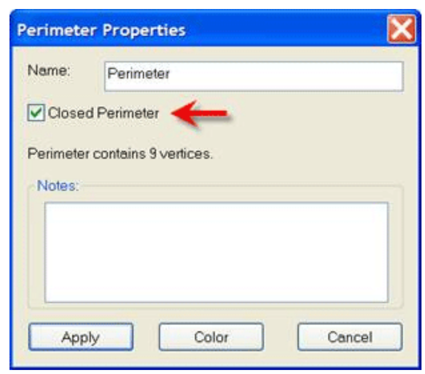


Figure 28-114. The Perimeter Properties window.

4. Set the scan properties for the appropriate perimeter. If the perimeter is closed the scan will be bounded by the perimeter. If it is open, the scan lines will be perpendicular to the perimeter.

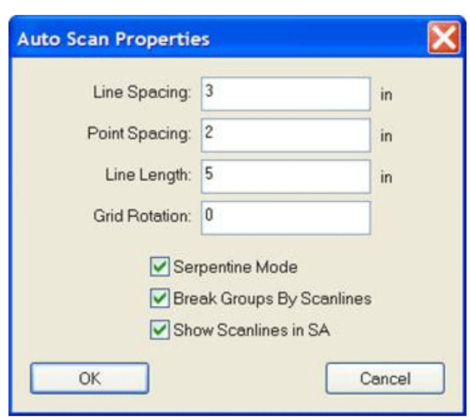


Figure 28-115. Auto Scan properties.

5. The user can specify the distance between scan lines and points per scan line (Figure 28-115). In addition the user can

also specify line length for open perimeters. The orientation of the lines to the perimeter can be controlled by the Grid Rotation field.

To measure a Perimeter, first select Active in the user interface (Figure 28-116) and then select a perimeter from the Treeview in SA. Once the perimeter is selected, select the measure button.

Figure 28-116. The Active setting.

The scan will start and lines will be constructed (if option was checked). Points will be spaced per user setting. A point will always be measured at the beginning and end of each line.

To stop a scan, press the cancel button in the progress dialog.

Target Tracking

The Sokkia Net1 is capable of tracking a prism target.

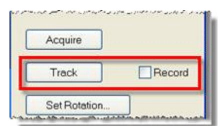


Figure 28-117. The track setting to track a prism target.

Simply select track and start moving the prism (Figure 28-117). The Net1 will track the prism. If the record option is check, measurements will be sent to SA while the prism is being moved.

Auto Measure

The Sokkia Net1 can also be used in the SA command Auto Measure. This command will drive the instrument to points in a group and allow a measurement to be recorded.

To perform the Auto Measure, select **Instrument>Automatic Measurement>Auto Measure**.

Select the group to measure and the dialog in Figure 28-118 will appear.

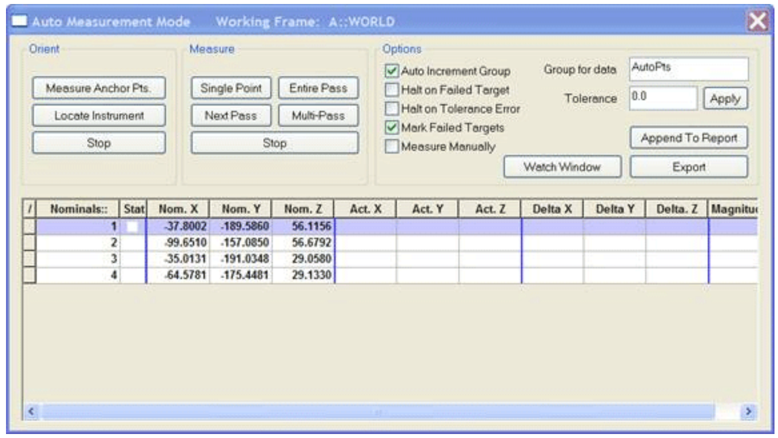


Figure 28-118. Auto Measure mode.

This will allow for a single point to be measured or an entire pass be performed. Choose Entire Pass and the instrument will drive to the first point.

Sight the target and press Record (**F3**) in the instrument interface. Once the measurement is recorded the instrument will drive to the next point.

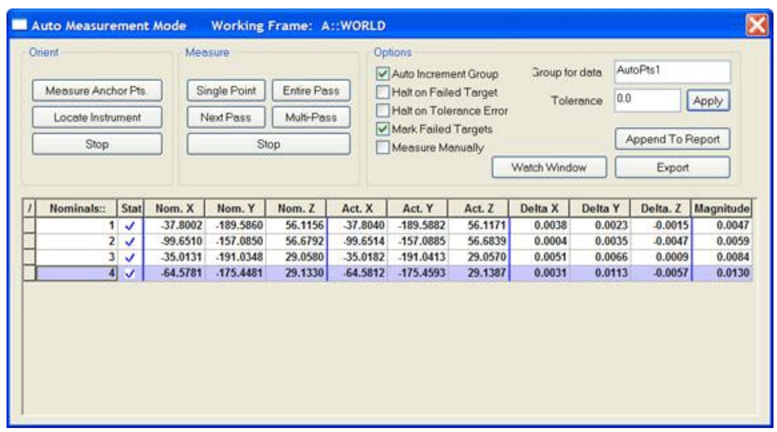


Figure 28-119. A completed auto measurement.

Leica Nova MS50

This guide may be used for initial configuration, connection and basic operation of the Leica Nova MS50 Total Station within SA. For more details on instrument operation and configuration, please contact Leica directly.

Hardware Setup

Set up the unit following the manufacturer's directions. Leica total stations may be connected either with a USB cable, wireless network adapter or Bluetooth connection. Ensure that you have the necessary cables, router or a Bluetooth adapter. Be sure the power is turned on and the battery is charged.

Software Setup

It is recommended that initial connection and testing be performed using a cable connection to your computer. To connect directly to the PC, you will need to install the instrument drivers. The current drivers SA supports can be downloaded from http://www.kinematics.com/ftp/SA/Install/Driver%20Downloads/TotalStations_Theodolites/Leica/Leica%20Nova%20MS50%20Drivers/.

To use the live remote video function available on this instrument, you will also need to install the VLC player application which can be downloaded from http://www.videolan.org/vlc/. Once installed, a Video button will appear on the ***Camera Interface*** dialog.

Direct Cable Connection

On the scope select **Instrument > Connections > All Other Connections**. Set the GeoCom connection to USB (to obtain the current IP address and port, press Cntrl..). Connect the USB cable and continue to the section *Running the Instrument*.

Wireless Connection

A connection can be established to the Nova by adding the device to a preexisting network or by building an Ad-hoc connection with your PC. For more detailed instructions, refer to the document "Connecting a Nova total station with a PC using a WLAN" on our download page. For a quick synopsis on a Windows 7 machine, follow these steps:

Building an Ad-hoc Connection on a Win7 PC

6. Turn on the wireless adapter on your PC and go to **Control Panel > Network and Sharing Center**.
7. Select **Setup a new connection or network > Set up a wireless ad hoc (computer to computer) network**.
8. When prompted, enter a network name (e.g. "Ad-hoc MS50) and a security type, if desired (no authentication, WEP and

WPA-2 are supported).

9. Select Save this network then Next to create the network.
10. Set the IP address to be compatible with the scope.

Setting the IP Address in the Adapter Properties

1. Turn on the wireless adapter on your PC and go to **Control Panel > Network and Sharing Center**.
2. Select **Manage wireless networks > Adapter properties** for the network you wish to connect to and for editing the properties of your ad-hoc network.
3. In the ***Wireless Network Connection Properties*** dialog, select Internet Protocol Version 4 (TCP/IPv4).
4. Select Properties and enter the IP address rather than obtaining one automatically. This address determines the computer's IP address on the network.

Configuring the MS50 Network Connection

1. Open the Windows CE control panel by selecting **User > Tools & Utilities > View ASCII files** and close the viewer.
2. Click on WLAN Settings in the taskbar and select the network you wish to connect to from the network list and click Connect. In some cases, you may need to set the IP address manually by doing following:

- Select **Start > Settings > Network and Dial-Up Connections** OWL221A1.
- Select Enable, and then Properties.
- Choose Specify an IP address, and enter the IP address for the total station on the network. The first three numbers of the IP address need to match the first three numbers entered for the PC. The last number must be unique from the PC.

3. Return to the Windows CE desktop, click on WLAN settings in the taskbar. In the ***Wireless Information*** tab, select the ad-hoc network that you created and select Connect.
4. Continue to the section *Running the Instrument*.

Connecting to Bluetooth

1. On the scope, select **Instrument > Connections > All Other Connec tions**.
2. Set the GeoCom connection to TS Bluetooth 1.
3. On your PC, go to your Bluetooth adapter and select Add a Device.
4. Select the device from the available Bluetooth connections and select next.

5. When prompted, enter the pairing code: **0000**. Once connected, a COM port number will be assigned and can be accessed through the properties of the device.

6. Continue on to the section *Running the Instrument*.

Configuring the Scope for Optimal Use with SA

1. Navigate to **Users > System Settings > Regional Settings**.

2. Check that the Hz angle display is set to *North anti-clockwise* and the V angle display is set to *Zenith angle*. This will ensure that the angles displayed on the scope match those of the recorded points in SA.

Starting the Interface

1. Select **Instrument > Add** and choose the Leica Nova MS50 Total Station from the ***Add Instrument to SA*** dialog (Figure 28-120).

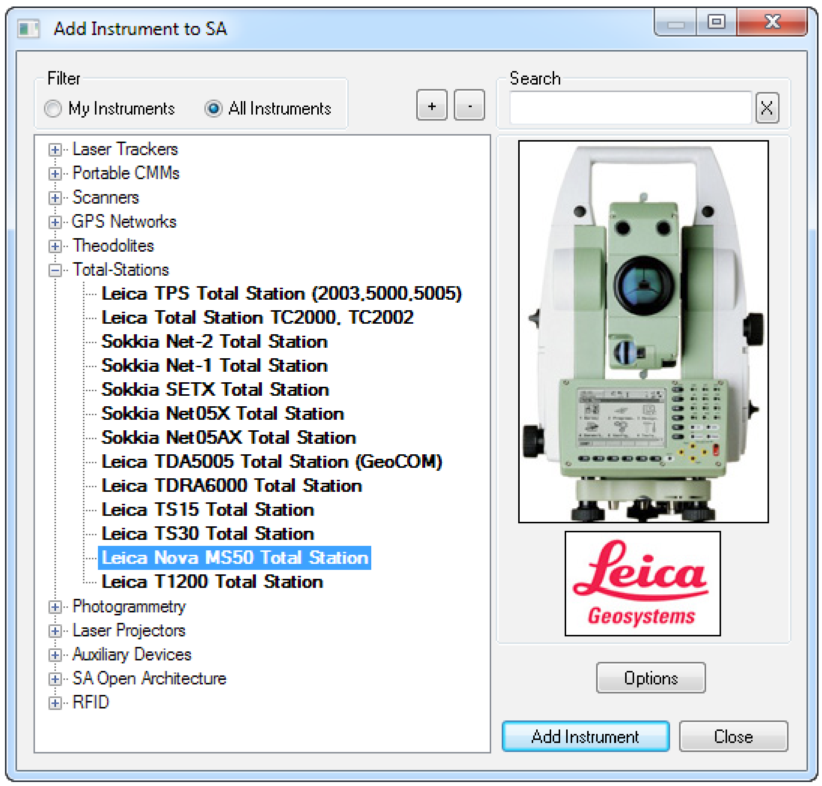


Figure 28-120. Adding the Leica Nova MS50 Total Station to a job.

2. Now run the instrument interface module under **Instrument > Run Interface Module** and choose **Theodolite Manager**.

3. Select New Setup and then Add (Figure 28-121).

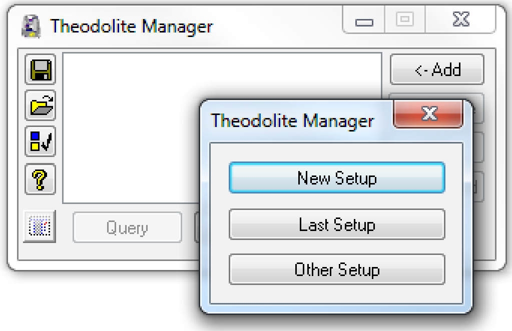


Figure 28-121. The Theodolite Manager dialog.

4. Select the instrument type and the COM Port. Set the COM Port to TCP if you are connecting with either a cable or wireless; or enter the port number for Bluetooth. Then select the instrument in the available SpatialAnalyzer list and click Connect (Figure 28-122).

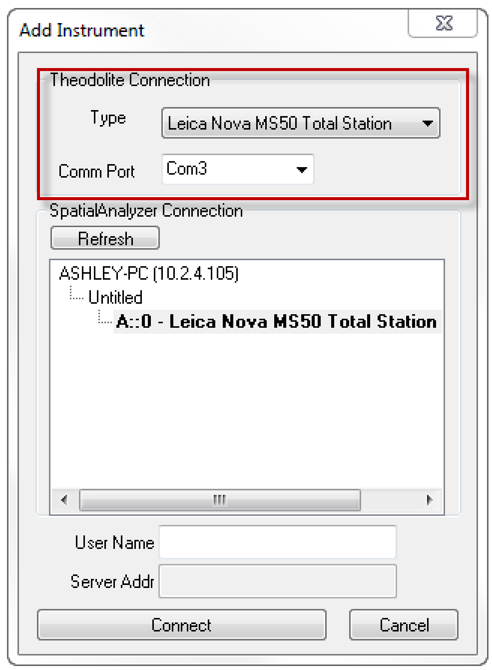


Figure 28-122. Adding a new setup.

Note: If connecting with a cable or wireless, enter the instrument's IP address in the Server Address box.

5. Select the instrument in the available SpatialAnalyzer list and click Connect.

Operations Specific to the MS50

Auto Measure SA Geometry

In addtion to scanning a predefined region by laying points out on a grid using auto measure, the MS50 can quickly scan an a perimeter and generate a point cloud within SA.

1. In the *Auto Measure SA Geometry* section of the instrument interface, ensure that the Revert button is *not* depressed. This button changes between scanning modes (Figure 28-

123).

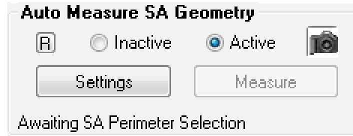


Figure 28-123. Auto measure SA Geometry section of MS50 Interface.

2. Click Settings to change the parameters of the scan. The density of the scan is defined by the vertical and horizontal angular increment set in decimal degrees. Once satisfied with the settings click OK (Figure 28-124).

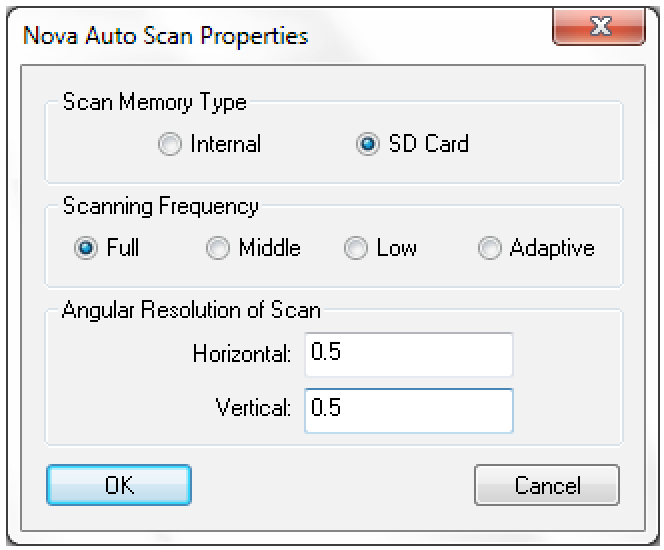


Figure 28-124. Auto measure SA geometry profile parameters.

3. Change the *Auto Measure SA Geometry* to *Active* by selecting the radio button. When you do so, a note will be shown at the bottom of the interface that states: *"Awaiting SA Perimeter Selection."*

4. Click on the desired perimeter by clicking on double-clicking on it in the tree. When selected, the perimeter *Collection::Name* will be displayed.

5. To begin the scan, click Measure. An example of this type of perimeter scan is pictured in Figure 28-125.

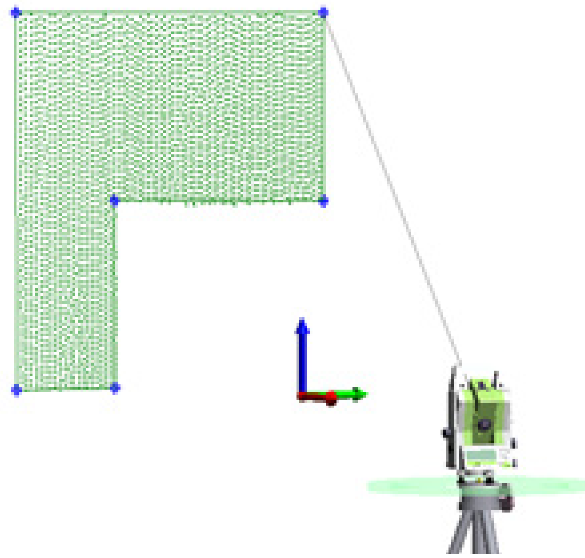


Figure 28-125. Perimeter scan using the Auto Measure SA Geometry measurement mode.

Importing Nova Scan Files into SA

If you choose to scan while not connected to a computer, you may import these files into SA at a later time. The Nova Scans are saved as *.SDB files on the SD Card or the internal memory. To do so, follow these steps:

1. Add an instrument to your SA job file to which you wish to import the scan data. Data will be imported into SA relative to this instrument.
2. Either connect to the live instrument using a cable or connect using the simulation mode.
3. Locate and move the *.SDB files to your local machine. If you are connected using the cable, you can use Windows Mobile Device Center to navigate to the SD card directly.
4. In Theodolite Manager, select your instrument and click Import.
5. Select the *.SDB data type, navigate to your data, and click Import.

Performing a Point to Point Auto Measure Scan

1. In the *Auto Measure SA Geometry* section of the instrument interface, ensure that the Revert button *is* depressed. This button changed between scanning modes.
2. Click the Settings button to change the parameters of the scan based on the line and point spacing in the ***Auto Scan Properties*** dialog (Figure 28-126).

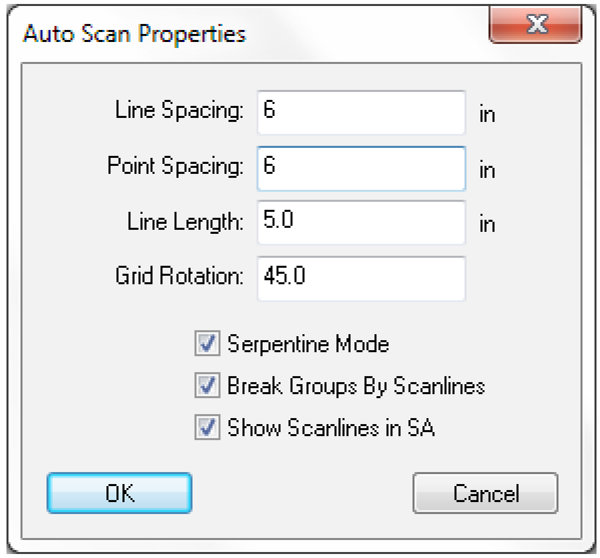


Figure 28-126. Auto Scan Properties dialog.

3. Change the *Auto Measure SA Geometry* to ***Active*** by selecting the radio button. When you do so, a note will be shown at the bottom of the interface that states: *"Awaiting SA Perimeter Selection."*
4. Click on the desired perimeter by clicking on double-clicking on it in the tree. When selected, the perimeter *Collection::Name* will be displayed.
5. To begin the scan, click Measure. An example of this type of perimeter scan is pictured in Figure 28-127.

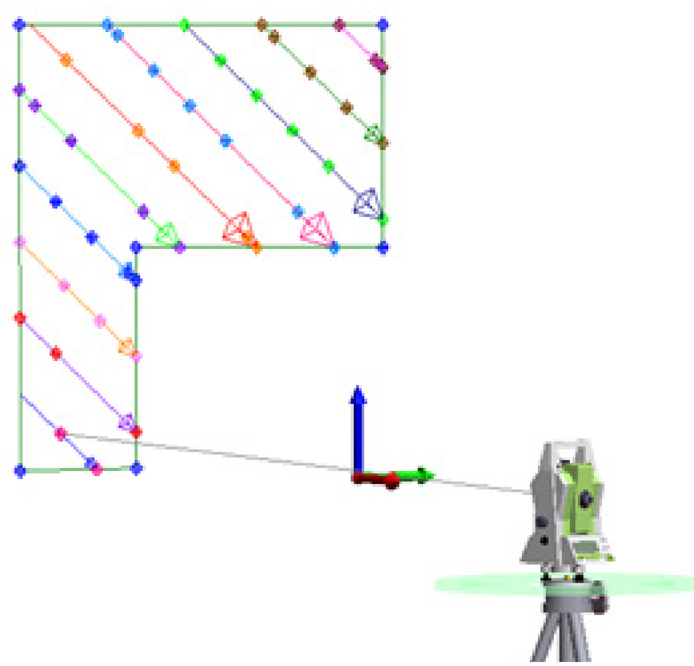


Figure 28-127. A point to point auto measure scan.

Photogrammetry

Aicon Move Inspect

This quick-start guide can be used for initial configuration, connection and basic operation of the AICON Move Inspect Photogrammetry system within SA. For more details on instrument operation and configuration, please contact AICON directly.

Hardware Setup

Set up the AICON MoveInspect system following the Manufacturer's directions. This will depend on the camera configuration and system type. Ensure you have the appropriate camera mounts, tripods, and cabling to power the unit.

Communication with SA is done through TCP/IP Ethernet connection so an appropriate network connection is necessary.

Software Setup

SA does not interface with the cameras directly but rather with the host application AICON MoveInspect. Therefore, MoveInspect must be installed, calibrated, and running on the local machine prior to any attempt to connect to SA.

Please contact AICON directly to obtain the correct version of MoveInspect for your camera system and install it following the manufacturer recommendations.

http://aicon3d.com

You can verify SA's current version compatibility by looking in the SA readme file available under the Help menu within SA.

Creaform HandyProbe

Starting the Interface

1. Start the Creaform HPS application.
2. Once started, ensure that C-Track is connected and calibrated per OEM guidelines (Figure 28-128).

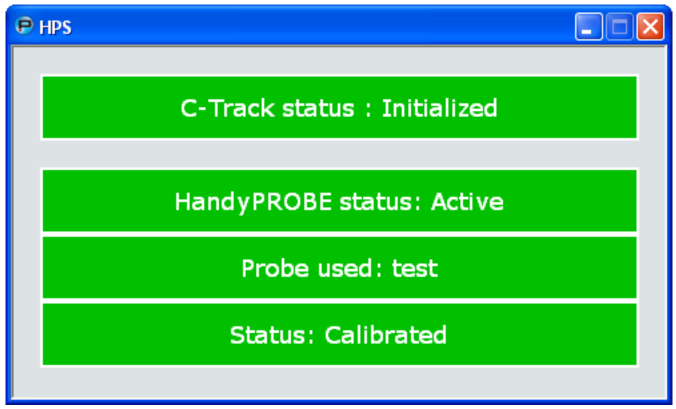


Figure 28-128. The HPS dialog.

3. Select **Instrument > Add** and choose the Creaform Handy Probe from the Instrument List.
4. Run the Creaform HandyProbe interface from the start menu's Interfaces menu.
5. The Creaform HandyProbe dialog box will open. Select your device, and the interface will start up (Figure 28-129).

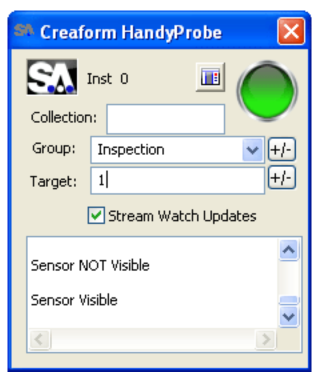


Figure 28-129. The HandyProbe interface.

Measuring with the Handy Probe

Button Functionality

- **Center Button.** A single press measures a single point. Holding the button will stream multiple points.
- **Check Button (left).** Increment the group name.
- **"X" Button (right).** Delete last measured point.

Watch Window

Using watch windows with the Handy Probe is very easy. Simply check the "Stream Watch Updates" option and open the watch window of choice (Figure 28-130).

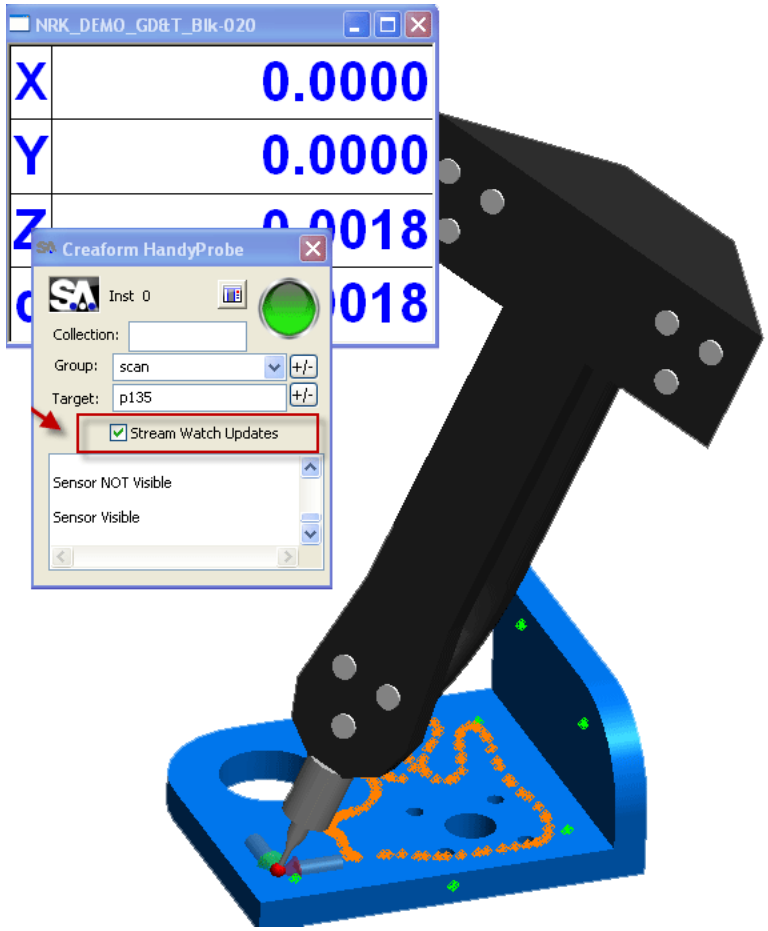


Figure 28-130. The HandyProbe in use with a watch window.

GSI V-STARS

Starting the Interface

1. Select **Instrument > Add** and choose the GSI V-Stars Photogrammetry System from the Instrument List (Figure 28-131).

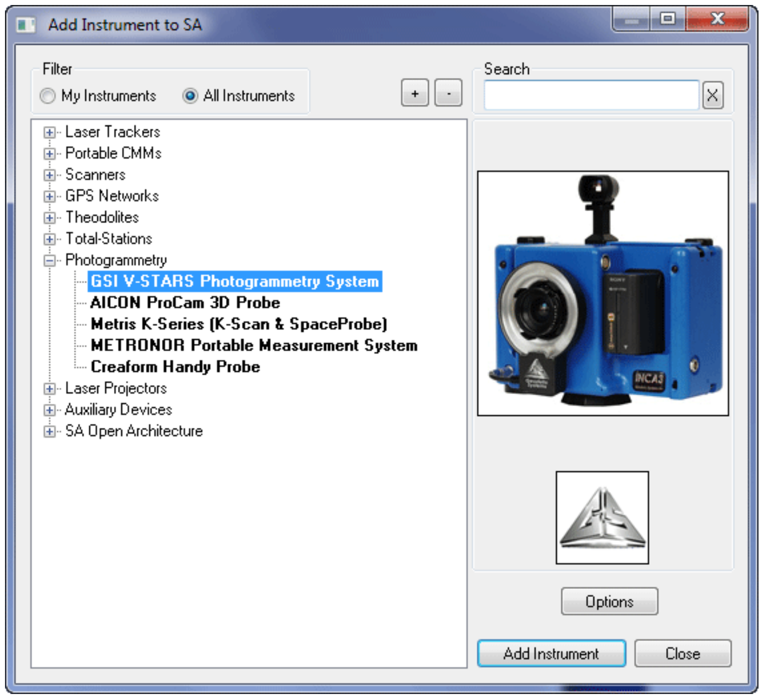


Figure 28-131. Adding a V-STARS camera system to SA.

2. Run GSI V-stars 4.7.A16 or later.

3. Start the instrument interface using **Instrument > Run Interface Module and Connect** or [icon] .

Interface Overview

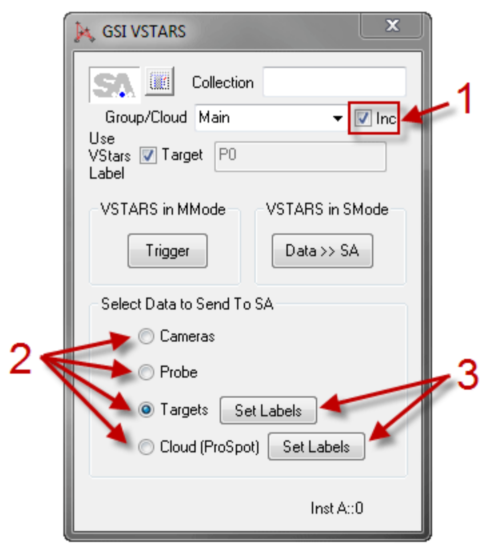


Figure 28-132. The V-STARS interface.

Data is sent to SA in 2 possible ways. If VSTARS is in M-Mode, then selected data is sent any time VSTARS is triggered. This can be done with the 'Trigger' shown (Figure 28-132), or in VSTARS. The **Continuous Trigger** option in VSTARS will result in the selected data type being sent to VSTARS upon each trigger event. If VSTARS is in S-Mode, the Data >> SA button shown above results in an immediate project import from VSTARS, and selected data is sent to SA.

Note the **Inc** checkbox (1). This is the Auto-Increment Group (or Cloud) name option. When checked, the group or cloud name will automatically increment after each data send to SA.

Select Data to Send To SA (2). These work as follows:

- **Camera.** If VSTARS is in M-Mode, the camera transforms are sent to SA, and all the M-Mode camera positions are shown in SA.
- **Probe.** If VSTARS is in M-Mode, then the most recent PROBE point is sent to SA once a trigger event occurs. Generally, the 'Inc' will be unchecked in this case. If VSTARS is in S-Mode, the project is imported immediately, and the most recent PROBE point is sent to SA.
- **Targets.** If VSTARS is in M-Mode, then all selected point types (see 3 below) are sent to SA once a trigger event occurs. Generally, the **Inc** checkbox will be checked in this case. If VSTARS is in S-Mode, the project is imported immediately, and all selected points are sent to SA.
- **Cloud (ProSpot).** If VSTARS is in M-Mode, then all selected point types (see 3 below) are sent to SA in the form of a point cloud once a trigger event occurs. Generally, the **Inc** checkbox will be checked in this case. Note that the cloud sent will carry then name shown in the 'Group/Cloud' edit box. If VSTARS is in S-Mode, the project is imported immediately, and all selected point types are sent to SA in the form of a cloud.

The Set Labels buttons (3) are used to set the VSTARS data types (labels) which will be sent when measuring either 'Targets' or 'Cloud (ProSpot)'. The Targets data selection defaults to "CODE", and the Cloud (ProSpot) data selection defaults to "_S". The selections denote strings. When any of the selected strings are found in the VSTARS label of a point, that point will be sent to SA in the selected data form ('Targets' means points, and 'Cloud (ProSpot)' means a point cloud).

Figure 28-133 shows the ***Set Target Labels to Use*** dialog. The dialog is the same for 'Cloud (ProSpot)'. All types are selected here for readability (un-selected labels are subdued).

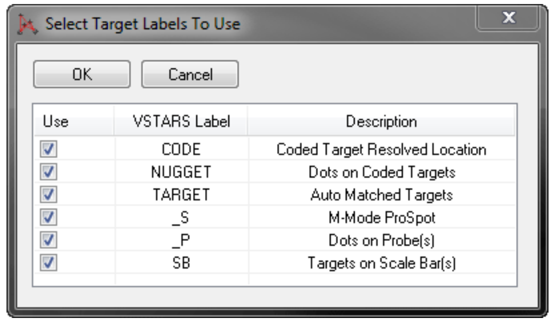


Figure 28-133. C

Metronor

Prerequisites

Metronor Server must be installed and running before SpatialAnalyzer will connect to the instrument.

Starting the Interface

1. Start Metronor Server (Figure 28-134).
2. Run Metronor Server and perform setup and field checks per OEM guidelines.

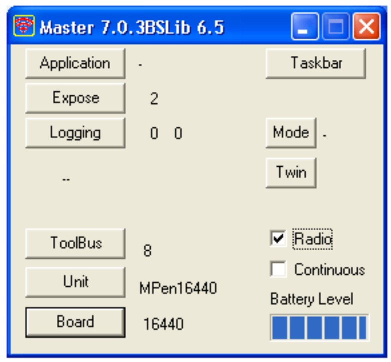


Figure 28-134. Metronor Server.

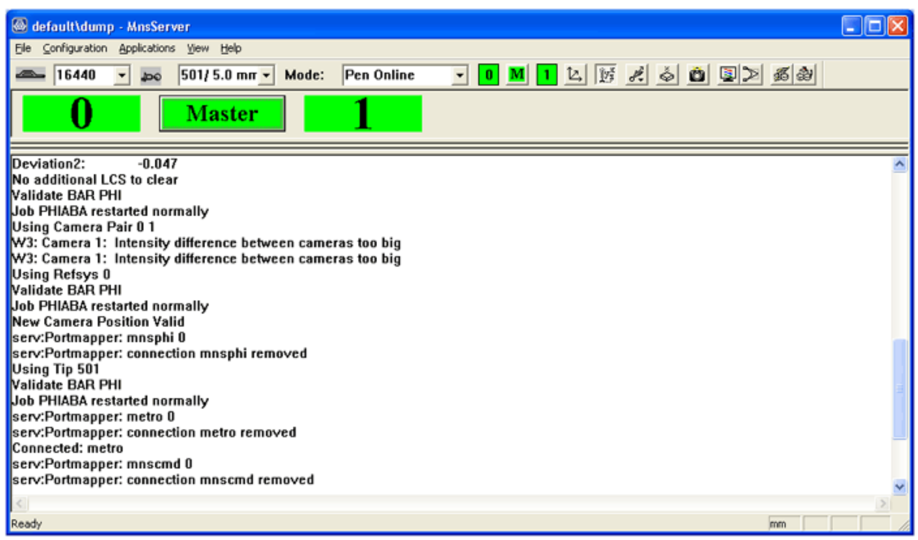


Figure 28-135. The startup process.

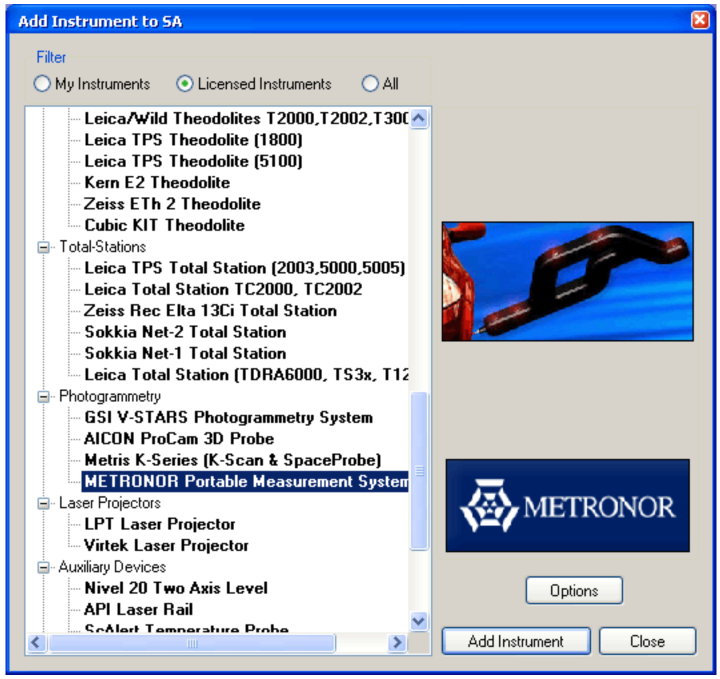


Figure 28-136. Adding the Metronor to SA.

3. Select **Instrument > Add** and choose the Metronor Portable Measurement System from the Instrument List (Figure 28-136).

4. Run the Metronor interface from the **Start > Programs > SpatialAnalyzer 20##.##.## > Interfaces** menu.

5. The Spatial Analyzer Metronor Logon dialog box will open and will ask what device you want to connect to (Figure 28-137). Select the instrument and type the server name (which will be either the computer name or the computer's IP address). Accept the default Server Port.

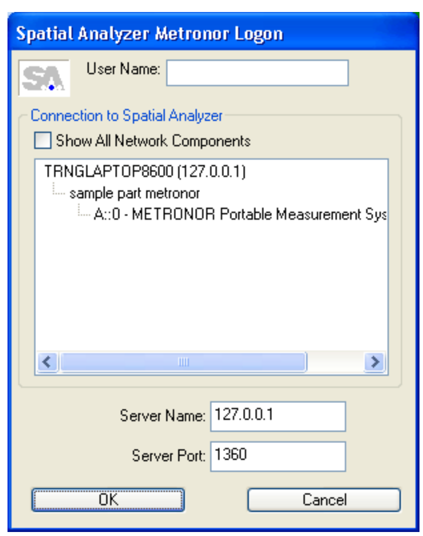


Figure 28-137. The logon dialog.

6. This will start the Metronor Interface.

Vicon Tracker

Prerequisites

Vicon Tracker must be installed and running before SpatialAnalyzer will connect to the instrument. SpatialAnalyzer and Vicon Tracker must be installed on the same computer.

Starting the Interface

1. Ensure Vicon Tracker is running and instrument is operating correctly per OEM guidelines.
2. Select **Instrument > Add** and choose the Vicon Tracker from the Instrument List.
3. Run the instrument interface using **Instrument > Run Interface Module and Connect** or

Instrument Interface Overview

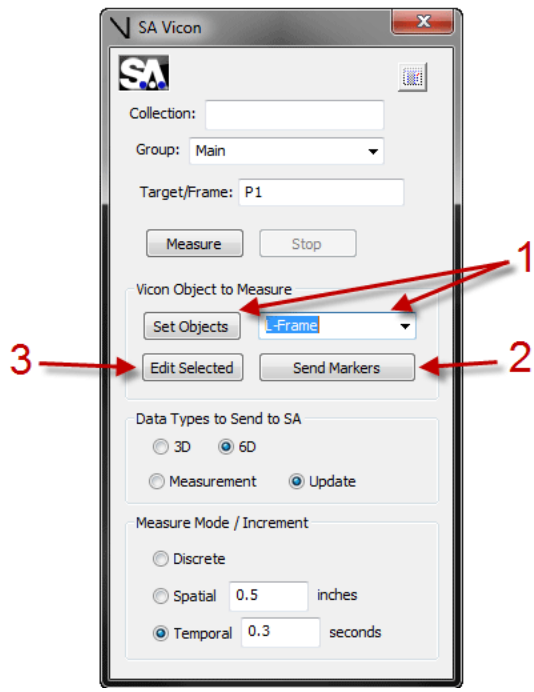


Figure 28-138. The Vicon interface.

Press Set Objects to populate the drop down combo box with all objects currently in view of the cameras in Vicon Tracker. For this, you must be sure all defined objects are in the view of the cameras. Here, an Object called 'L-Frame' is selected in the drop down.

Press Send Markers to send the sphere centers of all the Vicon markers in the selected object. This is for co-locating with other instruments in SA.

Press Edit Selected to add an offset frame/probe to the selected object (Figure 28-139).

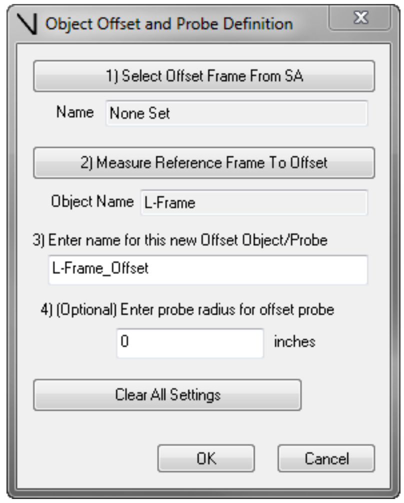


Figure 28-139. Defining an offset.

Just follow the steps in the dialog to create an offset frame or probe for the selected object.

1. Create a frame in SA on the object (or e.g. a probe which you have rigidly attached to the object) by measuring with an instrument, or creating by some other means.
2. Click 1) Select Offset Frame From SA. You'll be prompted to select the frame you have created, and the *Name* field will be populated with the name of the frame from SA.
3. Click 2) Measure Reference Frame to Offset to take a fresh 6-DOF measurement of the object you've selected. You'll see a suggested name appear.
4. In the *3) Enter name for this new Offset Object/Probe* field, type in a name if you'd like to use your own name.
5. In the *4) (Optional)* field, enter the probe radius for the offset probe.
6. Press OK to add this new offset frame/probe definition to the drop down combo box.

When your new offset object is selected, the offset frame will be sent to SA whenever you have "6D" Data Type selected, or the origin of the offset frame will be sent if you have "3D" Data Type selected. The probe radius will be added to all data sent from your offset object definition.

Scanners

Faro LS/Focus3D

Prerequisites

Before adding and running a Faro LS/Focus 3D Scanner in SA, the appropriate drivers must first be installed.

Download the drivers from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/Scanners/Faro%20LS%20Photon%20Scanner/ or install from the Instrument Drivers combined installer at http://kinematics.com/products/spatialanalyzer/spatialanalyzer-downloads.html

1. Unzip the dll's into the current SA install folder.
2. Place the `.bat` files in the current SA install folder.
3. Run `Unregister iQpen.bat`
4. Run `Register iQOpen.bat`

Running the Faro Scanner

1. Add a Faro Scanner LS to SA via **Instrument>Add** or using the icon (Figure 28-140). Select the Faro Scanner and press Add Instrument.

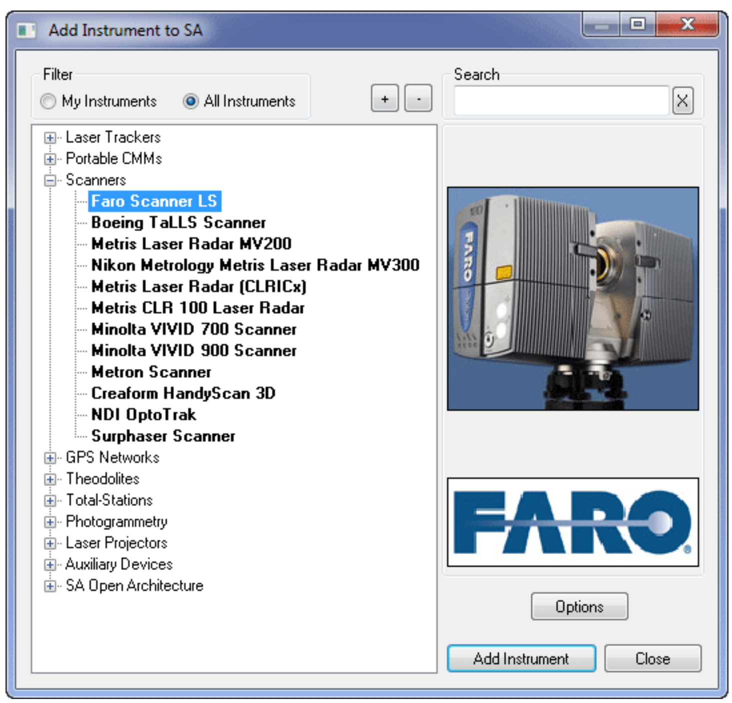


Figure 28-140. Adding a Faro scanner to the job.

2. Run the instrument interface module under **Instrument>Run Interface Module and Connect**.
3. The Faro Scanner LS Interface will now appear and be ready

for use (Figure 28-141).

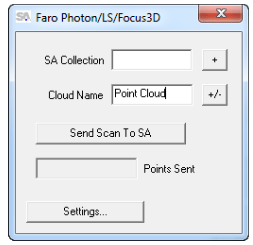


Figure 28-141. The Faro scanner interface.

Surphaser

Prerequisites

Before adding and running a Surphaser Scanner in SA, the appropriate drivers must first be installed.

Download the operating system-specific Surphaser drivers from ftp://ftp.kinematics.com/pub/SA/Install/Driver%20Downloads/Scanners/Surphaser/.

1. Unzip the drivers into a known location. Example: `C:\DrvWinUsb`
2. Plug in the Surphaser USB cable to the PC. Windows should recognize the USB device and automatically locate the drivers and install them. If this is performed correctly, the Surphaser scanner will be presented in your Device list (Figure 28-142).

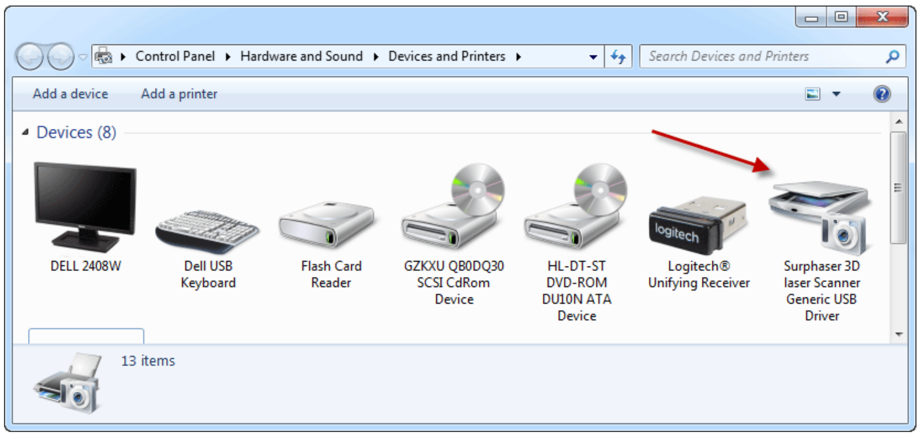


Figure 28-142. The Device list.

Running the Surphaser

1. Add a Surphaser to SA via **Instrument>Add** or using the icon. Select the Surphaser Scanner and press Add Instrument (Figure 28-143).

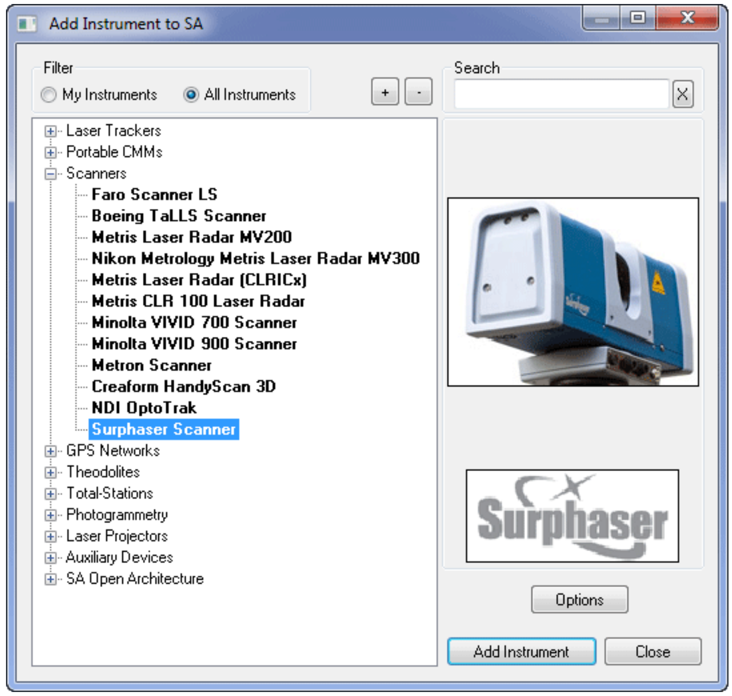


Figure 28-143. Adding the Surphaser to SA.

2. Run the instrument interface module under **Instrument>Run Interface Module** and choose Surphaser. Select the parameter file provided for the respective instrument and a log file (Figure 28-144).

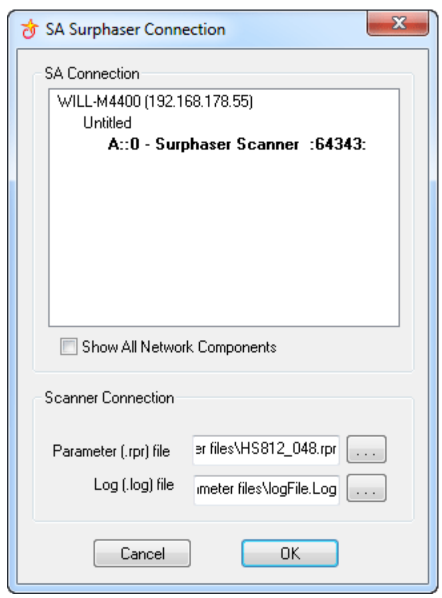


Figure 28-144. The Surphaser Connection dialog.

3. The Surphaser Interface will now appear and be ready for use (Figure 28-145).

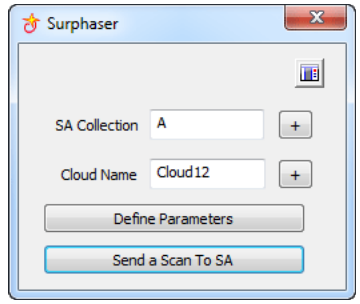


Figure 28-145. The main Surphaser interface.

Leica ScanStation P20

This guide may be used for initial configuration, connection and basic operation of the Leica ScanStation P20 within SA. For more details on instrument operation and configuration, please contact Leica directly.

Hardware Setup

Setup the unit following the manufacturer's directions. The ScanStation requires an Ethernet connection to the machine running SpatialAnalyzer 2014.02.04 or later. Be sure the Leica device power is on and that the batteries are charged.

Software Setup

You must have a FW license to activate the *API Data Access* toggle on the P20. You can toggle this on the scanner in **Status>System Information>Options**. The setting *API Data Access* must be activated (it is deactivated by default). Contact your Leica Geosystems representative for more information.

Starting the Interface

1. Select **Instrument>Add** and choose the Leica Geosystems ScanStation P20 from the ***Add Instrument to SA*** dialog (Figure 28-146).

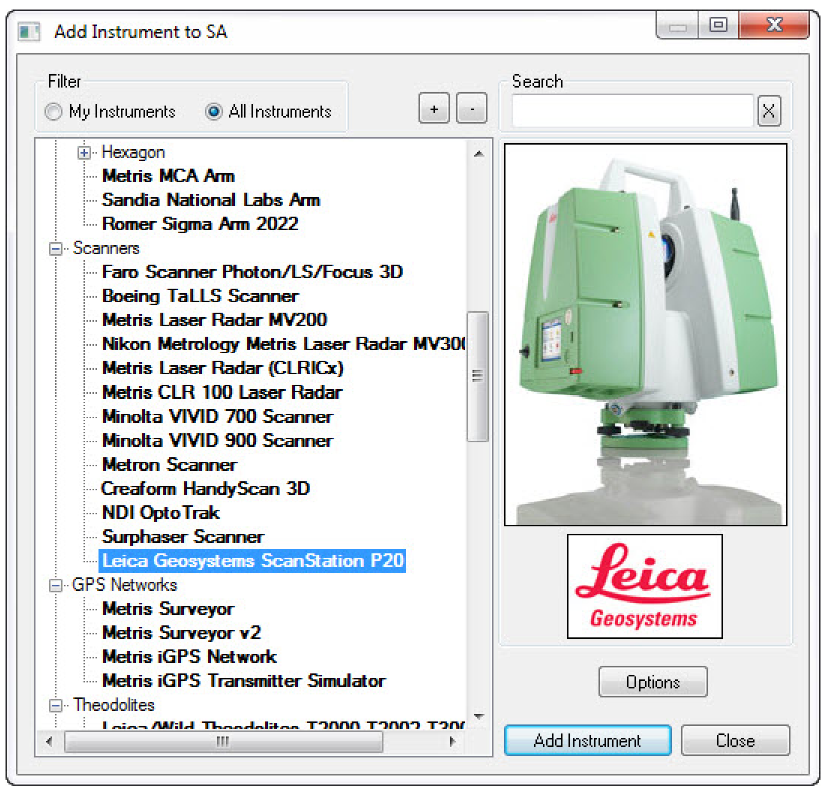


Figure 28-146. Adding the Leica ScanStation P20 to a job.

2. Now run the instrument interface module and connect using the running man icon .

3. Select Run Scanner (Figure 28-147).

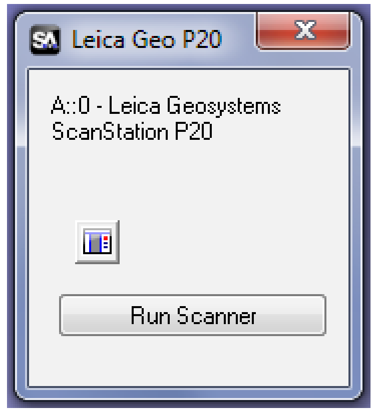


Figure 28-147. The run scanner dialog.

4. When the ***P20 Scanner Control*** dialog appears, select the instrument's serial number and click Connect Scanner. The scanner uses a DHCP connection so an IP address identification is not necessary. Then enter the project name, which specifies the directory to save the scan data and click Init Project (Figure 28-148).

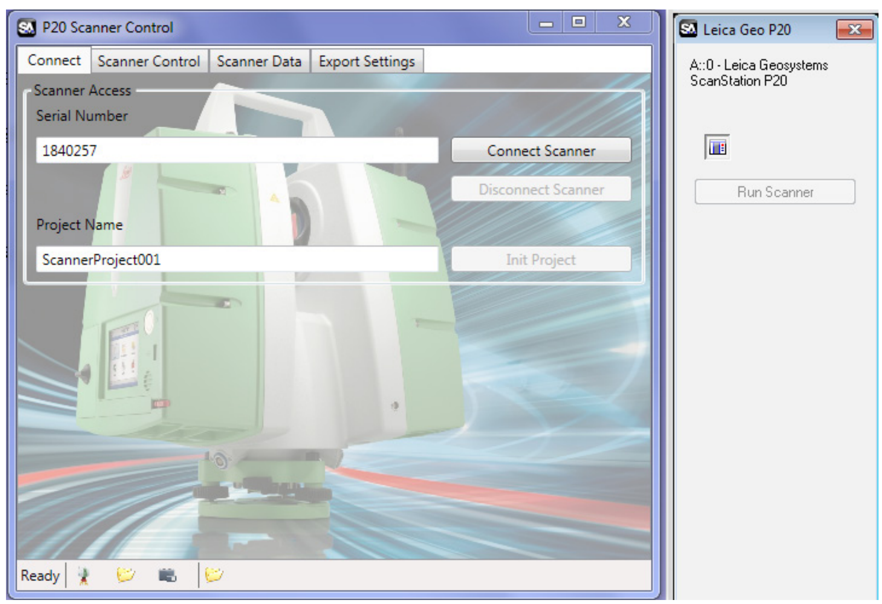


Figure 28-148. P20 Scanner control dialog.

Export Settings

The *Export Settings* tab (see Figure 28-149) in the ***P20 Scanner Control*** dialog defines the scan filter definitions and the scan transfer location on the local machine. The *ASCII Export Path* defines the location on the local machine where scan files will be transferred.

The advanced settings are master controls for filter options not typically changed for scans unless needed for a particular application.

- *Remove Intensity Overload Pixels*. This option is checked by default and removes any pixels that have an overload intensity such as pixels taken from a mirror or reflector. The location of the pixels are typically incorrect.
- *Remove Mixed Pixels*. This option is checked by default. It removes pixels that have a mix of return intensities indicating that the target is not reliable, typically occurring on edges of surfaces.
- *Remove Low Quality Pixels*. This option is unchecked by default but allows you to receive pixels with a low quality return.
- *Area Filter*. This is considered Leica's "Intelligent Area Filter". It will smooth out the data based upon the relative position of adjacent points in the scan. The degree that this is done is based upon the slider setting. To turn it off, slide the Area Filter slider to *Min*.

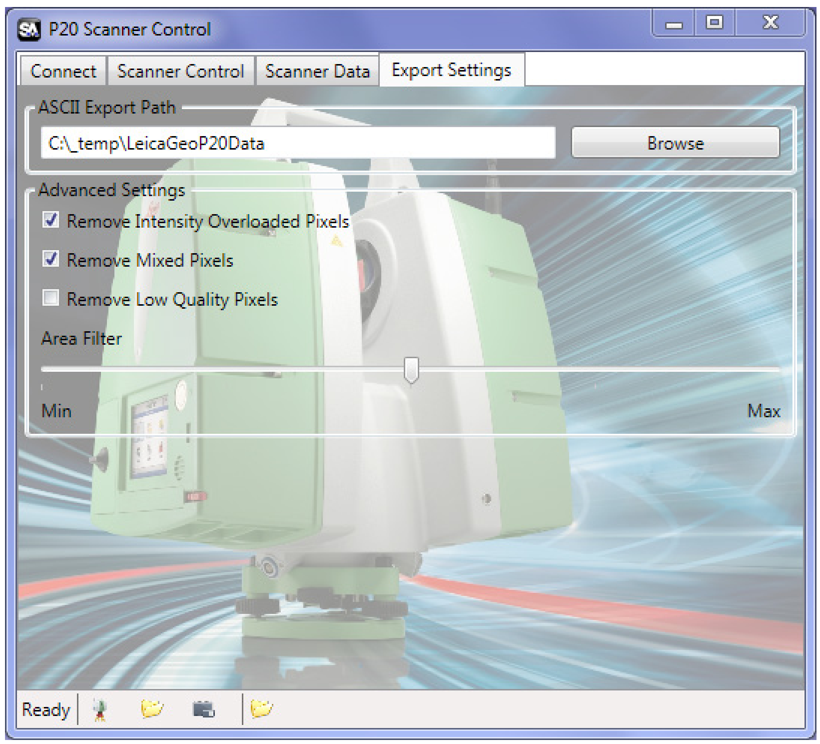


Figure 28-149. Export Settings tab.

Scanner Control

The *Scanner Control* tab defines the scan and controls (start/stop/pause/resume) (Figure 28-150).

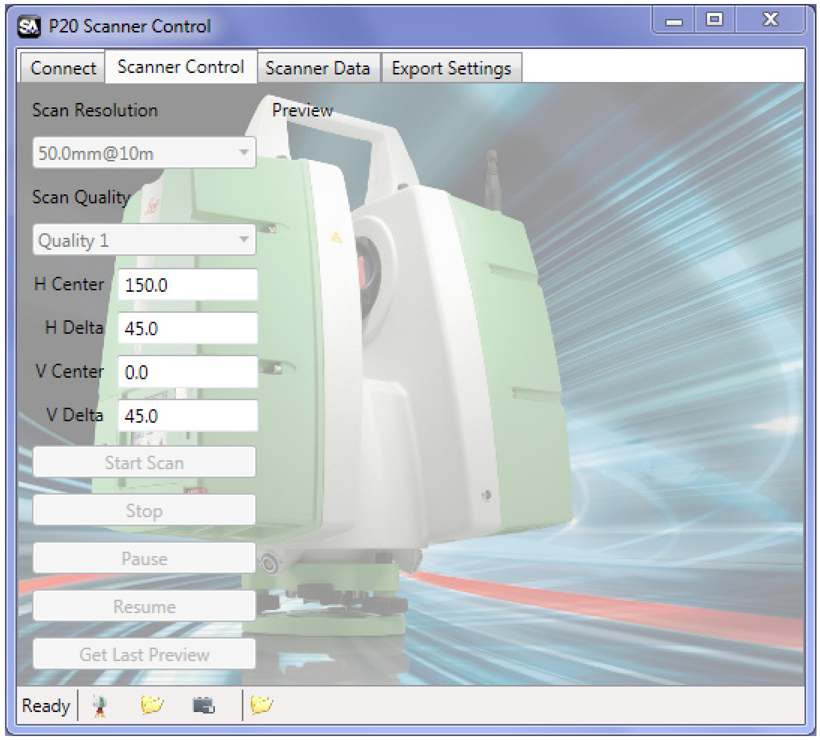


Figure 28-150. Scanner Control tab.

- *Scan Resolution*. This setting defines the density of the cloud. You can chose from 7 preset scan resolution settings defined as the distance between adjacent scan points at the distance specified. This represents the angular spacing of individual points. The points will be closer together at a closer distance and wider at a great distance based on this angular setting. The default setting when connecting to SA is the lowest resolution.
- *Scan Quality*. This setting defines accuracy for a particular point. Higher accuracy results in an increase in the overall measurement time. See chart below (Figure 28-151):

Scan time and resolution (hh:mm:ss)	7 pre-set point spacings (mm at 10 m)				
	Spacing	Quality level			
	mm	1	2	3	4
	50	00:20	00:20	00:28	----
	25	00:33	00:33	00:53	01:43
	12.5	00:58	01:44	03:24	06:46
	6.3	01:49	03:25	06:46	13:30
	3.1	03:30	06:47	13:30	26:59
	1.6	13:33	27:04	54:07	----
	0.8	54:07	1:48:13	----	----

Figure 28-151. Scan time and resolution chart.

- *H Center*. Defines the horizontal angle clockwise about the scanner start location to the center of the scan.
- *H Delta*. Defines the horizontal angular width of the scan about H center.
- *V Center*: Defines the Vertical angle about the scanner start lo-

cation to center of the scan.

- *V Delta*. Defines the vertical angular width of the scan about V center.

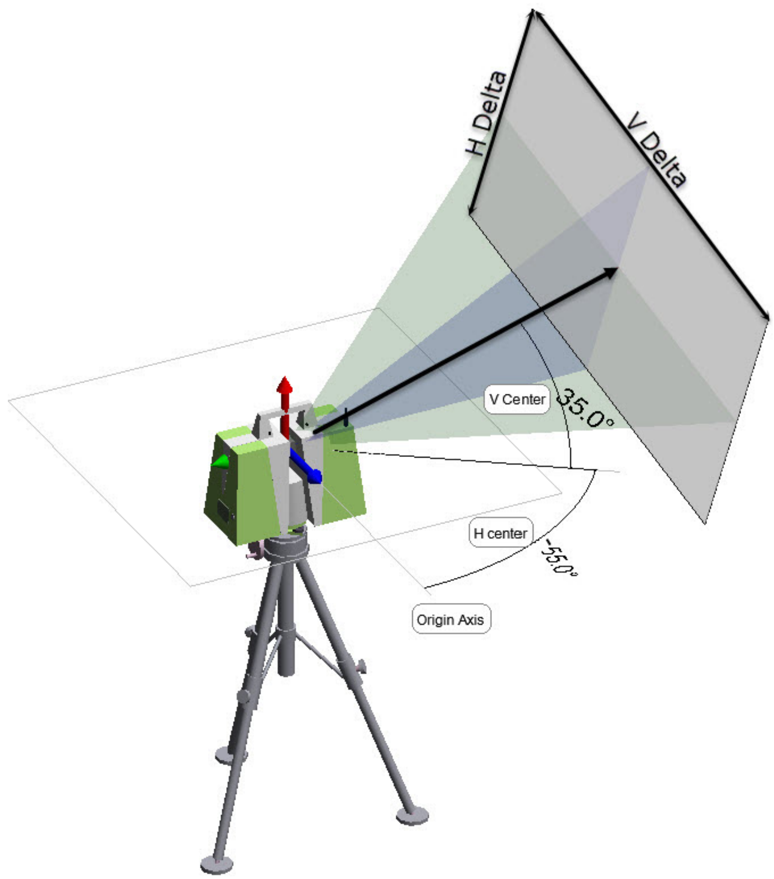


Figure 28-152. Scanning orientation.

Scanner Data

The *Scanner Data* tab provides a view of scan data on the instrument as well as transfer options (see Figure 28-153).

- Refresh Tree. This button updates the tree list on the interface with the latest scan data available from the instrument.
- *Tree View*. This section contains a static list of all the scans available from the instrument at the time the tree was last updated. To ensure you have the latest scan information, click Refresh Tree.
- *SA Cloud Data*. This information is sent with the scan and used to define it in the SA job once the transfer is complete.
- *Right Panel*. The right hand panel of this tab displays informa-

tion on the selected scan as well as a preview of that scan. The colors in the preview are based upon point return intensity.

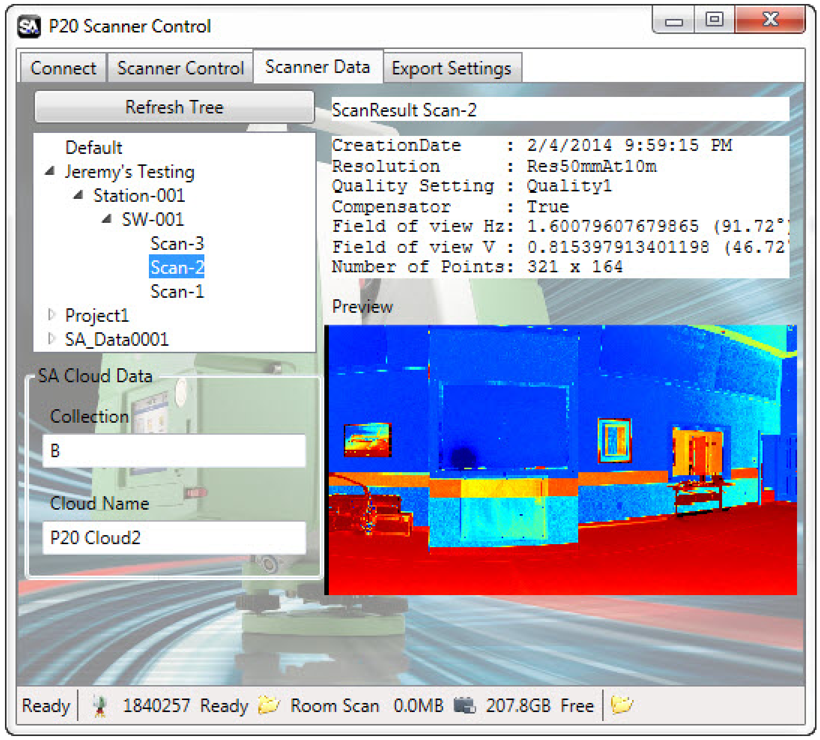


Figure 28-153. Scanner data tab.

Laser Projectors

LPT

Setup the projector following the manufacturer's directions. The LPT functions as a separate computer with a separate IP address and must be connected through a network connection using an Ethernet cable. Ensure that the unit is powered on.

1. Make sure the projector and the PC have compatible IP addresses. This can be done by attaching a monitor, keyboard, and mouse to the projector and verifying the Projector's IP address.
2. Open Windows Explorer on the PC. In the *Address* field, type the computer name of the projector. For example, **`\\LPT10587`**.
3. You will see a dialog box entitled ***Connect to LPT10587*** (for example). Enter your username and password. The default username and password are both **`Admin`**.
4. Once you are logged onto the projector, Windows Explorer will show the hard drives on the projector.
5. Run *RayTracer Operator* on the PC. (*RayTracer Operator* can be run on the projector but it is recommended to run it on the PC.)
6. On the *RayTracer Operator* Login screen, type **`Admin`** for the User ID, and **`Admin`** for the password.
7. You should have a database set up on the PC already, but if not, contact LPT for assistance. (To get started, you can run *RayTracer Administrator* to create a new database or to incorporate tools and parts created 'offline' (see "Using SA to Create Offline Tools and Parts" on page 838). Or, just copy a database from the projector to the PC, and use *RayTracer Administrator* to set it active.)
8. Be sure the database you wish to use is indicated in the *RayTracer Operator* login, and press OK.
9. Once *Operator* is running, check the 'Laser Name' list in the application's main window. Ensure that the projector you are connecting to is in the list, and that it is checked. If not, you must check it and 'take ownership' of the projector. Otherwise, commands will appear to succeed, but none will be carried out.

Running the Projector

1. Select **Instrument>Add...** and choose the LPT Laser Projector

from the Instrument List (Figure 28-154).

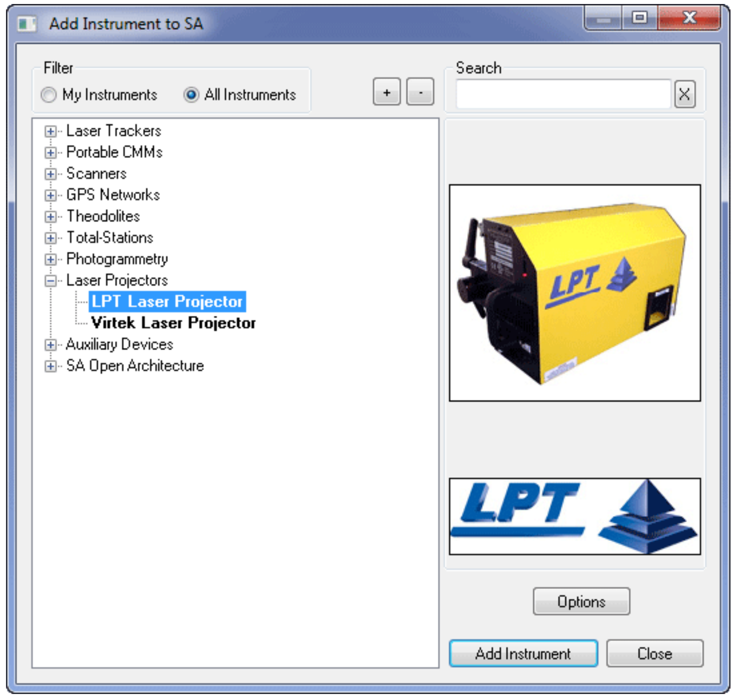


Figure 28-154. Adding an LPT to the SA job.

2. Select **Instrument>Run Interface Module**. Select **LPT Laser Projector** from the list.

3. In this dialog (Figure 28-155) two options appear: you can connect to a computer running *RayTracer Operator* or work Offline. Be sure to select **Connect to Operator**, and indicate the name of the computer running Operator. (The 'This Computer' button will fill in your pc's name in case you are running *Operator* on the PC as recommended.)

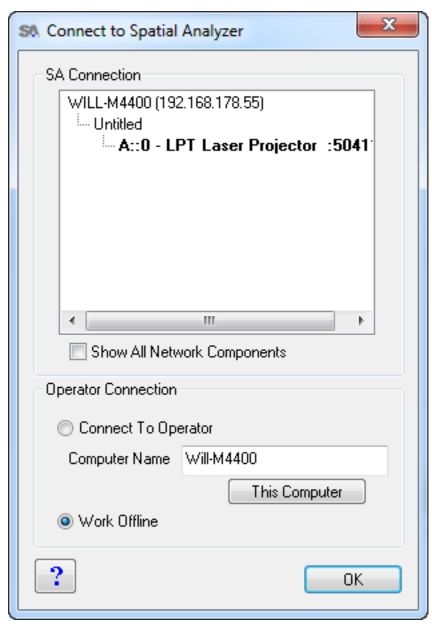


Figure 28-155. The Connect to SpatialAnalyzer dialog.

4. The progress report at the bottom of the interface window

will indicate actions as they take place.

Using SA to Create Offline Tools and Parts

1. Run SA and add an LPT projector. Then run the LPT Laser Projector Interface, and select *Work Offline* in the interface's ***Connect to SA*** dialog (see "Running the Projector" on page 836).
2. The interface will then run in *OFFLINE Mode*. You will select the path for the 'Offline Tool' and 'Offline Part' xml files in the interface. You can create as many tools and parts as you wish. It is highly recommended that tool and part 'sets' be created. In other words, for any set of parts, create a corresponding tool which represents the projector's location while projecting those parts. These tool and part sets can be organized in folders.

To Create a tool

- Set the path and file name for the tool in the 'Offline Tool' area of the interface. In SA, select **Instrument>Laser Projector>Alignment**. You'll be prompted to select the instrument (projector) and the Group (set of alignment points). The points will be written to the file indicated by the path in the 'Offline Tool' area of the interface.

To Create a Part

1. Set the path and file name for the part in the 'Add to Offline Part' area of the interface. If you are creating a part from anything other than points, be sure to set the Ply Filter in the interface (see section below for use of the Ply Filter). In SA, select **Instrument>Laser Projector>Project Objects**. You'll be prompted to select the instrument (projector), and the Objects to project (geometric entities, points, etc.). You will be prompted for a Ply Name for the object in the Part file. The objects will be written to the ply in the file indicated by the path in the 'Add to Offline Part' area of the interface, and codes to ensure uniqueness in the part file will be added to the ply name. You can continue to add objects (plies) to the same part file, and/or create new part files as desired.
2. Once desired set(s) of tools and parts have been created, use *RayTracer Administrator* to add these sets to a projector database.
3. During the creation of tools, you may wish to start up the projector (described below) and ensure the validity of the tool by actually aligning the projector to the tool. Then the parts corresponding to that tool location can be projected to ensure their validity. All this is described below.

Projecting within SA

Alignment

There are several possible ways to perform this action:

1. If you have been working with a projector in SA, and you are continuing work on a file, the projector's location/orientation will be persisted by SA and the interface. If this is the case, hit the 'Auto Align' button in the 'Align' section of the interface. Operator will pop to the forefront, and an Auto-alignment will take place. Operator's view area will go green upon a successful alignment. NOTE: For the following methods, be sure to use the 'Toggle FOV' (field of view), and 'Toggle +' (center cross) buttons in the interface to aid in the initial location of the projector.
2. If you are starting a new job, you can create alignment points (a tool) in SA, and align the projector to that tool. In the interface, be sure that 'Add to Projector Part' is selected in the 'Projections from SA' area (in 'Online' mode, if 'Add to Offline Part' is selected, Tools will be created in the last used offline tool.) Create a group of points in SA which you wish to use as the alignment tool. In the SA Instrument menu, select Laser Projector >> Alignment. You'll be prompted to select the instrument (projector), and the Group (set of alignment points). Once the group is selected, you will be prompted for the Tool Name. A tool by this name, containing the points from the selected Group, will be added to Operator's database. You will see the tool name and point appear in Operator if it is visible. Finally, the interface will instruct you to manually align the projector in Operator.... just maximize operator, and hit the 'Manual Alignment' button. Guide the projector to each indicated point, and left click when near the point. After 4 such points are Accepted, Operator will take over and finish the alignment. (If the alignment fails, Operator will give you an oportunity to override the alignment and continue, but you should check for the source of the failure - it is generally an rms error exceeded in the alignment calculation.) Once you have performed this manual alignment procedure, you should be able to Auto-align (as described above) so long as the projector is not moved, and the Tool in Operator is not changed.
3. If you have previously created Tools offline (as described above), you can use LPT RayTracer Administrator to add the tool(s) to your database, and they can then be used by selecting them from the 'Tool:' list in Operator. As always, a one time manual alignment will have to be performed in Operator, and

then Auto-align from the interface can be used until the projector is moved, or the Tool is changed.

Projecting

1. For projecting within the current Part, simply use the 'Next', 'Current', and 'Previous' buttons in the interface to toggle through plies. The 'Pause' and 'Resume' buttons also apply to the current Part.
2. Use LPT RayTracer Administrator to add any offline parts you have created (as described above) to the current database.
3. To add a part to the current database, and project it immediately: Be sure the projector has a valid alignment (Operator's view screen is green.) Be sure 'Add to Projector Part' is selected in the interface. Edit the name of the part if desired. (As with offline parts, you are free to continue to add projection objects (plies) to the part as many times as you wish.) If you are creating a part from anything other than points, be sure to set the Ply Filter in the interface (see section below for use of the Ply Filter). In SA, go to the 'Instrument' menu, and select **Laser Projector>Project Objects**. You'll be prompted to select the instrument (projector), and the Objects to project (geometric entities, points, etc.). If this is a new part, the part will be added to the current Database. The selected projection object(s) (i.e. the ply) will be added to the part and projected immediately. The ply name in the part will be "CommandedProjection_X", where X is a code to ensure uniqueness.

Using the Ply Filter

In the interface's 'Projections from SA' area is a 'Ply Filter' button. The button shows information about the current active filter option.

Filter Options Are:

- **None.** Just project everything as is
- **Minimum Angle.** Starting with the first point, look at all consecutive sets of 3 points in the ply, e.g. 1, 2, and 3. Make vectors 1to2 and 2to3. If the angle between 1to2 and 2to3 is less than the selected minimum angle, point 2 is eliminated, and the evaluation continues.
- **Number of Points.** Numerically spaced points are eliminated until the desired maximum number of points is reached.
- **Minimum Distance.** Starting with the first point, look at all consecutive sets of 2 points in the ply, e.g. 1 and 2. If the distance between 1 and 2 is less than the selected minimum distance, point 2 is eliminated, and the evaluation continues.

The ply filter is applied to any entity which comprises a continuous projection along some curve. (Anything except points.)

The current active filter (indicated on the filter button) is applied to all plies coming into the interface, whether they are to be projected immediately, added to the current database part, or added to an off-line part.

Assembly Guidance Laser Projector

Hardware Setup

Setup the unit following the manufacturer's directions with an unrestricted view of the work area.

Software Setup

You can download and install the latest drivers from http://www.kinematics.com/ftp/SA/Install/Driver%20Downloads/Projectors/AssemblyGuidance%20LaserProjector/ . You will need the following files:

- `LASERGUIDESDK_3_03_13.exe` or later
- `Register LaserGuide dll.bat`
- `Unregister LaserGuide dll.bat`

Installation

1. Run the file `LASERGUIDESDK_3_03_13.exe` or later on your machine. This will walk you through the installation process of the LaserGuideSDK used by SA to communicate with the device.
2. An install directory will be created under C:\Program Files (x86)\AGS\LaserGuideSDK as well as a shortcut link on the desktop to the test application for the AG projector.
3. Restart your computer.
4. Move the Register `LaserGuide dll.bat` into the SA install folder typically located here: C:\Program Files (x86)\New River Kinematics\SpatialAnalyzer 2013.12.10.
5. Double click *"Register LaserGuide dll"* to run the file from this directory. If you have previously registered from a different directory you will need to run the `Unregister LaserGuilde dll.bat` utility first.

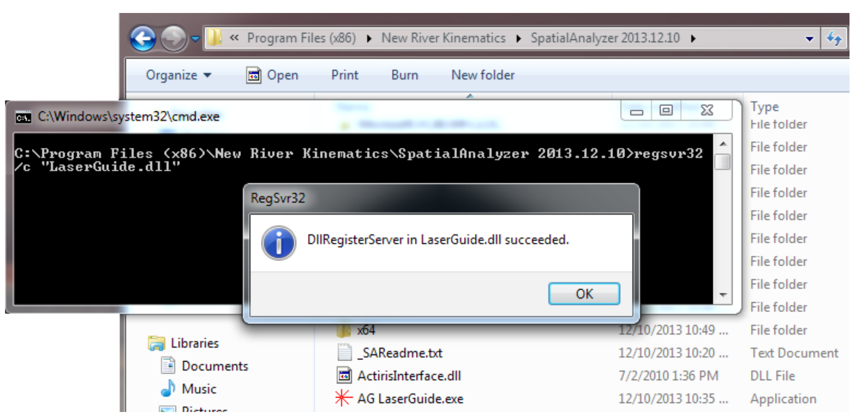


Figure 28-156. Windows registration.

Network Configuration

Be sure to create a fixed IP address on your computer that will be compatible with the IP address of your projector(s). The default address should be stamped on the front of your projector. For example, if your projector's IP address is 10.1.1.1, then you need to change your computer's IP address to 10.1.1.X; where X is between 1 and 255.

Starting the Interface

1. Select **Instrument>Add** and choose the Assembly Guidance Laser Projector.

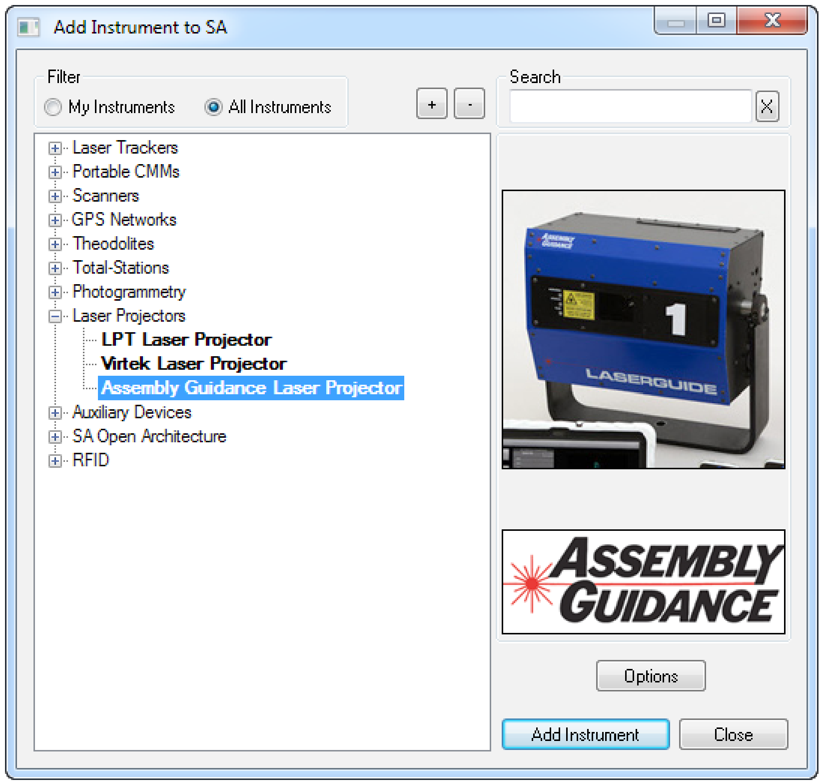


Figure 28-157. Adding an Assembly Guidance Projector to SA.

2. Now run the instrument interface module under **Instrument>Run Interface Module and Connect**.

3. Enter the projector's IP address and click the Connect >> button on the ***AG LaserGuide*** (see Figure 28-158) dialog. For more detailed connection information, you click on the Show Projector Window (see Figure 28-158) button to bring up the Assembly Guidance Contol Utilities.

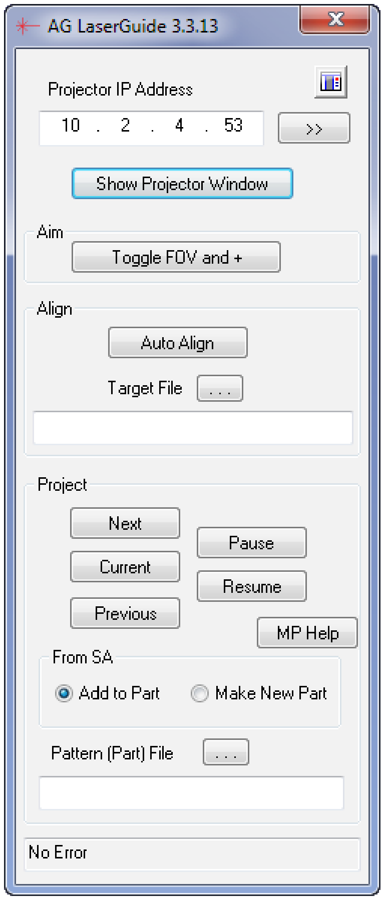


Figure 28-158. AG Laser-Guide dialog.

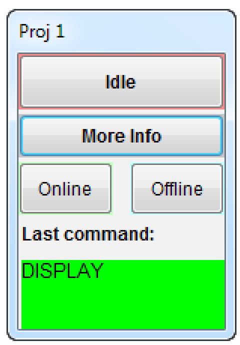


Figure 28-159. Projector window.

Aligning the Projector

There are two methods of aligning the projector in SA.: automatic and manual. To perform an auto alignment, use the browse button in the ***AG LaserGuide*** dialog (see Figure 28-158) to find a preexisting registration file to use. Click the Auto Align button, and the projector will auto measure the alignment points and the transformation from the measured points to the current point locations will be used to align the projector within SA.

To Perform a Manual Alignment

1. Measure six reflector positions with a separate instrument within the field of view of the projector. Located in the *Aim* section of the instrument interface is the Toggle FOV and +

button. This can be used to display the projector region and also to import a set of known location points.

2. Select **Instrument>Laser Projector>Alignment**.
3. Select the instrument to align and then the measured point group for alignment.
4. Browse to a location to save a Registration data file in .txt format.
5. Use the manual alignment utility pictured in (Figure 28-160) to point the laser at each registration point in order by name. To do so, click and hold while dragging the mouse. This drives the laser pointer relative to the cross hairs in the dialog and, in turn, drives the projector beam to the reflector point. Once aimed correctly, right-click to scan the target.

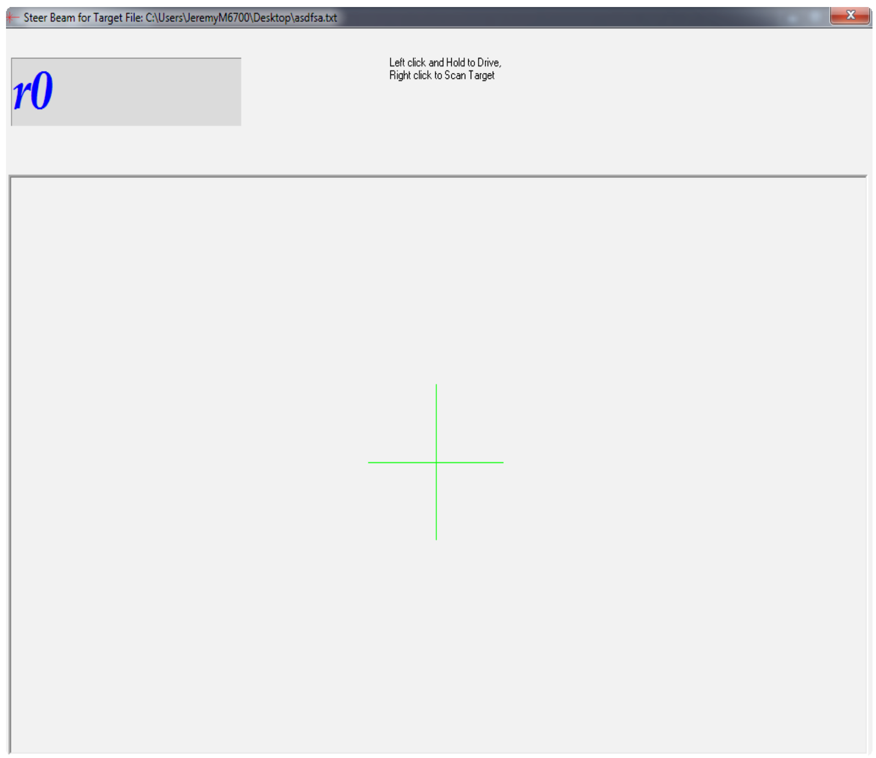


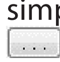
Figure 28-160. Steer Beam for Target dialog.

6. After each of the individual target locations have been identified in this fashion, these points will automatically measured and the alignment will be performed.

Point and Object Projection

The Assembly Guidance Interface provides the tools to project points as well as objects in SA. To project an object it must be added to a part file. These part files are then used to build the laser path definition for the projector. A single part may contain several patterns and additional patterns can be added to an existing part file.

Projecting Part Files

1. In order to project patterns from a previously created part file, simply browse to the file using the Pattern (Part) File button  in the interface.
2. Click Current to project the part shown in the selection path. You can utilize the Pause and Resume buttons to control projection of the current part file and the patterns it contains.
3. Select Next and Previous to toggle through part files in the same folder as the current part shown in the *Part File Edit* box.

Building/Editing Part Files and Projecting Objects

1. In the *From SA* section of the interface , select Add to Part or Make New Part.
2. Select **Instrument>Laser Projector>Project Objects**. You will then be prompted to select the instrument (if you have more than one in the job).
3. Now select the objects you wish to project. Doing so will create a file using the name of the first object selected for projection if you have Make New Part selected. If you chose Add to Part, it will add the selected objects to the current part file.

> **Tip:** Objects cannot be removed from a part file through SA.

Tutorials

CHAPTER 29

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. After completing a tutorial, you should feel more comfortable with using the covered areas of the software in real-world use.

Working With Frames

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will explore working with different coordinate frames, and see how the working coordinate frame affects the values reported in SpatialAnalyzer.
- **Areas Covered.** Creating points and geometry, geometry fitting, frame construction/activation, transformations, and point/object queries.
- **Time to Complete.** Approximately 30 minutes.

Creating Entities to Work With

For this tutorial, we want to create some points and geometry to play around with, so we'll start out by creating those entities first.

1. In SA, start a new job file by selecting **File>New** from the menu or choosing the New File Icon from the main toolbar.
2. Let's create a few points to work with in the workspace. To do this, go to **Construct>Point(s)>Enter** or press `Ctrl+P`.
3. In the ***Add Points to Model*** dialog (Figure 29-1), let's place

the points into a collection called *Nominal* and a point group called *NominalPoints*. We'll give the first point a name of *1*. Give the point an X coordinate of *15*, a Y coordinate of *10*, and a Z coordinate of *7*. We're entering these coordinates as Cartesian (XYZ) coordinates, so we'll leave that setting at the default:

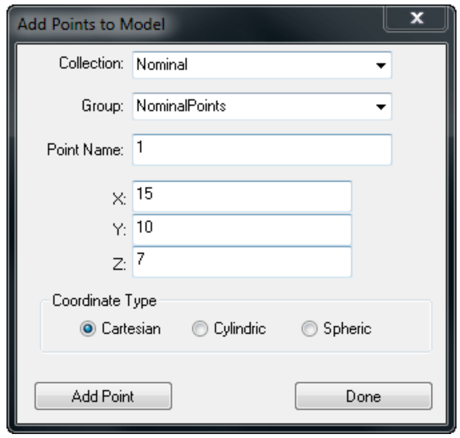


Figure 29-1. Adding a point.

4. Click the Add Point button, then click Done. In the graphical view, you should see a point created at the specified coordinate. If you don't see it, autoscale the view by clicking the Autoscale button , pressing **Alt+A**, or choosing **View> Zoom> Autoscale**.

5. In the tree view, you should see that the *Nominal* collection now exists. Since it did not exist when you created the point, SA created this collection for you.

6. In the tree, (if necessary) click the disclosure triangle next to *Nominal* to expand the list of items in the Nominal collection. Note that there is an item in the list named *Point Groups*. SA sorts items in the tree based on their category. Expand the *Point Groups* category, and you'll see that SA created the *NominalPoints* point group, because it did not already exist.

Note: Depending on how your copy of SA is set up, your active collection may not be named A.

7. Expand the *NominalPoints* point group to show all points contained within the point group, then click the triangle next to the single point to show the coordinate of the point named *1*. You should see the coordinates that you just entered (Figure 29-2).

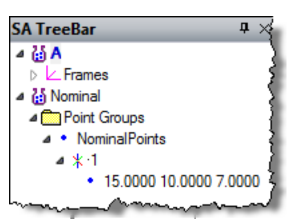


Figure 29-2. The newly-added point.

8. The default collection in your SA file is currently ac-

tive, because it is depicted in bold blue font in the tree. Let's activate our new *Nominal* collection. Right-click on the *Nominal* text in the tree and, from the context menu, choose **Make Active Default Collection**. Now, any geometry that you create will be placed into the active *Nominal* collection.

9. Now let's create another point. SA saves a history of the most recently used commands so that you don't have to select them from the menu again if you're repeating a single command several times. To use this history, hold down **`Ctrl+Shift+Tab`**. In the upper-left corner of the graphical view, a menu should appear showing your recent commands since you started SA, with the most recent command at the top (Figure 29-3).

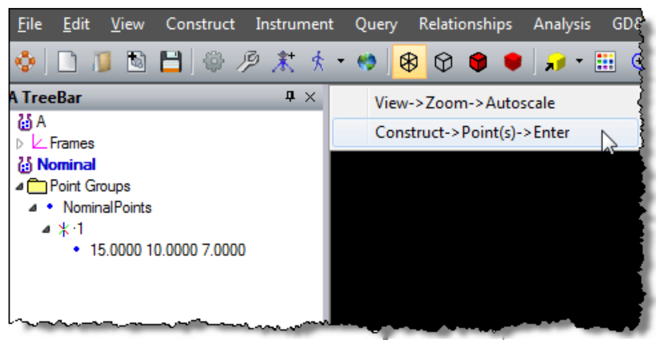


Figure 29-3. The list of recent commands.

10. Choose the **Construct>Point(s)>Enter** item from the list. Notice that SA remembers the last collection and point group that was used for this command. Enter *2* for the point name, and give it a coordinate of (*20, 20, 20*). Click the Add Point button.

11. After clicking the button, notice that SA automatically increments the point name by one. This behavior is found commonly in different parts of SA, and is intended to save time from typing, since incrementing names by one is so common. Change the coordinates to (*5,5,5*) and click the Add Point button again to create the third point, then click the Done button to close the dialog.

12. Now let's create a plane to work with. From the menu, select **Construct>Plane(s)>Enter**. The ***Plane*** dialog will appear (see Figure 29-4 on page 850), allowing you to specify properties for the newly created plane.

13. Leave the plane's name at the default of ***Plane***, and click the Transform button. Locate the plane at (*35, 0, 0*) and assign it an *Rx* value of *90°*. Click the Update button.

Note: The Autoscale keyboard shortcut of Alt+A will only work if the graphical view has the focus, by clicking once on it.

14. If necessary, autoscale or zoom out so you can see the newly-created plane relative to the points.

15. With the plane in view, set the *Rx* value back to *0°*, and change

Ry to *90°*. Again, hit the Update button to see the plane change orientation. Now, click the increment arrows next to the *Ry* field and notice how the plane is rotating around the active coordinate frame's Y axis. We'll set the *Ry* value to *160°*. Click the Update button, then click the "X" in the corner of the ***Plane Position*** dialog to close the window.

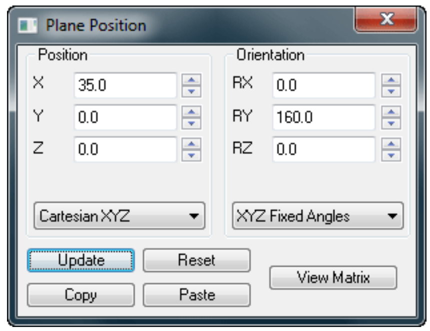


Figure 29-4. The Plane Position dialog.

16. In SA (as in mathematics), planes extend infinitely in two dimensions. So, the boundaries that are depicted in the graphical view are purely to give you an idea of the location and orientation of the plane--the actual plane is not bounded. Notice also that planes have an arrow drawn along their normal to indicate the direction of the plane. All geometry and surfaces have normal directions, which define the *positive side* of the geometry. All measurements with offsets in SA are always compensated relative to this positive side:

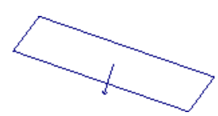


Figure 29-5. The positive side of a plane is indicated with an arrow.

17. The plane's normal direction can be reversed to face the opposite direction. In the ***Plane*** dialog, click the Reverse button.

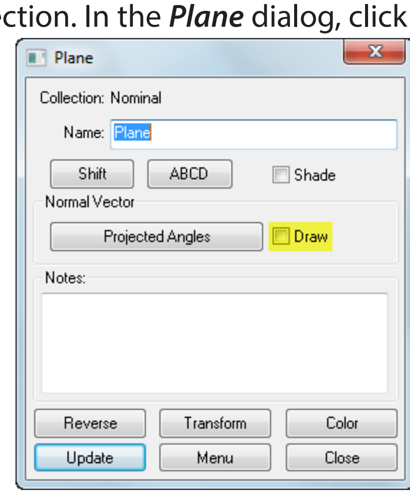


Figure 29-6. Selecting the option to draw a plane's normal vector.

Notice how the arrow flips to face the opposite direction. Turn off the plane's arrow so that it's no longer drawn in the graphical view by deselecting the **Draw** checkbox in the ***Plane*** dialog

(Figure 29-6). Close the dialog by clicking the Close button or the "X" in the corner of the window.

18. Finally, let's create a circle to work with. This time, rather than creating a circle and specifying its position and orientation, let's fit a circle to the three points that we created earlier. Orient your view so that you can see all three points. From the menus, choose **Construct>Circle(s)>Fit to Points** or press **Ctrl+Alt+C**. You will be prompted for the points that will define the circle fit. Hold down the **Shift** key, and click-drag to define a rectangle around the three points. In the bottom-left corner of the SA window, you should see a message saying "Picked 3 Points". If you look in the treebar, the selected points will highlight bold and blue, and they will change to a highlight color in the graphical view. Press **Enter** to accept the selection.

19. You will be presented with the ***Circle Fit*** dialog (Figure 29-7), which allows you to define the parameters and results of the fit operation. Above the box with the red background, you'll see your points (1-3) listed in a box, along with the resulting error of those points from the resulting circle. Since three points perfectly define a circle, our resulting circle will end up passing through all three points exactly. Notice that the *Planar* and *Radial* errors are zero, and the error magnitudes are also zero:

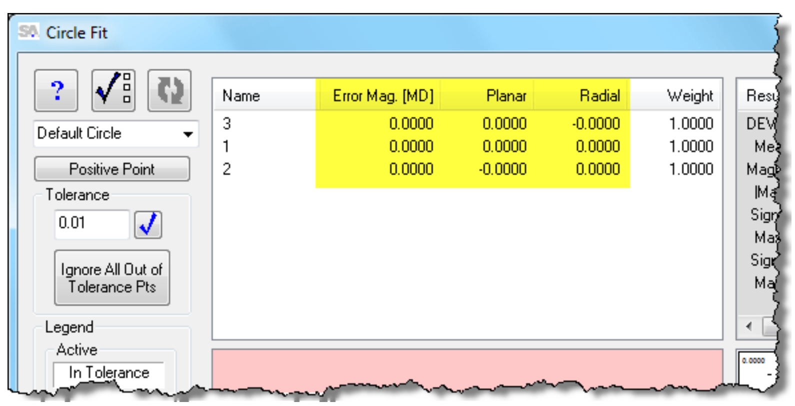


Figure 29-7. The Circle Fit dialog.

20. In the bottom-left corner of the ***Circle Fit*** dialog, ensure that only the **Make Cardinal Points** and **Make Geometry options** are selected, then click the OK button. SA will ask you for the name of the group in which you'd like to place the cardinal points. Enter *NominalPoints* and click OK.

21. The ***Circle*** dialog will appear. Give the circle a name of *MyCircle*. Notice the **Draw** checkbox again. Select it to see that circles have a normal direction as well, and notice that two points have been created from our circle fit: one on the circle's center, and another along the normal of the circle. These are the *Cardinal Points*. Turn the **Draw** checkbox back off, and click the Close button.

In the next section, we'll create some coordinate frames using a few different methods.

Creating Frames

Coordinate frames are crucial to many functions in SpatialAnalyzer and metrology in general. They define the locations and directions that are important to you or the features that you're measuring. SA has a number of powerful ways to create coordinate frames to meet all of the needs you might encounter on a day-to-day basis.

1. In the graphical view, note that the *World* frame is currently the working frame. The working frame is rendered with colored arrows representing each of the axis directions: red for X, green for Y, and blue for Z. In the tree, you will also notice that the working coordinate frame is rendered in a bold blue font.
2. Let's create a new coordinate system, offset it from the current *World* frame, and change its orientation. From the menu, choose **Construct>Frame>Enter**. Give the frame the name *Offset Frame*. This command creates a new coordinate frame with the same position and orientation as the working coordinate frame. We want to move our new frame. Click the Transform button to bring up the ***Frame Position*** dialog (Figure 29-8). Offset the frame by *10* units in X, Y, and Z, and also rotate the frame by *+45*° about the Z axis. Click the Update button to see the new frame move to the new position:

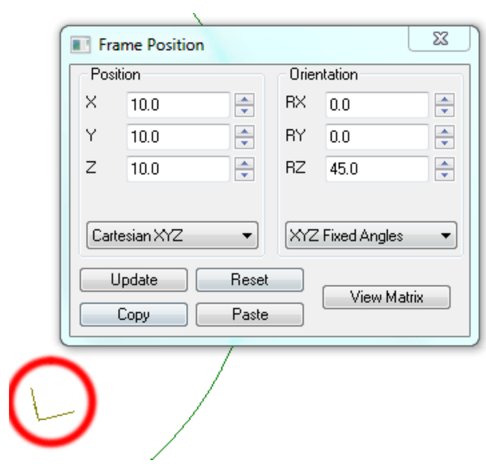


Figure 29-8. Creating a new frame.

3. Close the ***Frame Position*** and ***Frame*** dialogs, and verify for yourself visually that the new frame's origin is located at (*10, 10, 10*) in the active (World) frame's coordinate system. Verify also that your constructed frame is rotated 45° about the active coordinate frame's Z-axis. Notice that SA uses the right-hand rule. If the thumb of your right hand is pointed along

the axis of rotation, then curling your thumbs from one of the remaining axes to the other will define the sense of positive rotation. Our newly-created frame is drawn as a set of 3 orthogonal lines. The thicker colored arrows are only drawn on the coordinate frame that is currently active.

4. Let's activate our newly-created coordinate frame. Click on the A::World button in the WCF Toolbar (your button may look slightly different depending on the name of your default collection):

Figure 29-9. The WCF Toolbar.

5. From the ***Object Selection*** dialog, double-click the *Offset Frame* frame. The button's title will immediately change to reflect the new working frame, and the new frame will be rendered as active.

6. Now let's construct a frame on our circle. Objects can have frames constructed on them. The specific behavior is dependent on the type of object, but for a circle, the frame will be built with its origin on the center of the circle and its Z-axis along the circle's normal. From the menu, select **Construct>Frame>On an Object**. When prompted for the object, double-click the *MyCircle* circle. Name our new frame *Frame On Circle* and close the ***Frame*** dialog.

7. Finally, let's create a coordinate frame whose origin is at one of our circle points, whose X-axis points directly to the second, and whose Z axis clocks along the point defining the circle's normal. From the menu, choose **Construct>Frame>3 Points>Origin, X axis>Point on XZ Plane**. For the origin point, double-click the point in the tree named *Nominal::NominalPoints::1*. Note that SA uses a convention to denote the "complete location" of an entity. For objects, it includes the collection, followed by two colons, then the object name. For points, it includes the collection, followedby two colons, followed by the group name, followed by two more colons, and finally followed by the point name.

8. Let's pick the Point along X axis by clicking on it visually in the graphical view. We want to pick the *Nominal::NominalPoints::2* point, but how do we know which is which? From the menu, choose **View>Show Point Labels** (or press `Alt+L`). This will toggle on and off the display of labels for the points. Ensure that the Point along X axis prompt is still displayed, and in the graphical view, double-click the point labeled 2.

9. When prompted for the Point along XZ plane, double-click

the *Point on Normal* point. In the ***Frame*** dialog, give the frame a name of *Clocked Frame* and close the dialog.

Activating Frames

When analysis values are reported by SpatialAnalyzer, they are almost always reported in the working coordinate frame, unless otherwise specified. For example, when you query two points to determine the distance between them, you will be presented with the magnitude of the distance (in the current units), which will remain the same regardless of the working frame. However, you'll also get delta values along each active frame axis (dX, dY, dZ). In some cases, you can explicitly specify the frame to report results in, regardless of which frame is currently active.

1. With the *Offset Frame* frame still active, right-click on the *NominalPoints* point group and choose **Expand All Entries** from the context menu. You should now be able to view the X/Y/Z coordinates for all of the point group's points (Figure 29-10).

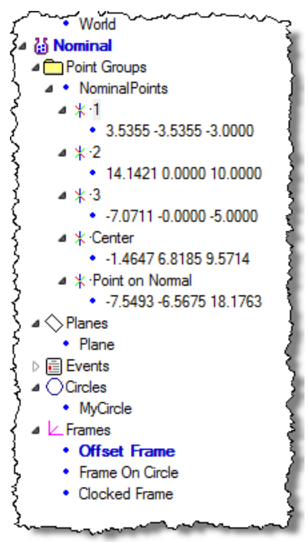


Figure 29-10. Viewing the point coordinates.

2. In the tree, right-click on *Clocked Frame* and choose **Make Working Frame** from the context menu. Notice that the point coordinates change immediately to reflect the new coordinate frame's position and orientation.
3. Let's look at the point coordinates further. Point *1* is located at (0, 0, 0) because we used it to define the origin of our working frame. Point *2* has Y and Z coordinates of 0 because the X axis points directly to it. Finally, the *Point On Normal* point has a Y coordinate of zero, since we defined the frame so that the point lies in the frame's XZ plane.
4. Next, activate the *Frame on Circle* frame. The *Center* point is at the origin, and the *Point on Normal* point is along the Z axis.

In the tree, right-click on *MyCircle* and choose *Properties* from the menu. In the ***Circle*** dialog, click the Transform button. Notice that the circle has position and orientation values of zero, since the active frame was created on the circle. (In other words, the frame was created at the circle's internal object origin). Close both dialogs.

5. Now we'll measure a few distances. From the menu, choose **Query>Point to>Point** (or press `Ctrl+D`). When prompted, double-click the *Center* point for the 1st point and *Point on Normal* for the second. In the ***Query Results*** dialog, the two points have dX and dY values of zero (since they both lie along the Z axis of the active frame). If you were paying close attention (or have a photographic memory), you'll notice that the distance between the two points is the same as the circle radius.

6. This time, activate the *Offset Frame* frame and follow the instructions in the previous step to compare the two points again. This time, notice that the two points have nonzero dX, dY, and dZ values, since they do not lie along the direction of any of the active frame's axes.

7. Let's determine the distance of one of our points from our plane. From the menu, choose **Query>Point to>Object**. Double-click *Point on Normal* for the point, and *Plane* as the plane. The ***Query Results*** dialog will display the distance of the point from the plane along the active frame's X, Y, and Z axes.

Conclusion

In this tutorial, we saw ways to create different geometric entities such as points, circles, and planes. We also explored a few methods of frame construction, and saw how results reported from commands can be influenced by the working coordinate frame.

Construct and Modify Objects

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to construct and modify objects.
- **Areas Covered.** Construct simple geometry, move an object, change object color and rename.
- **Time to Complete.** Approximately 10 minutes.

Creating and Modifying a Sphere

In this tutorial we want to become familiar with the basic object operations in SpatialAnalyzer.

1. Let's construct a sphere by using **Construct>Spheres>Enter**. The ***Sphere*** properties dialog will appear (Figure 29-11). Change the name to *MySphere*, then close the dialog.

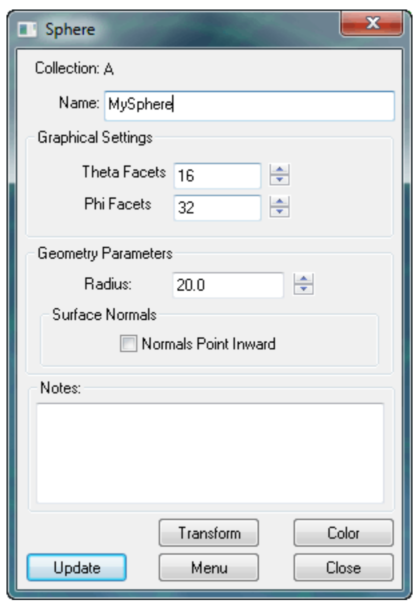


Figure 29-11. Constructing a sphere.

2. The newly-created sphere will display in the graphical view and the tree (Figure 29-12).

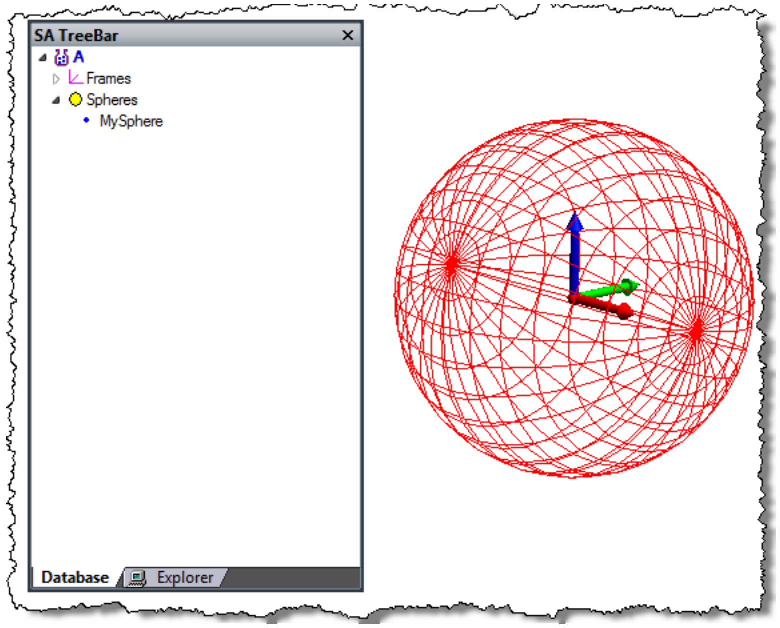


Figure 29-12. The newly-constructed sphere.

3. Now let's change the shading from wireframe to solid. Go to the toolbar and click the solid render icon  . The sphere will now appear shaded.

4. Change the color of the sphere to green. Double-click *MySphere* in the tree. The sphere properties box will appear. Select the color button and choose a green color from the color palette, then press OK.

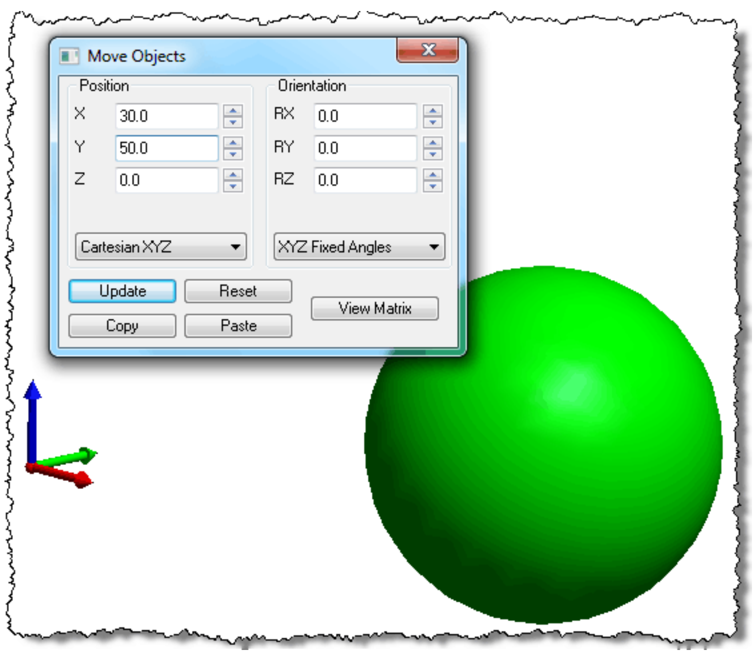


Figure 29-13. Moving the sphere.

5. Let's move the sphere to *30* inches in X and *50* inches in Y. Right-click *MySphere* and select **Move**. Enter in the values for X and Y and press Update (Figure 29-13).

6. Now copy the sphere and move it to *50* inches in Z. Right-click

MySphere and select **Copy**. Enter *50* for the Z position.

7. Now rename the sphere to *MySecondSphere*. Right-click *MySphere1* and select **Properties**. Change the name to *MySecondSphere* in the *name* field (Figure 29-14).

8. Now change the color to blue. This time change the active color to blue using the color icon . Once blue is selected use the color objects icon and single-click the sphere in the graphical view.

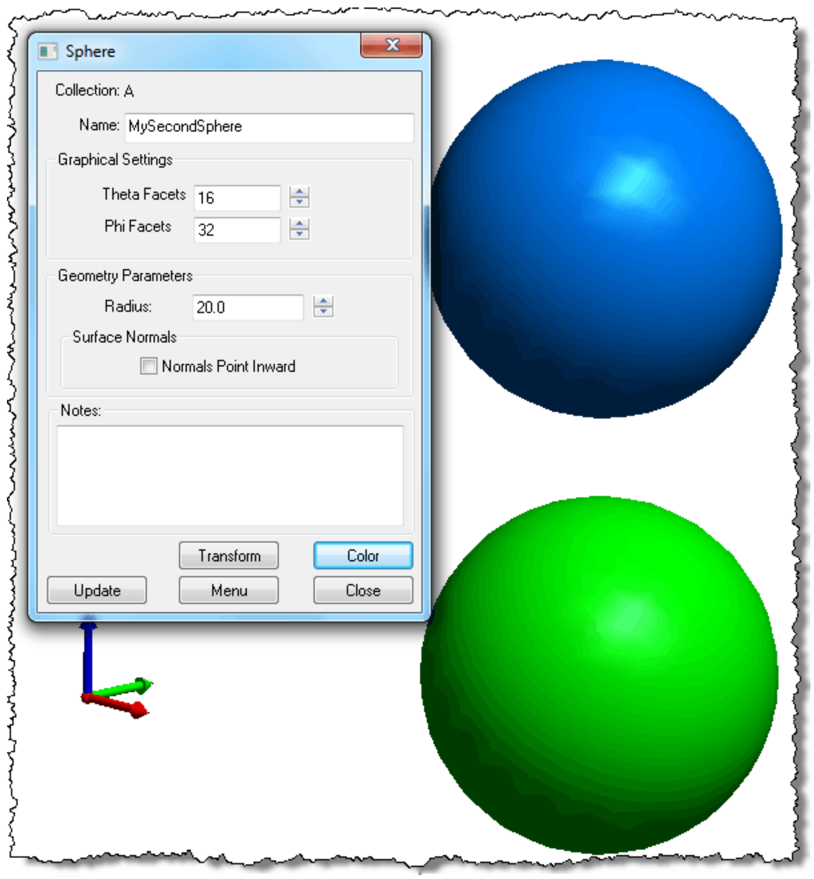


Figure 29-14. Setting the name of the sphere.

Conclusion

After completing this tutorial you should feel comfortable with creating, moving, and editing objects. Remember that most properties can be accessed by right-clicking an object in the tree or the graphical view.

View Control

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover basic view control functions.
- **Areas Covered.** Hiding and Showing of objects, background color, panning, zooming and rotating.
- **Time to Complete.** Approximately 15 minutes.

Changing Render Settings

Here we will change the render mode for the SA graphics from wire-frame to solid.

1. Open `Query Points to Surface.xit` from the Samples directory under **Help>Open Sample SA Files**. When a file is opened, surfaces are by default rendered in wireframe mode in order to open the file as quickly as possible.
2. The Render settings can be changed with the render mode icons . Try each of the Render settings. Now render the model solid by using the Solid icon (Figure 29-15).

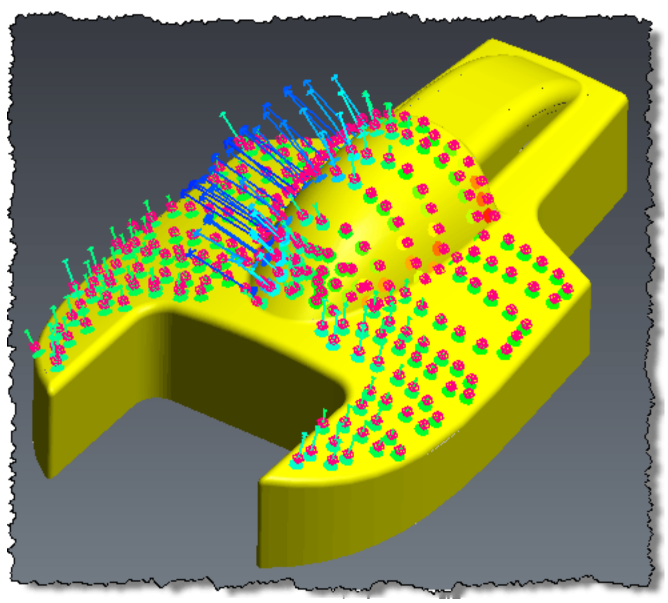


Figure 29-15. Rendering in solid-shaded mode.

Change Background Color

Let's change the background color from a gradient to a solid color.

3. Access the background color options by using the Back-ground button on the main toolbar.
4. In the ***Background*** dialog (Figure 29-16), select the **Solid** color radio button. Now select a solid color by clicking the current

solid color swatch. The color palette will display--choose a green color and press OK. Exit the background color dialog by pressing OK. After your eyes adjust...go back and change the background color to white by using the shortcut for white.

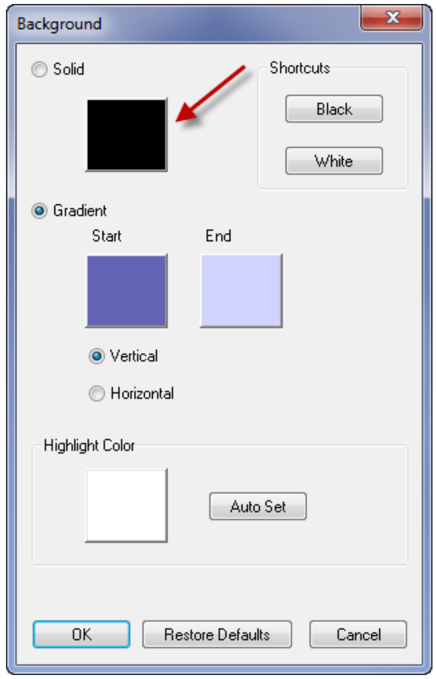


Figure 29-16. Changing the background color.

Hide and Show

Let's experiment with the ways to hide and show points, objects, and other SA entities.

1. Hide the vector group called *Boat Error_Vectors_Blotches*. Expand the *Vector Group* category in the tree. Now right-click *Boat Error_Vectors_Blotches* and uncheck **Show**. The object name will be greyed out in the tree (Figure 29-17). To show a hidden item, simply select **Show** again.

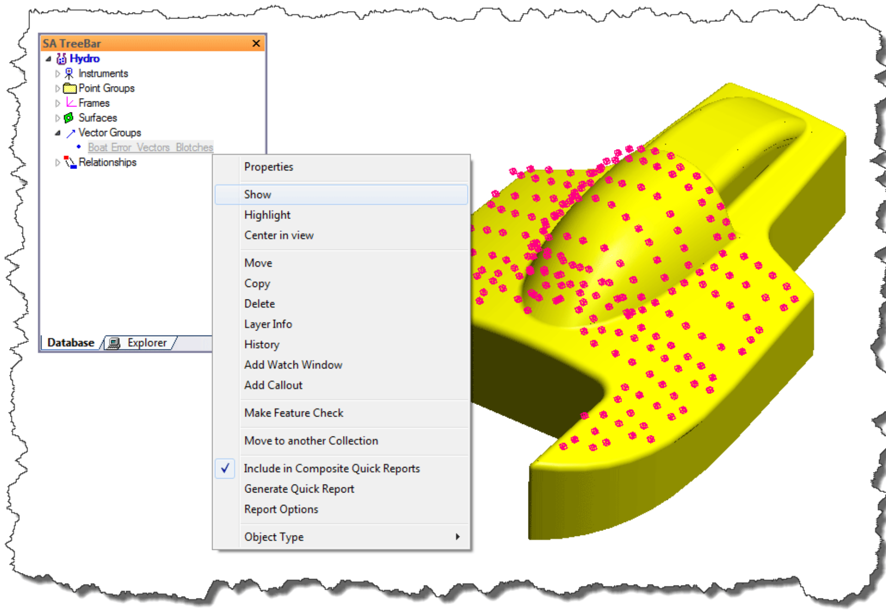


Figure 29-17. Hiding the vector group in the tree.

2. Now let's hide a few of the points. Choose **View>Hide Points** from the menu. Single-click a few points in the graphical view and press **Enter**. Now hide a single point in the Surface point group. Expand the Surface point group, right-click on a point and uncheck **Show**. Just like the above step for hiding the Vector Group, the point will be greyed out and the check box beside **Show** will be removed. Now hide all of the points by right-clicking the *Point Groups* category in the tree and selecting **Hide All**. Now all of the point groups in the collection are hidden (Figure 29-18).

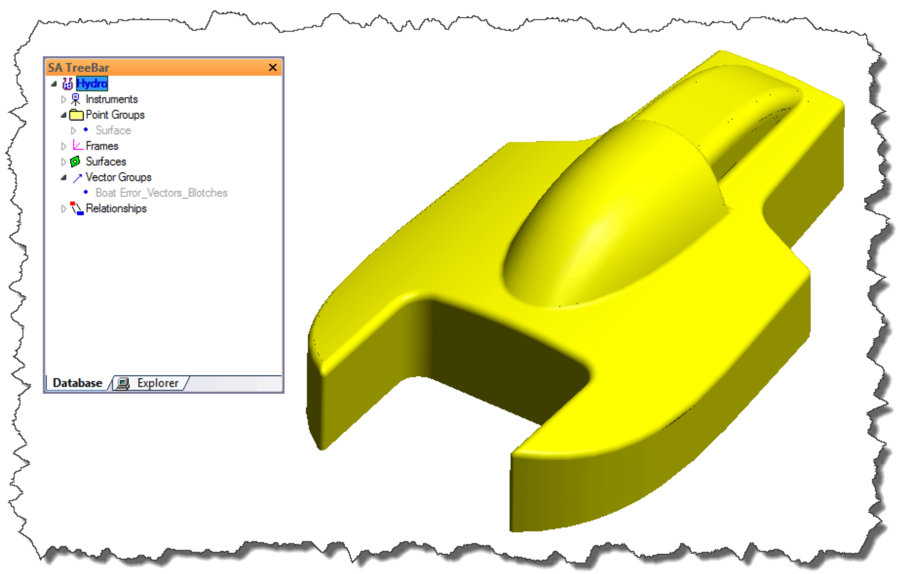


Figure 29-18. Hiding the entire point group.

3. Now let's hide the CAD model surface by using the menu item **View>Object Visibility>Hide Objects**. Select the surface by pressing **F2** and checking the surface category. Press OK to exit the F2 selection dialog and press **Enter** to complete the command.

4. Hide the remaining items in the *Hydro* collection by right-clicking *Hydro* and selecting **Hide All**.
5. Now to show an object you can simply reverse any of the above steps. Let's show everything by right-clicking *Hydro* and selecting **Show All**.

Zoom, Pan and Rotate

Now that we can change colors, render, and hide/show objects, let's learn how to manipulate the graphical view. See the Graphical View section for more information regarding Zoom, Pan and Rotate.

Zoom

1. Zoom in on the graphical view by rolling the scroll wheel forward on your mouse.
2. Zoom out by using the **Page Down** key.
3. Zoom back in by holding down **Ctrl** and left-click dragging the mouse.
4. Zoom back out by using menu item **View>Zoom>Out**.
5. Now let's put all visible objects back in the view by using Autoscale .

Pan

1. Use the arrow keys on the keyboard to pan the view up, down, left and right.
2. Now pan the view by holding down the **Shift** key while pressing the left mouse button and dragging the mouse.

Rotating

1. Now let's rotate the graphical view. Press and hold the right mouse button and drag the mouse. You will quickly notice that the view pivots about a particular point in space.
2. Let's change the rotation center by selecting the View Rotation Center button from the toolbar. Select one of the points at the corner of the model. Now rotate and you will see how the view pivots about the selected point.

Preset Views

Preset views are available which allow for quick orientation of the graphical view.

1. Press the drop down beside the icon and select **Top**. This will orient the view with respect to the *World* frame.

2. Saved presets can also be used. Now select the vector view preset view from the list.

Conclusion

We have now covered many ways to render objects, change background colors, show/hide objects and manipulate the graphics.

Basic Point Analysis

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to perform basic point analysis.
- **Areas Covered.** Query Point to Point, Query Point to Object and Query Points to Objects.
- **Time to Complete.** Approximately 20 minutes.

Creating Entities to Work With

For this tutorial, we want to create some points and geometry to play around with, so we'll start out by creating those entities first.

1. In SA, start a new job file by selecting **File>New** from the menu or choosing the New File Icon from the main toolbar.
2. Create a set of random points by using **Ctrl+Alt+Z** or **Construct>Point(s)>Layout>Random Points**. Use the default settings.
3. Now create a plane by using **Construct>Plane(s)>Enter** and accept the default settings.

Calculating Point to Point Distance

Calculating the distance between two points is a very common analysis operation and provides a good starting point for learning basic analysis in SA.

1. Using the points we created above, let's determine the distance between two points. Use **Query>Point to>Point** or **Ctrl+D** and select two points by single-clicking in the graphical view.
2. The distance between the two points will be reported in the ***Query Results*** dialog (Figure 29-19). The delta values for each component are expressed in the active coordinate frame. The point to point distance is calculated from the centers of the selected points. Any target offset, if present, will be ignored.

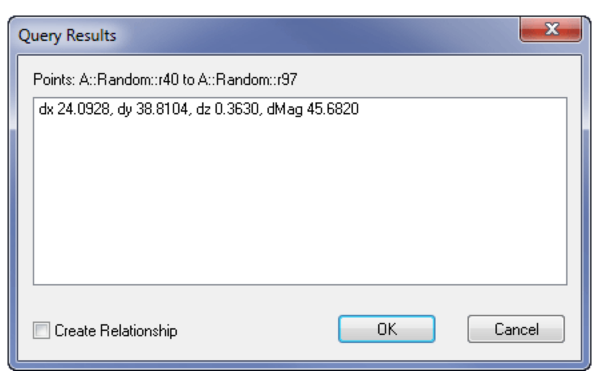


Figure 29-19. The query results dialog.

3. A record of the point to point query is stored in the SA tree in the form of an Event. This event can be viewed using the report bar or an SA report (Figure 29-20).

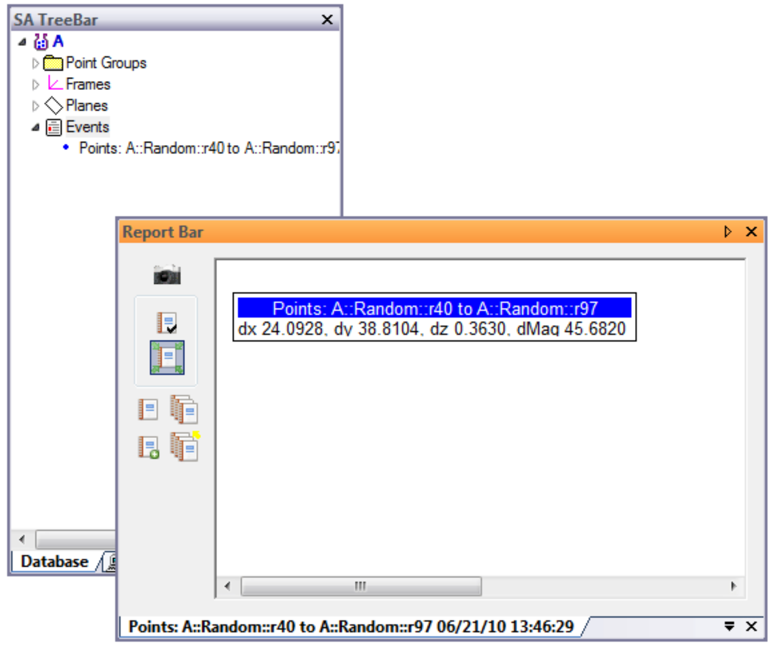


Figure 29-20. The query as an SA Event.

Calculating Point to Object Distance

Another common analysis operation is calculating the distance between a point and an object, which is reported as the closest distance between that point and the object. Let's calculate the distance between one of the random points and the plane we created earlier.

1. Select **Query>Point to>Object** from the menu. Select one of the random points and then the plane. The resulting distance between the point and the plane is displayed (Figure 29-21). Notice that the dx and dy values are zero. Since we're comparing a point to a plane, and the plane's normal is along the Z axis, there is no delta-x or delta-y component. (Said another way, the closest point on the plane to the selected point is directly below that selected point).

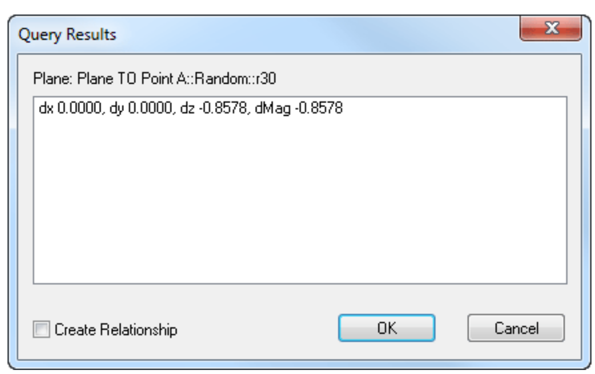


Figure 29-21. Querying a point to an object.

2. Like the Point to Point query, the results are stored in the tree as an event (Figure 29-22).

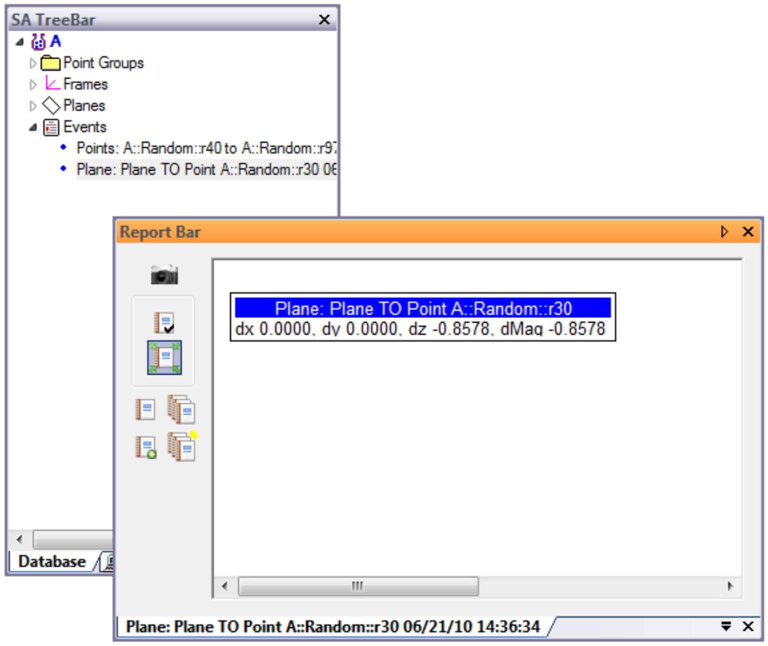


Figure 29-22. The Point to Object query as an event.

Calculating Multiple Points to Object Distances

During analysis operations it is often desireable to calculate the distance between multiple points and objects. To do this, a Points to Objects query can be performed. The output of this query is a vector group which gives a numerical and graphical representation of the point to object deviations.

1. Go to **Query>Points to>Objects** and select all of the random points. Then, select the plane when prompted for the objects. The ***Query Point to Surface Options*** dialog will appear (Figure 29-23). Here you can choose which direction the Vector should point and access general options. By default the vector (whisker) will point from the object to the point. Let's also create a point group with points that are offset toward the object by the target radius. (Think of these as the "contact points", or the points where the measurement tooling would have been touching the object being measured). Press OK.

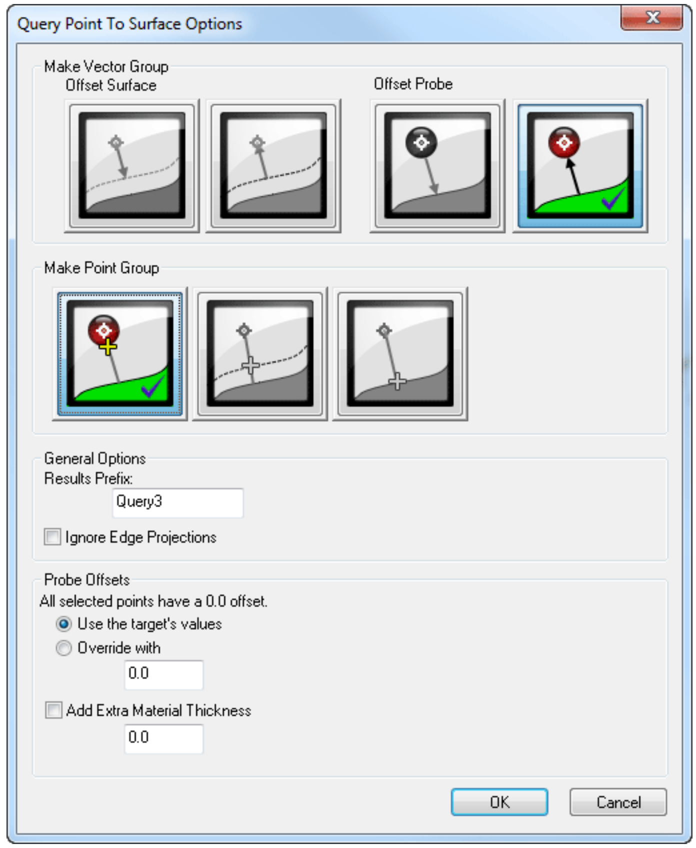


Figure 29-23. The options for querying points to objects.

2. A Vector Group will be created and the properties dialog will appear (Figure 29-24).

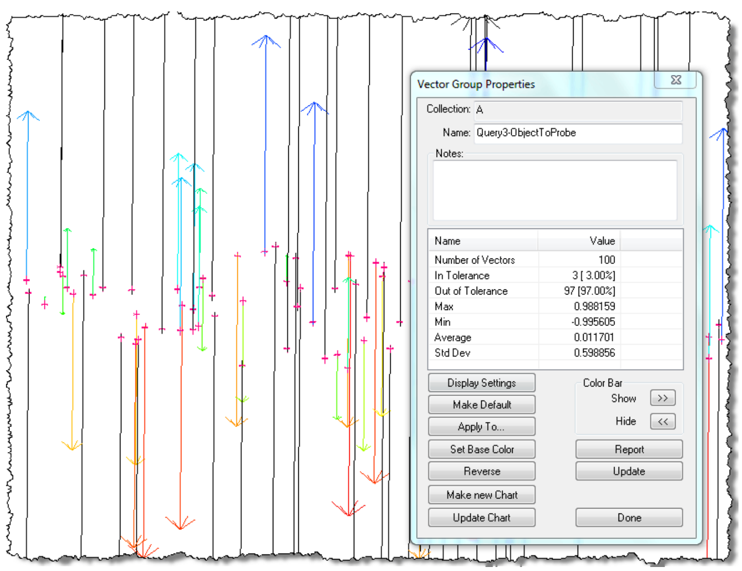


Figure 29-24. The vector group properties dialog.

3. To change the appearance of the vectors, click the Display Settings button. Here you can change the magnification, size, and style of the vector. Change the magnification, vector style and blotch size to match Figure 29-25.

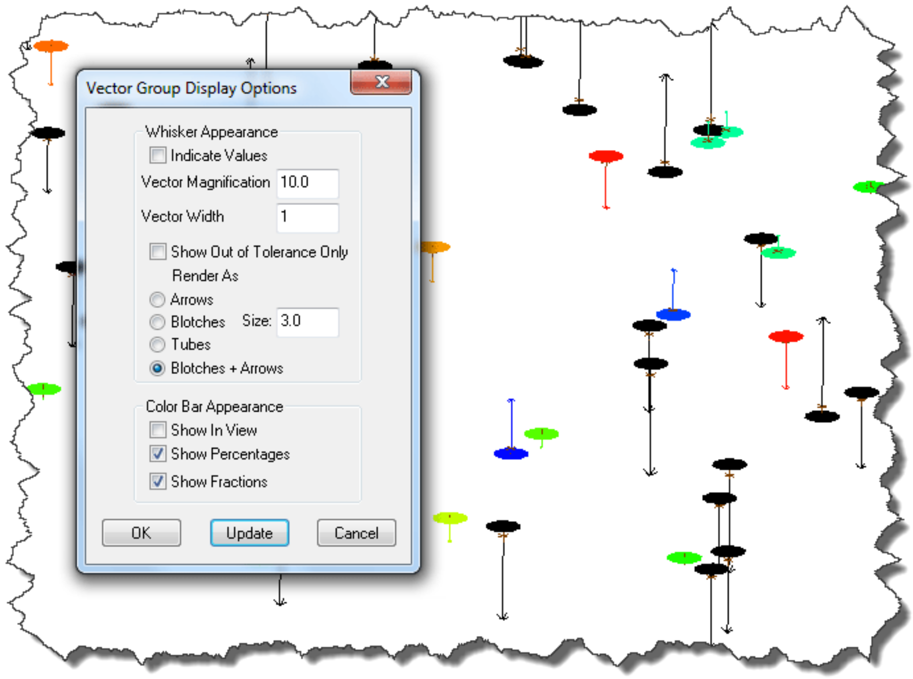


Figure 29-25. The Vector Group Display Options dialog.

4. Now change the colorization of the vectors and apply a tolerance for colorization. There are four different colorization methods (Figure 29-26). Choose **Toleranced (Go/No-Go)**.

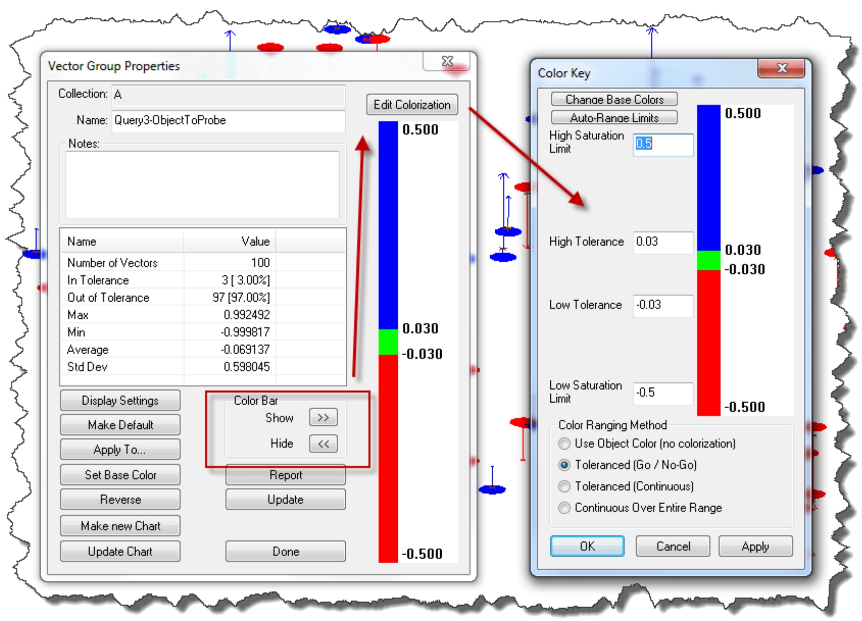


Figure 29-26. Colorization options.

Now the vector group reflects the display and colorization changes (Figure 29-27).

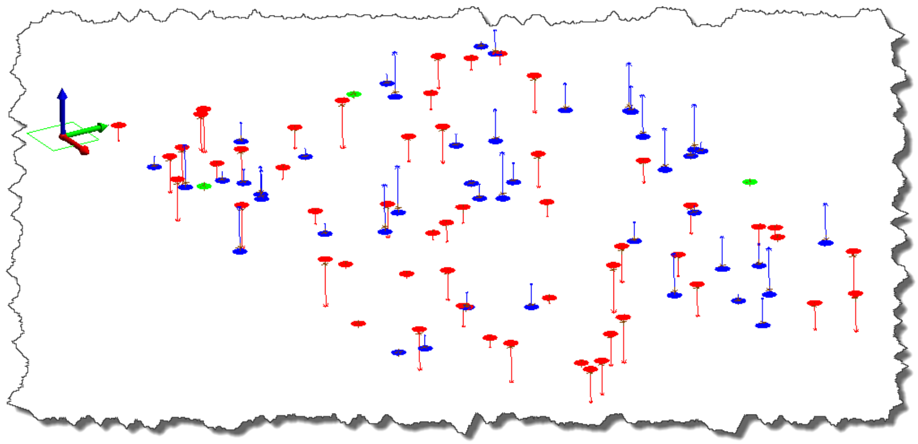


Figure 29-27. The result.

Best Fit Points to Points

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will explore the "best fit points to points" commands, a commonly-used functionality for relocating to an established coordinate system and aligning objects.
- **Areas Covered.** Renaming points using a name pattern, renaming points by auto-corresponding using inter-point distances, locating instruments using best-fit, fit weighting, and tolerance coloring zones.
- **Time to Complete.** Approximately 20 minutes.

Point Names Are Important

In this tutorial, we have a set of nominal points (fiducials), and we've measured those nominal points with a laser tracker. Our task here is to locate the instrument by fitting the measured points to the nominals.

1. We'll start by opening a tutorial file included with your installation of SA. In SA, choose **Help>Open Sample SA Files**. Choose the `Best Fit Points to Points.xit` file. This file contains a laser tracker which has measured a series of 7 points, each of which has a corresponding nominal point.
2. Our measured points currently match the names of the nominals, but in real life, you won't ordinarily be so lucky. Let's give random names to our measured points to simulate what you might encounter after a real survey. From the menu, choose **Edit>Rename>Points Using a Name Pattern**.
3. When prompted for the points, press **F2**. Check the *Station1* point group to select all points in the group, then click OK. Press **Enter** to accept the point selection.
4. For the *Name Pattern* field, enter %d. Enter a starting value of *5*, and click OK (Figure 29-28). The measured targets will be assigned names from *5* to *11*. Notice that the measured target names do not match the nominal point names.

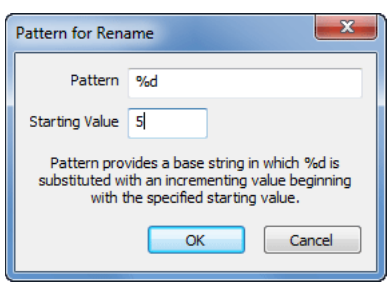


Figure 29-28. Renaming using a pattern.

5. Let's locate our instrument to the nominal points. In the tree, right-click the tracker and choose **Locate**. From the ***Locate Instrument*** dialog, choose **Best Fit** and click OK (Figure 29-29).

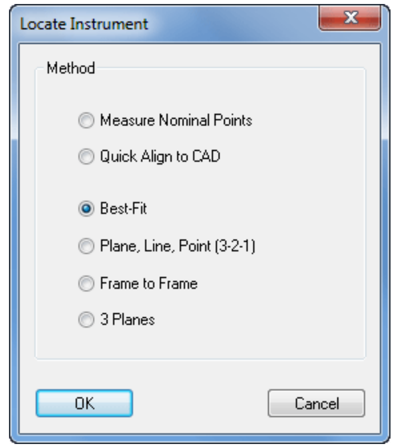


Figure 29-29. Locating using Best Fit.

6. You will be prompted to select the Nominals Group. Click one of the nominal points in the graphical view. Then click one of the instrument's points when prompted for the Measured Group. Nothing happens! The fit is not occurring because best fitting points requires that each nominal and corresponding measured point must have the same name.

7. You could rename each point manually, but there is a better way. We want to rename our measured points to match our nominal names. From the menus, choose **Edit>Rename>Points by Auto-Corresponding 2 groups>Inter-Point Distance**. Think of this command as performing a best fit "under the hood", then renaming the points based on their proximity to the reference points. When prompted for the reference group, pick the *Nominals* group. For the group to be renamed, pick the *Station1* group.

8. The default value of 0.1" (2.54 mm) works fine for our inter-point distance match tolerance. We know that each of our measured points should be within 0.1" (2.54 mm) of our nominals after a best-fit it performed, so all of the points within this threshold will be renamed. (Note that this function would probably not work if the nominal points were symmetrical. In that case, SA wouldn't know how to orient the measured points, and the resulting points may not be named correctly).

9. You will be presented with the proposed name changes. Click Accept to make the changes. The *Station1* point names now match the *Nominals* point names. We're now set to do a points to points best fit.

Fitting the Points

1. We saw how to use best fit from the instrument's Locate menu. Let's look at how to do it from the main menu. Choose **Instrument>Locate (Transform to Part)>Best Fit**. As before, pick *Nominals* for the nominal group and *Station1* for the measured group. The ***Best-Fit Transformation*** dialog will appear (Figure 29-30). At the top of this window, we're allowing 6 degrees of freedom for the fit: X, Y, Z, Rx, Ry, and Rz, which is what we want in this case. Notice the table at the bottom, which is currently displaying the nominal coordinates and resulting deltas. Click the **Actuals** checkbox to display the coordinates of the measured points. Notice that the fit hasn't yet been performed, and as a result the deltas are quite large. That's why the Re-fit button (with the two chasing arrows) and the fit results are highlighted blue.

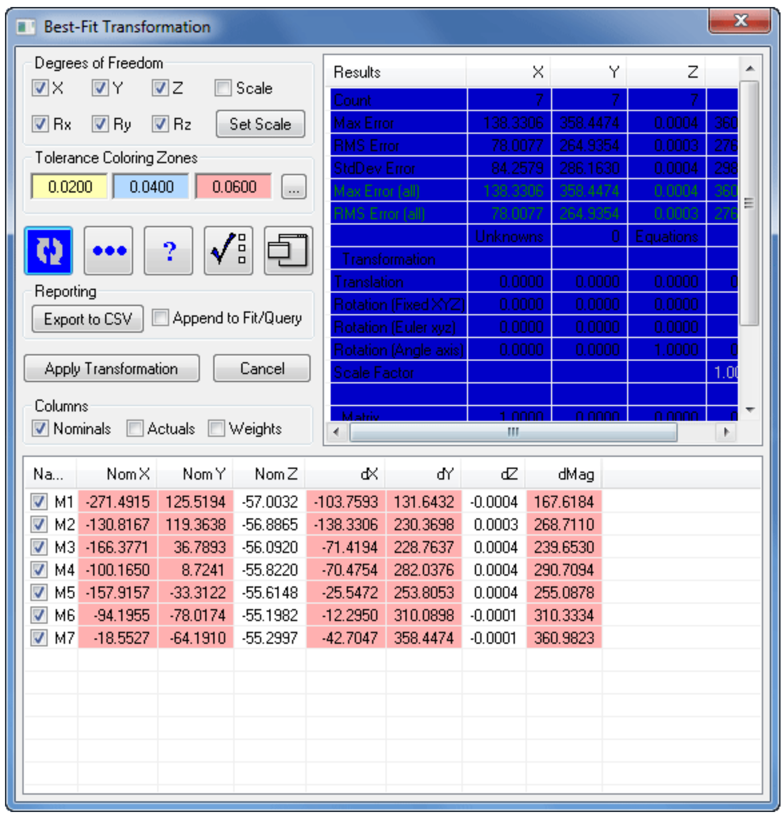


Figure 29-30. The Best-Fit Transformation dialog.

2. Click the Re-fit button to calculate the fit. Immediately, the results of the fit are displayed (Figure 29-31).

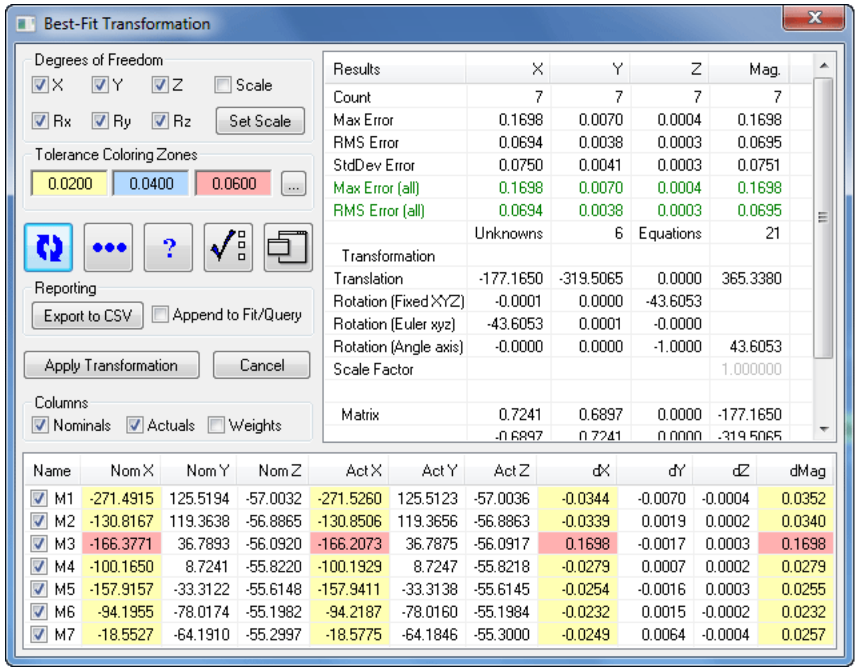


Figure 29-31. The fit results.

3. Notice that each field is color-coded based on the Tolerance Coloring Zones in the dialog. Deltas between 0.0000-0.0200" are not colored, deltas from 0.0200-0.0400" are yellow, 0.0400-0.0600" are blue, and 0.0600" and greater are red. Looking at the results of the fit, point *M3* has a very large dX value relative to the other points. This is highlighted by the red coloring. Click the "dX" column header twice to sort the values, with the highest value in the top row.

   It looks like our *M3* point has a significantly higher dX error than the other points, so that point is suspect. Let's temporarily remove it from the fit to see how the solution is affected. Uncheck the checkbox in the *Name* column for point *M3*. Since this changes the fit, you again need to click the Re-fit button.

4. Immediately, the errors drop dramatically, which backs up our theory that the *M3* point is not good. Let's give our tolerance warning indicators smaller values, for good measure. Click the ellipsis button ... in the *Tolerance Coloring Zones* section of the dialog. In the ***Tolerance Zone Coloring*** dialog (Figure 29-32), set the values to *0.005"*, *0.010"*, and *0.020"* respectively.

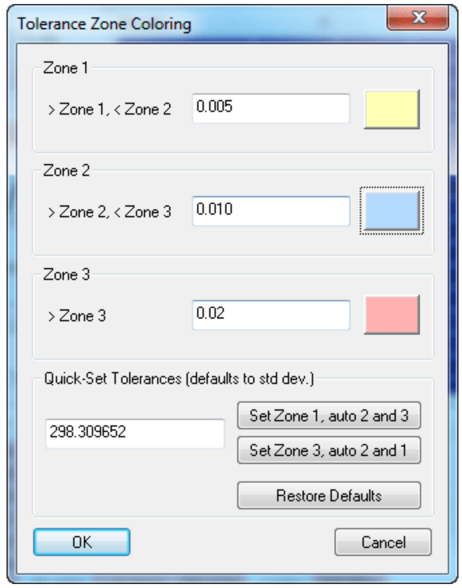


Figure 29-32. The Tolerance Zone Coloring dialog.

5. After the change, none of the rows are highlighted, so we know that all errors are within 0.005". In fact, we can easily see that the largest error is just under 0.002".
6. Now take a look at each of the individual components. The largest dX error is 0.0006", which we deem to be acceptable. The largest dZ error is 0.0004", which is also acceptable. But notice that two of the dY errors are -0.0011" and 0.0016". These are at least double the largest errors of any of the other components. Suppose that we believe that the Y values from those two points might be unreliable. The best-fit dialog allows us to weight these individual components to zero, so that they do not affect the fit. Let's do that now, starting with the point with the larger error.
7. Check the **Weights** checkbox so that we can see the individual component weights being applied. Double-click the *M2* row. In the ***Best-Fit Item*** dialog (Figure 29-33), click the 0 button under the Y component to set its component weight to zero and click the OK button.

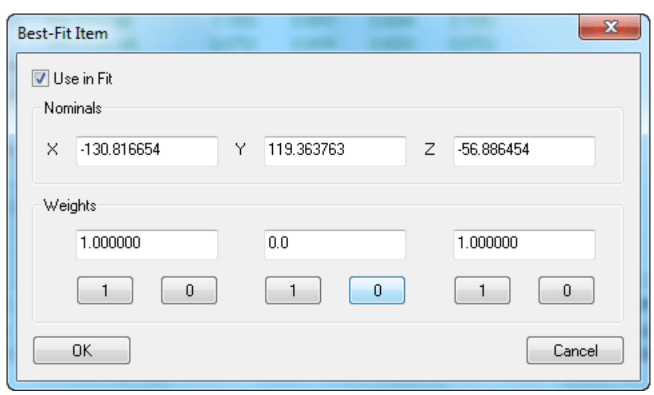


Figure 29-33. The Best-Fit Item dialog.

8. Click the Re-fit button. (Note that it's usually best to make

small changes and re-fit, so you can gradually see how the changes affect the solution). Now that the new solution is calculated, the new largest dY is under 0.001", so we may consider this to be acceptable.

9. Click the Apply Transformation button. The instrument immediately snaps to the nominals (Figure 29-34). We've now performed a fairly complicated fitting operation with ease. We've removed a point from the fit operation, and weighted an individual component from another point to zero to exclude it from the fit. As a result, we brought our errors from over 0.030" to under 0.001". Since most of the points and components are still in the fit, we can still be confident that the fit is good.

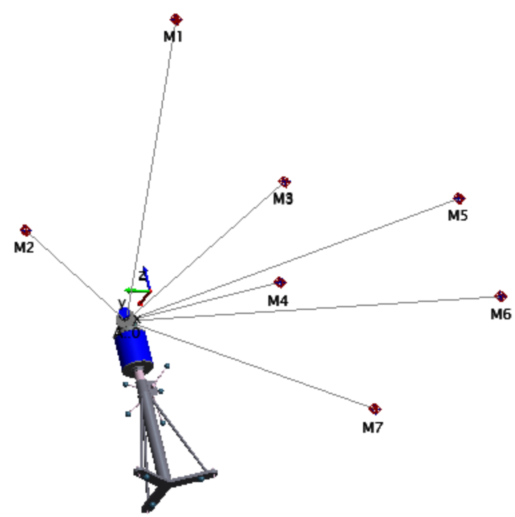


Figure 29-34. After the best fit.

Reviewing the Fit

1. Let's review the results of our fit. When the fit was completed, an event was created in the tree called *Best Fit Transformation*. Turn on the Report Bar by selecting **Reports>Report Bar Visible** from the menus (if necessary), and click the event in the tree to select it.

2. In the Report Bar, notice that you can view all of the results from the fit, including the points and components that were excluded from the fit or assigned special weights.

Best Fit Geometry from Points

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will explore how to best fit geometry to points.
- **Areas Covered.** Geometry Fitting.
- **Time to Complete.** Approximately 15 minutes.

A significant portion of metrology analysis involves fitting geometry to measured points. SA's interactive Geometry Fit Interface is able to fit a wide variety of geometric shapes to any set of measured, constructed, or cloud points. In this tutorial, you will learn how to best fit data to four of the most commonly created shapes: circles, cylinders, lines and planes.

1. To begin, open the tutorial file in SA included with your installation. Choose **Help>Open Sample SA Files** and select `Geometry Fit.xit`.

If you expand the items in the tree bar on the left, you will see that this file contains two separate instruments. The laser tracker measured points in the point groups **`Circle`** and **`Plane`**. The portable arm measured the points in the point group named **`Cylinder`**. These instruments were located to an established coordinate system using a best fit points to points transformation. This transformation was performed for visual purposes only and will not be demonstrated in this tutorial.

Best Fit a Plane to Points

There are several ways to fit geometry in SA. One is by selecting **Construct>Geometry by Fitting** (**`Ctrl+G`**), which is the general form of the command. If you already know the type of geometry you wish to create, a slightly faster method is to use **Construct>[Geometry Type]>Fit to Points**. This method will automatically select the proper geometry type for you. The keyboard shortcuts for geometry fitting are useful and easy to remember: **`Ctrl+Alt+[Letter]`**, where **`[Letter]`** is typically the first letter in the geometry name (**P** for plane, **C** for circle, etc). If either of these methods are used, you must explicitly select the points to use in the fit. The simplest option is to right-click on the appropriate point group and select **Fit Geometry** from the context menu. If you've measured features into different point groups (as you should in SA), this will fit the geometry to the entire point group, which is almost certainly what you want.

2. Right-click the `Plane` point group and select **Fit Geometry** from the context menu.

3. The ***Geometry Fit Interface*** will appear. Here you can control the fit tolerance, points used in the calculation, output parameters, and everything else related to the fit. Select the *Plane* option from the dropdown (Figure 29-35).

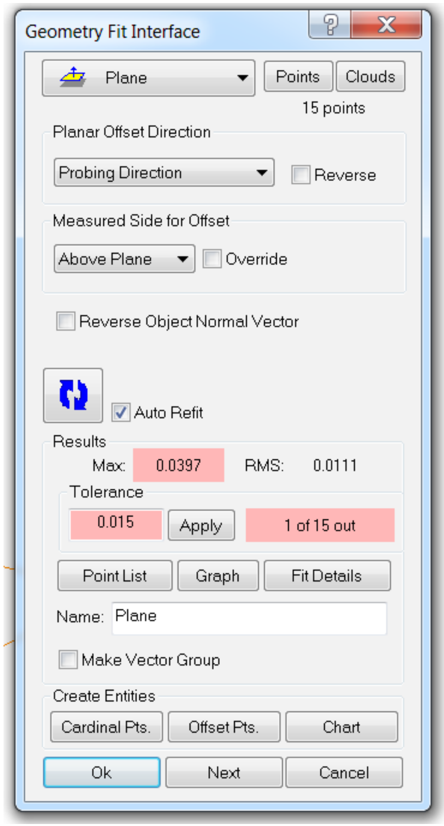


Figure 29-35. The ***Geometry Fit Interface***.

Fit Tolerance

4. Click on the Point List button. This will display the ***Geometry Fit Point Listing*** dialog (Figure 29-36) which shows you the fit error for individual points and permits the application of Tolerance Coloring Zones.

Tolerance Coloring Zones can be defined with values of your choosing and help you gauge how well the plane fits the selected data. You may change the tolerance zones as well as their colors by selecting the ellipses icon [...]. In this example, one point lies outside of the 0.015" tolerance zone. Four points are highlighted yellow because their error lies in the Limit 1 tolerance zone. All other points that are not highlighted are considered completely in-tolerance.

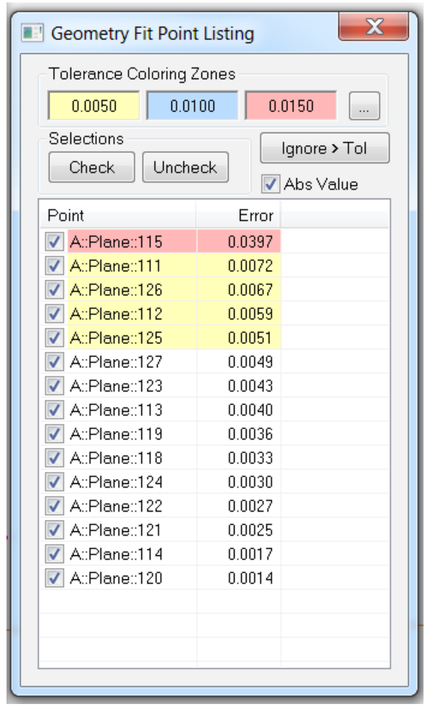


Figure 29-36. A geometry fit's point listing.

5. Let's uncheck the out-of-tolerance point highlighted in pink to remove it from the fit. Once unchecked, the fit will need to be recomputed. Press the recompute button, which will have a dark blue background indicating the currently displayed fit data is not reflective of the current settings. Evaluating the new fit results, all points are now considered in tolerance. Close out the point listing dialog box.

Controlling Probe Offset

By selecting which probe you are using while collecting data, you will allow for SA to automatically calculate the offset when fitting geometry. For this particular plane, a 1.5" SMR was used and as a result SA has already shifted the plane 0.75" to compensate for the probe radius (Figure 29-37).

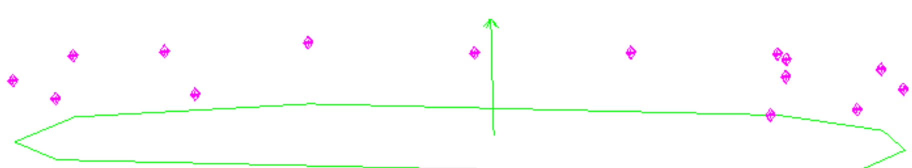


Figure 29-37. The plane is automatically offset from the measured points.

6. Click the Fit Details button. Among the statistics about the fit and the resulting geometry, the dialog also indicates that all points have offsets of 0.75". Close this window.

7. Check the *Override* checkbox (which overrides the target offsets to 0.0" by default) and notice how the plane now rests approximately on the measured points.

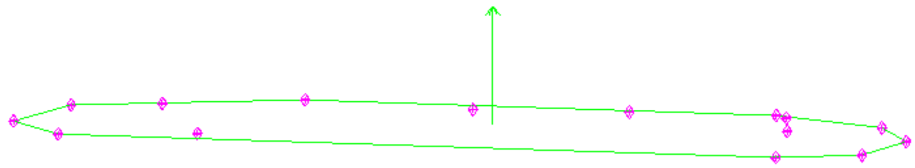


Figure 29-38. Overriding the offset to 0.0".

8. Uncheck the *Override* checkbox to again use the measured probe offset of 0.75".

In certain less common measurement situations (such as indirect measurement or when using a backing bar), you may wish to reverse the geometry offset direction or the direction of the resulting geometry. The *Reverse* checkbox in the *Planar Offset Direction* box and the *Reverse Object Normal Vector* checkbox are intended to be used for this purpose.

Creating Entities

It is sometimes desired to create probe compensated points—that is, points which represent the point of contact of the probe on the measured feature (instead of points representing the center of the probe). This can easily be accomplished by selecting the Offset Pts button.

Exercise

Now that you have discovered how simple it is to interact with the geometry fit interface when constructing a best fit plane to points, it's time to try an exercise utilizing three other geometric shapes with the remaining data on your own.

- Best fit a circle to its measured points.
- Best fit a cylinder to its measured points.
- Best fit a line to the cardinal points constructed from both the circle and cylinder.

If you need help or would like to check your work, follow the steps below.

Best Fit a Circle To Its Measured Points

1. All points for the circle fit were measured in the same point group. Right-click on the group in the tree and select **Fit Geometry**.
2. Click the Point List button. Several points are highlighted pink (Figure 29-39). Uncheck the point with the highest magnitude and recompute. Several points are still highlighted pink so uncheck the point with the highest magnitude once again and recompute. All points are now in tolerance (Figure 29-40).

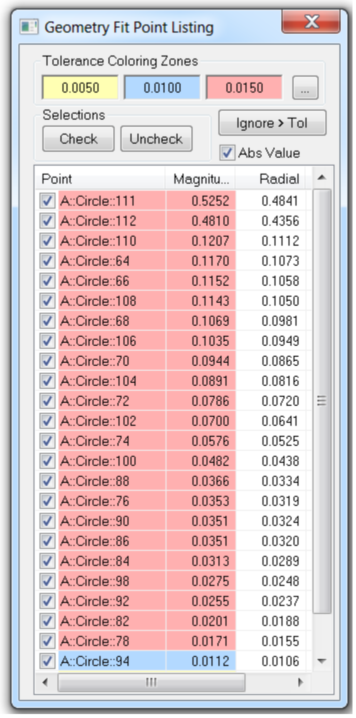


Figure 29-39. The point list for the circle, showing many out of tolerance points.

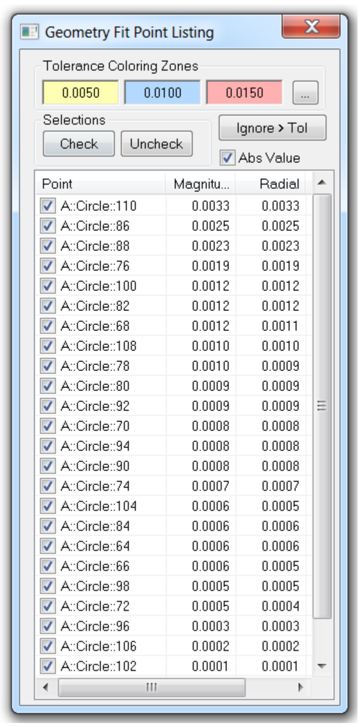


Figure 29-40. The fit improves dramatically after removing a few stray points.

3. Since the final step requires us to best-fit a line using cardinal points, click the Cardinal Pts button. The point group **`Fitted CircleCardinal Points`** has been created.

4. Click OK to accept.

Best Fit a Cylinder To Its Measured Points

1. All points for the cylinder fit were measured in the same point

group, so right-click on the group and select **Fit Geometry**.

2. Since the tolerance box lets us know that 0 of 16 points are out-of-tolerance (assuming the defined tolerance is acceptable to us), we do not need to look at the point list.
3. Click the Cardinal Pts button to create the point group **`Fitted CylinderCardinal Points`**.
4. Click OK to accept.

Best Fit a Line To The Cardinal Points Constructed from both the circle and cylinder.

1. The cardinal points constructed from the circle and cylinder are located in separate point groups, so choose **Construct>Lines>Fit to Points** or use the keyboard shortcut **`Ctrl+Alt+L`**. This will bring up the Geometry Fit Interface.
2. Select all points in both cardinal points groups by using either the F2 command or graphically clicking on the points themselves.
3. Of the five points, one is out-of-tolerance. Uncheck the point with the largest amount of error from the Point List dialog and recompute.
4. Click OK to accept.

Conclusion

This tutorial has covered the basics of geometry creation and how to create a best fit plane. The interface we covered is generic for all the geometry fits which makes fitting geometries extremely simple.

Creating Callouts

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to create callout views with graphical annotations.
- **Areas Covered.** Creating a Callout View, Creating Callout Annotations and Locking the view.
- **Time to Complete.** Approximately 10 minutes.

Creating A Callout View

1. In SA, start by opening the `Blower Shroud.xit` file from the Samples directory under **Help>Open Sample SA Files**.
2. Before we create a callout view, let's render the model in solid shaded mode by clicking the Solid icon .
3. Now create a callout view by clicking the Callout icon. *Callout 1* view will be created in the tree and will be active (Figure 29-41).

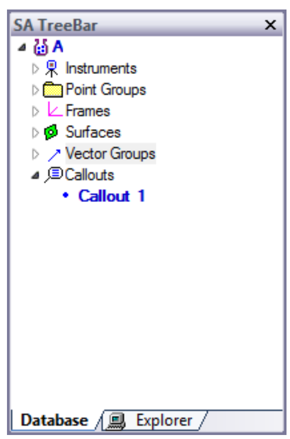


Figure 29-41. The newly-added callout view.

Creating Callout Annotations

1. Let's create a callout annotation in the graphical view. Start by clicking the arrow by the icon. A series of annotation types will appear. Let's start with a simple Point Comparison. Select **Point Comparison** from the drop down and select two points to compare. An annotation will appear and you can now place the annotation accordingly by left-clicking and dragging (Figure 29-42).

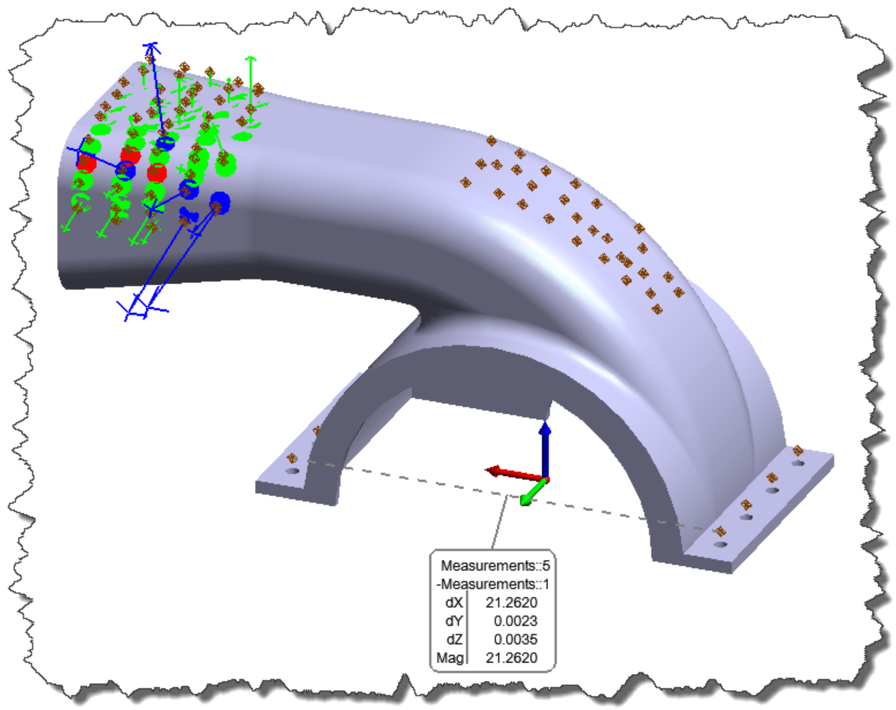


Figure 29-42. The point comparison callout.

2. Now let's create some annotations for a few of the vectors in the vector group. Start by clicking the arrow by the icon and select **Vectors**. When prompted select the vectors of interest. The ***Vector Callouts*** dialog will appear with view options (Figure 29-43). This dialog will allow you to select display options for the callouts. Once satisfied, press OK (keep callouts). Feel free to place the annotations accordingly by left-clicking and dragging (Figure 29-44).

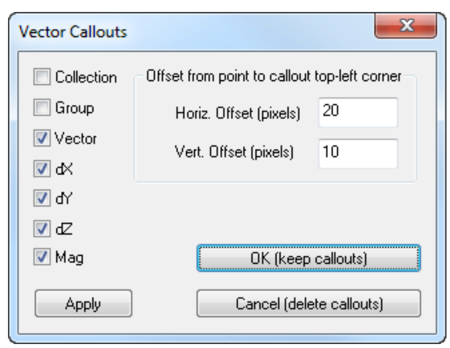


Figure 29-43. The Vector Callouts dialog.

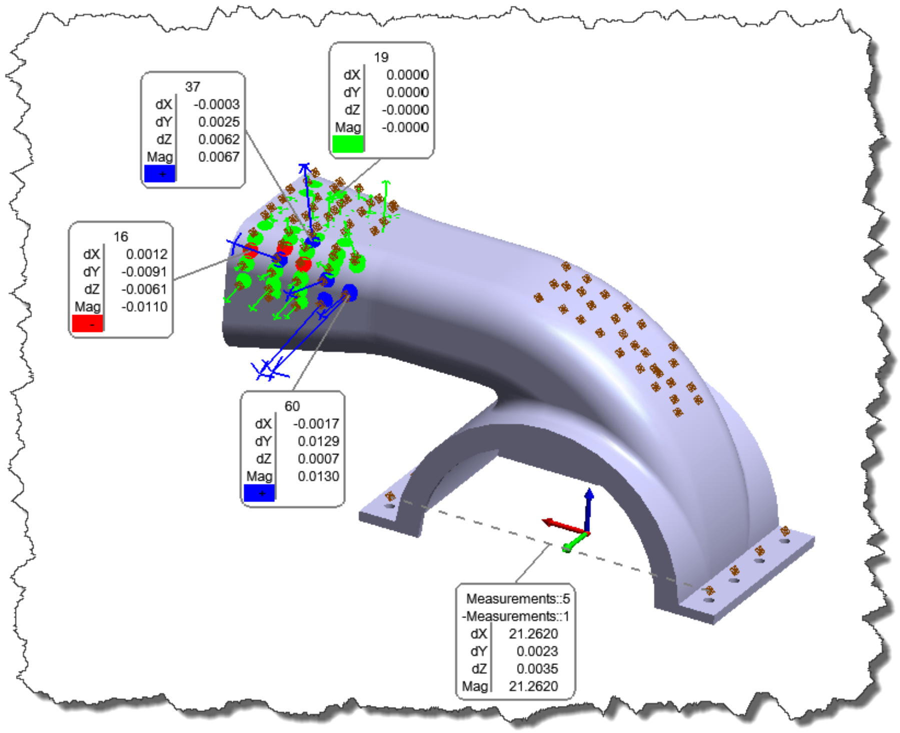


Figure 29-44. Vector callouts have been added.

Locking the Callout view

1. Right-click on *Callout 1* in the tree and select **Properties** (Figure 29-45).

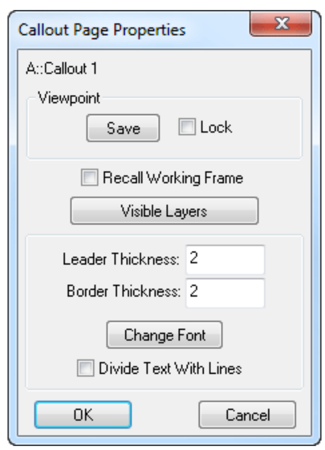


Figure 29-45. The Callout Page properties.

2. Select **Lock** in the *Viewpoint* section. When the viewpoint is locked, the view will be recalled when the callout is activated.
3. Double-click *Callout 1* in the tree. This will deactivate the callout.
4. Rotate and zoom the view to be different than the callout.
5. Now double-click *Callout 1* in the tree. This will activate the callout. Notice how the view returns to the locked position.

Inspection with GD&T

Note: This tutorial requires SA Ultimate or greater.

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to perform a GD&T inspection with nominal CAD and native GD&T annotations.
- **Areas Covered.** Importing CAD with GD&T annotations, creating Datums and Feature Checks, Associating/Trapping measurements and Reporting.
- **Time to Complete.** Approximately 15 minutes.

Importing CAD with GD&T annotations

The first step for this tutorial is to import the nominal CAD model with GD&T annotations. If your CAD models do not contain GD&T annotations, please refer to the tutorial on creating GD&T annotations.

1. In SA, start a new job file by selecting **File>New** from the menu or choosing the New File Icon from the main toolbar.
2. Import the native CAD model with GD&T annotations by selecting **File>Import>Direct CAD Access** or use the Auto Import button on the main toolbar.
3. Choose `Sample CAD.CATPart` from the Samples file folder in the SpatialAnalyzer install directory.
4. When the ***Direct CAD Access Settings*** dialog appears (Figure 29-46), make sure the **Annotations** option is selected and press OK.

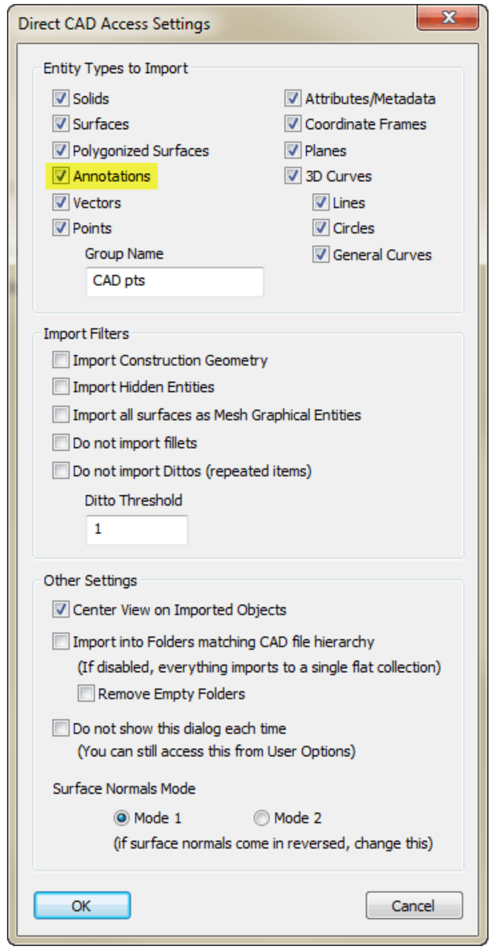


Figure 29-46. The Direct CAD Access settings.

5. Once imported, you will find the *Annotations* category is created with the imported annotations (Figure 29-47).

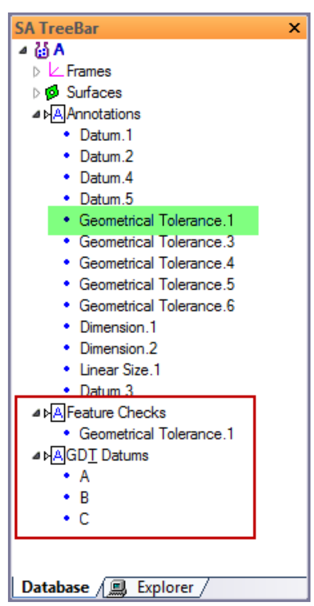


Figure 29-47. Imported annotations.

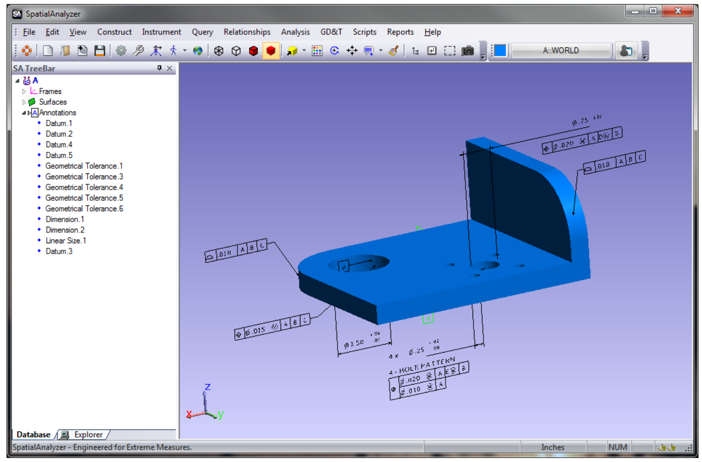


Figure 29-48. Annotations in the graphical view.

Creating GD&T Datums and Feature Checks

For this tutorial we will inspect only one of the GD&T Features. Right-click *Geometrical Tolerance.1* in the tree and choose **Make Feature Check**. You will notice two new categories are created, *GD&T Datums* and *Feature Checks*. These two new categories act as bins for the respective measurements.

Inspecting GD&T Datums and Feature Checks

1. The next step is to associate measured points to the respective GD&T Datum or Feature Check. For this tutorial we will assume that an instrument is present. So let's add an instrument and run the interface. See the Instruments section for information regarding adding and running an instrument. For this example we will use a Faro Arm.

2. Now that the instrument is added and running, simply right-click the collection containing the GD&T Datums and Feature Checks and select **Inspection>Inspect**. In this case we will right-

click collection *A* (Figure 29-49).

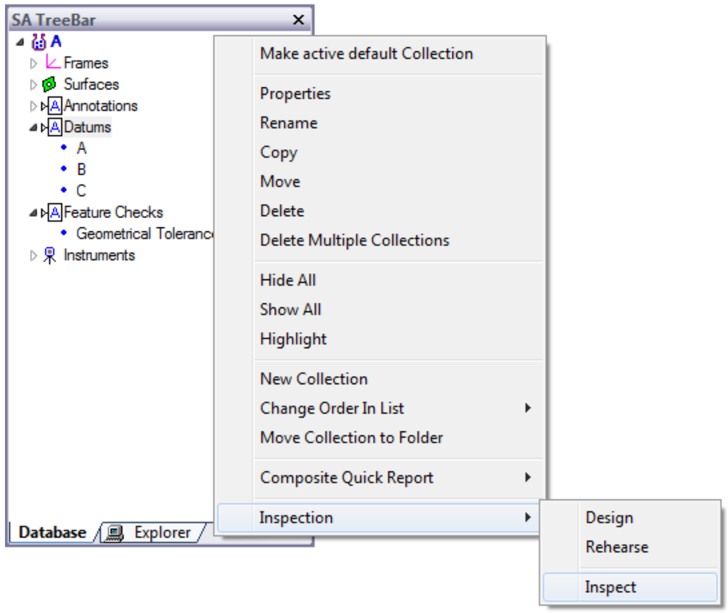


Figure 29-49. Selecting Inspect.

3. A dialog will display prompting what feature to measure. As points are measured, they are immediately associated to the GD&T Datum or Feature Check. Since an instrument is probably not available during this tutorial just use your imagination.
4. Once measurements are complete for a feature, press the next button [»] to advance to the next feature and continue measuring. Once all measurements are made the inspection is complete.
5. Since we did not take any actual measurements, lets open `GD&T with Arm.xit` from the Samples directory under **Help>Open Sample SA Files**.
6. In the `GD&T with arm.xit` file, all the GD&T Datums and Feature Checks have points associated.

Reporting the Results

1. To see the results of the Feature Checks evaluation, simply select the Feature Check of interest and the results will display in the Report Bar. If the report bar is not visible, turn it on using **Reports>Report Bar Visible**.

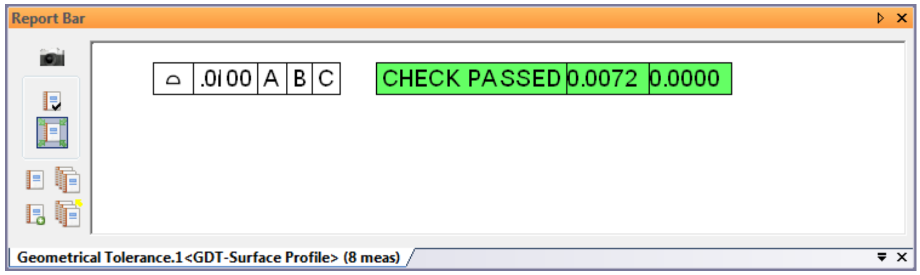


Figure 29-50. Feature check results for a surface profile check.

2. To report all results, right-click the *Feature Checks* category in the tree and select **Composite Quick Report>Generate**. All of the Feature Check results will be compiled into one report (Figure 29-51).

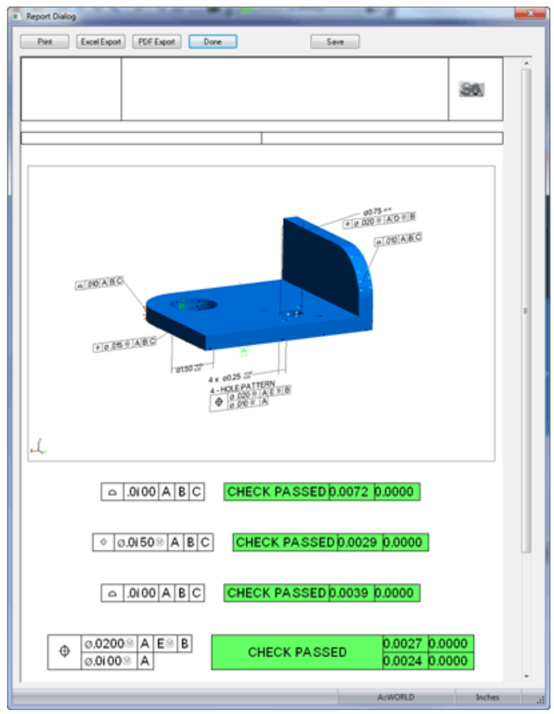


Figure 29-51. A composite quick report with GD&T feature check results.

Conclusion

In this tutorial, we covered how to import CAD with GD&T annotations and how to create, measure and report feature checks.

Reporting With SA Reports

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to create a report using the SA Report designer.
- **Areas Covered.** Creating an SA Report, dragging out items, changing report options.
- **Time to Complete.** Approximately 15 minutes.

Creating an SA Report

In this tutorial, our goal is to create a report comparing measured data to CAD. We will use a sample file and create the report from that data.

Note: This file contains data that was measured with a portable CMM arm. The measurements were compared to the nominal model using Query> Points to> Objects.

1. In SA, start by opening the `Query Points to Surface.xit` file from the Samples directory under **Help> Open Sample SA Files**.
2. Before we start the report, let's render the model in solid shaded mode by clicking the Solid icon .
3. Create an SA report by selecting **Report>Add SA Report**. The SA Report editor will appear, and the report is automatically stored in the tree under the SA Reports category (Figure 29-52).

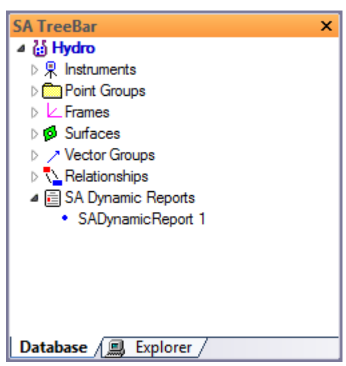


Figure 29-52. The newly-added report.

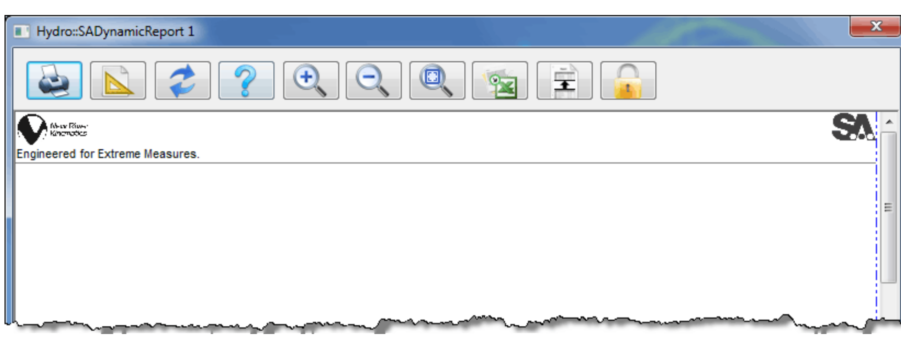


Figure 29-53. The Report Designer.

4. Let's add an item to the SA Report. Click and drag the *Boat Error_Vectors_Blotches* vector group into the report (Figure

29-54).

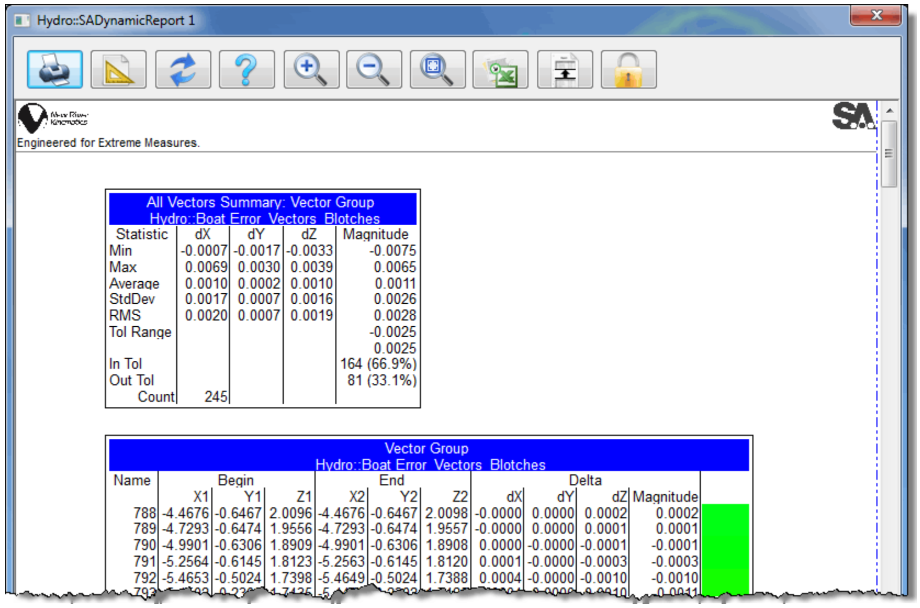


Figure 29-54. Adding a vector group to the report.

5. Now let's change the reporting options for the vector group so that only the vector magnitude will be displayed. Right-click anywhere on the vector group table and select **Report Options**.

6. Uncheck the **Point A**, **Point B** and **Delta** components and press OK. The SA Report will update and reflect your changes.

7. Let's add an image of the current graphics view to the report. Orient the SA graphics as you'd like, and press the Picture icon in the toolbar. The picture will be stored in the tree, as pictured in Figure 29-55.

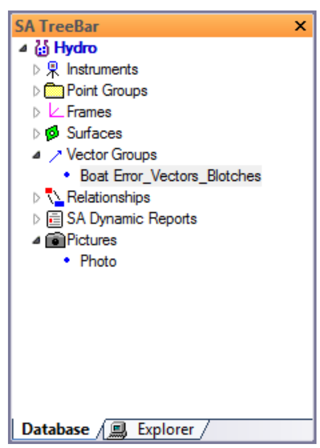


Figure 29-55. The screenshot has been added to the tree.

8. Click the photo, drag it over a table in the SA Report, and release the mouse button. Immediately, the report will relocate your tables so that they do not overlap. This is a feature that is on by default, but if you do not want this behavior, select the Global Reporting Options button . In the ***Report Designer***

Settings dialog, uncheck **Auto Vertical Overlap Adjust**. Now, the report will not automatically ensure that items do not overlap (Figure 29-56).

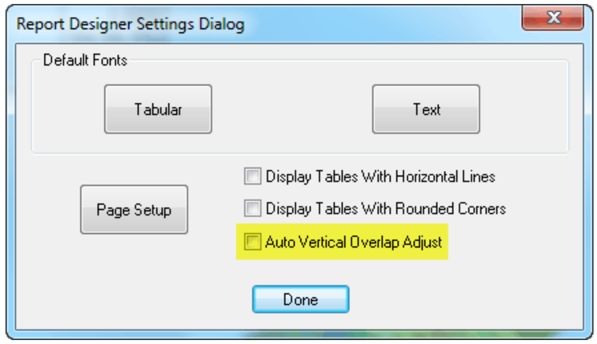


Figure 29-56. Ensuring that report items don't overlap.

9. The photo will have the same aspect ratio (width to height ratio) as the graphical view when it was taken. If you need to resize the image, hold down **Ctrl** and drag over the image, or grab a corner of the image to resize it as necessary. Drag the tables around to rearrange the report as you'd like (Figure 29-57).

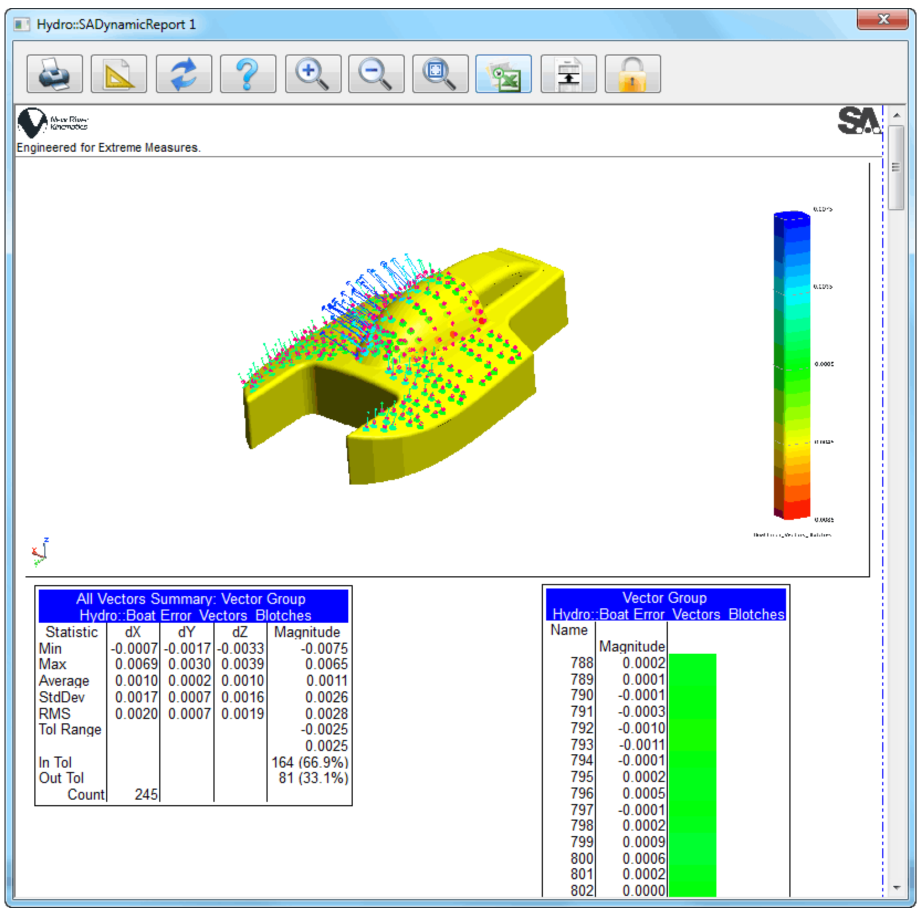


Figure 29-57. Arranging elements of the report.

10. The report is automatically saved into the tree. If you'd like, you can print the report, send it to a PDF, or export to Excel.

Creating Relationships

- **Skill Level.** Intermediate.
- **Description.** In this tutorial, we will explore the advantages of relationships, and look at how to create them.
- **Areas Covered.** Points to Objects Relationships, Groups to Objects Relationships, Relationship Reports.
- **Time to Complete.** Approximately 20 minutes.

Building the Relationships

Relationships are an extremely powerful concept that enable real-time examination of the positions and/or orientations between entities in SA. Using relationships, you can get dynamically updated comparisons between measured and nominal points, points and geometry, coordinate frames, and other entities. There are two sides to relationships: the reporting side, and the relationship minimization side. Relationship minimization is available in SA Ultimate and higher, and will be explored in a different tutorial.

1. We'll start by opening a tutorial file included with your installation of SA. In SA, choose **Help>Open Sample SA Files**. Choose the `Relationships.xit` file.
2. This file contains a sample CAD model of a hydroplane, along with a series of measured points along different surfaces of the model (Figure 29-58). Since this sample file already has constructed relationships, let's delete them to create them from scratch. In the tree, right-click on the *Relationships* category and choose **Delete All** from the menu. Confirm the deletion.

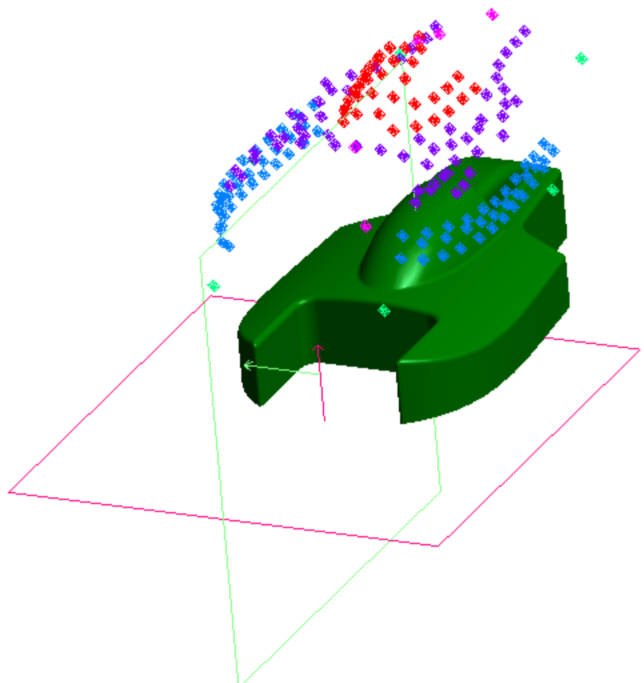


Figure 29-58. The measured points have not yet been aligned to the model.

3. Currently, our measured points are not aligned to the model. That's OK--for the purposes of this demo, we'll ignore that fact. We now want to compare our measured points to the CAD surface, and see how our measured part deviates from nominal. We'll use a series of Groups to Objects relationships to extract this information. In the menu, select **Relationships>Groups to Objects**. For the relationship name, enter *Canopy*.

4. The ***Query Point to Surface Options*** dialog will appear (Figure 29-59). This dialog allows you to specify how you want the relationship to compare the measured points to the surface. The set of buttons at the top of this dialog allow you to specify how the resulting deviations will be depicted in the view. All four options give the same numeric result--they just control how the resulting deviation vectors are displayed, and the sense (positive or negative) of the deviation. We want to know the deviation from the CAD to the offset point (probe contact point). Leave all of the other options at their defaults and press **Enter**.

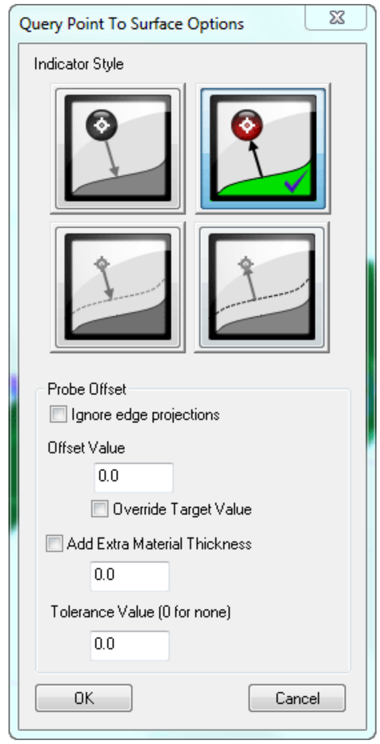


Figure 29-59. Query Point to Surface options.

5. When prompted for the groups, double-click the *Canopy* point group, then press **Enter**.
6. When prompted for the objects, double-click the *HYDRODECK* surface, then press **Enter** to complete the command.
7. Notice that the *Canopy* relationship has been added to the tree. If you expand the relationship, you'll see the absolute max error, RMS error, max signed error, and minimum signed error for the comparison of all of the points in the selected group to the selected surface.
8. Let's open up the Relationship Report so we can see this relationship's values update in real-time. Double-click the *Relationships* category. The ***Relationships Report*** dialog shows the desired relationship parameters for a given collection in real-time.
9. By default, only the relationship summary is displayed. Let's turn on more details. Right-click the *Canopy* relationship and choose **Report Options** from the menu. In the ***Report Options*** dialog (Figure 29-60), select the **Single Line** format:

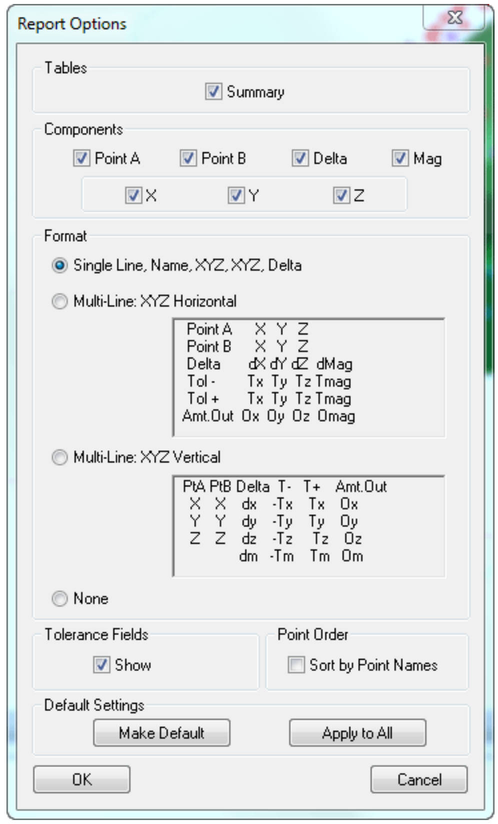


Figure 29-60. Report Options for the relationship.

10. Click OK to accept the change. In the ***Relationship Report*** window, you can now see the deltas and resulting deviations for each point, as well as a summary for the relationship.

11. Let's move the instrument up in Z. Since measurements are "tied to" instruments, the resulting measured points will move up in Z as well. Expand the *Instruments* category and right-click on the instrument. Choose **Properties** from the context menu. In the ***Instrument Properties*** dialog, click the Transform button. Click on the up arrow for the Z value to move the instrument up in Z, and watch as the Relationship Report updates all of the calculations. After moving the instrument up a little bit in Z, close the dialog and click OK in the ***Instrument Properties*** dialog. Notice now that our measured points are now well above the CAD surface.

12. Close the ***Relationship Report*** window by clicking the Done button. Let's create another relationship for the measurements of the sides of the Hydroplane. Choose **Relationships>Groups to Objects** and give the relationship the name Sides. Leave the projection options at the defaults and click OK.

13. This time, let's assign the point group to the relationship AFTER we've created it. This illustrates the idea that you can set up relationships in a nominal file prior to measuring anything, then assign measured points later. (You can even automatically assign them AS you measure by "Trapping Measurements"). Just press **Enter** to continue through the Groups prompt. As before, when prompted for the Objects, pick the *HYDRODECK* surface and press **Enter** to finish the command. Here, you've set the relationship up so that only the surface has been assigned--the groups involved in the relationship are not yet specified.

14. Expand the *Sides* relationship. Note that the tree indicates that there are *No Points Specified*. Let's fix that. Right-click on the collection, and in the context menu, choose **Associate Data>Points>Groups**. Double-click the *Top* group and press **Enter**.

15. Let's check to make sure that we've assigned the right objects to our relationship. Right-click the *Sides* relationship in the tree and choose **Highlight Entities**. The *HYDRODECK* surface will highlight, as well as the *Top* point group. We just made a mistake! We accidentally selected the *Top* group instead of the *Sides* group! No problem, we'll fix it. From the menu, select **View>Clear All Highlights** to remove the highlighting. Right-click the collection, and repeat step 14, only this time, select the *Sides* group.

16. The collection should now be set up correctly. Go ahead and set up a Groups to Objects relationship between the *Top* group and the *HYDRODECK* surface as well, and name it *Top*.

17. Notice the horizontal plane named *Tabletop Plane*. This part was surveyed on a flat tabletop, so it would be impossible to measure the bottom surface of the object. Instead, we created a nominal plane representing the top of the table, and measured a few points around the edges of the part to establish the table surface. Let's build a relationship between the *Plane* points and the *Tabletop Plane* plane, which we'll use in a later tutorial (Minimizing Relationships) to fit our measured data to the CAD model.

18. From the menu, select **Relationships>Points to Objects**. Give the relationship a name of *Tabletop*, and accept the default projection options. When asked for the points, double-click the 6 green points around the edge of the model (the ones in the *Plane* group). We could have done this using a Groups to Objects relationship, but we wanted you to see a different way to create a relationship. Press **Enter** to accept the points.

19. When prompted for the objects, double-click the horizontal *Tabletop Plane* plane. Press **Enter** to complete the command.

20. We also measured a scribe line passing through our part

which indicates the part's centerline. We measured this line by placing the probe directly over the scribe line and measuring the points. Repeat steps 18 and 19 one final time to create a relationship between the *Centerline* points and the *CL Plane* plane. Give the relationship a name of *Centerline*. However, this time, in the projection options, override the target offset value to zero (Figure 29-61). Since we measured a centerline and want to compare it to a plane defining the centerline of the part, we do not want to account for a probe offset. In other words, we want to compare the center of the probe to the plane directly, as seen at right.

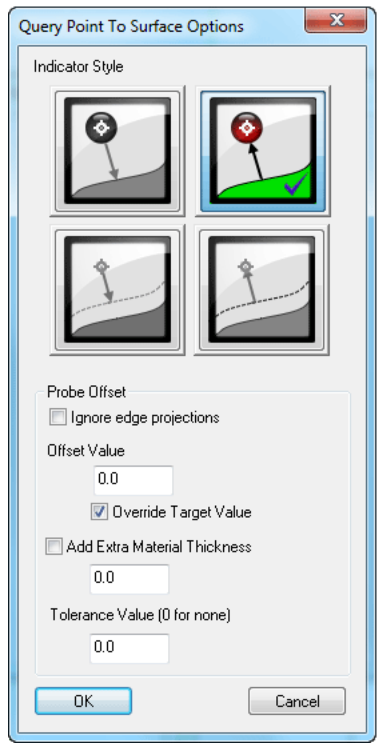


Figure 29-61. Overriding target offsets to zero.

21. We've now set up 5 relationships that show the real-time deviation between selected features and the corresponding measured points. As the instrument that measured the points is moved around, the relationships immediately recalculate and update the deviation values. This real-time re-calculation is very useful, particularly with instrument alignment and real-time building/part mating processes.

Conclusion

In the next tutorial, Minimizing Relationships, we'll explore the power of Relationship Minimization and discover how it can be used to perform a variety of complex alignments with constraints.

Minimizing Relationships

Note: This tutorial requires SA Ultimate or greater.

- **Skill Level.** Intermediate
- **Description.** In this tutorial, we will explore relationship minimization, an extremely useful capability used for alignment, assembly, and a host of other applications. We'll use relationships to locate our measurements relative to a CAD part, using a series of different situations.
- **Areas Covered.** Moving objects by minimizing relationships, local minima, Standard vs. Direct Search optimization, and relationship weighting.
- **Time to Complete.** Approximately 25 minutes.

Relationship Minimization

One of the great advantages of relationships is their ability to define constraints and alignment parameters. In traditional metrology software, degree-of-freedom constraints for alignments are usually explicitly specified. For example, if a pin is constraining an assembly, one might explicitly lock a degree of freedom along the radial directions of the pin, so that the pin only allows travel along its length.

Relationships, by their very nature, automatically take care of constraints when they are created. Different geometric setups will result in different constraints. For example, if a relationship is set up between a set of points and a plane, the plane will by its very nature constrain distance from the plane, but will allow the points to rotate in the plane's surface. As a result, a plane introduces a positional constraint along its normal, and rotational constraints in the plane of the plane. The ability to set up these constraints "naturally", without explicitly defining further constraint parameters, is one of the significant advantages of relationships over other alignment methods.

1. We'll start by opening a tutorial file included with your installation of SA. In SA, choose **Help>Open Sample SA Files**. Choose the `Relationships.xit` file.
2. In this file, five relationships have been set up for you. Three of these relationships are between the CAD model and the measurements of specific features of interest. The last two (*Tabletop* and *Centerline*) relate measurements to a table surface and center plane, respectively. To see how these relationships were created, see the Creating Relationships tutorial. Notice also how the measurements have not yet been aligned to the CAD model. That's one of the tasks we'll be accomplishing in this tutorial.

3. Let's align our measured data to the CAD model. From the menus, choose **Relationships>Move Objects by Minimizing Relationships**. In this case, we only want to move the instrument (and by association the targets measured by that instrument). So, when prompted for the Objects to Move, just press `Enter`. When prompted for the Instruments to Move, double-click the instrument and press `Enter`. The ***Minimize Relationships*** dialog will appear (Figure 29-62).

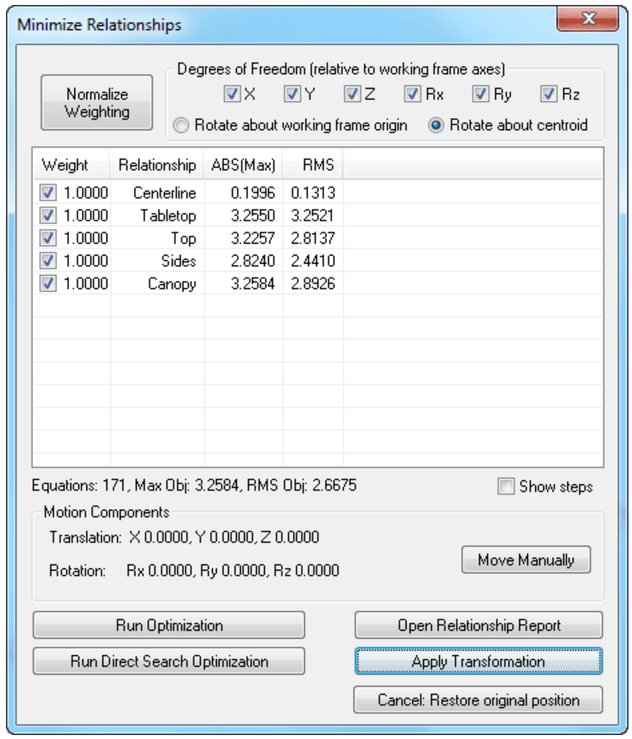


Figure 29-62. The Minimize Relationships dialog.

4. The process of relationship minimization finds the optimum position of the specified moving objects and instruments that results in the smallest RMS error. Let's first take a look at how our measured points fit to the CAD model, without including our tabletop or centerline in the solution. Uncheck the *Centerline* and *Tabletop* relationships to exclude them from the minimization process (Figure 29-63), then click the Run Optimization button. This is the "standard" method of minimizing relationships.

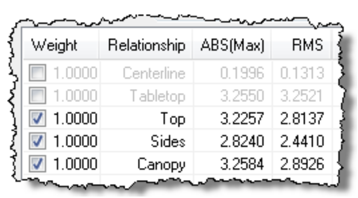


Figure 29-63. Excluding the Centerline and Tabletop relationships from the solution.

5. The instrument's position and orientation will be adjusted to arrive at a minimum error, which represents the ideal fit. Note that as the solution progresses, the *RMS Objective* value

decreases toward zero (Figure 29-64).

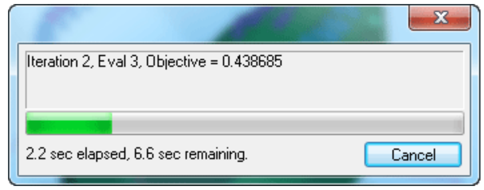


Figure 29-64. The objective progresses toward zero as the relationship solves.

6. The resulting absolute maximum and RMS errors are also displayed for each individual relationship in the ***Minimize Relationships*** window. The results look good, but we didn't get a good idea of what was going on. Let's move the instrument back up in Z. Click the Move Manually button, and in the ***Transform*** dialog, type *5* into the Z value and press **Enter**. Close the ***Transform*** dialog. Let's also choose an option so we can actually see the intermediate positions as the relationship is being solved. In the ***Minimize Relationships*** dialog, check the **Show Steps** checkbox (Figure 29-65).

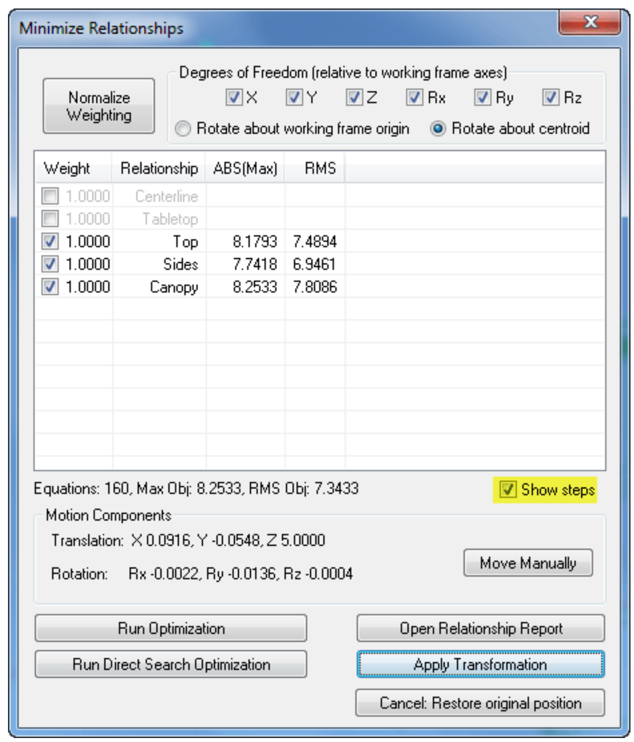


Figure 29-65. Turning on the Show Steps option.

7. Move the ***Minimize Relationships*** window to the side of the screen so that you can see the graphical view, then again click the Run Optimization button. Watch as SA solves the equations to minimize the error.

Local Minima

Under the hood, relationships are projecting measured points to the CAD surfaces and calculating the resulting deviations. If the measured points are too far from the CAD surfaces, or if the points are flipped from their true orientation, then depending on the geometry of the setup and the surfaces, it is possible for the solution to reach a local minimum, and stop prematurely.

1. Again, click the Move Manually button. In the ***Transform*** dialog, enter *180* for Ry and *-10* for Z. Notice that the measured data is now flipped upside-down relative to the CAD model. Close the ***Transform*** dialog, and again click the Run Optimization button.

2. The relationship solves and stops. Click the Apply Transformation button, and take a look at the graphical view. Clearly our points are flipped upside-down relative to the CAD model, and we have a large RMS error. What's going on here? What's happening is that the solution is getting caught in a local minimum. When minimizing relationships, you always want your measured points to have a "clean approach" to the geometry that you're relating to. This means that there shouldn't be any large rotations required that cause the relative orientation between the CAD and the measured data to "flip". Depending on the geometry of the setup, the solution could get caught up in the wrong minimum, as seen here.

3. Let's fix our local minimum problem so we can ensure that the relationship solves to the correct position. Click the Views button to go to the *Side* View (Figure 29-66).

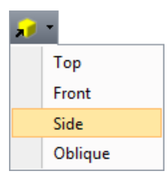


Figure 29-66. Selecting the predefined Side view.

4. From the menu, select **Instrument>Drag Graphically**. Hold down the right mouse button to rotate the instrument approximately 180 degrees (Figure 29-67).

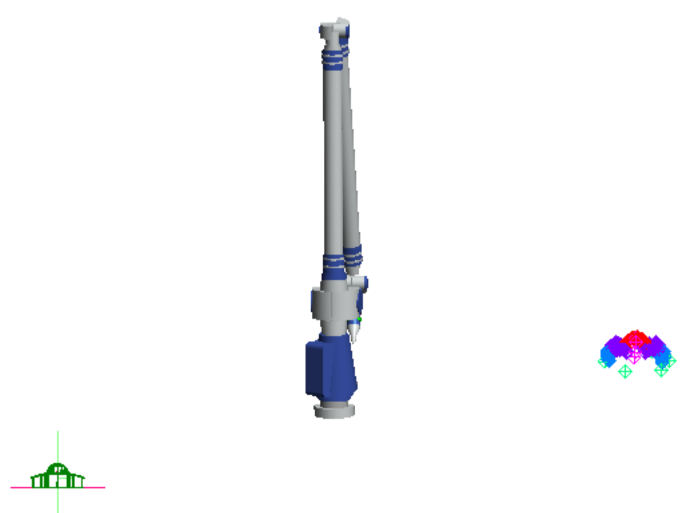


Figure 29-67. The instrument has been rotated about 180 degrees out.

5. Again, choose **Relationships>Move Objects by Minimizing Relationships**. As before, don't move any objects, but move the instrument. Ensure that the *Centerline* and *Tabletop* relationships are still deselected. Click the Run Optimization button. You'll notice that, after a few seconds, the solution comes to an abrupt and premature end--the points remain in their original positions. Because the initial condition for the solution is so far from the CAD model, a local minimum is being encountered.

6. Instead, click the Run Direct Search Optimization button. Direct Search Optimization is a more "brute-force" method of finding a minimum. It involves exploring the solution space more thoroughly to find a more optimized orientation, then moving to that intermediate position and checking again. This method is slower, and requires more time to solve (sometimes, significantly more time). Depending on the setup, it can also end up with a "flipped" result. However, it is less likely to get caught in a local minimum. After a few seconds of calculating--once the *Objective* value is under about 0.8"--click the Cancel button.

7. Notice that, rather than returning to the original position and orientation, the instrument and points stopped in their current position. This is a helpful behavior, because we can now switch to the traditional optimization method to close up the fit the rest of the way. Click the Run Optimization button, and the solution should be reached relatively quickly. When finished, click the Apply Transformation button. If you notice that your solution has solved to the incorrect orientation, move and rotate the instrument for a better starting condition and try again.

8. What is the lesson from all of this? The fact is, you'll avoid all of these issues with reaching local minima if you start with a good initial condition. As long as you manually orient the instrument so that the points have a relatively clean approach to the surface, and as long as they aren't too far away from the surface, the solution can be approached with no trouble. Many people will do a rough orientation using **Quick Align**, then fit with more data using relationships. In a situation like that, you won't encounter problems with local minima.

Incorporating the Tabletop and Centerline

1. Suppose that we now wish to incorporate the measurements of the tabletop to our fit. If we assume that the bottom surface is flat, and that the tabletop is flat, then our measured

tabletop points represent the bottom surface of the CAD part. Press **Ctrl+Tab** to reselect the last command, which should be the **Move Objects by Minimizing Relationships** command. As before, pick to move just the instrument.

2. This time, ensure that the *Tabletop* relationship is also selected, then click Run Optimization again. Our RMS error has increased from 0.0013" to 0.0022", but we've now allowed our measurements of the tabletop to influence the solution. Since the *Tabletop Plane* plane is facing directly along the Z axis, the relationship is only affecting the final solution's Rx, Ry, and Z results. X, Y, and Rz are not affected due to the geometry of the plane. (Rotating about Z or translating along X or Y will not affect the distance of the *Plane* points from the *Tabletop Plane* plane. Click the Apply Transformation button to accept the results.

3. Click the Views button in the toolbar to select the *Top* view, then zoom in on the CAD model and measured points. In the tree, right-click the *Point Groups* category header and choose **Hide All** from the context menu. Then, right-click the *Centerline* point group and choose **Show** from the menu (Figure 29-68).

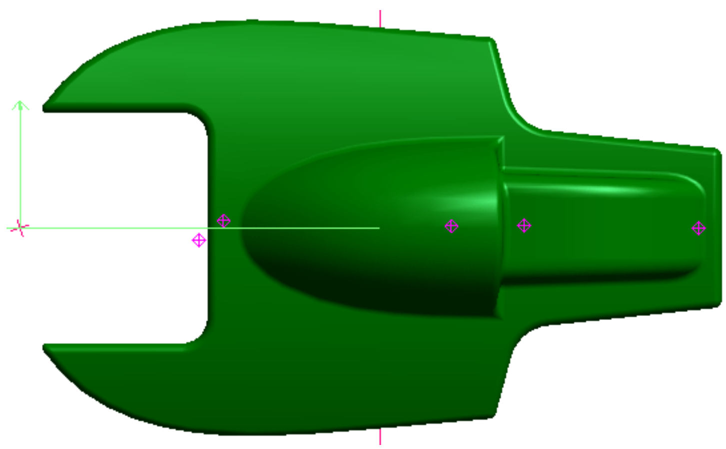
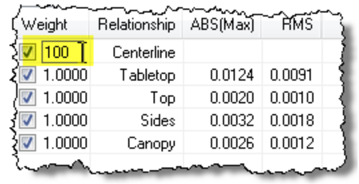


Figure 29-68. Showing the Centerline point group.

4. We haven't yet incorporated our *Centerline* relationship into the solution. Let's say, for sake of this tutorial, that it is very important to us that the measured centerline points are aligned well to the centerline plane. We still want the measured points of the CAD surface to influence our fit, but we consider the centerline measurements to be much more important. We can weight the *Centerline* relationship to have a much stronger influence in the overall solution.

5. Press **Ctrl+Shift+TAB** and choose the **Relationships>Move Objects by Minimizing Relationships** command from the list, and specify to move just the instrument. In the ***Minimize Relationships*** dialog, ensure that all five relationships are checked so that we incorporate them all into the minimization process.

6. Single-click the row with the *Centerline* relationship so that it is selected. Then, single-click the *Weight* column. Give the *Centerline* relationship a weight of *100 (*Figure 29-69*)*. This is equivalent to saying that we consider errors from the points in the *Centerline* relationship to be 100 times more important than errors from the points in the other relationships.



Weight	Relationship	ABS(Max)	RMS
100	Centerline		
1.0000	Tabletop	0.0124	0.0091
1.0000	Top	0.0020	0.0010
1.0000	Sides	0.0032	0.0018
1.0000	Canopy	0.0026	0.0012

Figure 29-69. Setting the weight of the Centerline relationship to 100.

7. Click the Run Optimization button. It may be difficult to see, but the solution has been adjusted to try to minimize the centerline errors with more effort than the other relationships, due to the higher weight assigned to the *Centerline* relationship. Notice that the RMS errors have jumped way up. As you can see, the points measured on the centerline were not measured carefully. Uncheck the *Centerline* relationship to again remove it from the minimization process.

8. This time, let's give the *Tabletop* relationship a little less weight, since it's not a direct measurement of the bottom surface of the part. Give it a weight of 0.5, and click the Run Optimization button. Now, the tabletop is still incorporated into the solution, but it is not considered to be as important.

Conclusion

We've seen here that it is important to give relationships a good "initial condition" to ensure that relationship minimization does not get caught up in a local minimum. We've also seen how relationship weighting can be used to assign more or less influence to different measured features when minimizing.

Geometry Relationships

- **Skill Level.** Intermediate
- **Description.** In this tutorial, we will cover how to create geometry relationships for reporting.
- **Areas Covered.** Creating Fit only, Compare only and Fit and Compare Relationships.
- **Time to Complete.** Approximately 20 minutes.

Fit and Compare Relationship

1. In SA, start by opening the `Geometry Relationship.xit` file from the Samples directory under **Help>Open Sample SA Files**.
2. We will first make a relationship that will take measured points, fit geometry to those points, and compare this geometry back to the nominal geometry. Navigate to **Relationships>Geometry Comparison>Fit and Compare to Nominal**. Select the nominal geometry and then select the points from the *Measured Points* point group that are above the red circle. Accept the default name for the relationship.
3. Double-click the *Fit Circle - Big Hole* relationship in the tree to expose the ***Geometry Relationship Report Options***. Here you can control tolerance, fit constraint and geometry fit settings (Figure 29-70).

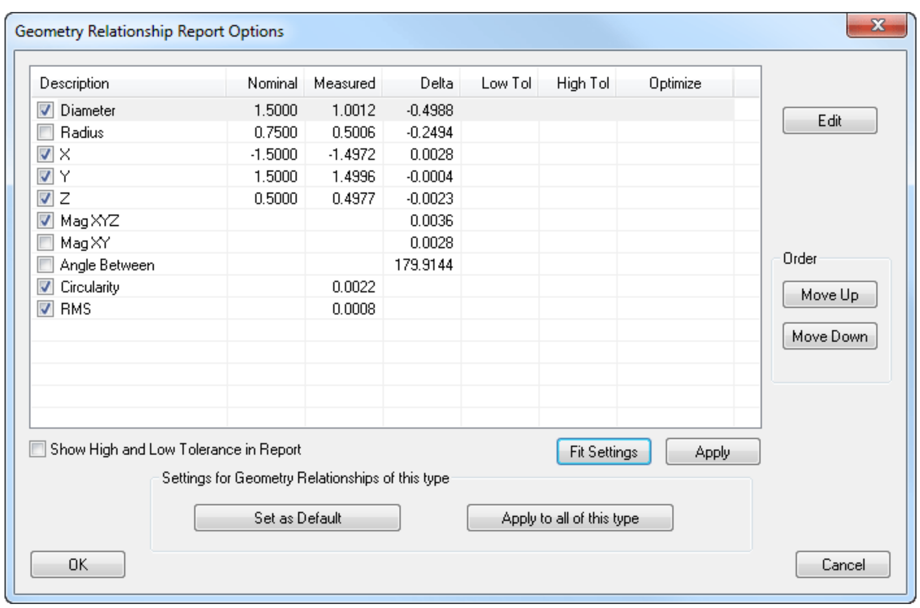


Figure 29-70. The Geometry Relationship Report options.

4. Let's change the Fit Settings so that circle is offset correctly. By default it was treated as an outside measured circle when in fact it was an inside measured circle (Figure 29-71).

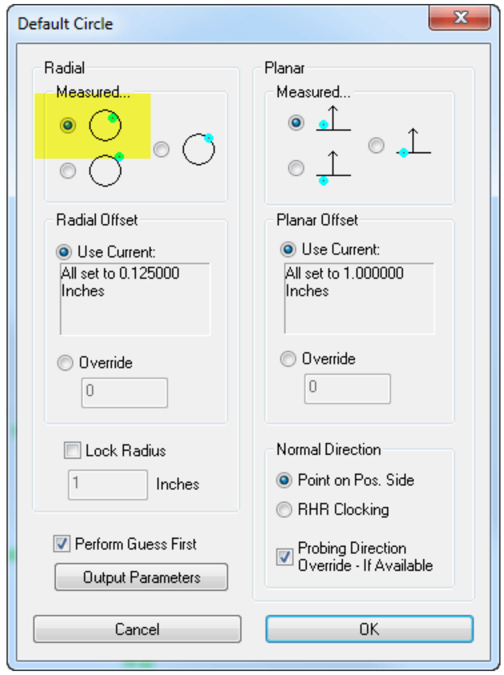


Figure 29-71. Setting the circle fit options to offset outward.

5. Let's set a tolerance for the X and Y components of 0.001". Double-click the *X* column in the properties dialog. The ***Criteria Properties*** dialog will appear. Set the tolerance for both high and low tolerances (Figure 29-72).

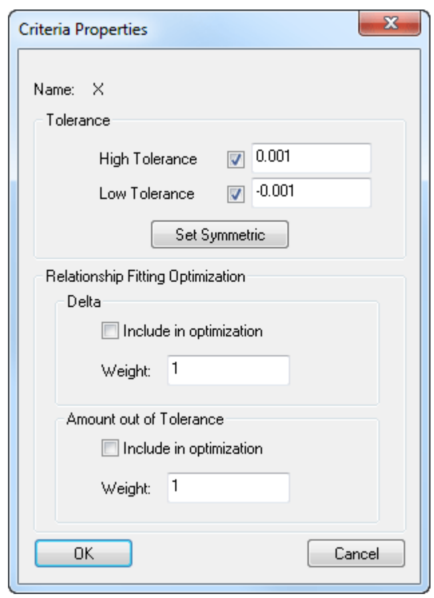


Figure 29-72. Modifying the high and low tolerances for the X attribute.

6. Now uncheck the **Z** and **MagXYZ** rows. Press OK to accept and close the dialog.

7. Right-click *Fit Circle - Big Hole* and select **Add Callout** (Figure 29-73).

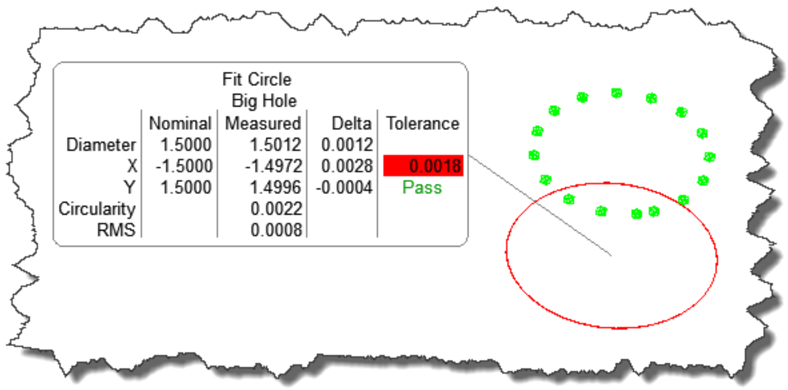


Figure 29-73. Adding the relationship callout to the circle.

8. The best part about a Geometry Relationship is its ability to dynamically update--if the points change, the geometry and reported values will update accordingly. Select **Instrument>Drag Graphically** and move the instrument around in the graphical view. Notice how the geometry and callout update automatically (Figure 29-74). Press **ESC** to cancel and restore the instrument position.

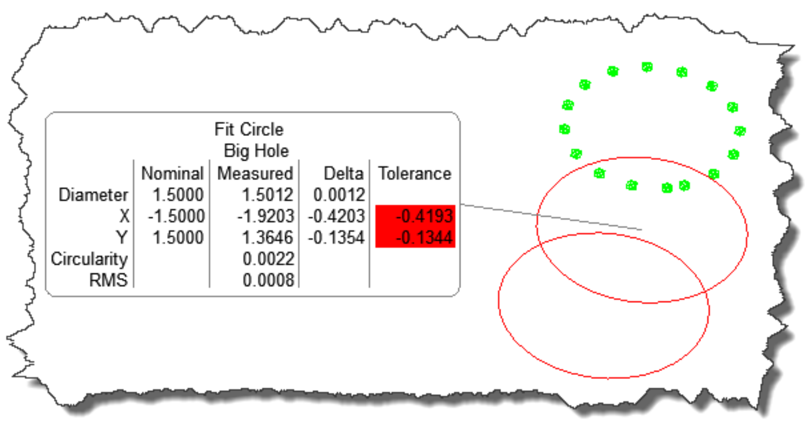


Figure 29-74. The callout automatically updates as the situation changes.

Fit Only Relationship

1. Fit Only relationships are similar to Fit and Compare to nominal. Select **Relationships>Geometry Comparison>Fit only** and select the group of points that are rectangular in shape. You will be prompted to select a geometry to fit to the selected points--select *Plane*. Accept the default name.

2. Right-click *Fit Plane* and select **Add Callout**. Fit only relationships utilize the same Geometry Relationship Report Options as Fit and Compare to Nominal, although with different reported items (Figure 29-75 and Figure 29-76).

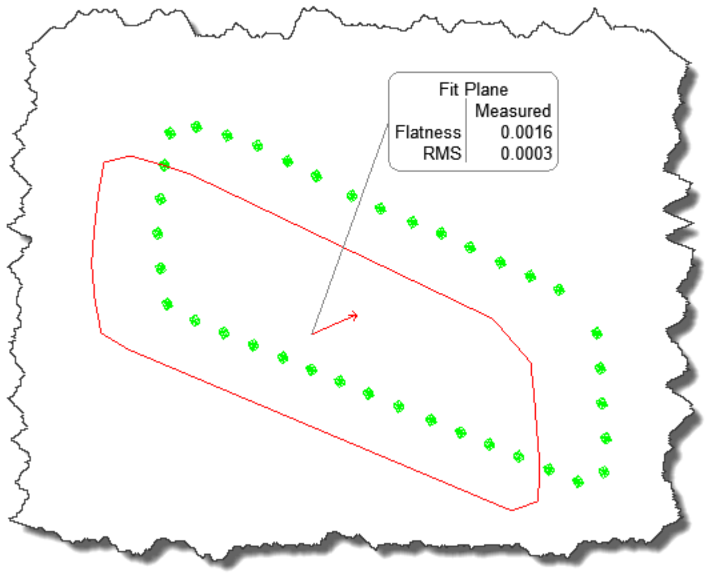


Figure 29-75. A fit-only relationship.

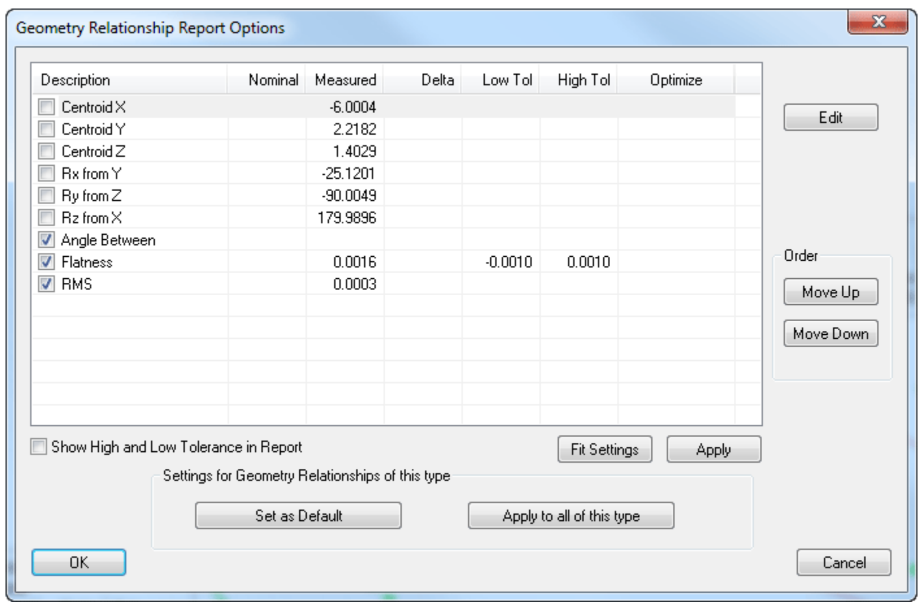


Figure 29-76. Geometry Relationship report options for fit-only relationships.

Compare Only Relationships

1. Now let's compare two circles. Select **Relationships>Geometry Comparison>Compare Only** and select the *Nominal::Small Hole* and *Measurements::Small Hole Measured* items from the tree and accept the default name.
2. Right-click *Circle: Small Hole Measured - Small Hole* and select **Add Callout**.
3. Compare only relationships utilize the same Geometry Relationship Options as well (Figure 29-77 and Figure 29-78).

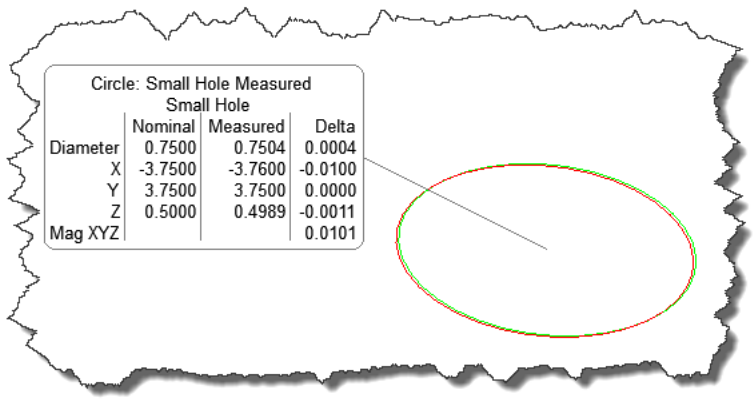


Figure 29-77. Compare-only relationships.

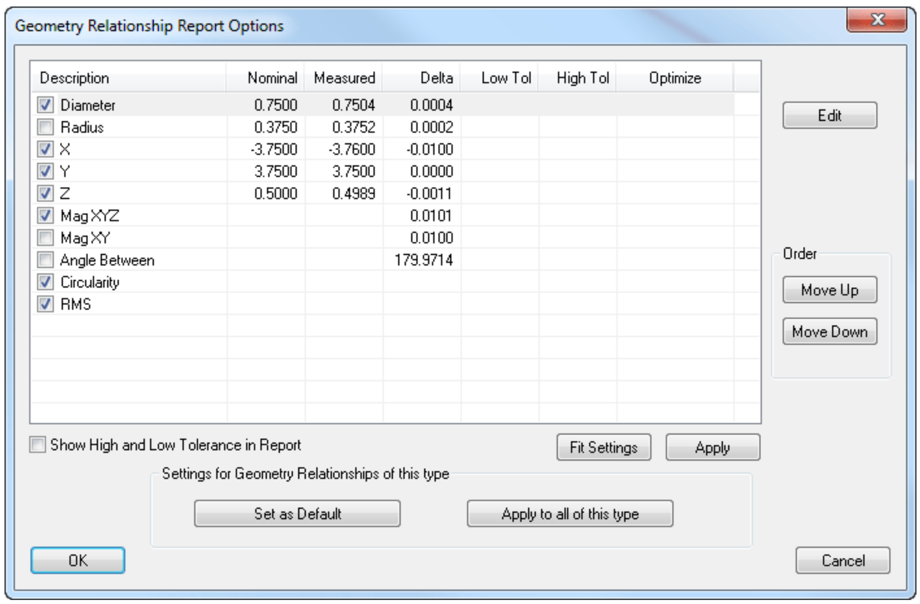


Figure 29-78. Geometry Relationship report options for compare-only relationships.

Conclusion

Geometry Relationships provide a quick and powerful method for creating and comparing geometry dynamically.

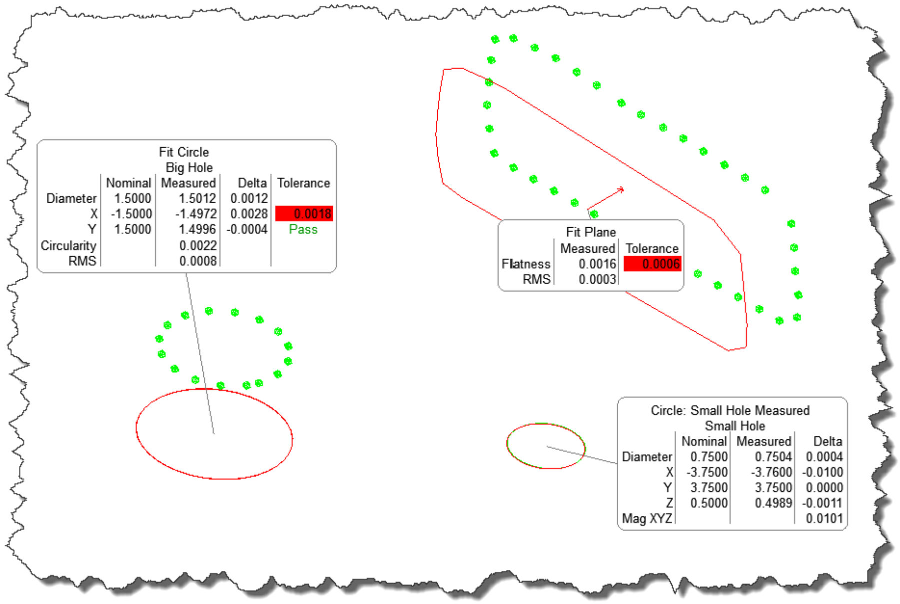


Figure 29-79. Various geometry relationships in a job.

Designing a GD&T Inspection

Note: This tutorial requires SA Ultimate.

- **Skill Level.** Beginner
- **Description.** In this tutorial, we will cover how to design a GD&T inspection routine for repeated use.
- **Areas Covered.** Importing CAD with GD&T annotations, creating Datums and Feature Checks and assigning nominal points and views.
- **Time to Complete.** Approximately 15 minutes.

Importing CAD with GD&T annotations

The first step for this tutorial is to import the nominal CAD model with GD&T annotations. If your CAD models do not contain GD&T annotations, please refer to the tutorial on creating GD&T annotations.

1. In SA, start a new job file by selecting **File>New** from the menu or choosing the New File Icon from the main toolbar.
2. Import the native CAD model with GD&T annotations by selecting **File>Import>Direct CAD Access** or use Auto Import from the main toolbar.
3. Choose `Sample CAD.CATPart` from the Samples file folder in the SpatialAnalyzer install directory. See Direct CAD Access for supported formats.
4. When the ***Direct CAD Access Settings*** dialog appears, make sure the **Annotations** option is selected and press OK (Figure 29-80).

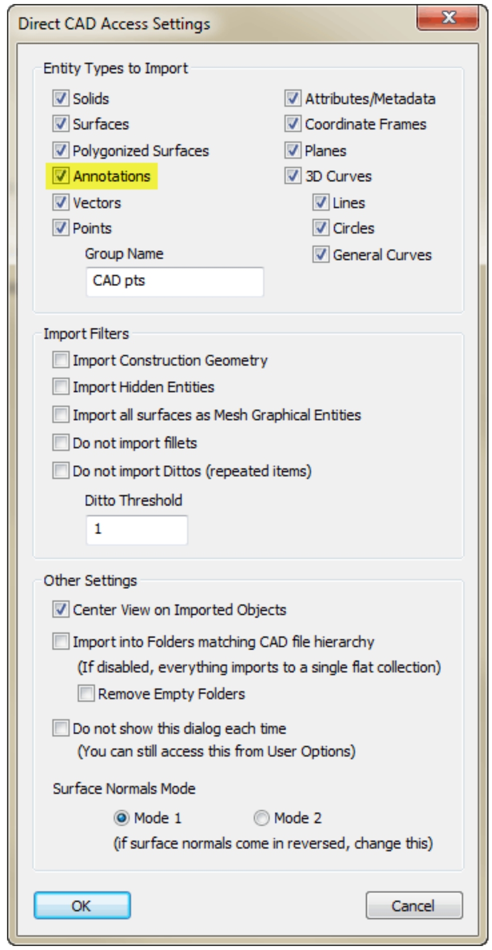


Figure 29-80. Ensuring that the Annotations option is selected in the Direct CAD Access import settings.

5. Once imported, you will find that the *Annotations* group is created with the imported annotations.

Creating GD&T Datums and Feature Checks

For this tutorial we will inspect only one of the GD&T Features.

1. Right-click *Geometrical Tolerance.1* in the tree and choose **Make Feature Check**. You will notice two new categories are created, *GD&T Datums* and *Feature Checks*. These two new categories act as bins for the respective measurements.

Designing the Inspection Routine

Now that the feature checks have been made, the inspection routine can be designed which will set views and nominal points for each GD&T Datum and Feature Check.

1. To design an inspection, simply right-click collection *A* and select **Inspect>Design**. A dialog will appear which contains options for the design of each GD&T Datum or Feature Check (Figure 29-81).

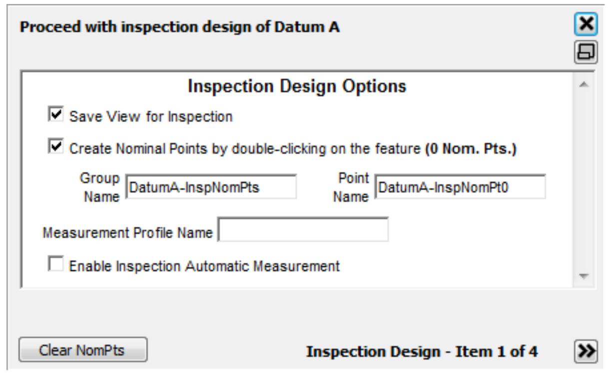


Figure 29-81. Options for designing an inspection.

2. The design dialog will guide you through the design of each GD&T Datum or Feature Check. So for *Datum A*, orient the view how you would like the user to see Datum A during inspection. Now let's put some nominal inspection points on the surface by clicking the areas for inspection on Datum A. Use the minimize button to minimize the dialog so more of the graphical view is available (Figure 29-82).

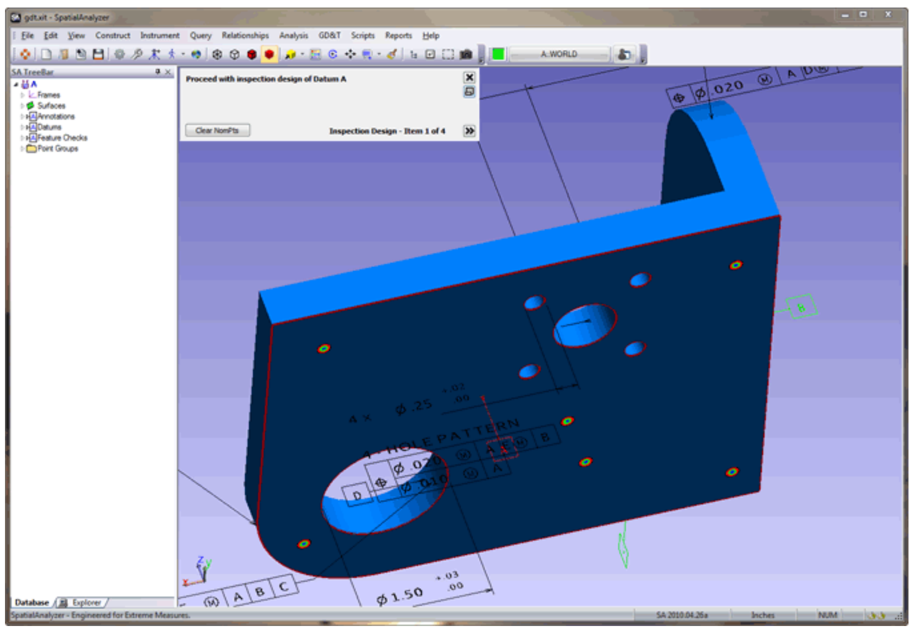


Figure 29-82. The options are now minimized.

3. Now that Datum A is designed, proceed to the next GD&T Datum or Feature Check by clicking [»] to advance. Do the same for the remaining GD&T Datums and Feature Checks. Once complete, close the dialog using [X].

Rehearsing a Designed Routine

Now that the inspection routine is designed, it can be rehearsed to check the work flow, nominal points and views.

1. To start Rehearse mode, right-click collection *A* and select **Inspect>Rehearse**. A dialog will appear with options regarding the rehearsal (Figure 29-83). During this rehearsal mode,

the mouse acts as an instrument. So with each mouse click, a measurement is simulated with some random error.

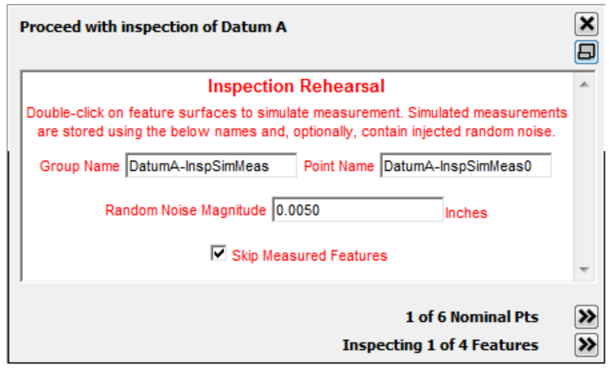


Figure 29-83. Rehearsing the inspection.

2. Now that the rehearse mode is started, let's click the dot for Datum A. With each click a measurement is simulated and the guiding dot will advance. Continue clicking the dots for Datum A. Once all the dots for Datum A are clicked, the inspection will advance to the next GD&T Datum or Feature Check.
3. Do the same as above for the remaining GD&T Datums and Feature Checks. Once complete, close the dialog using ☒.

Conclusion

We have just designed a routine that can be used to guide a user through an inspection process. A user can now simply right-click the collection of interest and select **Inspect**.

CMM Arm Inspection

- **Skill Level.** Intermediate
- **Description.** In this tutorial, we will cover a basic part inspection without CAD while using a Portable CMM Arm.
- **Areas Covered.** Add and Run an Instrument, Watch Windows, Collecting Measurements, Fitting Geometry and SA Reporting.
- **Time to Complete.** Approximately 30 minutes.

General Part to Inspect

For this exercise we will use a generic part in which we would like to determine the size of the large and small bore and the angle between the top face and side face (Figure 29-84).

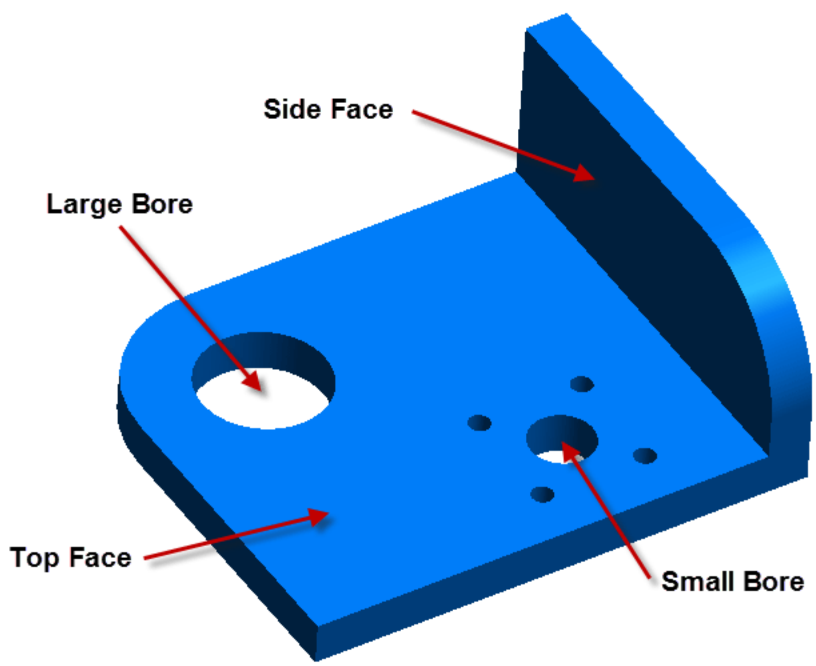


Figure 29-84. The part we'll be inspecting.

Adding and running the instrument interface

1. Start by adding the instrument. For this tutorial we will use an 8 ft Faro Quantum (7 dof) arm. Use the menu item **Instrument>Add** or the Add Instrument icon . Navigate the instrument list, select **FARO Arm USB 8ft & Quantum (7 dof)**, and then click Add Instrument at the bottom of the dialog (Figure 29-85).

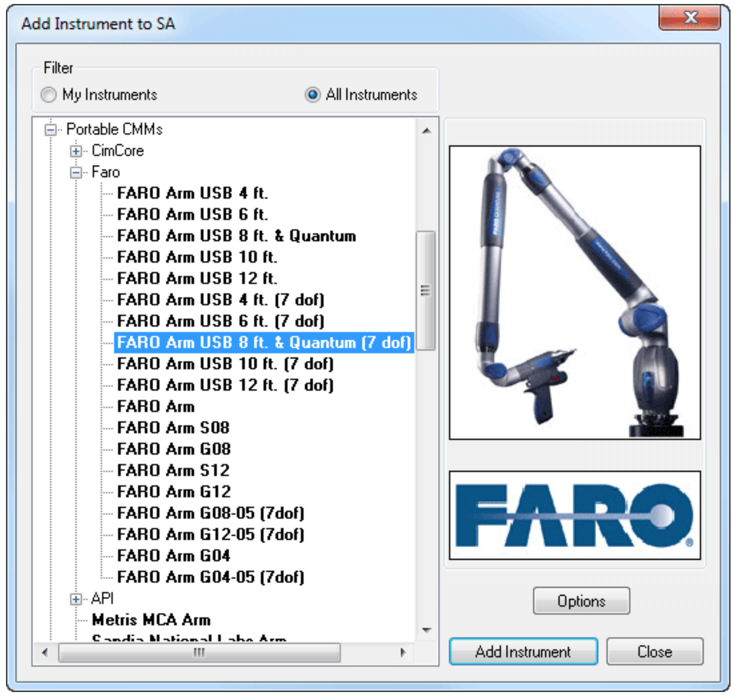


Figure 29-85. Adding the arm to the job.

2. With the instrument added, let's run the instrument interface. Simply use **Instrument>Run Interface Module and Connect** or use the Run Instrument Interface icon .

3. Once connected the instrument interface will appear. Now we are ready to measure!

Establishing Control Points

A good rule of thumb is to always establish a set of control points that can be used as a reference. These control points can be used to check drift and relocate the instrument.

1. Lay out three or more control points to measure. For our case we will measure four holes on the table by placing the probe in the hole.

2. Let's set the group and target name in the instrument interface so the group is called *Control* and the target name is *C1*.

3. Activate the discrete measurement mode by pressing .

4. Measure each of the four control points by placing the probe in the hole and pressing the green button on the arm. The target name in the instrument interface will automatically increment after each measurement.

5. Now check the measurements to for repeatability. Right-click the *Control* point group and select **Add Watch Window**. Move the probe to each measured point and note the deviation (Figure 29-86). The deviations should be extremely low--if not, the

control points should be re-measured.

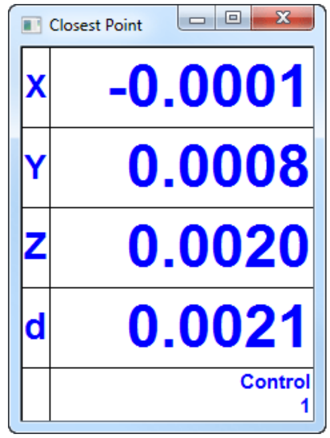


Figure 29-86. The closest point watch window.

6. Once measured, your graphical view should look similar to Figure 29-87.

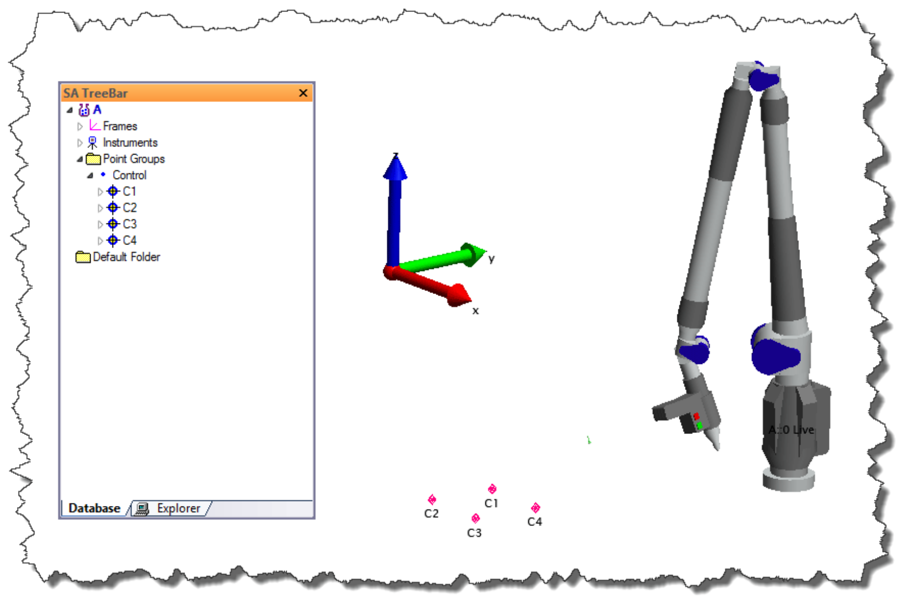


Figure 29-87. Measured control points.

Measuring the Bores

Let's start by measuring the Large and Small Bores. We will take a series of points in each bore and then fit cylinders to each set.

1. Set the group name to *Large Bore* and the target name to *1*.
2. Activate the discrete measurement mode by pressing .
3. Measure points inside of the large bore by pressing the green button to trigger the measurement. Once complete, close the discrete measurement dialog.
4. Change the group name to *Small Bore* and the target name to *1*.
5. Activate the discrete measurement mode by pressing .

6. Measure points inside of the small bore by pressing the green button to trigger the measurement. Once complete, close the discrete measurement dialog.

7. Your SA job should look like Figure 29-88.

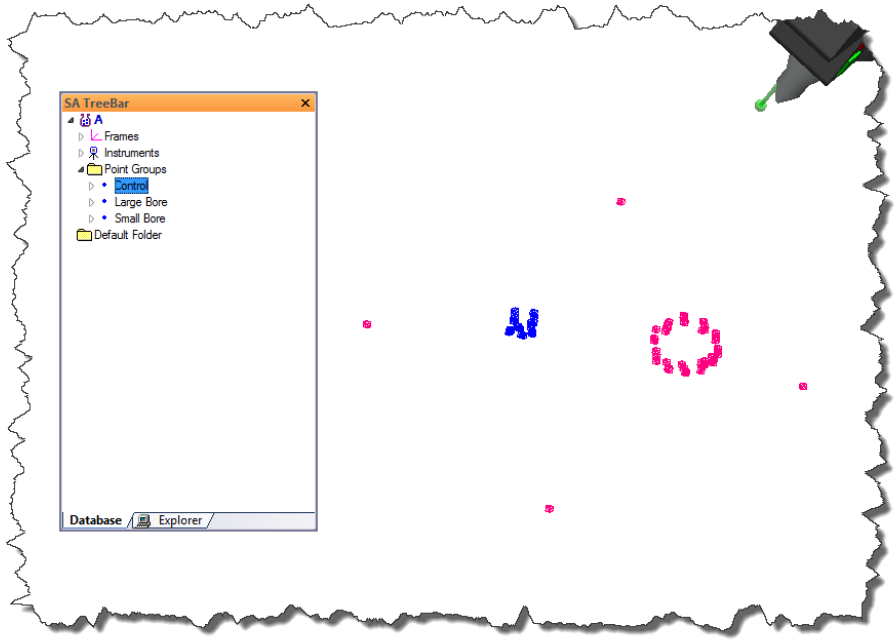


Figure 29-88. The large and small bores have been measured.

Measuring the Top and Side Face

1. Set the group name to *Top Face* and the target name to *1*.

2. Activate the Stream Points Mode by pressing .

3. Place the probe on the Top Face and press the green button to start the scan. Move the probe over the surface. When finished, press the red button to stop. Close the ***Stream Points*** dialog.

4. Set the group name to *Side Face* and the target name to *1*.

5. For the Side Face we will use the Plane mode. Activate this mode by pressing .

6. Take several points on the Side Face. After three points are measured, the plane fit results will appear. Once you are satisfied with the plane measurements, press Accept Plane.

7. Your SA job should look like Figure 29-89.

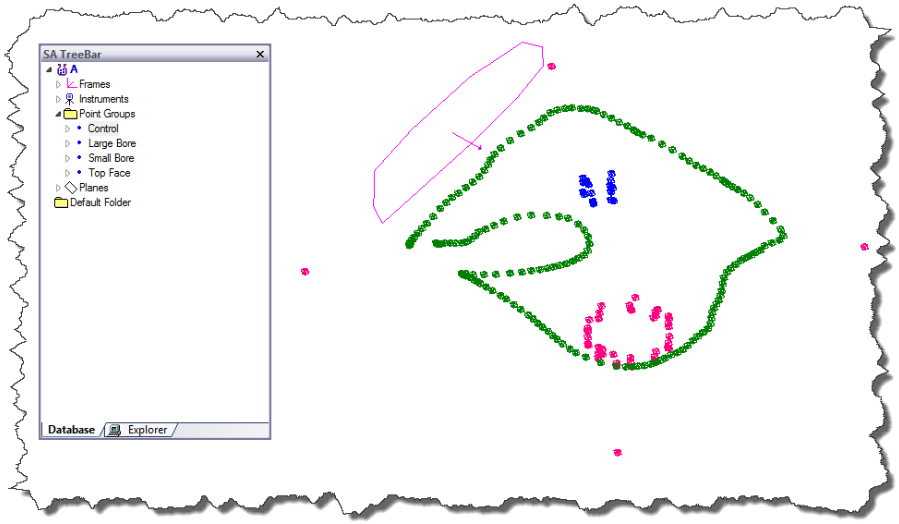


Figure 29-89. After the plane fit.

Fitting Geometry to the measurements

Now that the data is collected, let's begin the analysis. We will start by fitting cylinders to the bores.

1. Fit a cylinder to the Large Bore point group by using **Construct>Cylinder>Fit to Points**. Refer to the Best Fit Geometry from Points tutorial or Geometry Fitting section for more information regarding Geometry Fitting. Name the cylinder *Large Bore*.
2. Repeat the above for the *Small Bore* point group.
3. Now fit a plane to the *Top Face* point group using **Construct>Plane>Fit to Points**. Name the cylinder *Top Face*.
4. Your SA job should look like Figure 29-90.

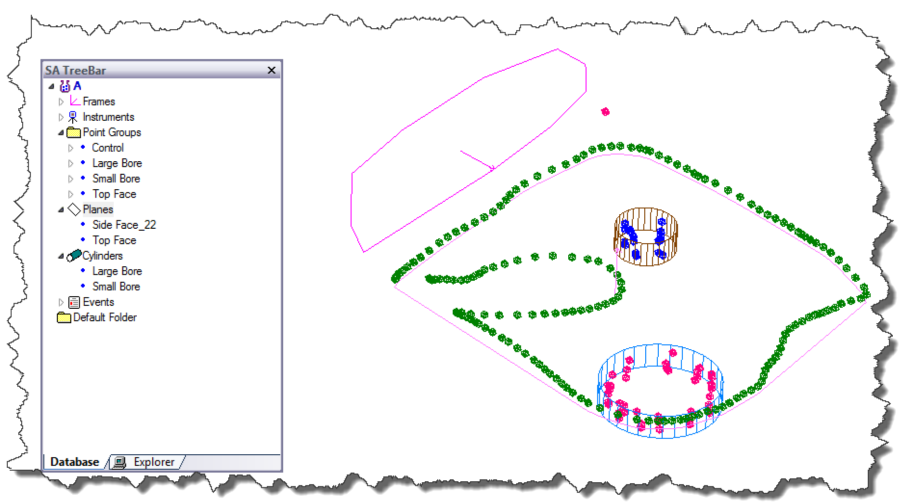


Figure 29-90. After the cylinder fits.

Reporting the results

With the geometry created we can now report the findings. For the report we will use SA Reporting.

1. Create an SA Report by selecting **Reports>Add SA Report**.
2. Orient the graphics to a view that would look good in the report.
3. Press to take a screenshot. This image will now be added to the tree. Click and drag the Photo into the SA report. Place the image so that it is centered at the top of the report.
4. Now drag the *Large Bore* cylinder onto the report. Right-click the cylinder and choose **Report Options**. Uncheck **Show Normal Vector** and **Show Projected Angles on Working Frame**. Repeat this for the *Small Bore* cylinder.
5. So far your report should look like Figure 29-91.

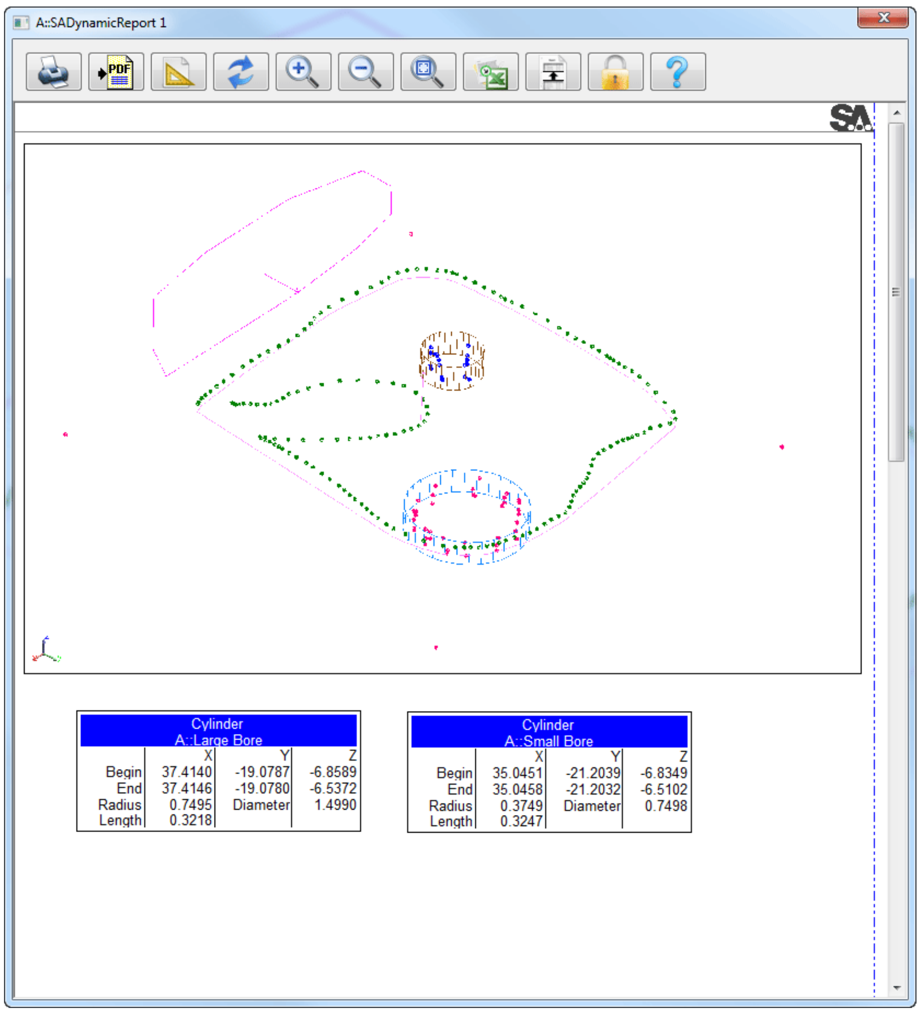


Figure 29-91. A screenshot and both cylinders have been added to the report.

6. Now determine the angle between the Top Face and Side Face. **Use Query>Plane>to Plane: Angle between normals**. Select the

two measured planes. Once completed the results will display and an SA Event will be created.

7. Drag the SA Event into the SA Report.

8. We have now reported our findings. The report will be saved in the tree. If desired, the report can be printed or converted into PDF format using the following icons:

9. The final report will look like Figure 29-92.

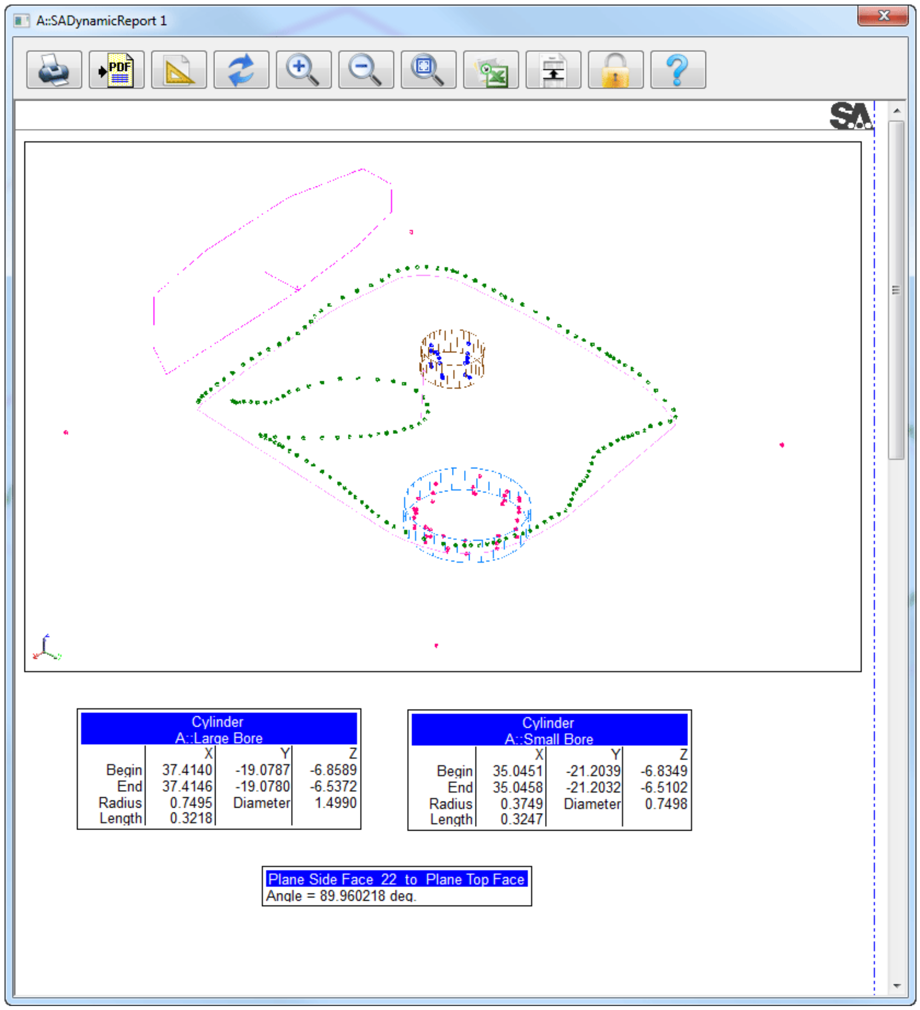


Figure 29-92. The final report.

Laser Tracker Inspection

- **Skill Level.** Intermediate
- **Description.** In this tutorial, we will cover basic inspection of simple geometries and look at their relationships
- **Areas Covered.** Add and Run an Instrument, Watch Windows, Collecting Measurements, Fitting Geometry and Dynamic Reporting.
- **Time to Complete.** Approximately 30 minutes.

General Part to Inspect

For this exercise we will perform a basic inspection of a desk and office. (Try to find one if you don't have one!) The office floor, walls and desk will serve as the measurement artifacts. We will calculate the angle between the walls and the floor and the height of the desk from the floor.

Adding and running the instrument interface

1. Start by adding the instrument. For this tutorial we will use a Faro Laser Tracker. Use the menu item **Instrument>Add** or click the Add Instrument icon . Navigate the instrument list, select the FARO Laser Tracker, and then select Add Instrument at the bottom of the dialog (Figure 29-93).

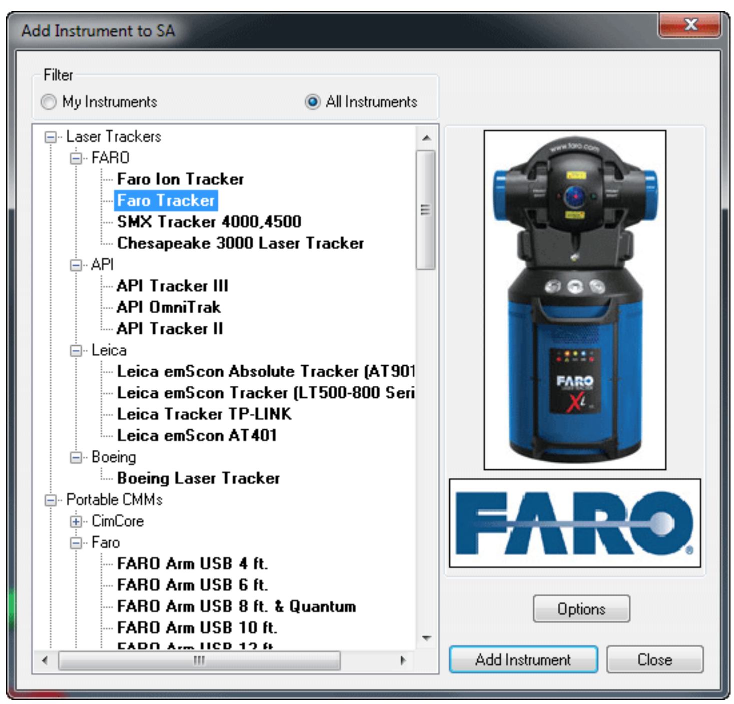


Figure 29-93. Adding the tracker.

2. With the instrument added, let's run the instrument interface.

Simply use **Instrument>Run Interface Module and Connect** or use the Run Instrument Interface icon . Once connected the instrument interface will appear. Now we are ready to measure!

Establishing Control Points

A good rule of thumb is to always establish a set of control points that can be used as a reference. These control points can be used to check drift and relocate the instrument.

1. Layout three or more control points in the room. For our case, we secured four SMR nests in various locations in the room.
2. Set the group and target name in the instrument interface so the Group is called *Control* and the Target name is *1*.
3. Make sure the correct targeting is selected in the Tracker interface. For our case we are using a 1.5" SMR. To change the active target, press and select the respective target (Figure 29-94).

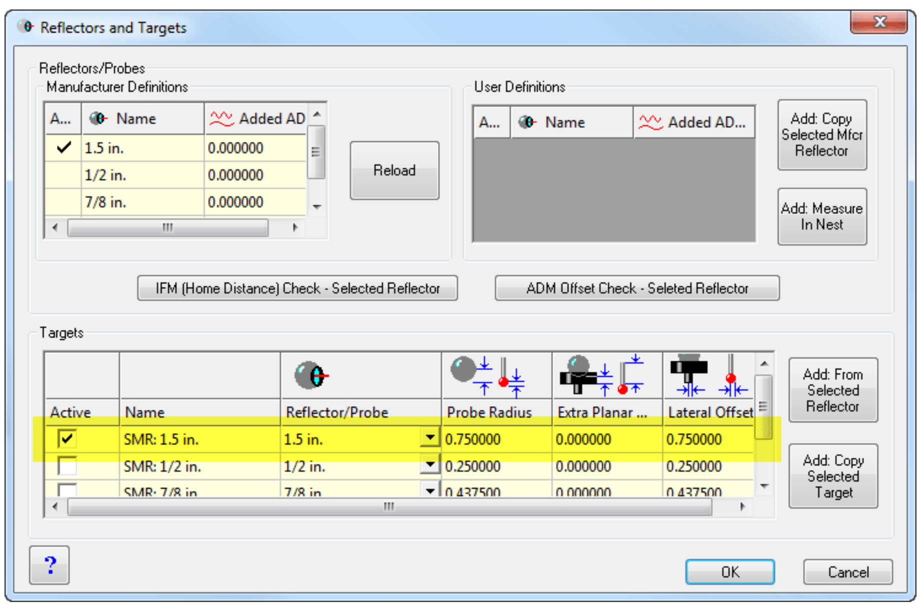


Figure 29-94. Selecting the proper target.

4. To acquire the points, select the **Stable Pt. to SA** measurement profile. By default, Quick Select 2 is set to Stable Pt. to SA.
5. Open the measurement profile parameters . Once opened, change the *Points* value to the number of control points you plan to measure (Figure 29-95). In our case we are measuring only four points. When finished press the Save button at the top.

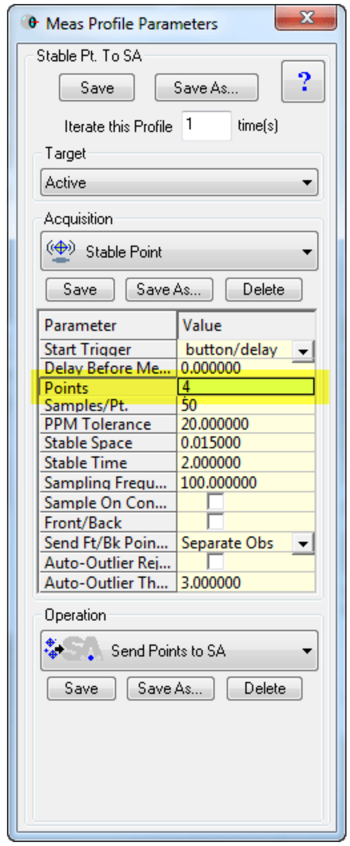


Figure 29-95. Specifying the number of points to measure in the profile.

6. Measure each of the control points, start the measurement mode by pressing the Measure button or **F3**. The measure dialog will prompt you when to move the target and when the target should be stable. The target name in the instrument interface will automatically increment after each measurement.

7. Now check the measurements for repeatability. Right-click the *Control* point group and select **Add Watch Window**. Now move the SMR to each nest and note the deviation. The deviations should be extremely low (Figure 29-96). If not, the control points should be re-measured.

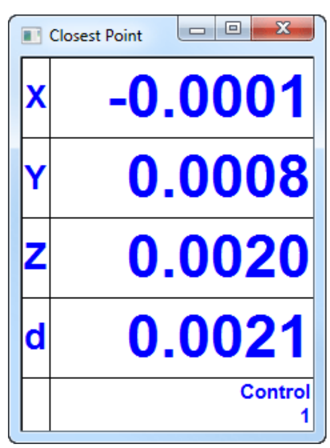


Figure 29-96. The watch window.

Measuring the Room

Now that a control network is established, let's move onto measuring the features of the room. We will use measurement modes that will create geometry as we measure. This improves the work flow and reduces the amount of post analysis. However, feel free to take measurements and then construct the geometry afterwards. Consult the Best Fit Geometry from Points tutorial or Geometry Fitting section for more information regarding Geometry Fitting.

1. First start by measuring the floor. Measure enough points to establish a good plane for the floor. In our case, we measured five points. Select the **Stable Pt. Plane** measurement profile. Open the measurement profile parameters, change the number of points to five, and select the **Show Fit Dialog** and **Send Measured Points** options (Figure 29-97). This way we can review the fit and capture the points that created the plane.

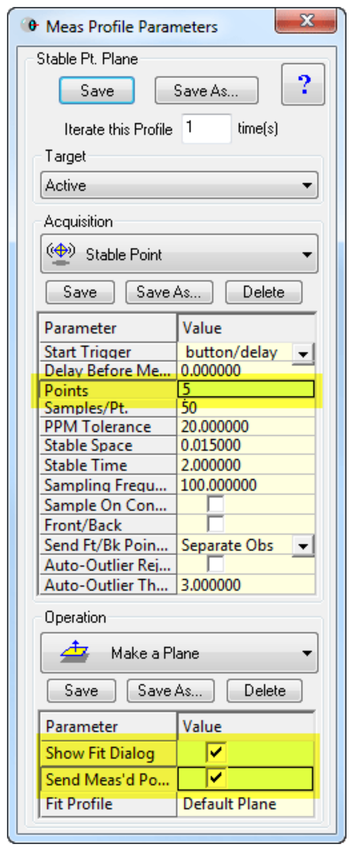


Figure 29-97. Changing the measurement profile.

2. Set the Group name to *Floor* and the Target to *1*.

3. Measure the floor by starting the measurement mode. The stable point acquisition mode will guide you as to when to move and will inform you when measuring. Once the five points are measured, the ***Plane Fit*** dialog will appear (Figure 29-98). Review the results and when satisfied press OK.

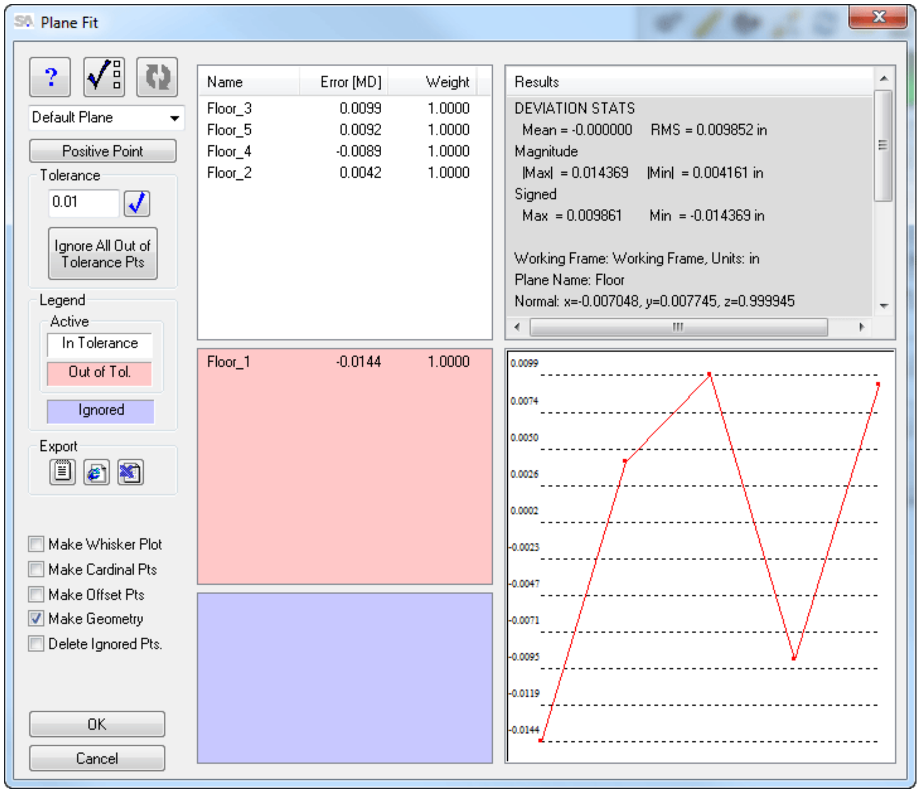


Figure 29-98. The Best Fit dialog.

4. The plane will be created and automatically offset the correct amount and direction. Your SA job should look similar to Figure 29-99.

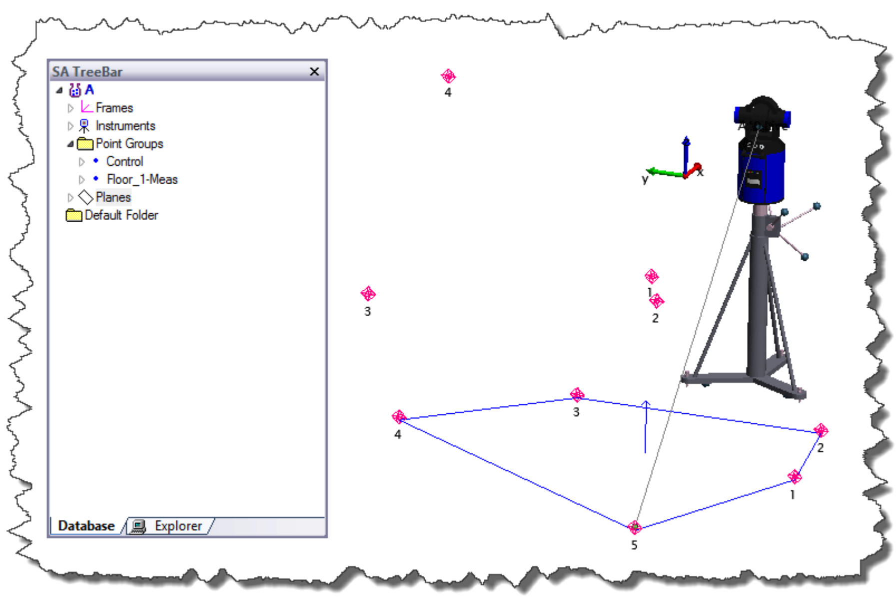


Figure 29-99. After the plane is created.

5. A good rule of thumb is to check instrument/part drift as often as possible. So at this point let's check to see if we have any drift. For these next steps we want the instrument to have drifted so we can practice relocating. So to simulate this, slightly bump or move the instrument.

6. Check the *Control* group by right-clicking the *Control* point group and selecting **Add Watch Window**. Now move the SMR to several of the SMR nests and note the deviation. It should be rather large since you bumped the instrument.

7. Now that we have established the need to relocate the instrument, right-click the instrument in the tree and choose **Jump Instrument to new location**. You will be prompted to choose a location method, select **No Thanks** and press OK. A new instrument has been added to the tree and the existing instrument has been hidden.

8. Set the Group to *Station2* and the Target to *1*. Now measure the control points using the same method as before.

9. With the control points measured, right-click the second instrument in the tree and select **Locate**. In this dialog select **Best-Fit**. You will be prompted to pick the nominal group and the measured group. Select the *Control* point group and the *Station2* point group, respectively.

10. The best-fit interface will appear (Figure 29-100). Press the button to perform the best-fit computation. Once satisfied with the result, press Apply Transformation. The new instrument will now be located with respect to the first instrument.

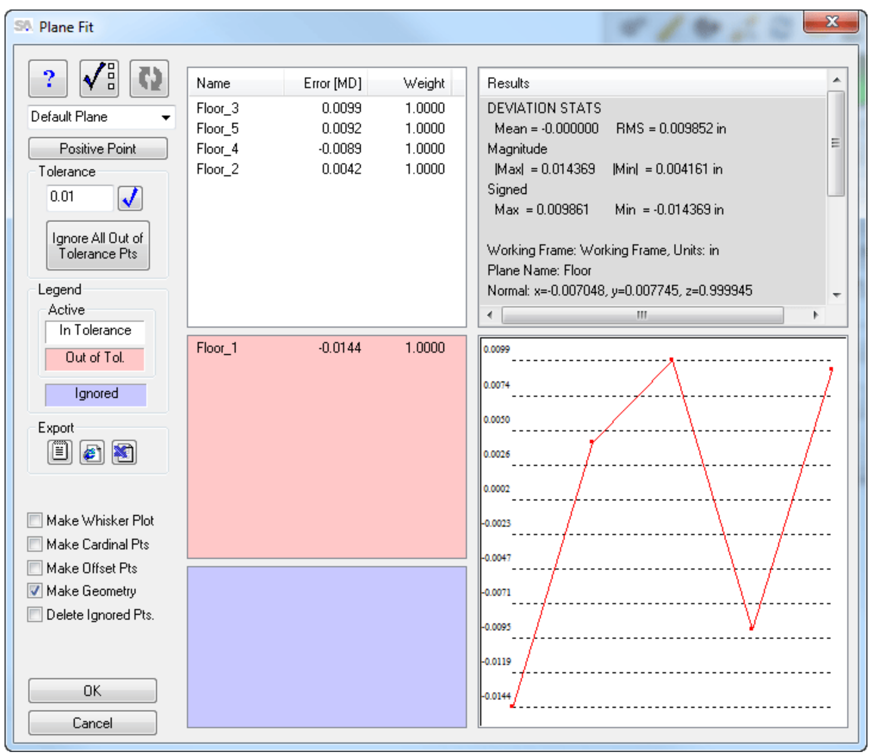


Figure 29-100. The plane fit dialog.

11. Proceed to measure two walls of the room, naming them *Wall1* and *Wall2*. Follow steps 1-3. When finished, your job should look similar to Figure 29-101.

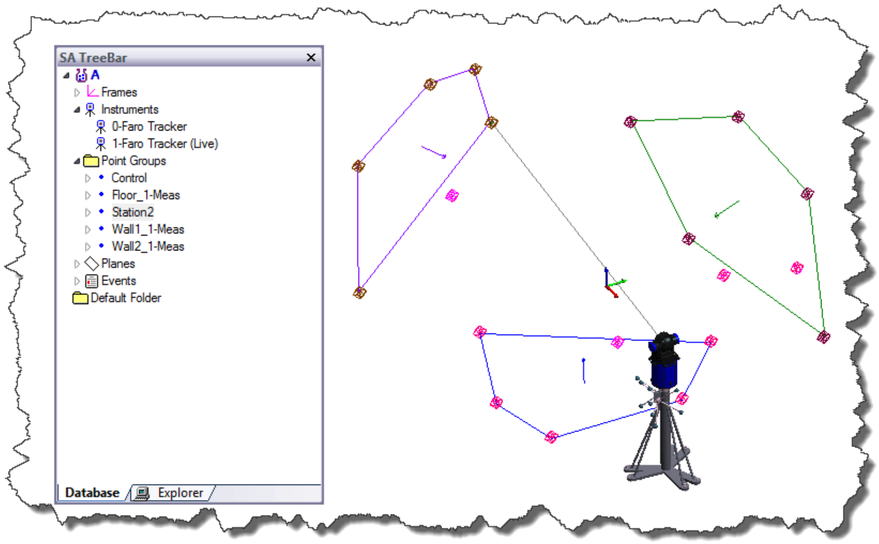


Figure 29-101. All three walls are now measured.

12. With the walls and floor measured, let's scan the top of a desk. Select the **Spatial Scan** measurement profile from the drop-down. Open the parameters and set the *Spatial Increment* to *6* inches. Set the Group name to *Desktop* and the Target to *1*. With the SMR on the surface of the desk, start the measurement mode. By default the measurement mode will stop when the beam is broken.

13. Your SA job should look similar to Figure 29-102.

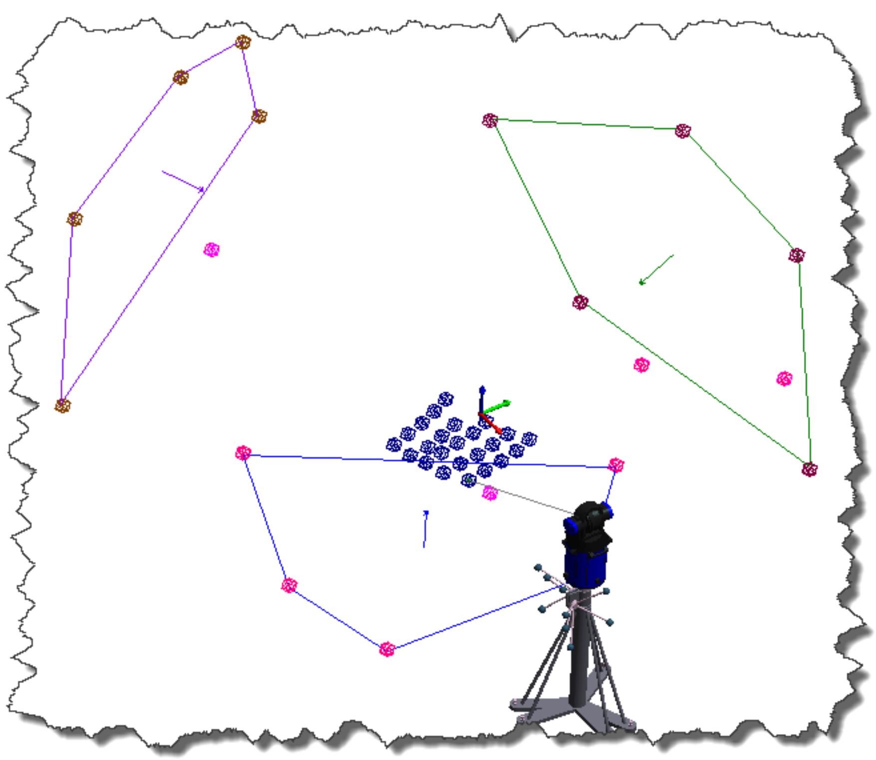


Figure 29-102. After the spatial scan.

Analyzing the data

With the data collected, we can now perform a few basic analysis functions. Let's determine the angle between the walls, the walls and floor and the maximum and minimum distance from the floor to the top of the desk.

1. To calculate the angle between *Wall1* and *Wall2*, use **Query>Plane>to Plane: Angle between normals**. Select the two measured planes. Once completed the results will display and an SA Event will be created.
2. Now do the same for each wall to the floor using the same query command as above.
3. To calculate the maximum and minimum distances from the floor to the top of the desk, use **Query>Points to>Objects**. When prompted, select the *Desktop* points and then select the *Floor_1* plane for the object. A vector group will be generated which will later be used in the SA Report.

Reporting the results

With the analysis complete, we can report the results using the SA Reporting functionality.

1. Create an SA Report by using **Reports>Add SA Report**.
2. Orient the graphics so that a screenshot can be taken and added to the report.
3. Press to take a screenshot image. This image will now be added to the tree. Click and drag the Photo onto the report. Place the image so that it is centered at the top of the report.
4. Now drag the three SA Events that were created for the Plane to Plane angle query. Click and drag each one onto the SA Report.
5. Now drag the vector group onto the SA report. Right-click the Vector Group table in the report and select **Report Options**. Uncheck all the components so that only magnitude is reported.
6. We have now reported our findings. The report will be saved in the tree. If desired, the report can be printed or converted into PDF format. Use the following icons:
7. Once complete the report should look similar to Figure 29-103.

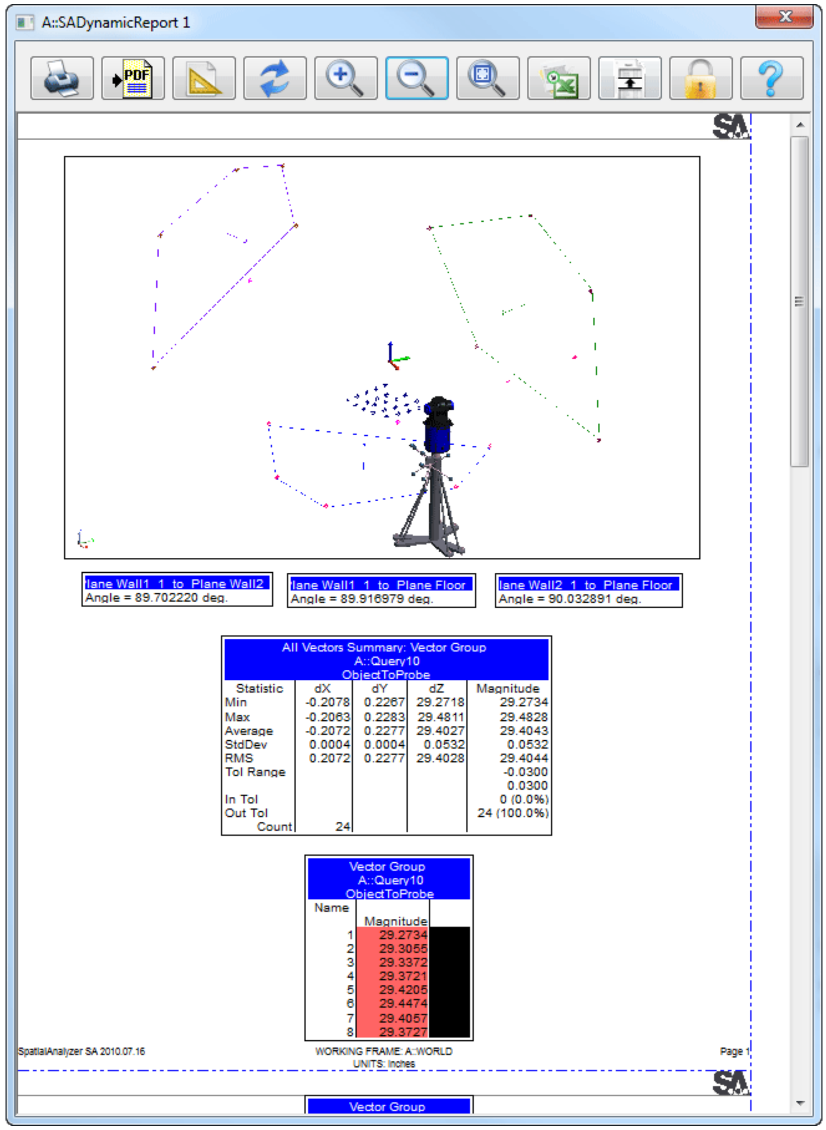


Figure 29-103. The final report.

Basic USMN

- **Skill Level.** Intermediate
- **Description.** In this tutorial, we will cover the basic process for performing a USMN network.
- **Areas Covered.** Unified Spatial Metrology Network.
- **Time to Complete.** Approximately 15 minutes.

USMN Sample File

1. We'll start by opening a tutorial file included with your installation of SA. In SA, choose **Help>Open Sample SA Files**. Choose the `USMN.xit` file. This file contains four instruments with common measurements (Figure 29-104). The goal is to bring all of these instruments into one common network and create a point group that represent the network as a whole. Traditional methods use a best-fit approach, but unfortunately error stack-up affects the instrument positions greatly. USMN uses instrument uncertainty and advanced optimization algorithms to simultaneously solve for all instrument positions, ultimately removing the error stack up and increasing the accuracy of the instrument network. When more than two instrument locations are present, it is recommended to use USMN.

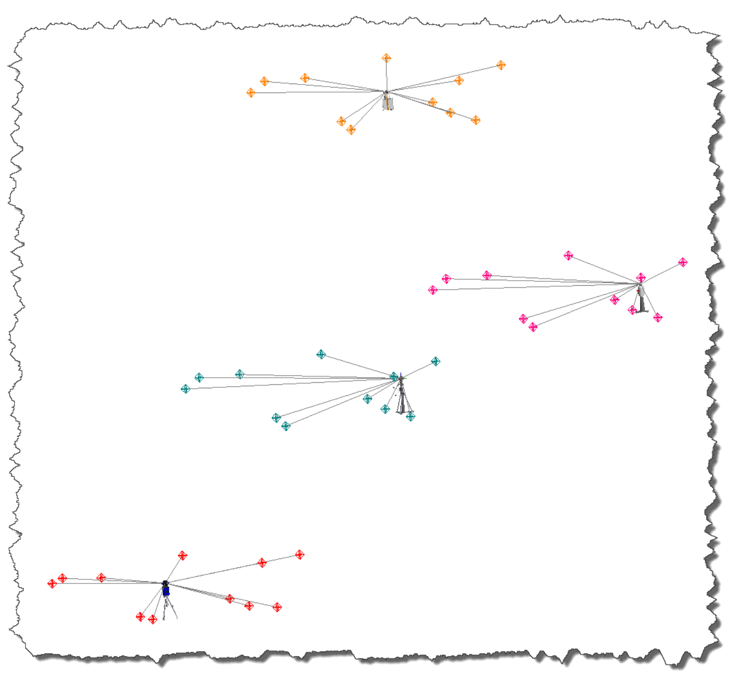


Figure 29-104. Prior to the USMN bundle.

2. To start the USMN process, navigate to **Analysis>Coordinate Uncertainty>Unified Spatial Metrology Network**. You will be prompt-

ed to select the instruments for the USMN network. Select the four instruments in the job. The USMN interface will be displayed (Figure 29-105), and you will have the controls and tools to create and analyze the measurement network. The next few steps will walk through this process.

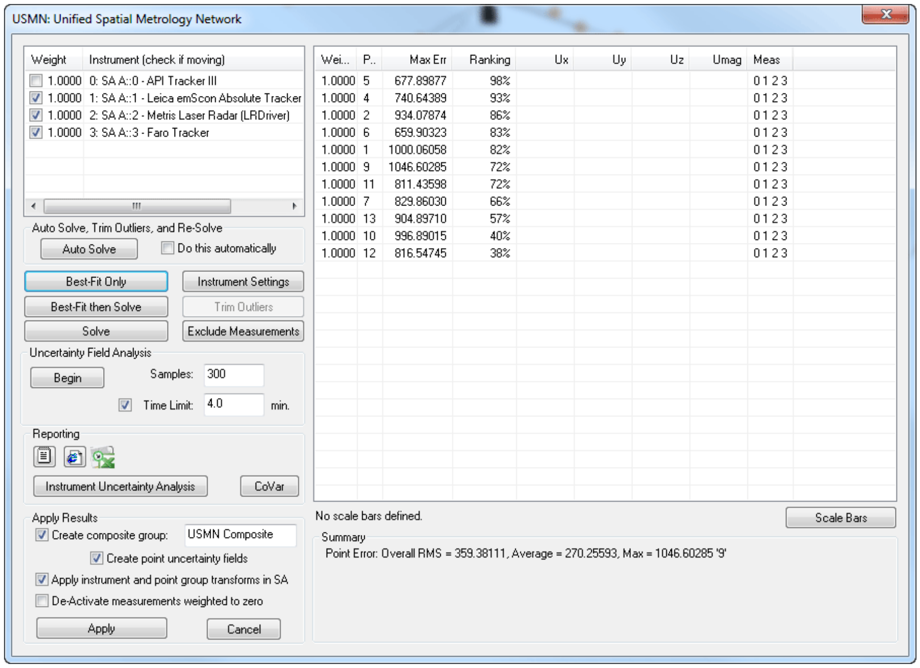
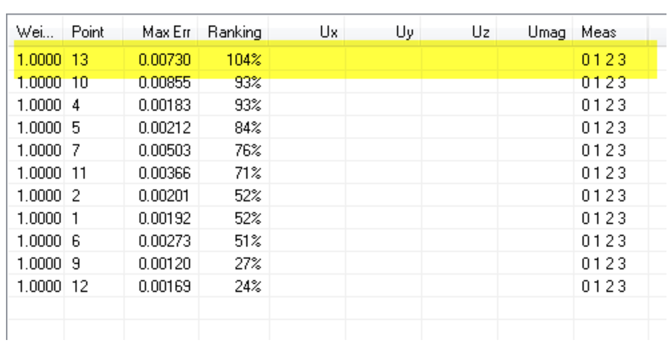


Figure 29-105. The USMN dialog.

3. Press the Best-Fit then Solve button. The instrument stations will be located to one another first by a best-fit. Their positions will then be optimized using an uncertainty model for each instrument.

4. Now note the *Max Error* and *Ranking* of each point (Figure 29-106). The ranking is an indication of the uncertainty of the point error in the network. Typically, points over 100% ranking are removed from the solution.



Wei...	Point	Max Err	Ranking	Ux	Uy	Uz	Umag	Meas
1.0000	13	0.00730	104%					0 1 2 3
1.0000	10	0.00855	93%					0 1 2 3
1.0000	4	0.00183	93%					0 1 2 3
1.0000	5	0.00212	84%					0 1 2 3
1.0000	7	0.00503	76%					0 1 2 3
1.0000	11	0.00366	71%					0 1 2 3
1.0000	2	0.00201	52%					0 1 2 3
1.0000	1	0.00192	52%					0 1 2 3
1.0000	6	0.00273	51%					0 1 2 3
1.0000	9	0.00120	27%					0 1 2 3
1.0000	12	0.00169	24%					0 1 2 3

Figure 29-106. Point details, including Max Error and Ranking.

5. Press the Trim Outliers button to remove points that exceed a specified ranking (Figure 29-107). In our case, we will remove any points over 100%.

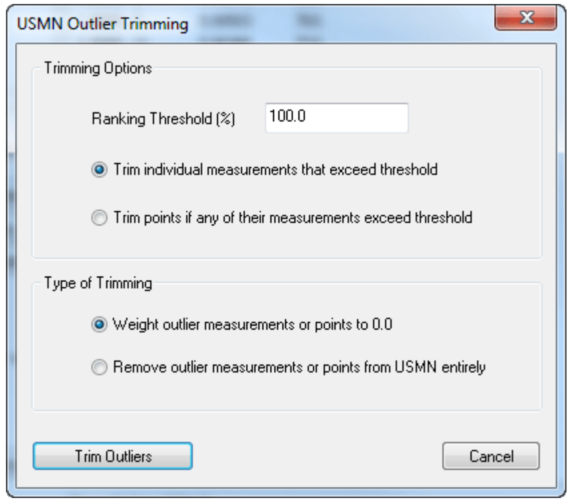


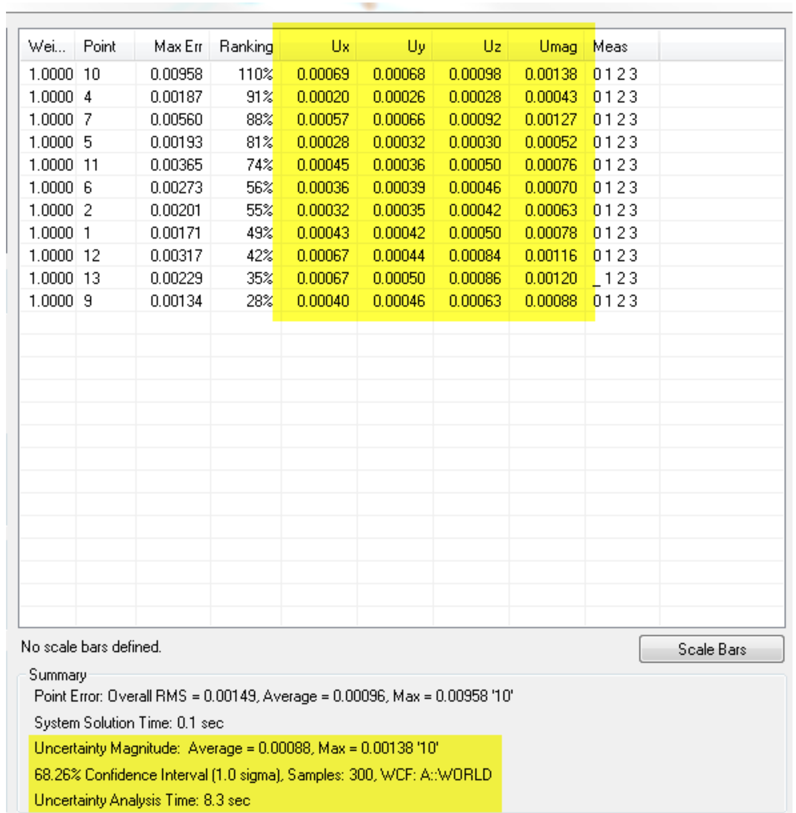
Figure 29-107. Trimming outliers.

6. Now press the Solve button to resolve the network for the remaining points. Now note the *Max error* and *Ranking* values (Figure 29-109). The rule of thumb is to only trim outliers one time, so we will leave point 10 with its new ranking of 110%.

Wei...	Point	Max Err	Ranking	Ux	Uy	Uz	Umag	Meas
1.0000	10	0.00958	110%					0 1 2 3
1.0000	4	0.00187	91%					0 1 2 3
1.0000	7	0.00560	88%					0 1 2 3
1.0000	5	0.00193	81%					0 1 2 3
1.0000	11	0.00365	74%					0 1 2 3
1.0000	6	0.00273	56%					0 1 2 3
1.0000	2	0.00201	55%					0 1 2 3
1.0000	1	0.00171	49%					0 1 2 3
1.0000	12	0.00317	42%					0 1 2 3
1.0000	13	0.00229	35%					_ 1 2 3
1.0000	9	0.00134	28%					0 1 2 3

Figure 29-108. New ranking values after outliers have been trimmed.

7. Once the outliers have been trimmed, compute the uncertainties of the network by pressing the Begin button under the *Uncertainty Field Analysis* section. Once completed the uncertainties can now be reviewed (Figure 29-109).



Wei...	Point	Max Err	Ranking	Ux	Uy	Uz	Umag	Meas
1.0000	10	0.00958	110%	0.00069	0.00068	0.00098	0.00138	0 1 2 3
1.0000	4	0.00187	91%	0.00020	0.00026	0.00028	0.00043	0 1 2 3
1.0000	7	0.00560	88%	0.00057	0.00066	0.00092	0.00127	0 1 2 3
1.0000	5	0.00193	81%	0.00028	0.00032	0.00030	0.00052	0 1 2 3
1.0000	11	0.00365	74%	0.00045	0.00036	0.00050	0.00076	0 1 2 3
1.0000	6	0.00273	56%	0.00036	0.00039	0.00046	0.00070	0 1 2 3
1.0000	2	0.00201	55%	0.00032	0.00035	0.00042	0.00063	0 1 2 3
1.0000	1	0.00171	49%	0.00043	0.00042	0.00050	0.00078	0 1 2 3
1.0000	12	0.00317	42%	0.00067	0.00044	0.00084	0.00116	0 1 2 3
1.0000	13	0.00229	35%	0.00067	0.00050	0.00086	0.00120	_ 1 2 3
1.0000	9	0.00134	28%	0.00040	0.00046	0.00063	0.00088	0 1 2 3

No scale bars defined.

Scale Bars

Summary

Point Error: Overall RMS = 0.00149, Average = 0.00096, Max = 0.00958 '10'

System Solution Time: 0.1 sec

Uncertainty Magnitude: Average = 0.00088, Max = 0.00138 '10'

68.26% Confidence Interval (1.0 sigma), Samples: 300, WCF: A::WORLD

Uncertainty Analysis Time: 8.3 sec

Figure 29-109. Calculating composite point uncertainties.

8. Now apply the results of the USMN. Each of the four instruments should now be aligned (Figure 29-110).

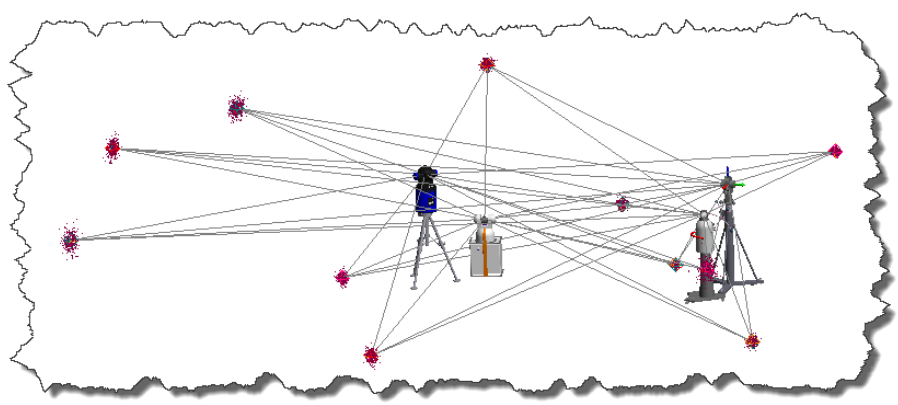


Figure 29-110. After the USMN bundle.

9. A USMN Composite point group will be created. This point group will represent the optimized locations of the common points and represents the instrument network as a whole. Figure 29-111 is a close up look at a composite point versus the measured points.

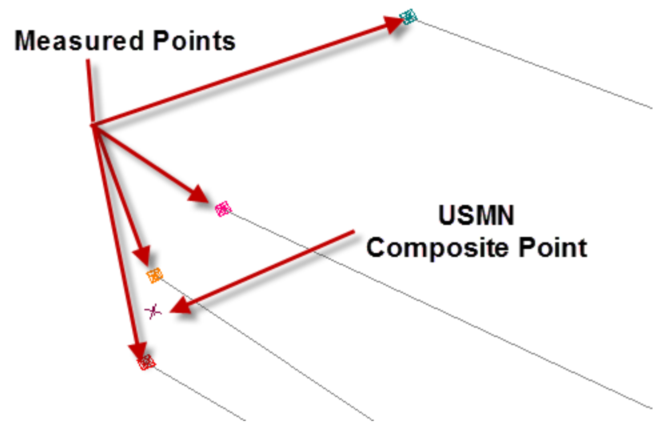


Figure 29-111. The position of the USMN composite point relative to the measured points.

Conclusion

We have learned how to bring multiple instruments into one network easily and with more accuracy than best-fit methods alone. For more information on USMN, see the USMN section.

Running Multiple Laser Trackers

- **Skill Level.** Advanced
- **Description.** In this tutorial, we will cover how to run two laser trackers simultaneously.
- **Areas Covered.** Adding Instruments, Running Instrument Interfaces, IP Addresses, Watch Windows
- **Time to Complete.** Approximately 30 minutes.

Configuration Requirements

When running multiple instruments simultaneously, network configuration must be considered. For this exercise we will use two laser trackers which have different IP addresses. We will configure the PC with dual IP address to accommodate. Please refer to the IP Address Basics section for more information on setting multiple IP Addresses. For this example we used one PC with a network router connecting the two instruments to the PC.

Adding the Instruments

1. Select **Instrument>Add** or the Add Instrument icon . Navigate the instrument list and select your Laser Tracker type, then select Add Instrument at the bottom of the dialog.
2. Repeat the above step and add a second Laser Tracker.
3. Now run the instrument interface by selecting **Instrument>Run Interface Module** and select **Laser Tracker**. The ***SA Network Browser*** will appear (Figure 29-112). Select the first laser tracker under your SA file name and press OK. Depending on your laser tracker type, you might be prompted to enter the respective IP address. If prompted, enter now. Now the instrument interface is up and running for the first instrument.

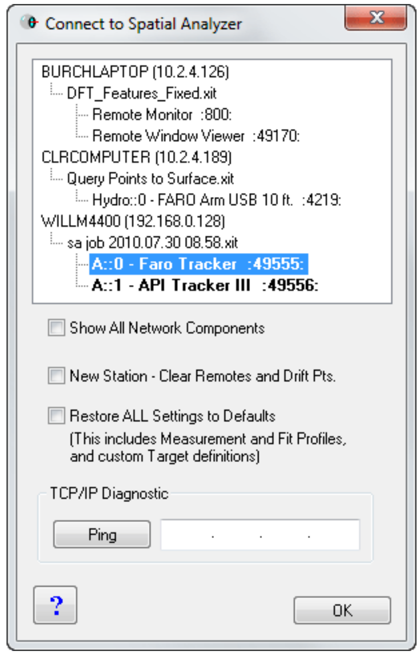


Figure 29-112. Selecting the appropriate instrument to connect to.

4. Repeat the above step for the second laser tracker (Figure 29-113).

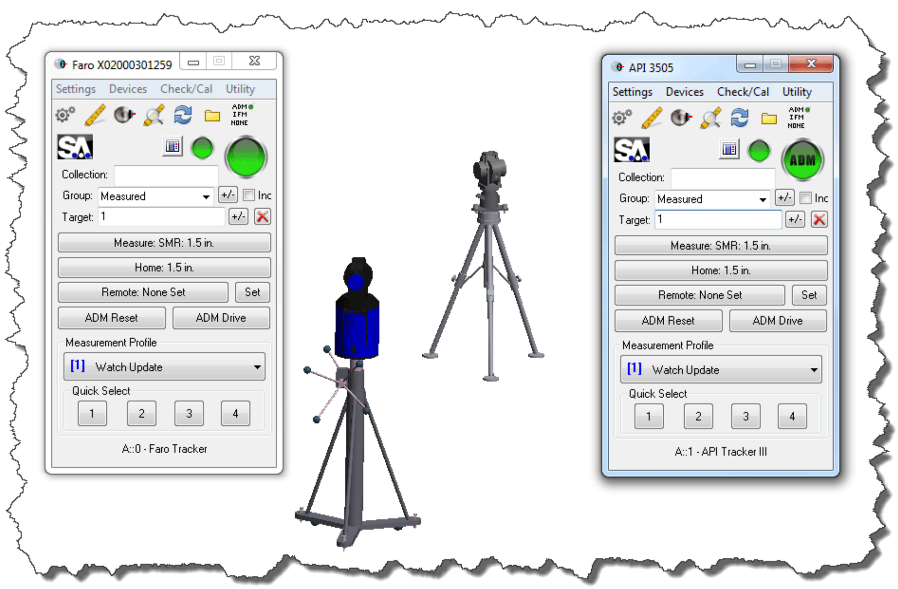


Figure 29-113. Starting a second interface.

Common Network

With both instruments running, we will need to bring the instruments into a common coordinate system.

1. With the first tracker, measure a set of common points that can be seen from both instruments, and name the group accordingly.
2. Repeat this process for the second instrument.

3. Now best-fit the second laser tracker to the first laser tracker. Right-click the second laser tracker in the tree and select **Locate**. Choose **Best-Fit** and choose the first tracker's control points as the Nominal Group and the second tracker's control points as the Measured group. Accept the best-fit and now the two laser trackers are located to one another (Figure 29-114).

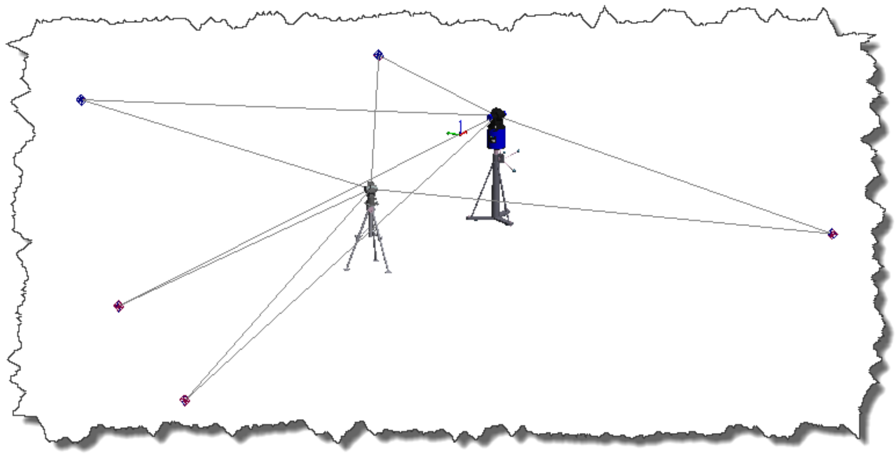


Figure 29-114. Fitting the two instruments together.

Working with both instruments simultaneously

To demonstrate working with multiple instruments simultaneously, let's measure a plane and then watch the deviation of the plane with both laser trackers.

1. Measure points on the floor with one of the laser trackers.
2. Create a plane from the measured points.
3. Now add a watch window using **View>Watch Window>Add Point to>Objects**. Select the first laser tracker and then select the plane. Accept the default projection options.

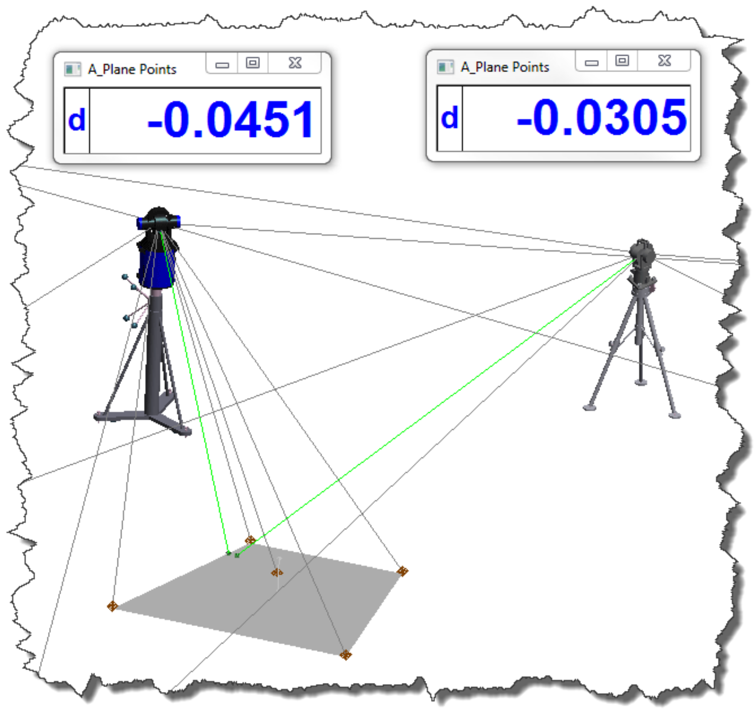


Figure 29-115. Watching two entities simultaneously.

4. Repeat step 3 for the second laser tracker (Figure 29-115).
5. Now we are monitoring the deviation with one single object using two instruments.

The OPC Interface

CHAPTER 30

SA supports the OPC standard, enabling a wide-variety of possibilities for real-time communication between itself and other applications or process control hardware. This chapter discusses the capabilities and use of this interface.

What Is OPC?

OPC is an acronym that originally stood for "OLE for Process Control", but has since been renamed to "Open Platform Communications". It is a standard originally developed in 1996 to enable communication of real-time factory data between various hardware devices and software applications from different manufacturers.

OPC as a communication standard enables various devices in a manufacturing environment to communicate among themselves—for instance, allowing real-time measurement data to be monitored by other hardware, or to track part progress through an assembly line in a real-time database.

SA & OPC

The OPC DA (Data Access) server functionality is available in SA Ultimate (64-bit only). SA's implementation is based on version 2.05 of the OPC DA specification. Any client that complies with this specification can communicate with SA's server—that is, that client has the ability to read from and write to SA's OPC server.

Prerequisites

In order to activate and use the OPC DA server in SA, the following prerequisites must be satisfied:

- The OPC Core Components Redistributable (x64) 3.0 package must be installed on the system,
- 64-bit SA Ultimate must be installed,
- SA's OPC server must be registered and enabled, and
- The OPC server DCOM and network settings must be configured for remote client access.

The OPC Foundation (http://www.opcfoundation.org) provides a compliance test client for the data access 2.05 specifications to all of its members. A sample client which can be used to try out new SA functionality can be downloaded for free from one of the following websites:

- www.softing.com/opc (select OPC Classic options)
- www.matrikonopc.com/products/opc-desktop-tools/opc-explorer.aspx

Configuring the OPC Server

Tip: It is recommended that you create a system restore point prior to changing the system configuration.

Step 1: Creating a DCOM User

Windows 7 Home Premium

Note: For Windows 7 Home Premium, this must be an administrator account.

1. Login to Windows 7 under an administrator account.
2. Under the Windows Control Panel > User Accounts, select Manage another account. Then click Create a new account.
3. Add a new administrative user **OPCUser**, password **nrkopc**.

Windows 7 Professional

1. Under the Windows Control Panel, select Administrative Tools, then select Computer Management.
2. Expand the **Local Users and Groups** item in the tree.
3. Right-click **Users** and select **New User...** from the context menu.
4. Enter **OPCUser** for the *User Name* and **nrkopc** for the password. Enter a description as well (Figure 30-1).
5. Click Create, then Close.

6. Add the new user to the **`Distributed COM Users`** group (Figure 30-2).

Step 2: Disable Windows Firewall

1. Under Windows Control Panel, select Windows Firewall.
2. From the sidebar, select Turn Windows Firewall On or Off.
3. Select the *Turn off Windows Firewall (not recommended)* option.

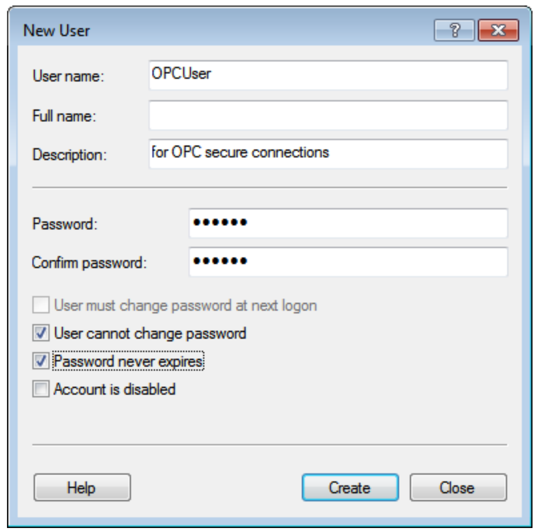


Figure 30-1. Setting up an OPC user account.

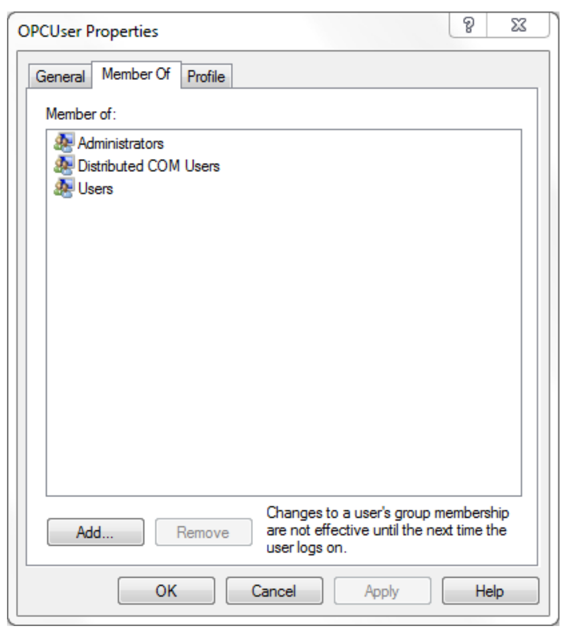


Figure 30-2. Making the new user a member of the **`Distributed COM Users`** group.

Step 3: Adjust User Account Control Settings

1. As administrator, type **`uac`** in the start menu.
2. Set the UAC to *Never Notify*, then click OK.

3. Restart the computer.

Step 4: Install SA

1. Log into the **`OPCUser`** account.
2. Install SA as usual.
3. If SA automatically starts at the end of the install, close it.

Step 5: Install OPC Core Components

1. Run the OPC Server Utility (`C:\Program Files (x86)\New River Kinematics\SpatialAnalyzer <version>\x64\OpcServerUtility.exe`).
2. If the core components are missing on your system, the utility will notify you—continue with the following. Otherwise, skip to step 6.
3. To install the missing components, you download the OPC Core Components Redistributable (x64) package from the OPC Foundation website (https://opcfoundation.org/).
4. After installing the core components, restart your computer.

Step 6: Register the NRK OPC DA Server

1. Login to the **`OPCUser`** account.
2. Ensure SA is not running, then run the OPC Server Utility (see Step 5, item 1).
3. Click the Register SA OPC Server button.

Step 7: Enable the OPC Server in the SA Configuration

1. Run SA Ultimate (x64) or greater.
2. In SA's ***User Options*** dialog, select the *Machine Configuration* tab.
3. Turn on the *Enable OPC Server* option. The server will now run in the future whenever SA is running.

Step 8: Test the Server with a Local Client

1. Install a client onto the machine. For instance:
   - www.softing.com/opc (select OPC Classic options)
   - www.matrikonopc.com/products/opc-desktop-tools/opc-explorer.aspx
2. Start the client application.

3. Verify that the **NRK OPC DA Server** is presented in the list of local v2 data acess servers.

4. Connect to the SA OPC server and browse its tags.

5. Add items to the client's group and verify that data is updating as appropriate.

6. Once proper local operation is verified, close the local OPC client and SA Ultimate before continuing on.

▶ **Note:** At least one tag (**NRK SA Version**) is always available in the OPC server's address space. If you add an instrument to your job file, SA automatically adds the instrument's coordinates to the address space.

Step 9: Configuring System-wide DCOM Settings

1. Ensure you're logged in as **OPCUser**.

2. Run the OPC Server Utility (step 5, item 1).

3. Click the View DCOM Configuration button.

4. Right-click on **My Computer** and select **Properties** from the context menu.

5. In the *Default Properties* tab, ensure that the ***Enable Distributed COM on this computer*** option is enabled. The *Default Authentication Level* should be set to ***Connect***, and the *Default Impersonation Level* should be set to ***Identify*** (Figure 30-3).

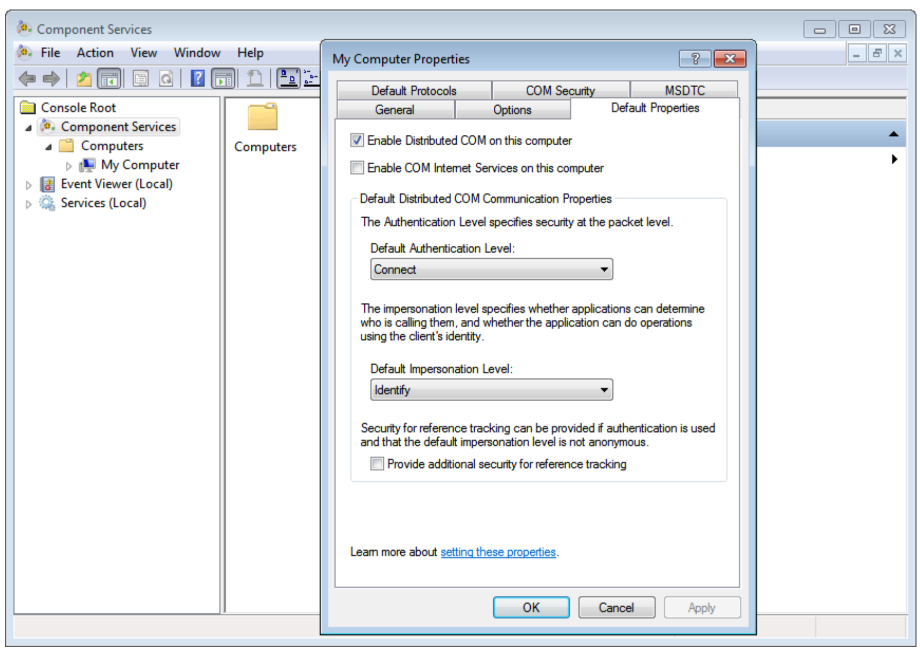


Figure 30-3. Setting the computer's default component services properties.

6. Select the *COM Security* tab. Click the Edit Default... button in the *Access Permissions* group.

7. In the ***Access Permission*** dialog, add permission for **Everyone** and the **ANONYMOUS LOGON** users, then click OK.

8. Click the Edit Limits... button in the *Access Permissions* group.

▶ **Note:** Both of these are required for OPC functionality.

9. In the ***Access Permission*** dialog, add permission for **`Everyone`** and the **`ANONYMOUS LOGON`** users, then click OK.
10. Click the Edit Default... button in the *Launch and Activation Permissions* group.
11. In the ***Launch and Activation Permission*** dialog, add permission for **`Everyone`** and the **`ANONYMOUS LOGON`** users, then click OK.
12. Click OK again to close the ***My Computer Properties*** dialog.

▶ **Note:** Do not adjust system-wide COM security Launch and Activation permissions limits (the Edit Limits... button).

Step 10: Configure the OPC Server's Specific DCOM Settings

1. In the ***Component Services*** window, open the properties of the **`NRK SA OPC DA Server`** (Figure 30-4).

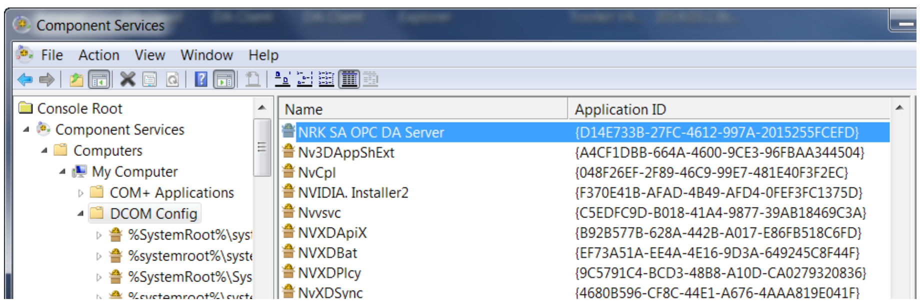


Figure 30-4. Opening the properties for the NRK SA OPC DA Server component.

▶ **Note:** The OPC clients are not allowed to launch SA. For this purpose the *Local Path* parameter is not configured and the *Location* tab's parameters are not set.

2. In the *General* tab, set the *Authentication Level* to ***Default***.
3. Verify that the *Security* tab's *Launch and Activation Permissions* are set as indicated in Figure 30-5 (no launch is allowed).

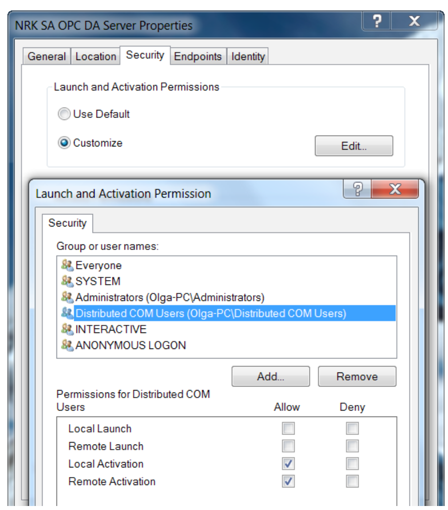


Figure 30-5. Settings for the *Security* tab.

4. Verify that the *Security* tab's *Access Permissions* are set as indicated in Figure 30-6.

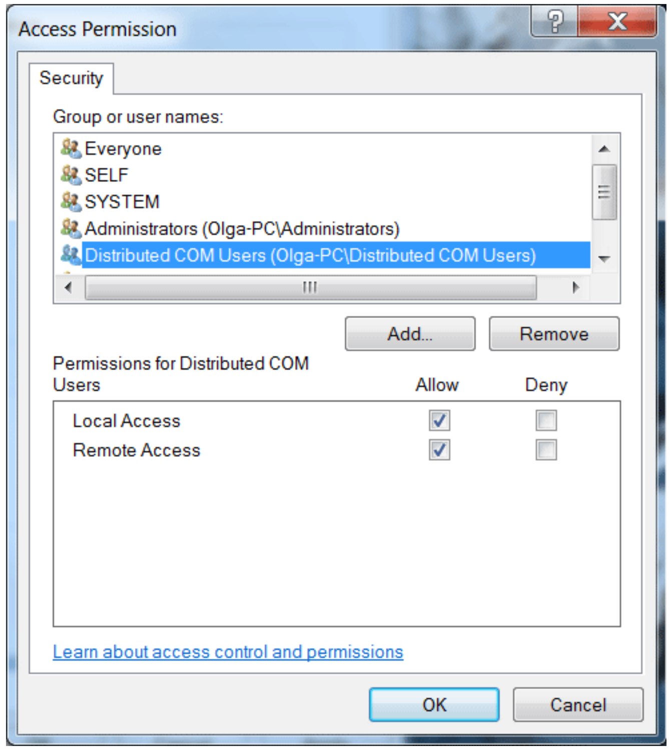


Figure 30-6. The *Security* tab's *Access Permissions*.

5. In the *Endpoints* tab, verify that *DCOM Protocols* and *Endpoints* are set to defaults (the default is TCP/IP).
6. Set the *Identity* to ***Interactive User*** to allow displaying of the SA GUI elements on the server side.

Refer to the following section to configure the client side.

Configuring the OPC Client

Step 1: Create an OPC User Account & Install Components

1. Follow the instructions described in "Step 1: Creating a DCOM User" on page 942.
2. Follow the instructions described in "Step 5: Install OPC Core Components" on page 944.

Step 2: Configure System-Wide DCOM Settings

1. Ensure you're logged in as **`OPCUser`**.
2. Run the OPC Server Utility (step 5, item 1).
3. Click the View DCOM Configuration button.
4. Right-click on **`My Computer`** and select **Properties** from the context menu.

5. In the *Default Properties* tab, ensure that the ***Enable Distributed COM on this computer*** option is enabled. The *Default Authentication Level* should be set to ***Connect***, and the *Default Impersonation Level* should be set to ***Identify*** (Figure 30-3).
6. Select the *COM Security* tab. Click the Edit Limits... button in the *Access Permissions* group.
7. In the ***Access Permission*** dialog, add permission for **Everyone** and the **ANONYMOUS LOGON** users, then click OK.
8. Click the Edit Limits... button in the *Launch and Activation Permissions* group.
9. In the ***Launch and Activation Permission*** dialog, add permissions for **Everyone**, **Distributed COM Users**, and **ANONYMOUS LOGON**, then click OK.

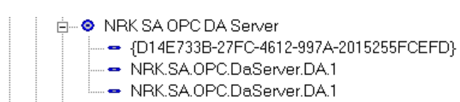
Step 3: Test the OPC Server Connection

1. Restart the computer.
2. Install a client onto the machine. For instance:
   - www.softing.com/opc (select OPC Classic options)
   - www.matrikonopc.com/products/opc-desktop-tools/opc-explorer.aspx
3. Start the client application.
4. Browse the remote OPC server to verify that data is being received.

Using SA's OPC Server

Once the server is up and running, the server's identifications (class ID, description, and program ID) are displayed in the client (Figure 30-7).

Figure 30-7. The SA OPC DA server identifications.



The following parameters are available to the OPC server:

- Active coordinate values for each live instrument as a 3-entry array of doubles (XYZ).
- Active coordinate values for each live instrument as separate values (X, Y, and Z).
- The current SA version.
- The full name of each instrument.

The tag name for each coordinate includes the collection and instrument ID for the associated instrument—for example, **`A::0.x`** (Figure 30-8).

Figure 30-8. Tags for single-coordinate items in the OPC server.

Item	Value	Quality	TimeStamp
A::0.x	-195.745613286698	GOOD	10:18:53.401
A::0.y	9.02854736148172E-14	GOOD	10:18:53.402
A::0.z	-172.072930905314	GOOD	10:18:53.403
NRK SA Version	SA 2014.04.07b (x64)	GOOD	10:04:37.883

Point coordinates are presented with A::0.xyz tag names in the array format (Figure 30-9):

Figure 30-9. A coordinate as an array.

Item	Value	Quality	TimeStamp
A::0.xyz	[0,2] (-195.745613286698, 9.02854736148172E-14, -172.072930905314)	GOOD	10:18:53.404
NRK SA Version	SA 2014.04.07b (x64)	GOOD	10:04:37.883

MP Support

Measurement Plan commands have been added to support interfacing with SA's OPC server—both setting and retrieving values from the server's address space. These commands include:

- Set OPC DA Tag Value Double
- Get OPC DA Tag Value Double
- Set OPC DA Tag Value Integer
- Get OPC DA Tag Value Integer

These commands are detailed in the MP Command Reference. The "set" commands will create a double or integer in the server's address space under the specified tag name. This value can be retrieved from the address space using the corresponding "get" command, or can be updated (overwritten) by specifying another "set" command.

This Page Intentionally Left Blank.

Keyboard Shortcuts

APPENDIX A

The following page is a list of default keyboard shortcuts that ship with SpatialAnalyzer. To become a more efficient user of SA, it is recommended that you try to become familiar with as many of these shortcuts as possible.

KEYBOARD SHORTCUTS

General

Action	Shortcut
Repeat Command	Ctrl + Tab
Command History	Ctrl + Shift + Tab
Graphical Selection Mode	F3 (During Selection)
List Selection	F2
Undo to Restore Point	Ctrl + Z
Create Restore Point	Alt + Space
Pick Working Frame	Alt + W
User Options	Ctrl + U

View

Action	Shortcut
Show Point Labels	Alt + L
Hover	Ctrl + H
Cloud Display Control	Ctrl + T
Refresh	F5

File

Action	Shortcut
Open File	Ctrl + O
Save File	Ctrl + S

Edit

Action	Shortcut
Object Properties	Alt + Q
Copy then Transform	Ctrl + C
Delete Objects	Delete
Delete Points	Alt + D
Add User Note	Ctrl + N

Geometry Fit

Action	Shortcut
Fit Geometry	Ctrl + G
Fit Line	Ctrl + Alt + L
Fit Circle	Ctrl + Alt + C
Fit Plane	Ctrl + Alt + P
Fit Cylinder	Ctrl + Alt + Y
Fit Sphere	Ctrl + Alt + S

Graphical View

Action	Shortcut
Rotate (Orbit)	(Drag)
Zoom In/Out	Pg Up/Pg Dn
	Scroll Up/Down
	Ctrl + + Drag Up/Down
Pan	↑ ↓ ← →
	Shift + (Drag)
Recenter View	(Click to Center)
Zoom to Selection	+ Drag Rectangle
Autoscale	Alt+A
Clipping Planes	Alt+X
Previous View	Ctrl + Alt + ←
Next View	Ctrl + Alt + →

Queries

Action	Shortcut
Point to Point	Ctrl + D
Group to Group	Ctrl + W

Instruments

Action	Shortcut
Add Instrument	Alt + I
Point at a Point	(Double-click on point)

Geometry Construction

Action	Shortcut
Construct Points	Ctrl + P
Layout Random Points	Ctrl + Alt + Z
Patch Normal Shift	Ctrl + M
Construct Line 2 Points	Ctrl + L

Analysis

Action	Shortcut
Best Fit Points to Points	Ctrl + B
USMN	Alt + U

Math Reference

APPENDIX B

Although not at all essential to get benefit out of the software, this section provides background knowledge on some of the math behind SpatialAnalyzer.

Linear Least Squares Adjustment

Least squares is a procedure for adjusting observations containing random errors. It was developed in the latter part of the eighteenth century by Karl Gauss. It was not used much until computers became commonplace because of the lengthy calculations required. SpatialAnalyzer uses it in many routines (transformations, best-fit lines-planes-circles etc), to provide the best and most consistent answer for fitting problems.

For a group of equally weighted observations (or measurements in our case), the basic condition that is enforced in least squares adjustments is that the sum of the squares of the *residuals* is minimized. Residual is another term for the difference between a measured value and its most probable (or "correct") value. For 3D coordinate metrology, one example of a residual would be the difference between the true X-coordinate for a point and the actual X-coordinate measurement.

The method of least squares is a criterion for fitting a specified model to observed data. In this case, the specified model is the reference set of points (or expected values), and the observed data are typically your measurements of the reference set of points. Let's use a common use of the least squares adjustment as an example. The least squares line routine is a commonly used method for computing the best-fit line from a series of measurements of a linear function. The distance your car travels when you're driving at a particular speed is an example of a linear function. In this example, you are trying to drive your car at a constant speed and measure how far you have gone and the time it took you to get there. While driving, it is likely that a little error is going to creep into the process—you're not going to be able to . That error is generally referred to as random error. Least squares adjustment techniques are really good at dealing with this type of error and are able to produce a reasonable and consistent answer in spite of it. An example is shown in the figures. The time is plotted on the X-axis and the distance on the Y-axis.

The points are the paired measurements made of time verses the distance traveled, which is linear in this case because you were trying to maintain a constant speed. To find what is considered the "best line" through the data the least squares line routine is used. In this case, the residual that gets minimized is the difference (squared) from a measured point to the line that best describes the distance you traveled verses the time it took. The least squares line routine will find a line, such that the sum of the distances (squared) from the data points to the line is the minimized. The figure shows a line that best fits those points; it was solved for with the best-fit line routine in SpatialAnalyzer.

The slope of the best-fit line is actually the average speed that you traveled on your trip. This line is the most consistent way of determining where you probably where at any point in time during your trip.

The least square line adjustment was able to use this set of measurements even through each point had some error, and it produced a high-quality and reliable answer. This ability of dealing with measurements that have at least some component of random error and still being able to produce consistent and representative answers is why the technique is used in SpatialAnalyzer.

Basics of Spatial Transformations

Homogeneous transformations—transforms for short—can be used to represent the position and orientation of one coordinate frame (system) with respect to another coordinate frame. A general transformation matrix can be conveniently represented as a four by four matrix; however, only twelve of the sixteen numbers are typically used in this application[1]. This representation is unique—i.e., there is only one transformation matrix that unambiguously describes a particular relative position and orientation. The problem with representing all transforms as a matrix is that this representation is overconstrained. We know, for example, that a rigid object in space has only six degrees-of-freedom—three positional and three rotational. You simply don't need all twelve numbers in the 4x4 matrix to represent the relative position and orientation. Let's look at other more compact representations.

First of all, we need to define which coordinate frame is described relative to the other. Let's assume we have frames **A** and **B**, and that we wish to describe frame **B** relative to **A** (Figure B-1).

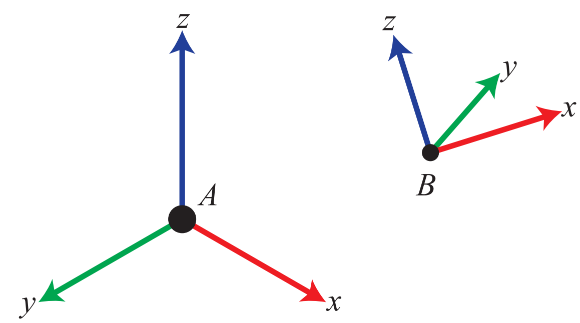


Figure B-1. Two coordinate frames in space.

Position is easy—we simply describe the x, y, and z components of the origin of **B**. Note these terms are the distances along the x, y, and z axes of frame **A**. Also note the position of **A** relative to **B** would not, in general, be the negative of these values. (This would only be the case if the corresponding axes for the two frames were all parallel). Rotation is not as easy as position. To describe rotations, certain standard conventions have been adopted. These are by no means the only possible methods. The following sections will discuss different viewpoints that are commonly used to discuss transforms, then we will discuss simple unit rotations, and go on to develop compound rotation methods.

The old viewpoint debate: Are we describing or transforming?

This section only becomes necessary because of the language that individuals use to discuss transforms. This is one of the most common sources of errors in dealing with spatial transformations. Transforms can be used in two different ways:

- **As a descriptor.** This is the way we will always talk about transforms in SpatialAnalyzer. This is the description of one frame

1 The four-by-four matrix is typically used because it is symmetric. Numerical routines can generally be made more efficient when the numerical pieces are symmetric.

relative to another. For example, frame **B** relative to frame **A**.

- **As a transform operator.** In this case we ask the question: what operations we do to frame **A** to rotate and translate it to the position and orientation of **B**? In some cases, it's natural to think about transforms in this way. For example, if you want to describe the position of a box as it moves, it's natural to think about the rotations and translations that it is subjected to over time.

The final result is a little counter-intuitive—but essentially, it doesn't matter which view point you use—they both produce the exact same results. You just need to be aware of the fact that despite the language used, the math is the same. Many people incorrectly interpret one as the inverse of the other.

Unit Rotation Matrices

Before we can discuss compound rotations, we need to develop the building blocks of unit rotations. These are simply rotations that occur about any given principal coordinate axis (X, Y, or Z). They are as follows:

$$R_x(\theta) = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta \\ 0 & \sin\theta & \cos\theta \end{bmatrix} \quad R_y(\theta) = \begin{bmatrix} \cos\theta & 0 & \sin\theta \\ 0 & 1 & 0 \\ -\sin\theta & 0 & \cos\theta \end{bmatrix} \quad R_z(\theta) = \begin{bmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Or, for the particular example of an X rotation of 30°, a Y rotation of 60°, and a Z rotation of 90°:

$$Rx(30^\circ) = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0.866 & -0.500 \\ 0 & 0.500 & 0.866 \end{bmatrix} \quad Ry(60^\circ) = \begin{bmatrix} 0.500 & 0 & 0.866 \\ 0 & 1 & 0 \\ -0.866 & 0 & 0.500 \end{bmatrix} \quad Rz(90^\circ) = \begin{bmatrix} 0 & -1 & 0 \\ 1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Compound Rotations

Using the building blocks of the above unit rotations, we can now proceed to develop more complex compound rotations. As we stated earlier, a rotation matrix is unique. However, the descriptive methods we often use to describe a rotation are not. The same physical orientation can be described in an infinite number of ways. We'll explore the most common below.

Fixed Angle XYZ

In this case, we will start with frame **A** and **B** coincident. We will then rotate frame **B** about the X-axis of **A** (X_A), then about the Y-axis of **A** (Y_A), then about the Z-axis of **A** (Z_A). We can express this as:

$$R_{xyz}(30^\circ, 60^\circ, 90^\circ) = R_z(90^\circ)R_y(60^\circ)R_x(30^\circ) = \begin{bmatrix} 0.000 & -0.866 & 0.500 \\ 0.500 & 0.433 & 0.750 \\ -0.866 & 0.250 & 0.433 \end{bmatrix}$$

Fixed Angle ZYX

In this case, we again start with frame **A** and **B** coincident. We then rotate frame **B** about the Z-axis of **A** (Z_A), then about the Y-axis of **A** (Y_A), then about the X-axis of **A** (X_A). We can express this as:

$$R_{zyx}(90°, 60°, 30°) = R_x(30°)R_y(60°)R_z(90°) = \begin{bmatrix} 0.000 & -0.500 & 0.866 \\ 0.866 & -0.433 & -0.250 \\ 0.500 & 0.750 & 0.433 \end{bmatrix}$$

Euler Angle XYZ

In this case, we will start with frame **A** and **B** coincident. We will then rotate frame **B** about the X-axis of **B** (X_B), then about the Y-axis of **B** (Y_B), then about the Z-axis of **B** (Z_B). Note that the axes of frame **B** rotate during the operations. We can express this as:

$$R_{x'y'z'}(30°, 60°, 90°) = R_x(30°)R_y(60°)R_z(90°) = \begin{bmatrix} 0.000 & -0.500 & 0.866 \\ 0.866 & -0.433 & -0.250 \\ 0.500 & 0.750 & 0.433 \end{bmatrix}$$

Euler Angle ZYX

In this case, we will start with frame **A** and **B** coincident. We will then rotate frame **B** about the Z-axis of **B** (Z_B), then about the Y-axis of **B** (Y_B), then about the X-axis of **B** (X_B). Note that the axes of frame **B** rotate during the operations. We can express this as:

$$R_{z'y'x'}(90°, 60°, 30°) = R_z(90°)R_y(60°)R_x(30°) = \begin{bmatrix} 0.000 & -0.866 & 0.500 \\ 0.500 & 0.433 & 0.750 \\ -0.866 & 0.250 & 0.433 \end{bmatrix}$$

Euler Angle ZYZ

In this case, we will start with frame **A** and **B** coincident. We will then rotate frame **B** about the Z-axis of **B** (Z_B), then about the Y-axis of **B** (Y_B), then again about the Z-axis of **B** (Z_B). Note that the axes of frame **B** rotate during the operations. We can express this as:

$$R_{z'y'z'}(90°, 60°, 90°) = R_z(90°)R_y(60°)R_z(90°) = \begin{bmatrix} -1 & 0 & 0 \\ 0 & -0.500 & 0.866 \\ 0 & 0.866 & 0.500 \end{bmatrix}$$

Equivalent Angle Axis

This representation is a little different from the others given in that it does not result from the combination of unit rotations. This convention stems from the fact that for any given rotation there exists a unique axis of rotation and magnitude of rotation. If we rotate about a unit vector K and amount θ we find that the following matrix results:

$$R(K, \theta) = \begin{bmatrix} K_xK_xv_\theta + \cos\theta & K_xK_yv_\theta - K_z\sin\theta & K_xK_zv_\theta + K_y\sin\theta \\ K_xK_yv_\theta + K_z\sin\theta & K_yK_yv_\theta + \cos\theta & K_yK_zv_\theta - K_x\sin\theta \\ K_xK_zv_\theta - K_y\sin\theta & K_zK_yv_\theta + K_x\sin\theta & K_zK_zv_\theta + \cos\theta \end{bmatrix}$$

where $v_\theta = (1 - \cos\theta)$ and K_x, K_y, K_z are the components of unit vector K. Through an inverse analysis we can extract the (K, θ) for any rotation matrix.

Quaternions (also known as Euler Parameters)

A Quaternion is simply another way of expressing the information that is inherent in the equivalent angle-axis representation. Namely,

$$\varepsilon_1 = K_x \sin\left(\frac{\theta}{2}\right) \qquad \varepsilon_2 = K_y \sin\left(\frac{\theta}{2}\right) \qquad \varepsilon_3 = K_z \sin\left(\frac{\theta}{2}\right) \qquad \varepsilon_4 = \cos\left(\frac{\theta}{2}\right)$$

End User License Agreement

APPENDIX C

SOFTWARE LICENSE AGREEMENT OF NEW RIVER KINEMATICS, INC.
("MANUFACTURER")
NOTICE-- READ BEFORE INSTALLING THIS PACKAGE

CAREFULLY READ THE TERMS AND CONDITIONS OF THIS AGREEMENT BEFORE INSTALLING THIS PACKAGE. INSTALLING THIS PACKAGE INDICATES YOUR ACCEPTANCE OF THESE TERMS AND CONDITIONS. IF YOU DO NOT AGREE WITH THE TERMS AND CONDITIONS OF THIS AGREEMENT, PROMPTLY RETURN THIS PACKAGE TO THE PLACE OF PURCHASE FOR REFUND IN THE AMOUNT YOU PAID.

THIS AGREEMENT, entered into by and between NEW RIVER KINEMATICS, INC. ("Licensor"), a Virginia corporation having principal offices at 436 McLaws Circle, Williamsburg, Virginia 23185, and the undersigned customer ("Licensee") determines, the rights and licenses granted to Licensee in the Licensed Software (hereinafter defined) supplied by the Licensor.

Definitions

As used in this agreement, the following definitions, not in alphabetical order, apply:

- **Activated Software.** Software for which Licensee has paid applicable license fees and charges and for which Licensor has delivered a Security Device.
- **Effective Date.** Date of execution of this Agreement by Licensee or in the absence of any other ascertainable date or, at the option of Licensor, means the date of installation of the Software by Licensee.
- **Licensed Software or Software.** Licensor's software, selected by Licensee, in executable code only, as licensed under this Agreement.
- **Licensed Documentation.** means all the documentation, in printed or electronic form, that is delivered from Licensor to Licensee and related to the Licensed Software.

- **Licensed Product.** Licensed Software and the Licensed Documentation, collectively, including the Security Device when provided by Licensor with Activated Software.
- **Maintenance Fees.** Fees payable for Software Maintenance of Activated Software.
- **Security Device.** Device issued by Licensor and licensed to Licensee to enable Licensee's Use of Activated Software to achieve functionality for which Licensee has paid Licensor applicable license fees and charges.
- **Software Maintenance.** Corrections, revisions, patches, service packs, updates, and subsequent releases of the Licensed Product made available by Licensor from time to time.
- **Unactivated Software.** Software for which no license fees and charges have been paid by Licensee and for which no Security Device has been issued, thus limiting authorized use of the Software to Viewer Mode.
- **Use.** Licensee's right to install, use, access, run, or otherwise read the Software into and out of memory in strict accordance with the Licensed Documentation and all the terms of this license granted from Licensor to Licensee pursuant to this Agreement.
- **Viewer Mode.** Use of Software only for purposes of evaluating the Software or viewing a file or other work product not created by the Unactivated Software.

License Grant

Subject to the payment of the license fees and charges to Licensor, Licensor grants to Licensee a limited, nonexclusive and nontransferable license for Use of the Licensed Product during the Agreement's term defined in Section 4. Licensee agrees that it has no right, power or authority to make any modifications to or unauthorized copies of the Licensed Product.

At Licensor's request upon the expiration or other termination of Licensee's authorized Use of the Software, Licensee will return to Licensor all Licensed Documentation, including the Security Device.

License Fees, Charges and Taxes

Licensee shall pay Licensor's license fees and charges for the Licensed Product in effect at the time of Licensee's acceptance of this Agreement. No license fees and charges apply when the Licensed Software is Unactivated Software.

The license fees and charges, taxes and other applicable charges are due and payable upon installation of the Activated Software unless separate arrangements for deferred payment are made with Licensor prior to installation.

Term

Unless otherwise terminated or cancelled as provided by this Agreement, the term of Licensee's Use of Activated Software shall commence upon first Use of the Software and shall continue for five (5) years from the most recent version of Activated Software delivered to Licensee. Licensee acknowledges that if Licensee at any time fails to extend the term of Software Maintenance by payment of applicable Maintenance Fees, no versions of Activated Software will be obtained following the expiration of the latest period for which Software Maintenance has been purchased. Notwithstanding the foregoing, Licensor will have the right prior to the end of any term for which Software Maintenance has been purchased to notify Licensee that Licensor will not provide Software Maintenance after that term, and in that event, the term of Licensee's Use of Activated Software will extend five years from the date of the latest version of Activated Software delivered to Licensee prior to expiration of the then term of Software Maintenance.

Unless otherwise terminated or cancelled, the Term of Licensee's Use of Unactivated Software is five (5) years from the date of first Use of Unactivated Software.

Protection of Licensed Product

Licensee acknowledges and agrees that the Licensed Product and all permitted copies are Licensor's exclusive property and a valuable trade secret of Licensor. Licensee will not disclose or make available to third parties the Licensed Product or any portion or any information from Licensor marked ''confidential'' without Licensor's prior written approval. Licensor reserves title to, ownership of, and all proprietary rights to the Licensed Product, as well as any related work product and major or minor releases of the Software, if any. Licensee shall promptly and in good faith take all action reasonably necessary, advisable, or requested by Licensor to assure compliance with this Section 5 by all employees and agents of Licensee.

Upon reasonable notice during the Agreement's term and within six (6) months after termination or expiration of the Agreement, Licensor may arrange for the inspection and review of Licensee's computer systems to assure compliance with all terms and provisions of this Agreement during Licensee's normal business hours and in such a manner as not to interfere unreasonably with Licensee's operations. In the event such audit discloses that the number of permitted copies has been exceeded, Licensee shall promptly pay to Licensor the appropriate license fees and charges for the additional computers or users.

Prohibited Uses

Licensee shall not:

- At any one time, use, share, or use concurrently more than one installation or copy of the Software on any one computer, including but not limited to a workstation, terminal, or other digital electronic device, except that Unactivated Software is not subject to such restrictions;
- Modify, translate, reverse engineer, decompile, disassemble, reduce the Software to a humanly perceivable form, or create derivative works based upon the Licensed Product, or cause or permit another to do so;
- Remove any proprietary notices, labels, copyright marks, or trademarks on the Licensed Product;
- Modify, adapt, rent, lease, commercially host, sublicense, redistribute, or lend the Software;
- Copy or attempt to copy the Software for any purpose whatsoever, except as expressly permitted by the terms of this Agreement.
- Attempt to access or use Activated Software, except by means of the Security Device licensed by Licensor to Licensee.
- Attempt to alter, modify, reprogram or otherwise tamper with any Security Device issued by Licensor, regardless of whether the Security Device has been licensed to Licensee.

Support Services and Updates

For Activated Software Licensor will provide Licensee with all Software Maintenance made available by Licensor for the Software during the period of time for which Licensee has purchased and paid for Software Maintenance. Initially, Licensor will provide Licensee Software Maintenance through the expiration date of the Security Device first provided by Licensor to enable functionality of the Activated Software. Subject to the provisions of Section 4, Licensee's right to receive Software Maintenance can be extended for succeeding periods of time upon payment of then applicable Maintenance Fees. If Licensee allows Software Maintenance to lapse, Licensor may, but need not, reinstate and continue Software Maintenance upon Licensee's payment of any and all Maintenance Fees for the period or periods of time during which Software Maintenance lapsed, plus a reinstatement fee of 25% of such previously unpaid Maintenance Fees.

As part of Software Maintenance purchased by Licensee, Licensor shall provide support services to attempt to correct any error, malfunction or defect in the Activated Software's functionality or provide an alternative process, provided that Licensee gives Licensor prompt notice of the problem and sufficient information to either diagnose or recreate such problems. Licensee acknowledges that the Licensed Software is complex and may not be error free and that all errors, if any, may not be correctable or avoidable.

Warranty

Licensor warrants that, for a period of sixty (60) days after delivery the Licensed Product to Licensee:

- Under normal Use and service, the media on which the Licensed Software is delivered (if so delivered otherwise than electronically) shall be free from defects in material and workmanship, and
- The Licensed Product will meet Licensor's then current published specifications for the Software.

If If the Licensed Product fails to meet the media warranty of Paragraph 9(a)(1) and Licensee gives Licensor written notice thereof during the applicable warranty period, Licensor shall replace such media. If the Licensed Product fails to meet the warranty of Paragraph 9(a)(2) and Licensee gives Licensor written notice thereof during the applicable warranty period, Licensor's sole obligation shall be to provide technical services to attempt to correct the failure, provided that Licensee gives Licensor detailed information regarding the failure and Licensor is able to duplicate or view the same. Licensee acknowledges that the Licensed Product is complex, may not be error free, and that all errors, if any, may not be correctable or avoidable.

EXCEPT AND TO THE EXTENT EXPRESSLY PROVIDED IN PARAGRAPH 9 AND IN LIEU OF ALL OTHER WARRANTIES, THERE ARE NO WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, ANY WARRANTIES OF MERCHANTABILITY, INFORMATIONAL CONTENT, FITNESS FOR A PARTICULAR PURPOSE OR SYSTEM INTEGRATION, OR NON-INFRINGEMENT OF THIRD PARTY RIGHTS RELATING TO THE LICENSED PRODUCT. Some jurisdictions do not permit the exclusion of implied warranties or limitations on applicable statutory rights of the consumer, and, as such, some portion of the above limitation may not apply to Licensee. In such jurisdictions, Licensor's liability is limited to the greatest extent permitted by law.

Licensee shall be solely responsible for the selection, Use, efficiency, and suitability of the Licensed Product and Licensor shall have no liability thereof.

The warranty provision of this Paragraph 9 do not apply if the Software has been subject to modification by other than Licensor or on Licensor's instruction, has not been installed, operated, repaired, or maintained in accordance with instructions supplied by Licensor, is subject to accident or abuse, or is used in ultra hazardous activities.

Licensor warrants that the Software is date compliant. The duration of this warranty and remedies available to the Licensee for breach of this warranty shall be limited to repair or replacement of the Software where such non-compliance is discovered and made known to Licensor in writing within ninety (90) days after acceptance. Nothing in this warranty shall be construed to limit any rights or remedies Licensee may otherwise have under this Agreement with respect to other defects.

Despite any other provision hereof to the contrary, Licensor's liability of any kind shall never exceed the amount of the license fees, charges and taxes described in Section 3, and the Licensor's liability expressly excludes incidental or consequential damages to Licensee, including lost profits, even if Licensor is on notice of, and has express knowledge of, the potential loss or damage.

Despite any other provision hereof to the contrary "UNACTIVATED SOFTWARE", AS HEREIN DEFINED, IS SUBJECT TO NO WARRANTY, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, ANY WARRANTIES OF MERCHANTABILITY, INFORMATIONAL CONTENT, FITNESS FOR A PARTICULAR PURPOSE OR SYSTEM INTEGRATION, OR NON-INFRINGEMENT OF THIRD PARTY RIGHTS RELATING TO THE LICENSED PRODUCT.

Negation of Proprietary Rights Indemnity

THERE IS NO WARRANTY AGAINST INTERFERENCE WITH LICENSEE'S QUIET ENJOYMENT OF THE LICENSED PRODUCT OR AGAINST INFRINGEMENT, AND LICENSOR HAS NO LIABILITY TO LICENSEE FOR THE INFRINGEMENT OF PROPRIETARY RIGHTS BY THE LICENSED PRODUCT OR ANY PORTION THEREOF.

Termination/Cancellation

Licensor has the right to terminate or cancel this Agreement in accordance with other provisions of this Agreement and also if:

- Licensee fails to pay Licensor any license fee or charges when due; or
- Licensee is in default of any other provision hereof and such default has not been cured within ten (10) days after Licensor learns of the default and gives Licensee written notice thereof; or
- Licensee becomes insolvent or seeks protection, voluntarily or involuntarily, under any bankruptcy law; or
- Licensee attempts to assign or sublicense its rights under this license.

In the event of any termination or cancellation, Licensor may:

- Declare all amounts owed hereunder to Licensor to be immediately due and payable;
- Require that Licensee cease any further use of the Licensed Product or any portion thereof and immediately destroy, erase from any temporary RAM and permanent memory, and return to Licensor all copies of the Licensed Product, including the Security Device; Upon Licensor's request, an authorized representative of Licensee shall provide a written certification that warrants compliance with this requirement.
- Cease performance of all of Licensor's obligations hereunder without liability to Licensee; and
- Electronically render the Software of no further Use by Licensee.

UPON ANY TERMINATION, EXPIRATION, OR CANCELLATION OF THIS AGREEMENT, DUE TO BREACH OR OTHERWISE, LICENSOR SHALL IN ALL EVENTS HAVE ALL RIGHTS, POWER AND AUTHORITY TO EXERCISE ELECTRONIC SELF-HELP AND TO EXERCISE ALL RIGHTS, REMEDIES AND RECOURSE TO WHICH IT MAY BE ENTITLED, WHICH SELF-HELP

SHALL INCLUDE WITHOUT LIMITATION, ALL ELECTRONIC MEANS AVAILABLE TO LICENSOR.

Limitation of Liability

IN NO EVENT SHALL LICENSOR BE LIABLE TO LICENSEE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES OF ANY KIND, REGARDLESS OF THE FORM OF THE ACTION, ARISING OUT OF OR RELATED TO THIS LICENSE AGREEMENT OR THE PERFORMANCE OR BREACH THEREOF, EVEN IF LICENSOR HAS BEEN ADVISED OF THE POSSIBILITY THEREOF. LICENSOR'S LIABILITY TO LICENSEE HEREUNDER, IF ANY, SHALL IN NO EVENT EXCEED THE TOTAL OF THE LICENSE FEES PAID TO LICENSOR HEREUNDER BY LICENSEE. IN NO EVENT SHALL LICENSOR BE LIABLE TO LICENSEE FOR ANY DAMAGES RESULTING FROM OR RELATED TO ANY FAILURE OF THE SOFTWARE PRODUCTS, INCLUDING, BUT NOT LIMITED TO, LOST DATA TIME, PROFITS, DELIVERY DELAY OR LICENSOR'S PERFORMANCE OF SERVICES UNDER THIS LICENSE AGREEMENT. SOME JURISDICTIONS DO NOT ALLOW THE LIMITATION OF LIABILITY FOR PERSONAL INJURY, OR OF INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THIS LIMITATION MAY NOT APPLICABLE TO LICENSEE. Some jurisdictions do not permit the exclusion of limitation of liability for consequential or incidental damages, and, as such, some portion of the above limitation may not apply to Licensee. In such jurisdictions, Licensor's liability is limited to the greatest extent permitted by law.

General

Any claim arising out of or related to this Agreement must be brought no later than one (1) year after it has accrued.

This Agreement is the sole agreement between the parties relating to the subject matter and supersedes all prior understandings, writings, proposals, representations, or communications, written or oral, of either party. Only a writing executed by the authorized representatives of both parties may amend this Agreement.

No Assignability. This Agreement and the licenses granted hereunder cannot be transferred or assigned by Licensee without the prior written consent of Licensor. Any attempted assignment or sublicense by Licensee.

Severability and Survivability. If any part of this Agreement is held invalid or unenforceable, that portion will be construed consistent with applicable law as nearly as possible to reflect the original intention of the parties and all other portions of this Agreement remain in full force and effect. In the event of any expiration, termination or cancellation of this Agreement, provisions that are intended to continue and survive shall do so.

This Agreement is binding upon the Licensor's and Licensee's successors and assigns.

Failure of either party to enforce any provision of this Agreement will not be deemed a waiver of the provision or the right to enforce that provision.

Licensee shall not export or re-export, or allow the export or re-export of the Software or any copy, portion or direct product of the foregoing, in violation of any export laws, restrictions, national security controls or regulations of the United States or any other applicable foreign agency or authority. By accepting this license Agreement, Licensee represents and warrants that it is not located in, under control of, or a national or resident in a United States embargoed country, or on the United States Treasury Department's list of Specially Designated Nationals or the United States Commerce Denied Person's List or Entity List.

This Agreement shall be interpreted in accordance with the substantive laws of the Commonwealth of Virginia. In an action arising out of or relating to this Agreement or the Licensed Product, both parties consent to the nonexclusive jurisdiction of the federal and state courts located in the City of Roanoke, Virginia. The prevailing party of any dispute shall recover its reasonable attorneys' fees and costs incurred in litigating, arbitrating, or otherwise settling or resolving such dispute.

During the Agreement's term, Licensee agrees to regularly back-up its Licensed Computer on a separate medium. Licensee acknowledges that any failure to do so may significantly decrease its ability to mitigate any harm or damage arising from any problem or error in the Licensed Products or the provision of services under the Agreement.

NOTWITHSTANDING ANYTHING TO THE CONTRARY CONTAINED IN THIS AGREEMENT, IT IS SPECIFICALLY ACKNOWLEDGED AND AGREED THAT NO REPRESENTATION, WARRANTY OR OTHER ASSURANCE IS MADE, DIRECTLY OR INDIRECTLY, THAT (I) ALL OR ANY PORTION OF THE LICENSED PRODUCT COMPLIES WITH ANY FEDERAL, STATE OR LOCAL LAWS, RULES, REGULATIONS OR ORDINANCES, AS THE SAME MAY BE CHANGED FROM TIME TO TIME, OR (II) LICENSEE'S USE OF THE LICENSED PRODUCT WILL RESULT IN LICENSEE'S COMPLIANCE WITH ANY FEDERAL, STATE OR LOCAL LAWS, RULES, REGULATIONS, OR ORDINANCES, AS THE SAME MAY BE CHANGED FROM TIME TO TIME.

The Software is a ''commercial item,'' as defined at 48 C.F.R. § 2.101, consisting of ''commercial computer software'' and/or ''commercial computer software documentation," as defined in 48 C.F.R. § 12.212 or 48 C.F.R. § 227.7202, as applicable. Any use, modification, reproduction, release, performance, display, disclosure or distribution of the Software by the United States government, its agencies or instrumentalities shall be governed solely by the terms of this Agreement and shall be prohibited except to the extent expressly permitted herein.

Licensee's Acceptance

Licensee will be deemed to accept the terms of this Agreement and agree to perform obligations of Licensee under this Agreement upon installation of the Software or Use of the Software or by any other means, electronic or otherwise, prescribed by Licensor and accepted by Licensee.