SpatialAnalyzer

Tutorials

SpatialAnalyzer

Tutorials

2023.04.30

SPATIALANALYZER TUTORIALS

Hexagon Metrology, inc.
436 McLaws Circle
Williamsburg, VA 23185
+1 (757) 565-1500
www.kinematics.com

TABLE OF CONTENTS

TABLE OF CONTENTS

Basic Operations

CHAPTER 3

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Spatial Analyzer Setup and Configuration:

Basic Operations

Installation and Configuration

Installing Spatial Analyzer (SA)

SA is a Windows only application. To run SA on a mac a windows compatibility environment must be run through Boot Camp or a similar application.

The current version of SA, as well as a beta testing version, can be downloaded from the kinematics.com webpage here:

https://kinematics.com/download/downloadindex.php

To install the application double click on the SpatialAnalyzer ####.exe file and follow the directions.

Displaying Ribbon Menus

Starting in 2019 a ribbon menu was added to SA. This ribbon menu was designed to incorporate the best of the classic menus and the Toolkit menu and streamline SA workflow. The display of the ribbon menu is optional and enabled with a check box option on the Display tab of the Users Options "**Show Ribbon Bar**" (Figure 3-21):

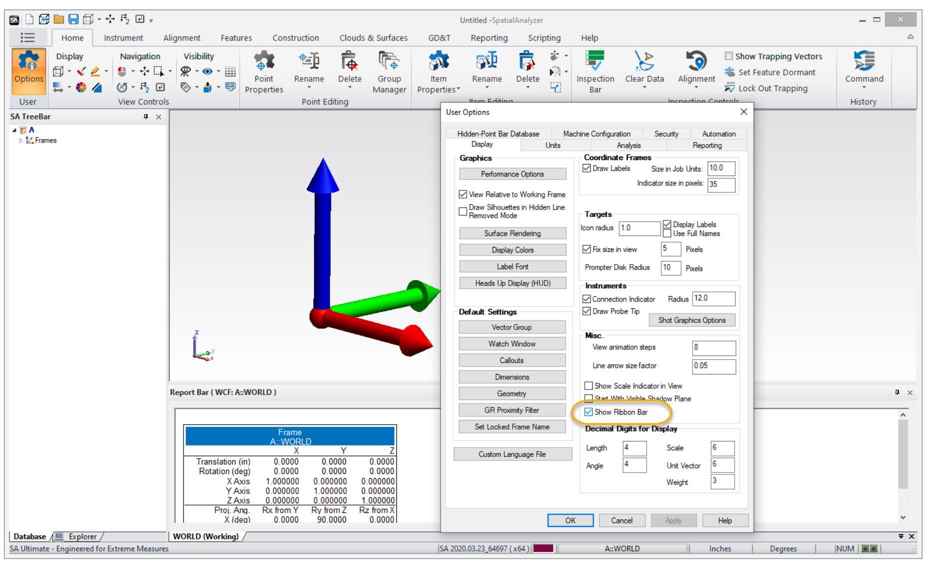


Figure 3-1. Default SA Window. The On/Off control for the Ribbon menu is found in the User Options on the Display Tab.

The following tutorials will use the Ribbon menu display. Buttons within the tabs are referenced as [Tab Name]>[Section Name]>[Button Name]>[Sub Button Name]. So as an example, you can rename a point by going to the **Home**(the tab to select)**>Point Editing** (ribbon section to look in)**>Rename** (the large icon). You can also rename many points using a naming patter by going one level deeper **Home>Point Editing>Rename>Points using a Name Pattern** , indicating to use the drop down button under Rename and select from within.

Customizing New SA Job Files

Much of SA can be customized. Some basic settings are saved in the registry and will persist when SA is closed and reopened. However, much of the settings are saved within a job file. To customize how a new job file looks you can save a template by doing the following:

1. Configure a job file such that it has the settings you would like to see when SA opens.
2. Select File>Save as Read-Only SA Template (Figure 3-2) and save the file within the C:\Analyser Data\Templates directory as "`default.xit64`".

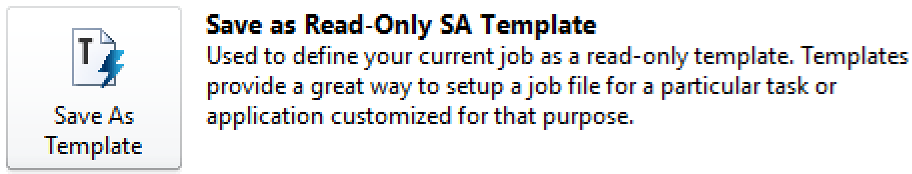


Figure 3-2. Saving a Job File as a Read Only Template

Additional template files can be saved and opened as needed but the default template is used for newly created job files automatically.

Basic Operations

Navigating the Graphical View

Here is a summary of the mouse-clicks and keyboard shortcuts for navigating the graphical view (Figure 3-3).

Graphical View

Action	Shortcut
Rotate (Orbit)	(Drag)
Zoom In/Out	Pg Up/Pg Dn
	Scroll Up/Down
	Ctrl + + Drag Up/Down
Pan	↑ ↓ ← →
	Shift + (Drag)
	(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 + →

Figure 3-3. KeyBoard Shortcuts for Graphical control

Navigating the Graphical View Tutorial

- **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.

When a new file is opened, the CAD surfaces are displayed in wireframe mode by default. This avoids the wait that might be required if the surfaces were to be solid rendered in the view – something that could be a lengthy operation in a large, complex file.

2. The Render settings can be changed with the render mode icons available on the Home >Display section(Figure 3-4).

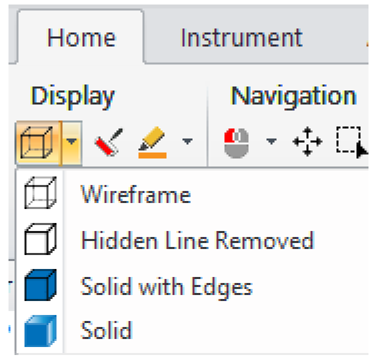


Figure 3-4. Render mode controls

Try each of the Render settings. Now render the model solid by using the Solid icon (Figure 3-5).

- **Wireframe.** Geometry is displayed as edges only, and hidden edges are visible.
- **Hidden Line Removed.** Geometry is displayed as edges only, but hidden edges are not displayed.
- **Solid + Edges.** Solid surfaces are rendered in addition to edges.
- **Solid.** Solid surfaces are rendered without edges.

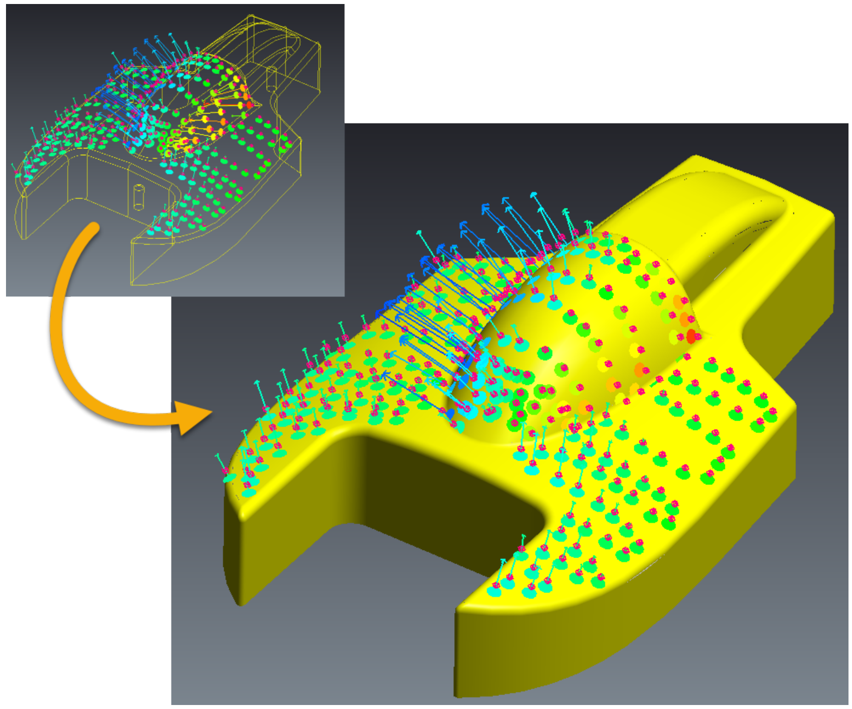


Figure 3-5. Rendering in solid-shaded mode.

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 Background button on the main toolbar also in the display section of the Home tab .

4. In the ***Background*** dialog (Figure 3-6), 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.

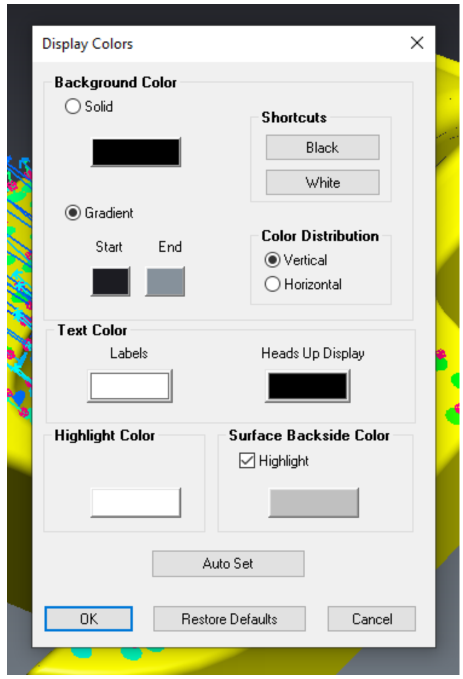


Figure 3-6. Changing the background color.

Hide and Show

Let's experiment with the ways to hide and show points, objects, and other SA entities. In addition to the menu controls in the Visibility section you can directly interact with the parts in the tree or the graphics:

5. Hide the vector group called *Boat Error_Vectors_Blotches from the tree*. 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 3-7). To show a hidden item, simply select **Show** again.

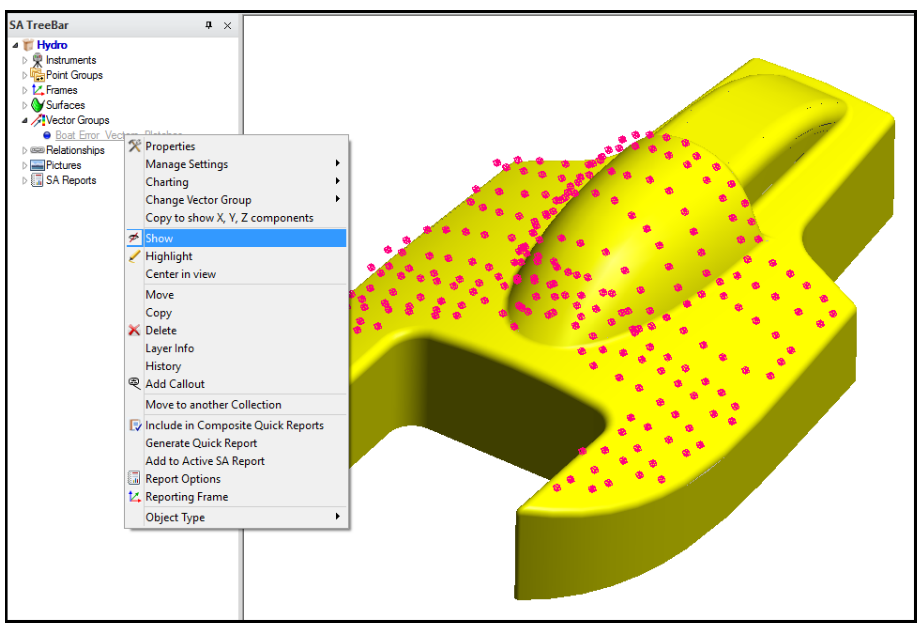


Figure 3-7. Hiding the vector group in the tree.

TRANSLUCENCY

6. Right-click on the dish surface in the graphical view and select Translucency. Change the setting to Translucent and click OK.

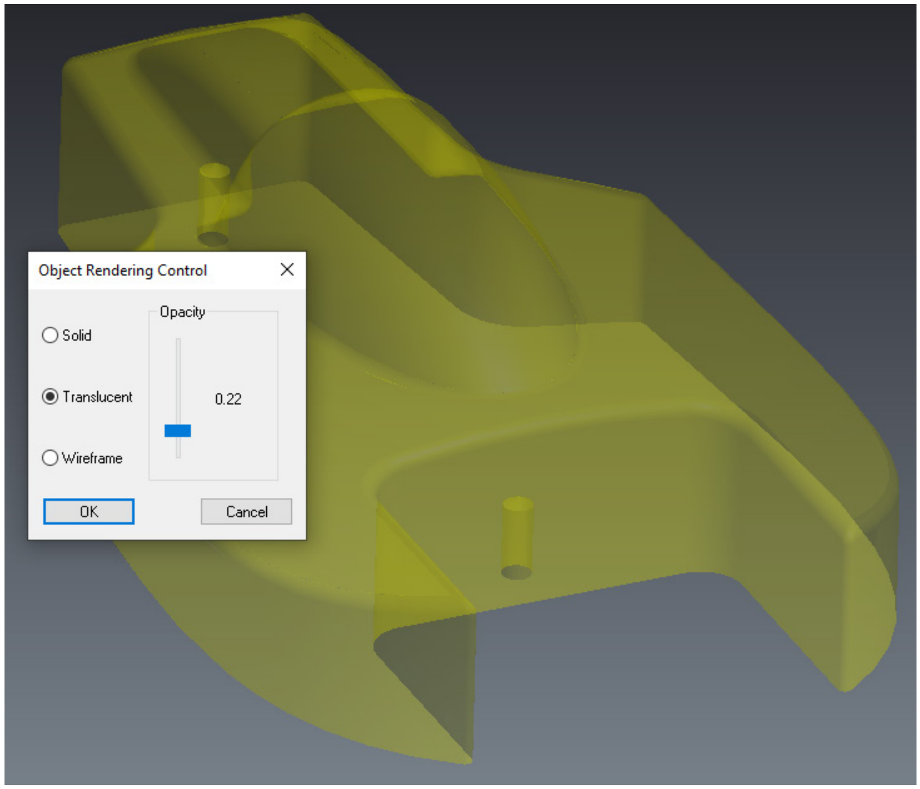


Figure 3-8. Changing CAD translucency so you can see through it.

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 placing your cursor over a zoom point of interest and rolling the scroll wheel forward on your mouse.
2. Zoom out by using the **Page Down** key.
3. Left-click in a region in which you'd like to zoom, drag a rectangle around that region, then release the cursor in the opposite corner to fit the zoom to your rectangle.
4. Now let's put all visible objects back in the view by using Autoscale ✣ . You can also use the hot key (Alt+A).

Pan

5. Try panning by clicking and holding the center scroll wheel on the mouse. Then proceed to drag your mouse. If you don't have a middle mouse button you can pan by holding down

the **Shift** key while pressing the left mouse button and dragging the mouse.

6. You can also use the arrow keys on the keyboard to pan the view up, down, left and right.

Rotating

7. 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.
8. 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.

9. Press the drop down beside the icon and select **Top**. This will orient the view with respect to the *World* frame.
10. 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.

Point Properties and Data Management

Keywords

- **Point Group.** A group of points that share a commonality. This could be a set of points that all define a specific feature, or all make up a group of fiducials.
- **Measured Point.** A point that was measured from an instrument and will have all metadata associated with it. Measured points are automatically tied to an instrument and will move with it any time an instrument's alignment is adjusted.
- **Constructed Point.** A point that is created manually in a job file or a point that is imported.
- **Target Offset.** The amount that a point will needs to be shifted prior to checking its deviation from an object. This is typically established when the point is measured via the instrument interface.

Opening a file to play with

1. In the Ribbon Bar, go to the HELP tab. In the Documentation section, choose Documentatoin>Sample Files.
2. Open the SA file Best *Fit Points To Points.xit*.

VIEW POINT PROPERTIES

3. Expand the Points Groups category in the tree to show Station1 and Nominals point groups.
4. Now expand the Station1 and Nominals point groups to display the points in each. Take note of the icons for each point (Figure 3-9).

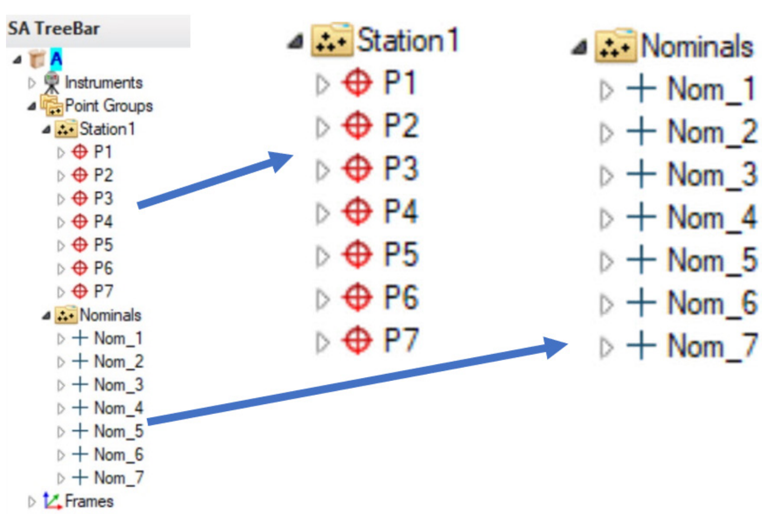


Figure 3-9. Measured vs. Constructed Points.

(NOTE: The red icons for the Station1 point group indicate they are measured points from an instrument. The blue icons for the Nominals point group indicates they are constructed or imported points.)

5. Single left-click on the white triangle next to P1 located in the Station1 point group to see its X, Y, and Z values relative to the working frame.
6. Open the full target information for P1 by double-clicking it in the tree or graphically. Right-clicking to open the properties is also an option (Figure 3-10).

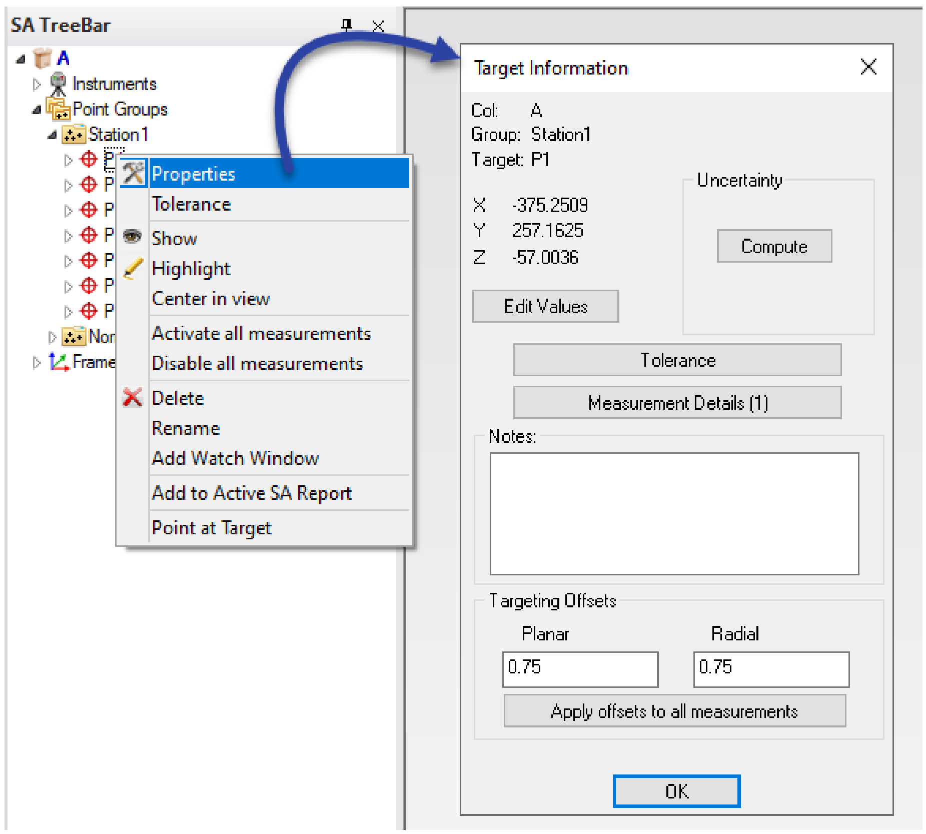


Figure 3-10. Right-click to Open Point Properties

7. Select Measurement Details, to open the metadata for P1. This will show you all the information that is attached to this point.
8. After reviewing select Done to close the dialog.
9. Notice at the bottom of the Target Information dialog there is a place where the planar and radial target offsets can be changed. Select OK to exit the target information dialog.

POINT MANAGEMENT

SA handles off sets by always recorded the center of the probe and recording the offsets as part of the point's properties. This allows SA to applying those offsets when a geometry fit or query is performed and use the direction vector of each point to object comparison to determine the exact off set direction. This ensures that the most accurate offset is always used (Figure 3-11).

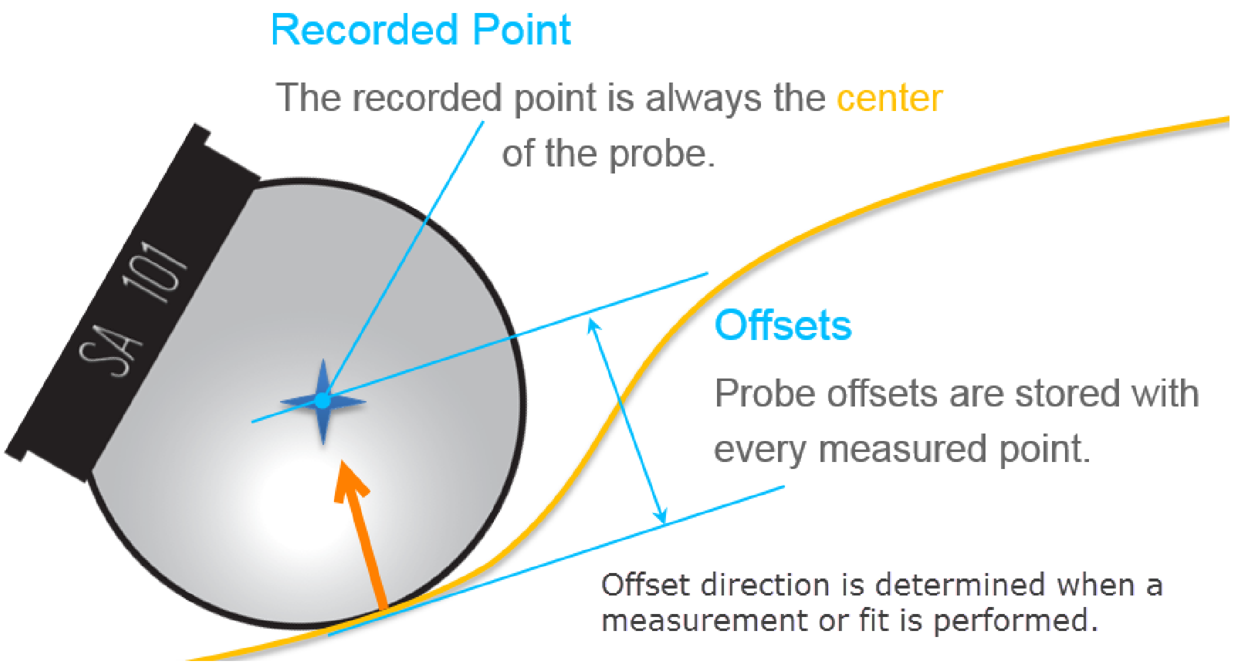


Figure 3-11. Offsets Recorded with a Point

Both Planar and Radial offsets are saved in a points properties. The planar offset is the primary offset used for most everything. Anytime you measure a surface the query direction provides the "planar direction for comparison. Radial offsets are secondary, only being used as part of a geometry fit where such as in a pin-net measurements of a hole. In this case the circle can be fit first, in plane, and then the radius can be adjusted as needed (Figure 3-12).

These values are set automatically for you when you select the correct tooling in the instrument interface and measure from within SA, but they can easily be adjusted.

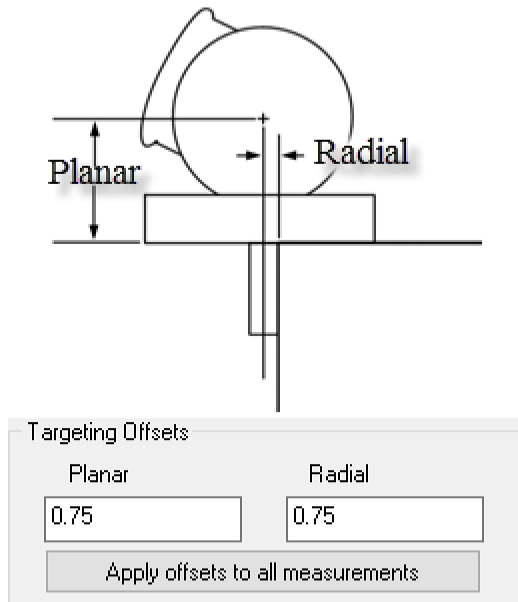


Figure 3-12. Planar and Radial Offsets

We noticed earlier that the point offset for P1 was 0.75" planar and radial. Let's change those point offsets to 1" for every point in the Station1 point group.

10. Right-Click on the Station1 point group in the tree and select "Set Point Properties".

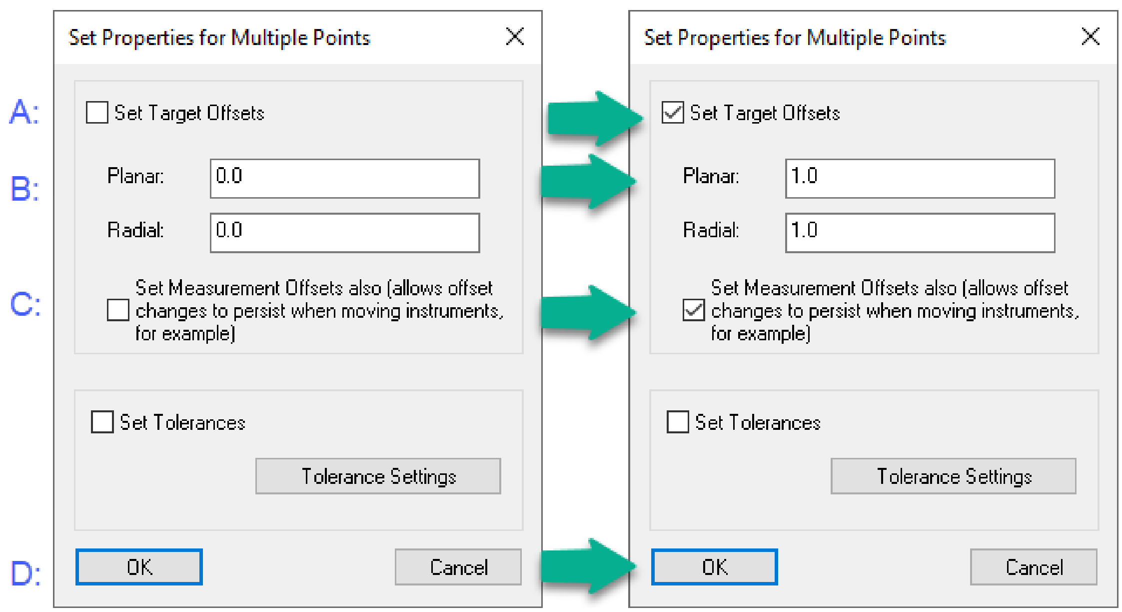


Figure 3-13. Set Properties for Multiple Points at one time.

- **A.** Select the top checkbox to change the target offsets.
- **B.** Type "1.0" in the textbox next to "Planar and Radial" to change the offsets to the desired value.

- **C.** Select second checkbox to apply the settings to the measurements.
- **D.** Click OK to update the points.

11. Check the properties of any point in the Station1 point group to ensure the planar and radial offsets show 1", respectively. Then Click Ok to close.

12. In the Ribbon Bar, select Group Manager. This will show you every point group in the SA job file and its values.

13. Here we can rename multiple points simultaneously. (Ensure to have the Station1 point group selected). Press *Ctrl+H* to bring up the "Replace" dialog.

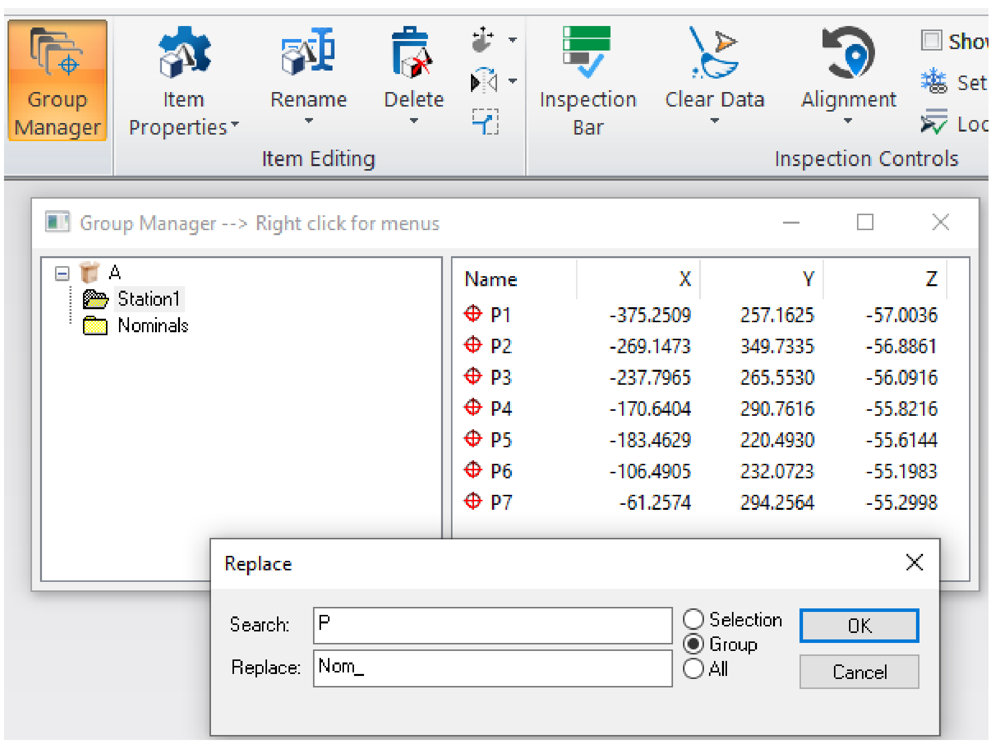


Figure 3-14. Renaming Points Using Group Manager

- **A.** Type "P" in the search field.
- **B.** Type "Nom_" in the replace field.
- **C.** Click OK.

EXPORT/ IMPORT POINTS

14. Right-Click on the Nominals point group in the tree and select "View Point List".

15. In the next dialog click "Export to Text file".
16. You will be prompted to select every point in the list shown, select "Yes".
17. Next you will need to choose a location to save the exported point list. Choose the desktop, the default file name will be "Point List.txt", click Save.
18. In the next dialog, the Ascii Export dialog, you can change some of the export settings, but here we will select OK.
19. Finally, you will see your saved point list in notepad. You can close this window and return to the job file.
20. Now, at the top left corner of the SA Job file, click on the file menu icon and follow the steps laid out in the below image to import the Ascii File.

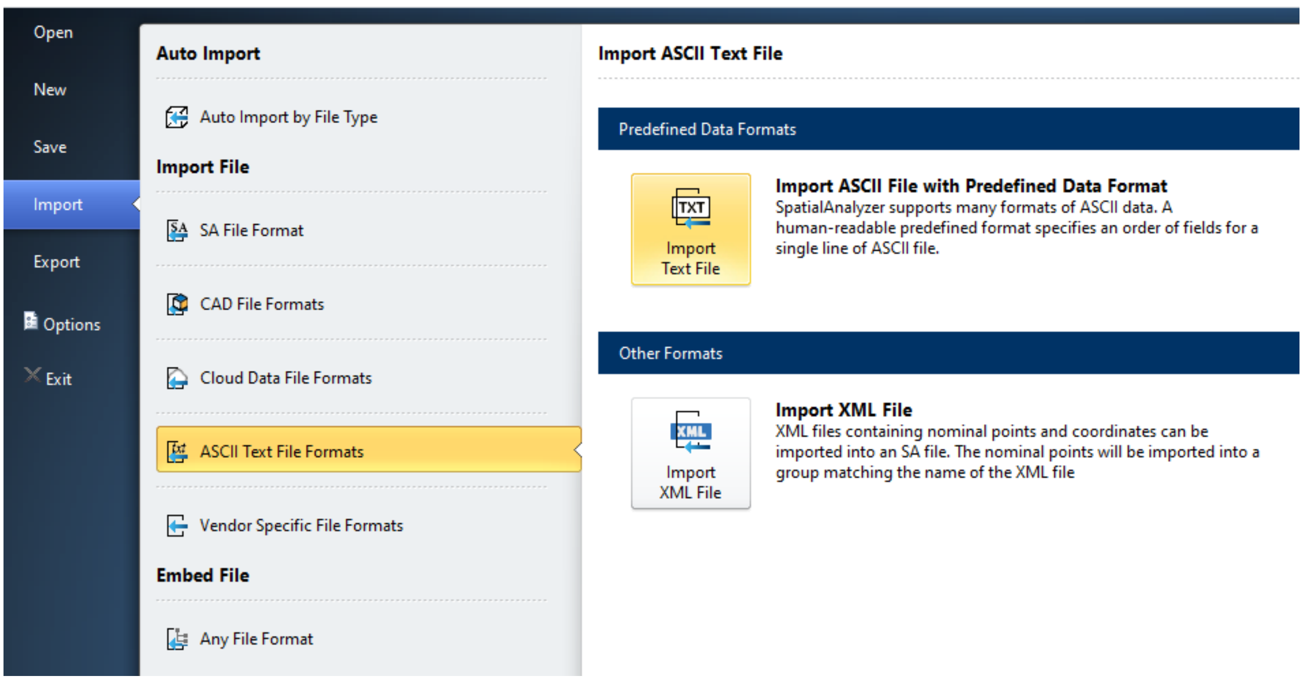


Figure 3-15. Importing Points from an AScii Text file.

21. When prompted, select the file that you would like to import and choose the Point List.txt file that you previously saved to your desktop.
22. The Ascii Import window will open. All the settings should be set correctly by default, so select Import at the bottom of the dialog.
23. An import report will open to notify you that 7 points were imported correctly. Click OK. NOTE: In the tree there should

now be another point group named Point List.txt and should include the 7 points that were exported.

Coordinate Systems In SA

Overview of How to Control Coordinate Systems

Spatial Analyzer provides a lot of flexibility when it comes to defining a base coordinate for a job file. There are lots of terms for this in different software packages including:

- Spatial Reference System
- Coordinate Reference Systems
- Coordinate Reference Frame
- Frame of Reference
- 6D location

SA uses the shortest and simplest term and calls a coordinate reference frame simply a "Frame". Each job file has a base reference for the job file which is called the "WORLD" frame. Any time you open a new job file you will find that it already has a frame in it called "WORLD" which defines the base reference for the job file (Figure 3-16).

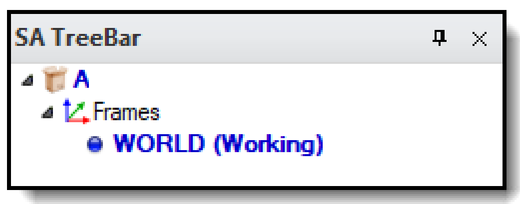


Figure 3-16. The World Frame is the Base reference for the entire job file.

Also notice that this WORLD frame is marked in bold blue letters and labeled "(Working)". That means that this frame is used to define the reported locations of anything else in the job file. So if you were to create a cylinder and wanted to report its location you are immediately given its location in a Cartesian coordinate system with the WORLD frame defining the origin of this reference.

Building a Cylinder at a specific location

1. To build a cylinder navigate to the Construction Tab and in the Build manually drop down select Cylinder.

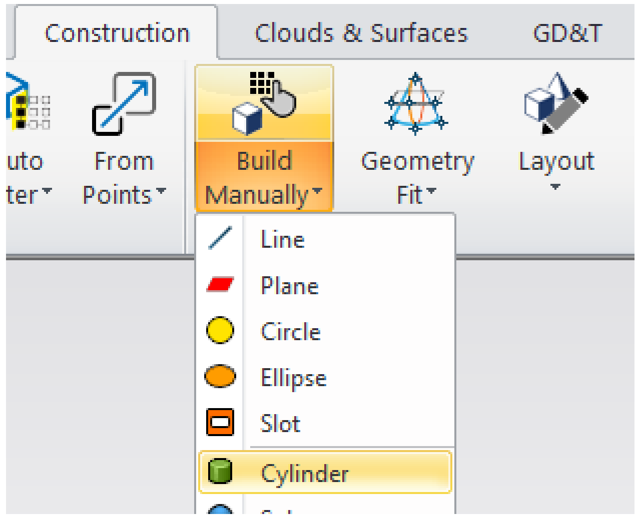


Figure 3-17. Build a Cylinder by entering its coordinates.

2. The Properties of the newly created cylinder will pop up. To accept the default properties, select close at the bottom right of the window.

In the example below the cylinder shown is 24" away in X and shifted 9" away in Y and its origin is perfectly on the XY Plane with respect to the World Frame (Figure 3-18).

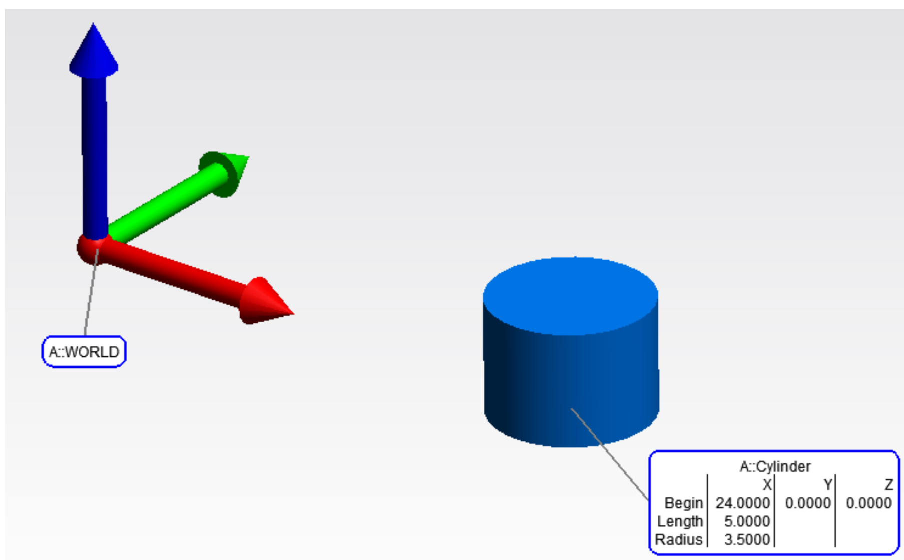


Figure 3-18. Location of a cylinder in 3D space with respect to the World frame.

Changing the Working Frame

What is unique about SA is that at any time you can select any frame in the job file, right-click on it and select Make Working Frame and it becomes the working frame for the job. This changes the reporting of all objects in the job file to now report their location with respect to the new "Working Frame". In the image below a new frame was added to the prior example file without editing the location of either the World frame or the Cylinder. Once the New Frame was marked as Working the cylinder's coordinates were updated to reflect its position relative to the New Frame.

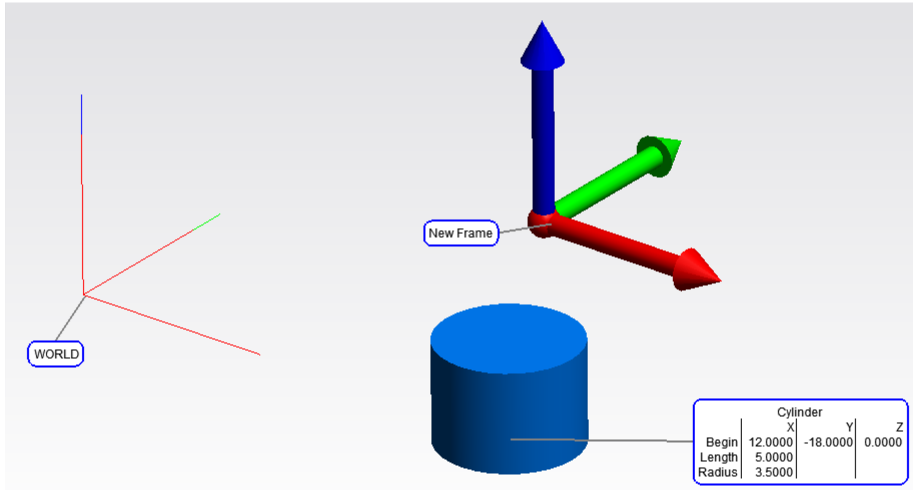


Figure 3-19. Resulting change in location of a cylinder's coordinates when a new frame is set as Working.

Its easy to tell which frame is the current working from because it is both clearly displayed in the graphics (only the working frame is drawn in 3D while the other frames in the job are shown as lines) and marked in the tree:

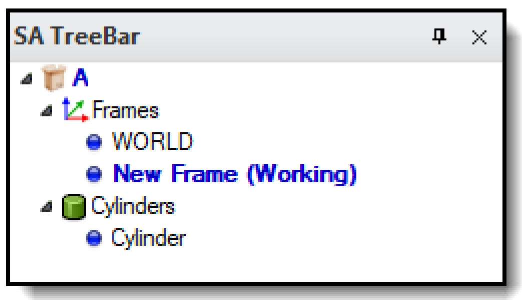


Figure 3-20. The Working frame in the tree is marked in Bold.

Setting A Reporting Frame

There are times when its helpful to be able to report the location of an object with respect to a frame that is not the current working frame. This can also be done and is set within the object's properties by editing the "Reporting" Frame.

Frames can be created in many ways, including being directly measured from any 6D probing device, and define a location. Including reporting a position and rotation in space they can also be used for alignment.

Working With Frames Tutorial

- **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.

3. In SA, start a new job file by selecting **File>New** from the menu or choosing the New File Icon from the Quick Access Toolbar.

4. Let's create a few points to work with in the workspace. To do this, go to the Construction Tab and Press Build Manually press **Ctrl+P**.
5. In the ***Add Points to Model*** dialog (Figure 3-21), 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:

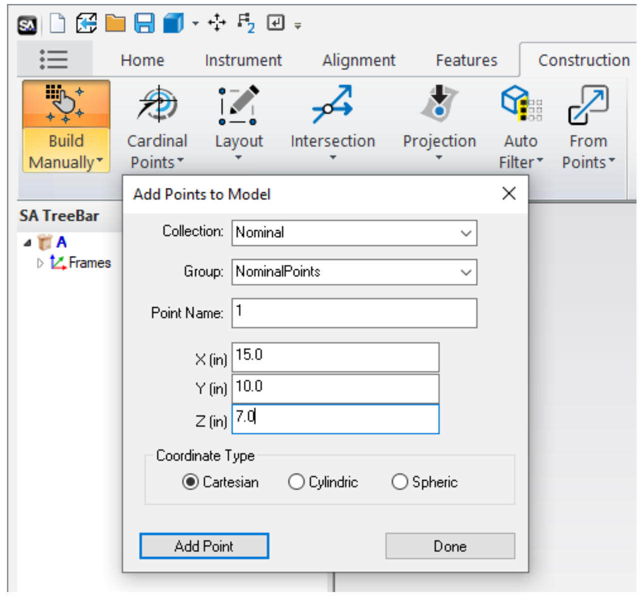


Figure 3-21. Adding a point.

6. 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 in the Quick Access Toolbar ✣, pressing **`Alt+A`**.

7. 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.

8. 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.

9. 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 3-22).

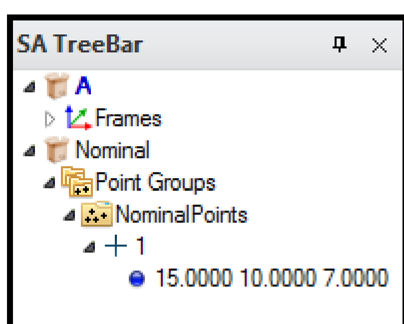


Figure 3-22. The newly-added point.

10. The default collection in your SA file is currently active, 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.

11. 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 3-23). You can also access this control directly on the Home Tab

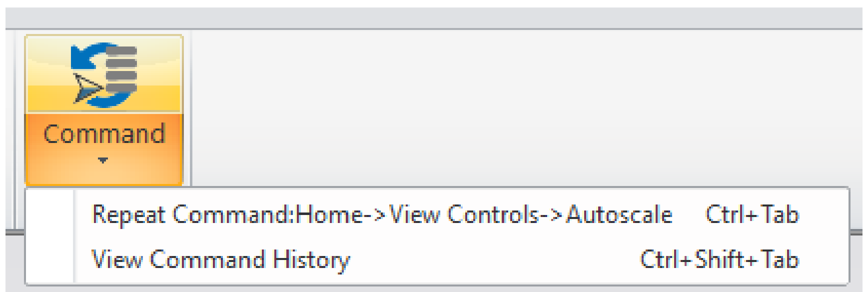


Figure 3-23. The list of recent commands.

12. Choose the **Construct>New Points>Build Manually** button 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.

13. 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.

14. Now let's create a plane to work with. From the menu, select **Construct>Plane(s)>Enter**. The ***Plane*** dialog will appear (see Figure 3-26 on page 29), allowing you to specify properties for the newly created plane.

15. 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.

16. If necessary, autoscale or zoom out so you can see the newly-

Note: The Autoscale keyboard shortcut of Alt+A will only work if the graphical view has the focus, by clicking once on it.

created plane relative to the points.

17. 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.

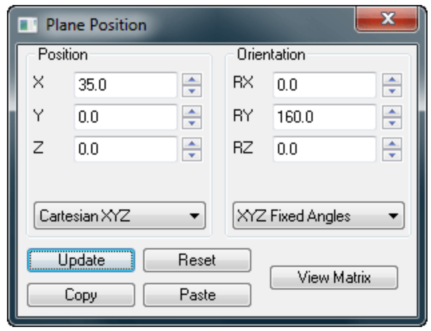


Figure 3-24. The Plane Position dialog.

18. 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:

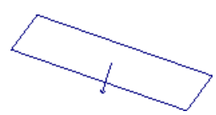


Figure 3-25. The positive side of a plane is indicated with an arrow.

19. The plane's normal direction can be reversed to face the opposite direction. In the ***Plane*** dialog, click the Reverse button.

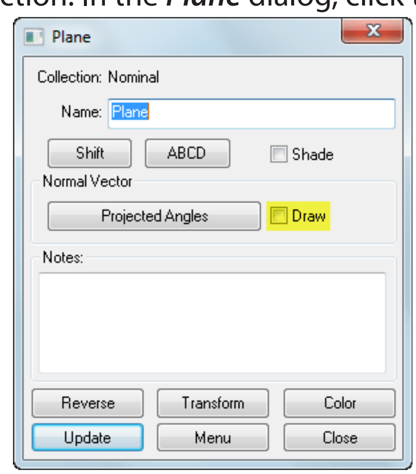


Figure 3-26. Selecting the option to draw a plane's normal vector.

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 3-26). Close the dialog by clicking the Close button or the "X" in the corner of the window.

20. 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.

21. You will be presented with the ***Circle Fit*** dialog (Figure 3-27), which allows you to define the parameters and results of the fit operation. Since three points perfectly define a circle, our resulting circle will end up passing through all three points exactly. Notice that the *Max* and *RMS* errors are zero:

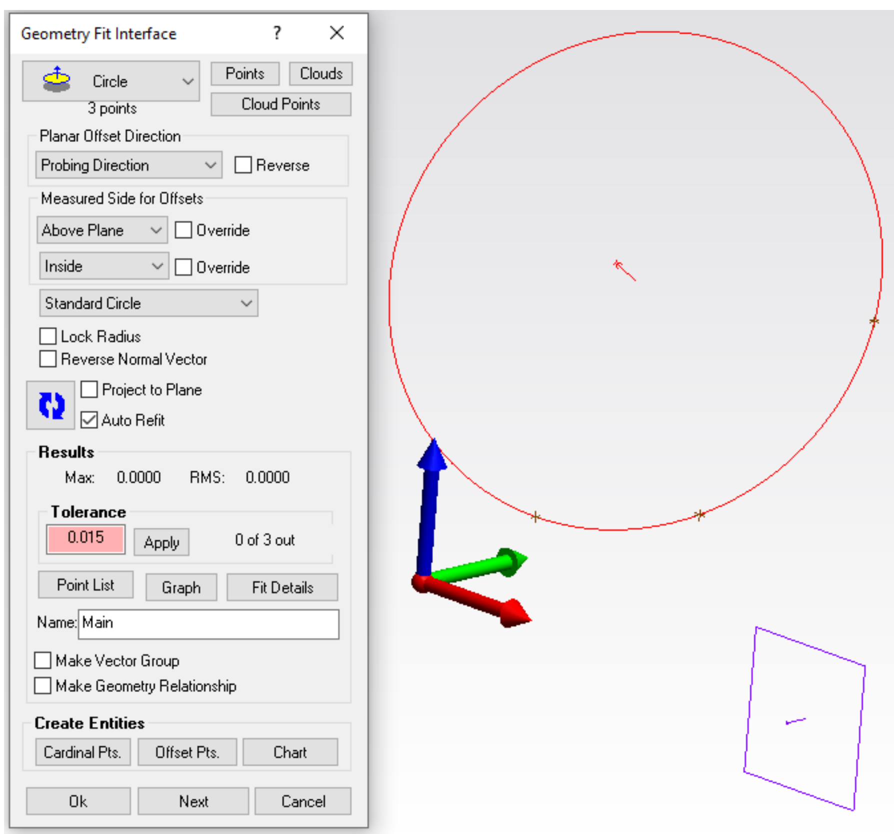


Figure 3-27. The Circle Fit dialog.

22. In the bottom-left corner of the ***Circle Fit*** dialog, press the **Cardinal Pts** 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.

23. Press OK again and 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 3-28). 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:

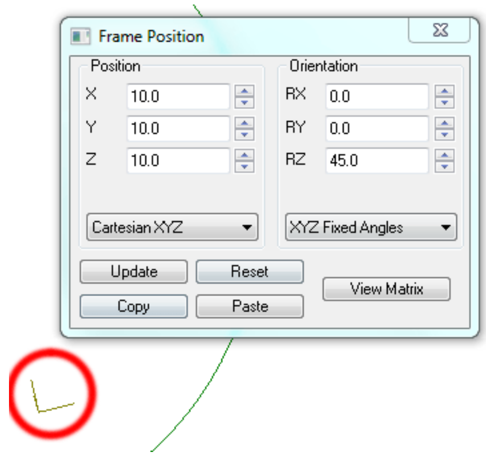


Figure 3-28. Creating a new frame.

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).

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 *My-Circle* 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 3-29).

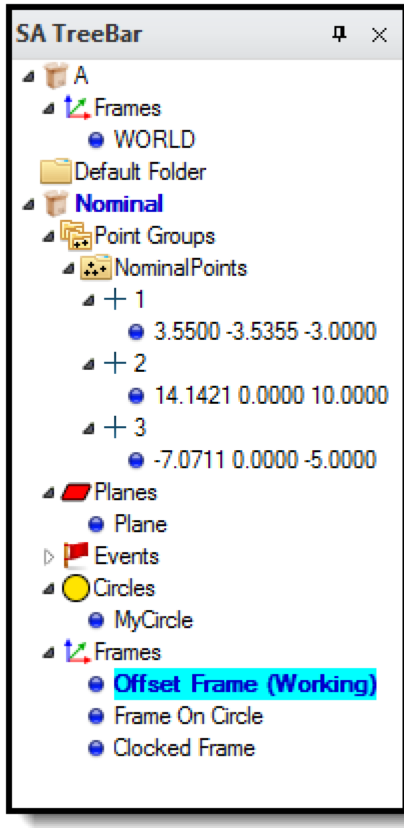


Figure 3-29. Viewing the point coordinates.

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.

This Page Intentionally Left Blank.

Instrument Operations

CHAPTER 4

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Instrument Connection and Operation

Connecting an Instrument

Spatial Analyzer (SA) is a software designed for measurement applications. It provides a window into a 3D world where measured point locations can be viewed from different angles and comparisons can be made between where measurements are in space. These measurements can be either imported or measured directly by an instrument.

In order to connect an instrument in SA, navigate to the **Instrument** tab of the Ribbon menu and Press the Add button.

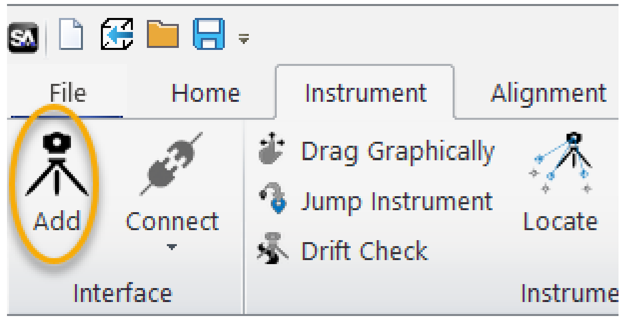


Figure 4-1. Instrument Add Button.

When you add an instrument to SA it defines an instrument station placement in the application's 3D world. Measurements from this instrument station are linked to this instrument model.

If you have connected to an instrument before then more than likely all you have to do to connect to that instrument again is to press the Connect button. However, initial setup and configuration requires a few more steps depending on the instrument type for that reason we have a selection of quickstart guides which were designed to help connect a particular instrument.

For connection information regarding a specific instrument refer to the appropriate chapter of the **Instruments Manual.**

Each chapter has an overview section regarding the type of instrument and the common functions used by all the instruments of that type and at the end is a selection of mode specific quickstart guides with specifics on how to connect.

For tutorials with specific instrument types see:

"Basic Tracker Operation Using the Instrument Toolbar" on page 38

Basic Tracker Operation Using the Instrument Toolbar

This tutorial walks through basic connection and operation using a Leica AT960 as an example.

ADDING AN INSTRUMENT MODEL TO SA

1. Open a blank SA Job File.
2. Go to the INSTRUMENT tab. In the INTERFACE section, select the add instrument icon, .
3. The ADD INSTRUMENTS TO SA dialog will open. Follow steps in Figure 4-2 below.

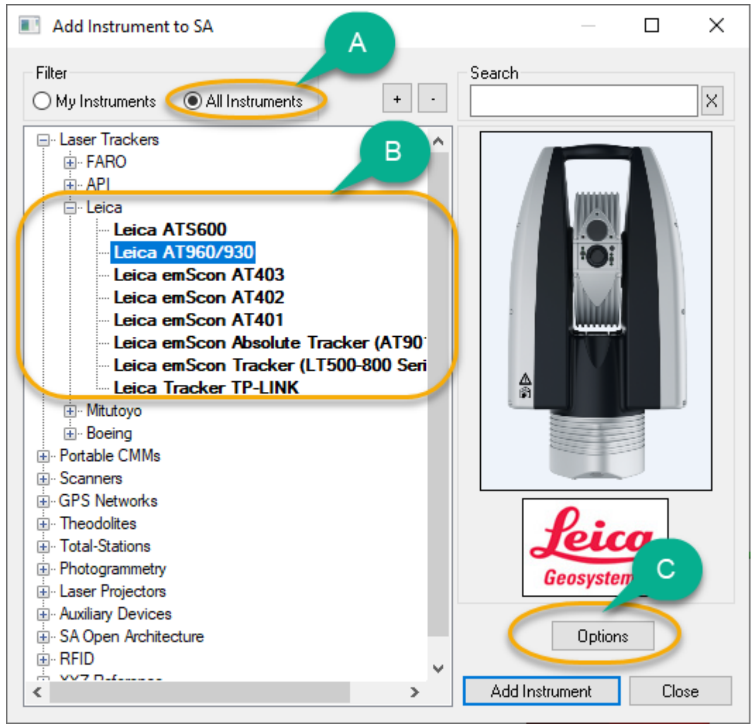


Figure 4-2. Add Instrument Options.

- **A.** Select the **All Instruments** filter if it is not already selected.
- **B.** Single left-click the tracker you would like to add to the job file. The Leica AT960 is selected here, but you can also make a selection from the or Faro or API sections as well.
- **C.** Selection Options from below the instrument image.

4. When you select the Options button, as displayed in the figure above, the Add Instrument Options dialog will open. Follow the steps in Figure 4-3 below.

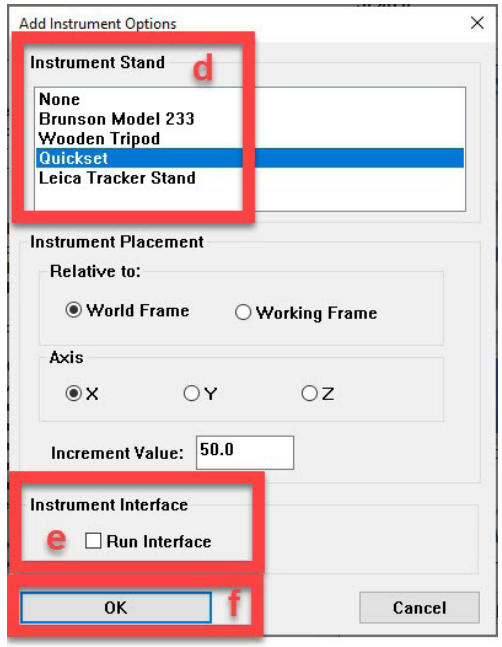


Figure 4-3. Add Instrument Options.

- **d.** Select Quickset as the instrument stand.
- **e.** Ensure the **Run Interface** checkbox is UNCHECKED.
- **f.** Click OK.

The instrument placement can be changed when necessary. The defaults will suffice for this exercise, though in real world use, this would just be a preliminary position prior to locating your instrument to a part or reference system. The current selection will position our instrument relative to World Frame, 50 inches along the X axis. So, our instrument's position once added, will be X=50, Y=0, and Z=0.)

5. Select Add Instrument button to add the instrument model to the job file.

CONNECTING TO AN INSTRUMENT

For this tutorial we will connect to a tracker in simulation mode, which does not require an actual instrument.

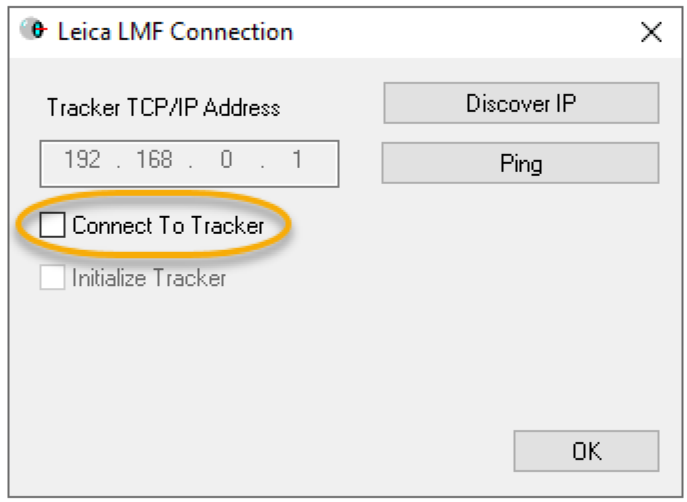
6. In the INSTRUMENT tab In the INTERFACE section, select the dropdown in the Connect icon,  .
7. Select **Laser Trackers** from the dropdown.
8. In the Connect to SpatialAnalyzer dialog, select the tracker model you just added. Click OK.

Each tracker move will involve adding a new instrument model to the job file. These represent tracker station placements and you could have many within a single job file.

9. In the Tracker Connection dialog, uncheck the Connect to

Tracker checkbox. *This will ensure that our computer connects to the instrument in simulation mode. When actually connecting to your instrument you will want this box checked.* If you were to connect to a live instrument, this dialog povides the ability to adjust the IP address and ping the instrument.

Figure 4-4. Connect to Tracker Dialog.



10. Click Ok.

INSTRUMENT TOOLBAR

The instrument toolbar is a simplified instrument interface which allows the user to name points, define tooling and initiate measurements easily. See image below for explanation (Figure 4-5).

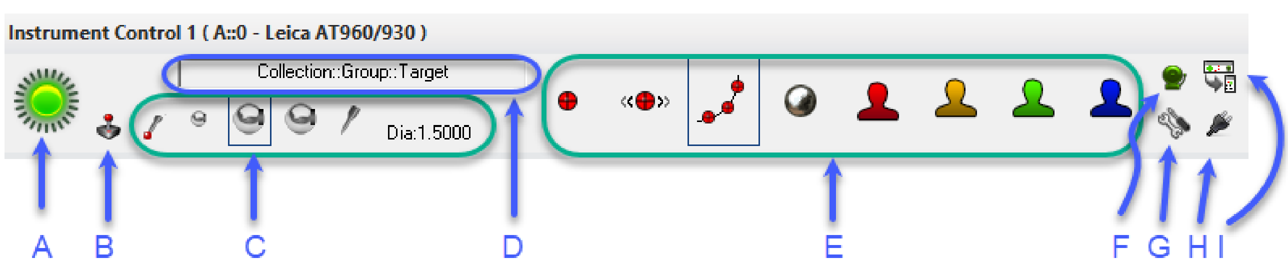


Figure 4-5. Instrument Toolbar Controls

- **a.** Beam Status indicator
- **b.** Steer Tracker Head control
- **C.** Tooling Quick-Select
- **D.** Target Name
- **E.** Measurement Profiles
- **F.** Alarms
- **G.** Checks/Utilities
- **H.** Battery Status

- **l.** Access button to the full instrument interface.

Depending on who might have last used the tracker, you may see the full instrument interface when you connect, because this status is saved. This full interface can be undocked from the SA such that it appears as a stand alone application (Figure 4-6). If so you will see it in the windows task bar as a reflector icon. To switch back to the Instrument Toolbar use the button on the bottom right of the interface.

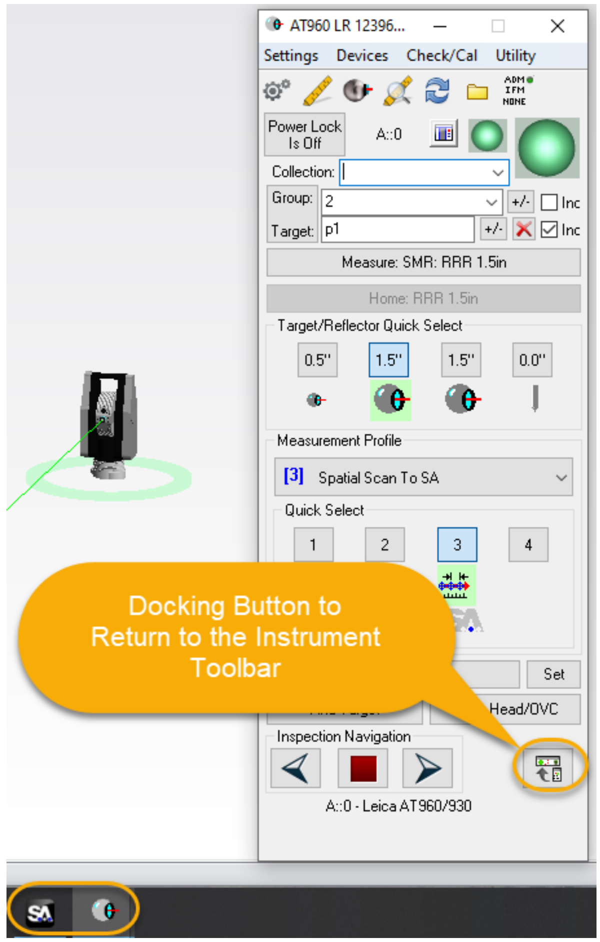


Figure 4-6. The Full Tracker Interface shown as a separate application.

11. Click the red beam status icon. It will turn green to emulate the action of locking on to a reflector.
12. Click in the target name box to edit the point name (Figure 4-7).

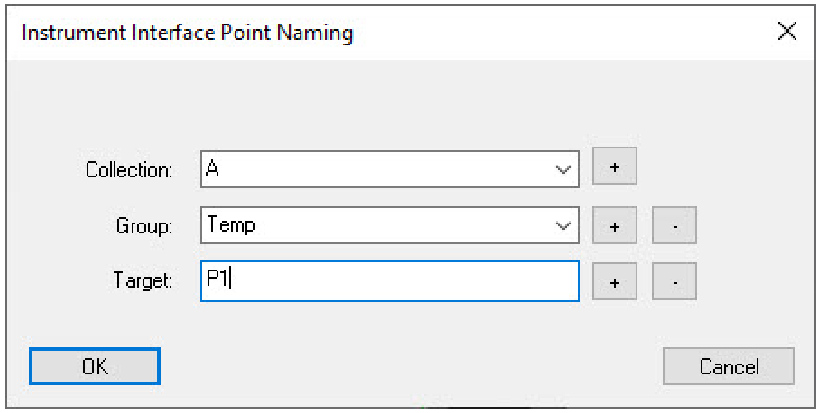
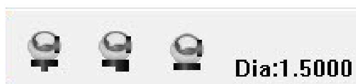


Figure 4-7. Instrument Interface Point Naming Control.

13. In the Instrument Interface Point Naming window make sure the collection letter is A, Group is Temp, and Target is P1. Once you have entered that information click OK. (The + and – symbols on the right side of the dialog allow you to increment or decrement the group and target names.)

14. In the Tooling Quick-select section, select the SMR 1.5" option.



15. Now, in the Measurement Profile section select the single point option . A point will be measured and the X,Y, and Z of that point will flash in the top left corner of the graphical view (HUD) along with the point name.

16. In the treebar you will see a new Point Groups category has been created. Expand that category and see the new point group "Temp" has been created which includes point "P1".

TOOLING DEFINITIONS

We can change the tooling quick selects so that they show the most frequent tools we use for easy access. To change them we need to right-click on the icon in the tooling quick select section that we would like to change. That same action will allow us to build new tooling definitions as well.

17. Right-click on the SMR 1.5" tooling definition in the instrument toolbar. Then select the drop-down to see the available tooling choices. If you do not see what you are looking for you can define new target .

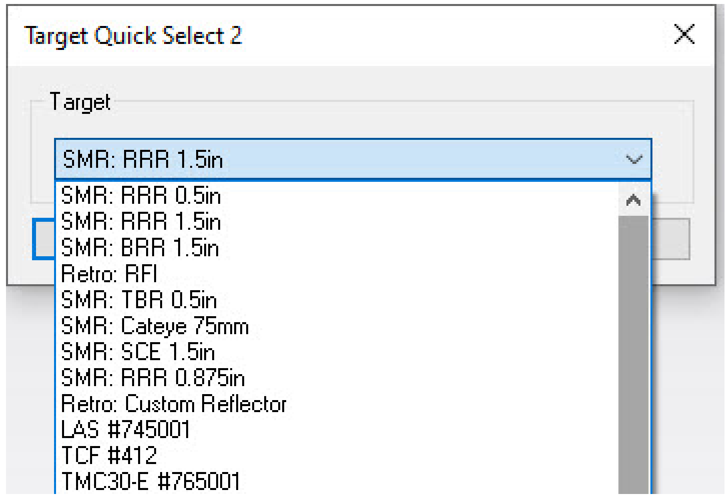


Figure 4-8. Target Quick Select.

18. Select **define new target** and click OK.

19. The Reflectors and Targets Dialog will open. We will define a few new tooling definitions. See Figure 15-5.

The top-left table will show the manufacturer definitions which are those that come directly from the instrument. The larger table at the bottom are the targets defined within SA that can be assigned as a quick-select and used during measurement acquisition.

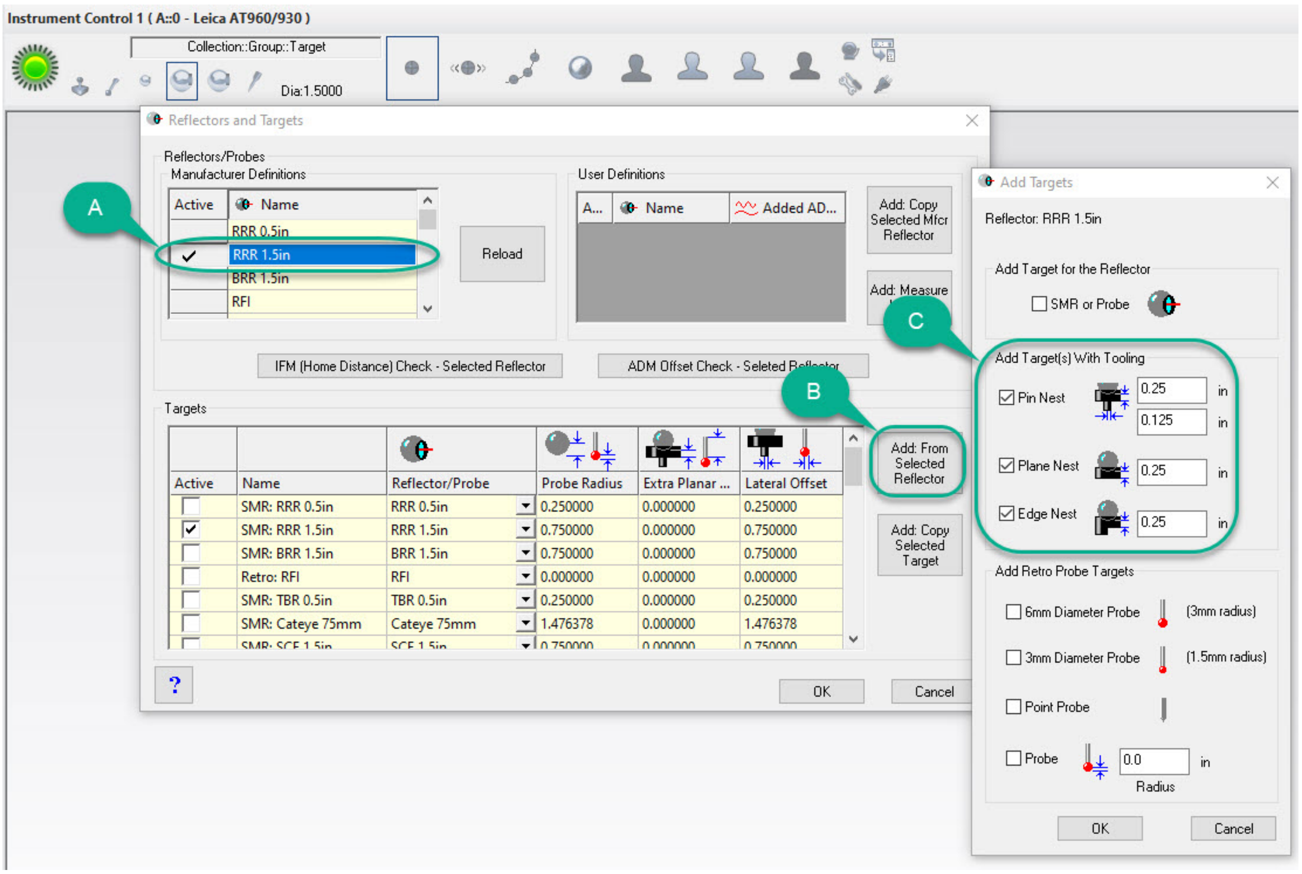
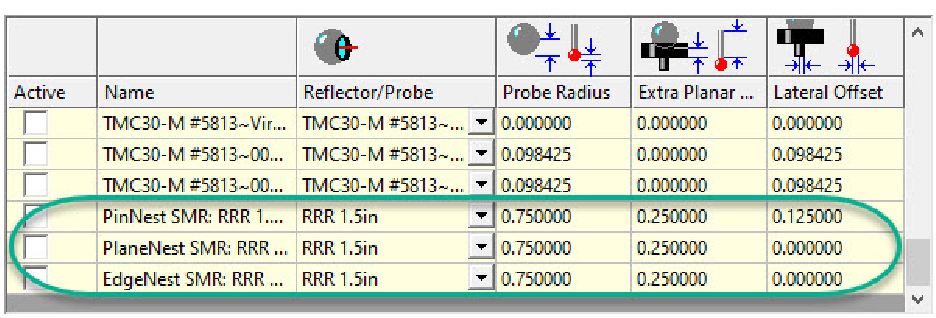


Figure 4-9. Building Targets within the Reflector/Targets Database

- **A.** Select the RRR 1.5in in the Manufacturer's Definition section.
- **B.** Click Add: From Selected Reflector.
- **C.** Click the three checkboxes shown. This will create target definitions that includes a Pin Nest, Plane Nest, and Edge Nest. Ensure the offsets for each tooling coincide with the physical tooling that will be used. Click OK when done.

Notice that the three tooling definitions that were created are now in the large table at the bottom of the Reflectors and Targets Dialog. Check the probe radius, extra planar offset, and lateral offset to ensure the values reflect the offsets you are wanting to add.



Active	Name	Reflector/Probe	Probe Radius	Extra Planar ...	Lateral Offset
☐	TMC30-M #5813~Vir...	TMC30-M #5813~...	0.000000	0.000000	0.000000
☐	TMC30-M #5813~00...	TMC30-M #5813~...	0.098425	0.000000	0.098425
☐	TMC30-M #5813~00...	TMC30-M #5813~...	0.098425	0.000000	0.098425
☐	PinNest SMR: RRR 1....	RRR 1.5in	0.750000	0.250000	0.125000
☐	PlaneNest SMR: RRR ...	RRR 1.5in	0.750000	0.250000	0.000000
☐	EdgeNest SMR: RRR ...	RRR 1.5in	0.750000	0.250000	0.000000

Figure 4-10. Newly Constructed Target Definitions.

20. Click OK to exit the database.

MEASUREMENT PROFILES

When taking measurements, we have the choice of how we would like to acquire the points. We can also change the behavior of the measurement profiles. Below are the four most common measurement profiles (Figure 4-11).

- **Single Point.** When pressed takes a single point measurement.
- **Stable Point.** When pressed will start a mode that allows you to hold the SMR stable within a certain space for a certain amount of time and a measurement will be taken. This allows us to take measurements without someone repeatedly hitting the measure button.
- **Spatial Scan.** When pressed will start a mode that takes incremental measurements as the SMR is moved a certain distance. This is a continuous measurement and can be very useful when taking a multitude of measurements very quickly on a part.
- **Tooling Ball.** Used when measuring tooling balls typically for alignment purposes. This allows us to measure around the tooling ball to create a sphere, which in turn generates a center of that sphere which is the measured center that can be used for alignment.

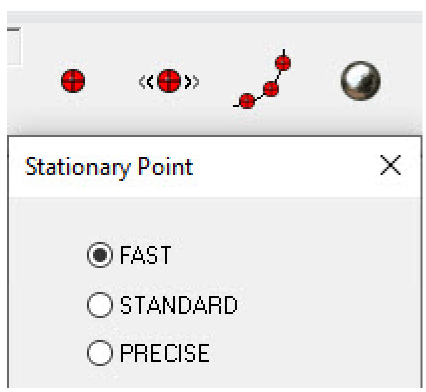


Figure 4-11. The 4 Basic Measurement Modes in the Toolbar.

To change the parameters of these measurement profiles, just right-click the icon and edit the parameters. For the single and stable point modes, there are three selections:

- **Fast.** This mode takes .5 second worth of samples.
- **Standard.** This mode takes two seconds of samples.
- **Precise.** This mode takes five seconds of samples.

What this means is that the Leica 960 operates at 1000 Hz; therefore it takes 500 samples during fast mode operation.

The Red, Yellow, Green and Blue silhouettes represent a user defined measurement profile. You can right-click either one of them, click a drop down, and choose from a variety of other predefined measurement options. When chosen they will be assigned to that silhouette.

Utilities and Checks

The final icons in the toolbar is the alarm icon and Checks/Utilities. See below.

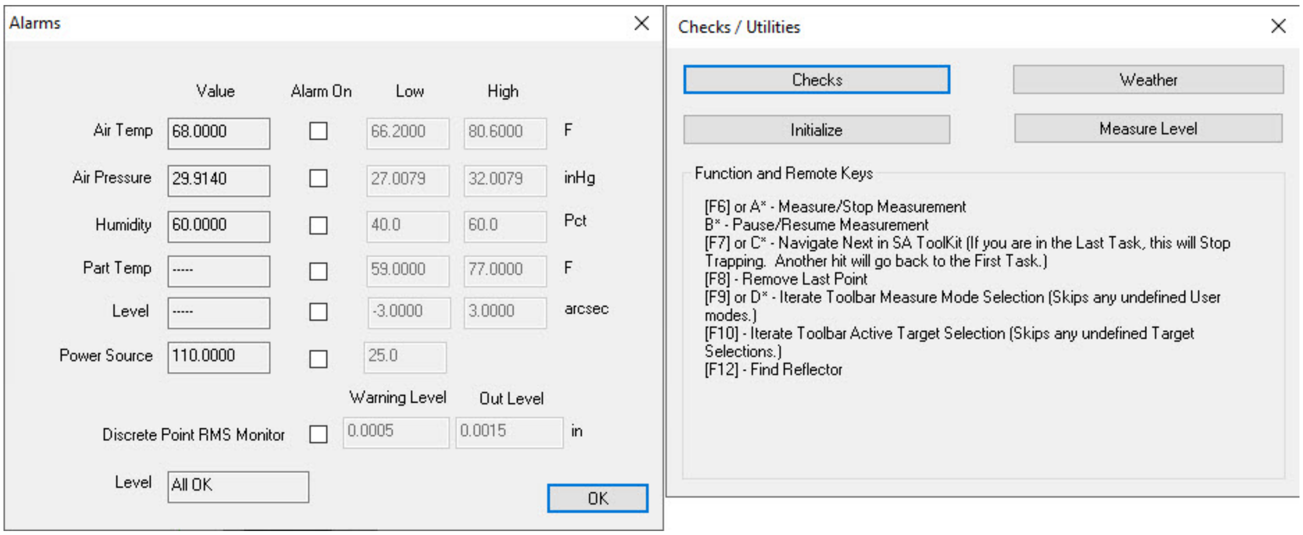


Figure 4-12. Alarms Checks and Utilities Controls Available from the Toolbar.

21. Press the Alarms button and then check the **Discrete Point RMS Monitor** checkbox. You will now be notified in the graphics if a measured point exceeds the set threshold.

This Page Intentionally Left Blank.

Instrument Operations

CHAPTER 5

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Basic Alignments

Measure Nominal Points

1. Open the SA file containing your reference points needed for alignment or import the points into an existing file.
2. In the Ribbon Bar, go to the INSTRUMENT tab and follow the steps in (Figure 5-1):

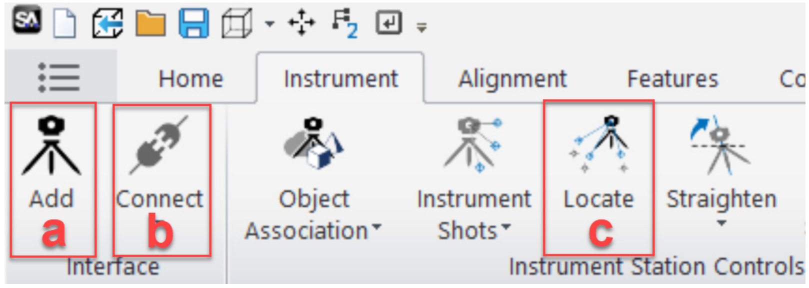


Figure 5-1. Add, connect, and locate an instrument.

Benefits

This process prevents the user from having to extract center points

1. Open the SA file containing your reference points needed for alignment or import the points into an existing file.
2. In the Ribbon Bar, go to the INSTRUMENT tab and follow the steps in FIGURE x-1:

a. Select Add.

b. Select Connect.

c. Select Locate.

3. Choose **Measure Nominal Points**.
4. Select the Nominal Point Group. This is the previously established reference group.

DIALOG SETTINGS

5. Check that your optional settings are correct for your job (remember that these settings will vary depending on the job) (Figure 5-2):

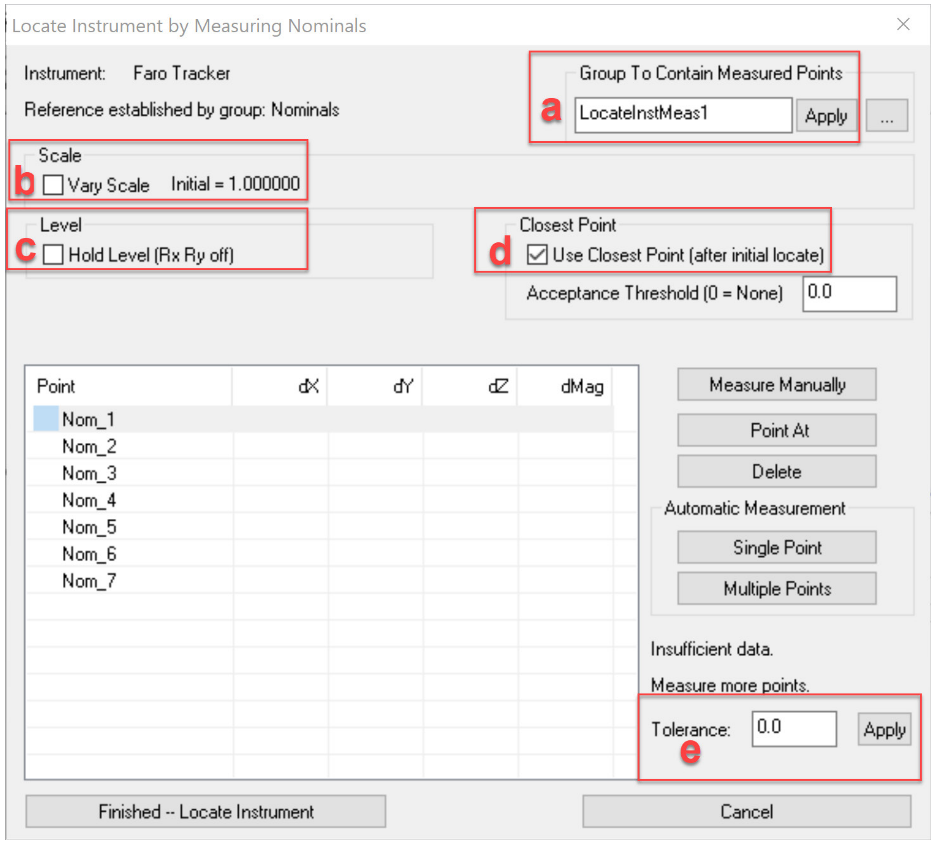


Figure 5-2. Locate Instrument by Measuring Nominals dialog.

a. Change the group name for the group to contain measured points and click Apply.

b. Check the Vary Scale box if you would like to "float the scale" (See Temperature Compensation module for details).

c. Check the Hold Level box to make sure that a leveled instrument stays parallel to the Z-axis.

d. The Closest Point box is checked by default to ensure that after three points are measured and an initial locate is performed, points may be measured without selecting the row first and SA will automatically give the deltas between its closest corresponding point.

e. Change the Tolerance box to display the dMag in RED and assign a point as out of tolerance. Click Apply after changing the tolerance.

RECORDING POINTS

6. If you are measuring out of order, then click on the row of the first point that you would like to measure. There are several options for taking measurements, as shown in (Figure 5-3):

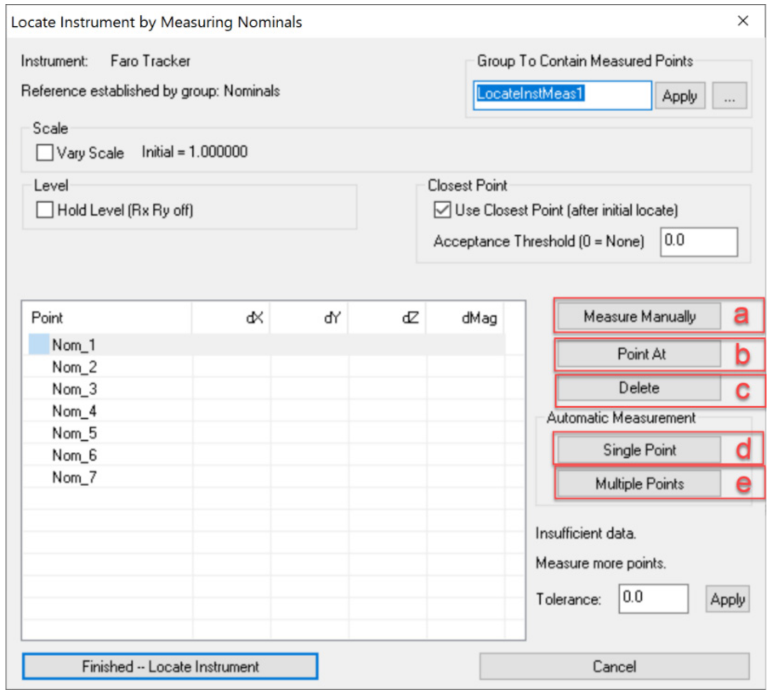


Figure 5-3. Measuring in the Locate Instrument Measure Nominals dialog.

a. Measure Manually. When clicked, the selected target in the list is measured. If the target already exists, an additional observation is added.

b. Point At. Points the instrument at the selected target and attempts to lock on. This applies only if the instrument supports pointing/target acquisition.

c. Delete. Deletes the selected target.

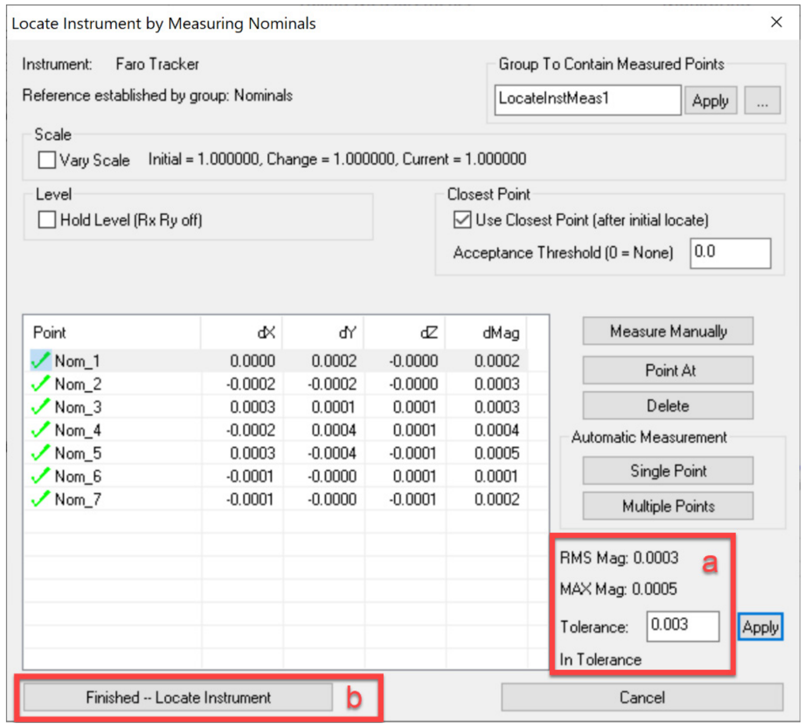
d. Single Point. Attempts to point at, acquire, and measure the selected target.

e. Multiple Points. Attempts to consecutively point at, acquire, and measure all points in the list.

COMPLETING THE ALIGNMENT

7. Once all of the points have been measured (or all points that can be seen), then check the MAX and RMS and ensure that the deviations are within tolerance. If so, then choose Finished-Locate Instrument (Figure 5-4).

Figure 5-4. Checking RMS and Max magnitudes and locating the instrument.



a. RMS and Max statistics are displayed here.

b. Finished – Locate Instrument locates the instrument to the nominal group.

REPORTING THE ALIGNMENT

8. In the SA Treebar, expand the Events category and double-click on the last item added. This will display your alignment results in the Report Bar (Figure 5-5).

Instrument Locate by Measuring Nominals				
Instrument				A::0 - Faro Tracker
Nominal Group				A::Nominals
Results	X	Y	Z	Mag
Transformation				
Translation (in)	-0.0003	-0.0009	-0.0006	
Rotation (deg)	0.0003	0.0001	-0.0002	
Point	dX (in)	dY (in)	dZ (in)	dMag (in)
Nom 1	0.0000	0.0002	-0.0000	0.0002
Nom_2	-0.0002	-0.0002	-0.0000	0.0003
Nom 3	0.0003	0.0001	0.0001	0.0003
Nom_4	-0.0002	0.0004	0.0001	0.0004
Nom 5	0.0003	-0.0004	-0.0001	0.0005
Nom 6	-0.0001	-0.0000	0.0001	0.0001
Nom_7	-0.0001	-0.0000	-0.0001	0.0002
RMS Mag	0.0003			
MAX Mag	0.0005			
Current Scale	1.000000	Fixed		
Level Lock	RxRy ON			
Working frame				A::WORLD
In Tolerance				

Figure 5-5. Locate Measure Nominals Report Table.

Drift Check Verification and Re-Alignment

A Drift Check is a crucial part of the measurement process since it gauges how much your instrument has moved relative to the part you are measuring. Without recorded drift checks, your data is unreliable.

Key Points

- Ensure that you have at least three control points.
- Perform drift checks frequently and on regular intervals. The frequency will need to increase the tighter the tolerances.
- Get to know your instrument and your working environment to establish an expected threshold for a drift check. If you have an idea of how accurate your instrument is under your expected working conditions, it will become second nature to accept and continue or add a new instrument and re-align.
- All drift checks are stored in the EVENTS category in the treebar for traceability and reporting purposes.

Getting Started

This tutorial is for a points based inspection and alignment.

1. Open a new SA file.
2. Add and Connect your instrument. You will need the SA demo part for this exercise, but not the CAD model.

SETTING UP DRIFT POINTS WHEN YOU ARE NOT ALIGNING TO A REFERENCE SYSTEM

In this exercise we are not aligning to the part, but simply measuring some of the as-built conditions of the part. Therefore, we do not have a reference system to tie in to but we still need to measure some initial points to check drift throughout the inspection and at the end.

3. Select the Instrument Control bar, name the Group Reference Points and the Target 1 (Figure 5-6).

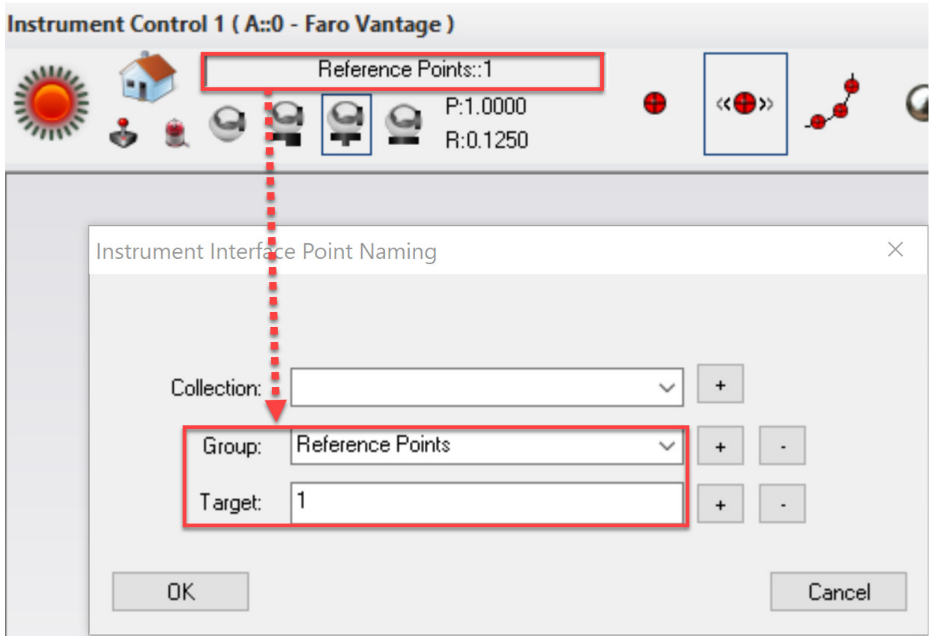


Figure 5-6. Naming Reference Points

4. Measure the six points shown below in Figure 5-7, ensuring that the probe is as stable as possible.

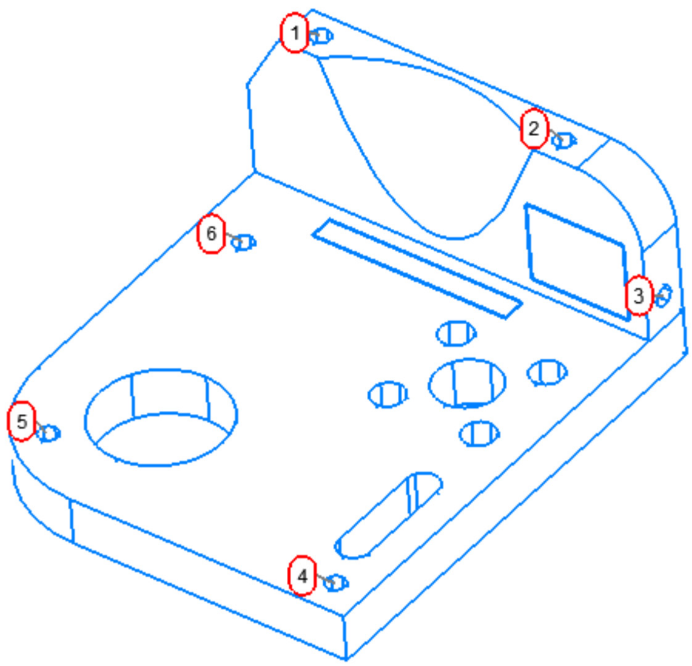


Figure 5-7. Six reference points to be used as drift checks

Before beginning your inspection, you may want to ensure that your setup is stable, and the reference points were measured properly. We recommend that you check the deviation of each point in real time. There is no need to record these points since you have not begun the measurement process yet.

5. In the tree bar, right-click on the point group A::REFERENCE POINTS and select Add Watch Window.

6. Place the probe in each hole and make sure the deviation is what you would expect. If it is not, delete this point group, correct your setup and begin again at step 3.

7. Measure the three planes in FIGURE X*3 by following the steps below:

a. In the **Features** tab, type `Plane 1` in the name box.

b. Check the box **Repeat for Inspection**.

c. Click the Plane button.

d. Measure this plane.

e. In the Inspection Bar, click Next.

f. Measure Plane 2 and click Next.

g. Measure Plane 3.

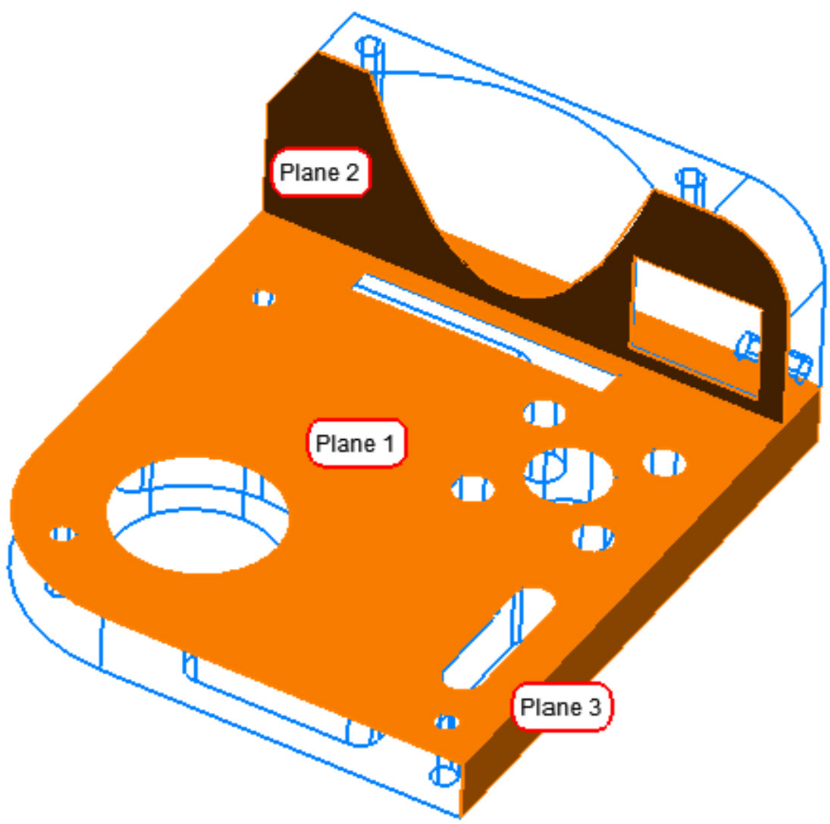


Figure 5-8. Three datum planes.

PERFORMING A DRIFT CHECK

The occurrence of a drift check during an inspection will vary. Many metrologists check drift at specific time intervals if their job will take a long time to complete. If your environment is temperature controlled and very stable, you may choose to check a drift every hour. If you work in an unstable environment you may want to check a drift every ten minutes. If you measure a great number of points and you would like to immediately validate those, then checking a drift at varying time intervals may be ideal. Or perhaps a crane moved overhead, and you want to check stability. There is not a one-size-fits-all standard for checking drift in portable metrology.

8. In the Ribbon Bar, navigate to the **Instrument** tab, and select **Drift Check** in the *Instrument Station Controls section*.

9. Select the A::REFERENCE POINTS point group as the Drift Point Reference Group.

10. Measure points 1-6.

11. Look below to Figure 5-9 to find the RMS and MAX errors. The RMS error is ________. The MAX error is _______. For training purposes, let's say that this drift is acceptable.

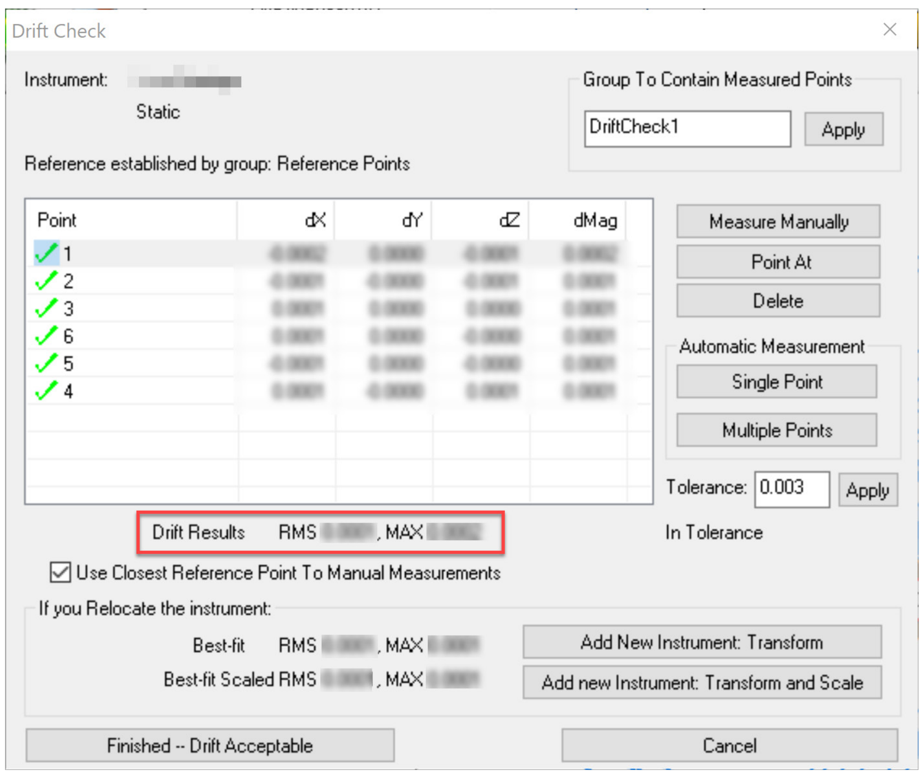


Figure 5-9. The Drift Check Dialog

12. Click Finished – Drift Acceptable in the bottom-left of the dialog.

13. Measure the two circles in Figure 5-10 (below):

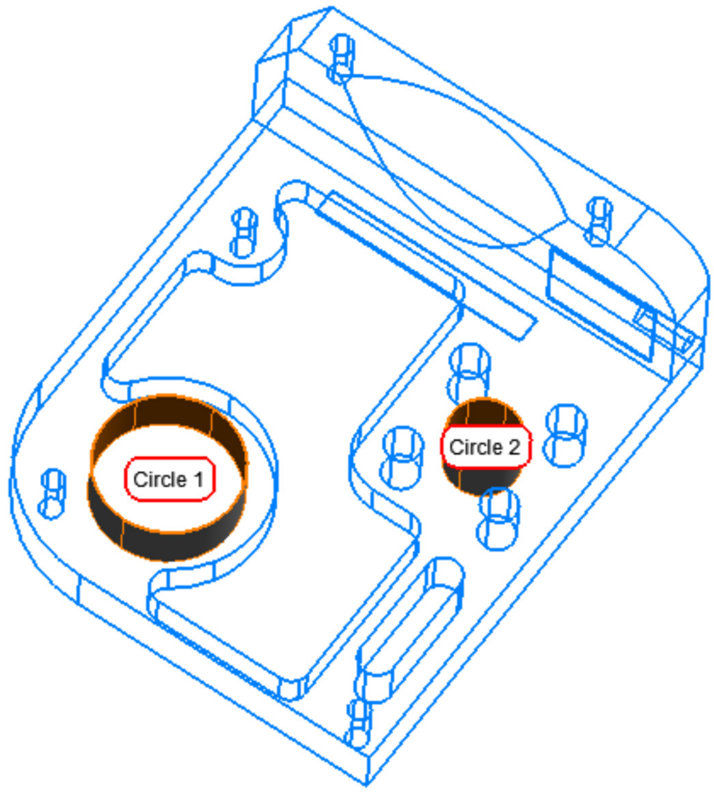


Figure 5-10. Measure two circles as shown.

WHEN A DRIFT CHECK FAILS

1. To simulate a loss of drift, move the part slightly.
2. Repeat steps 8-10.
3. The RMS Error is ________. The MAX error is _______. These errors should be much higher than if a loss of drift occurred naturally over time or due to temperature change.

Since we have lost stability, a new instrument must be added and aligned to the original reference points established by the first instrument station. There is a way to perform these steps automatically by using the "bad" drift points.

4. In the Drift Check dialog, select Add new Instrument: Transform.
5. The **Locate Instrument by Measuring Nominals** dialog will appear. This prevents you from having to re-measure your reference points again.
6. If you are satisfied with the alignment, click Finished – Locate Instrument.

You must now re-measure Circle 1 and Circle 2 since there is no way to prove that drift was lost after these features were measured.

7. Right-click on Circle 1 and select D**elete Associated Points**.
8. Click Yes to confirm deletion.
9. Repeat step 20-21 for Circle 2.

THE FINAL DRIFT CHECK

10. Right-click on the instrument in the graphical view and select **Drift Check**.
11. Select the point group A::DRIFT RE-LOCATE as the drift point reference group.
12. Measure points 1-6.
13. Ensure that the drift is acceptable before clicking Finished – Drift Acceptable in the bottom-left of the dialog.

Using Best-Fit to Align Point Measurements

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.

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.

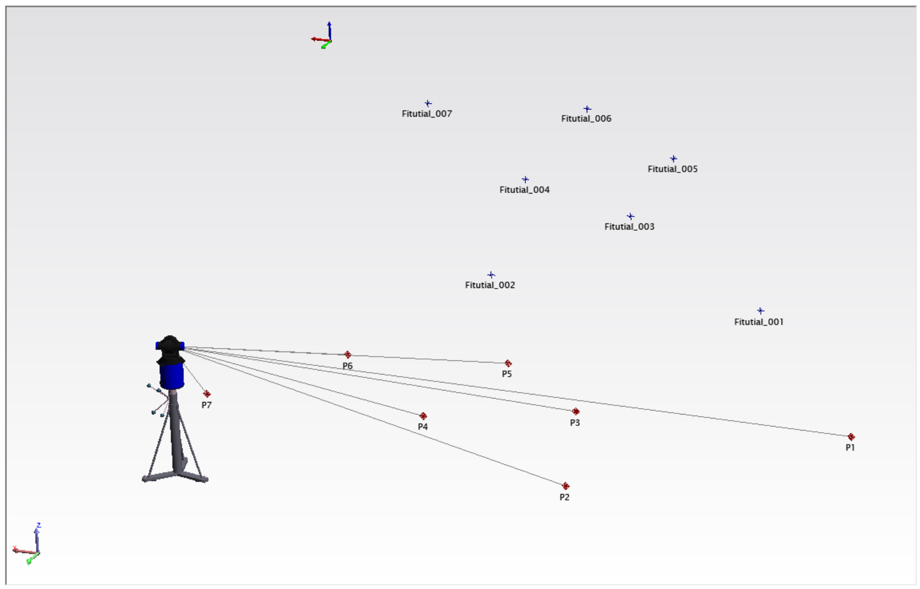


Figure 5-11. Best Fit Points to Points Example file

1. 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 (Figure 5-12).

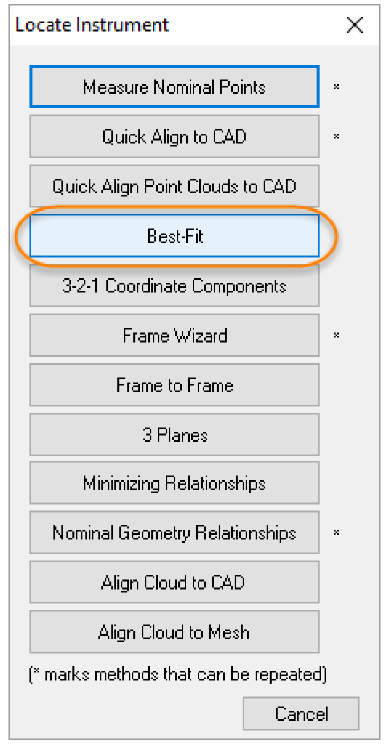


Figure 5-12. Locating using Best Fit.

2. You will be prompted to select the ***Nominals Group***. Click one of the nominal (fiducial) points in the graphical view. When you do, the prompt will ask for the ***Measured Group***, and you should then click one of the measured points.

The Best Fit alignment operates by fitting together points within two separate point groups by fitting together points with the same names. In this file the points are not named the same within the two point groups. As a result you will see the following dialog open (Figure 5-13):

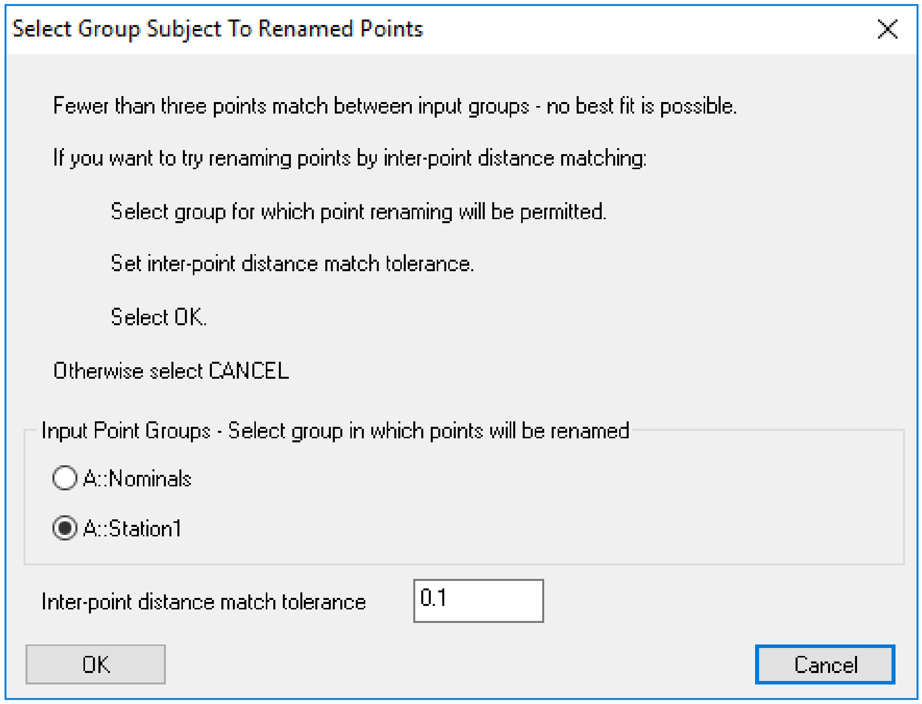


Figure 5-13. Automatic rename option provided by the Best Fit alignment.

There are a number of ways to rename points for this type of alignment but this tool uses an advanced **Inter-point distance** calculation to identify the relative placement of points in the two groups and usually does a pretty good job of matching up the names.

3. Press OK and accept the renaming of the Station1 points, and the Best Fit dialog should open.
4. 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).
5. 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.
6.

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 5-14). 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.

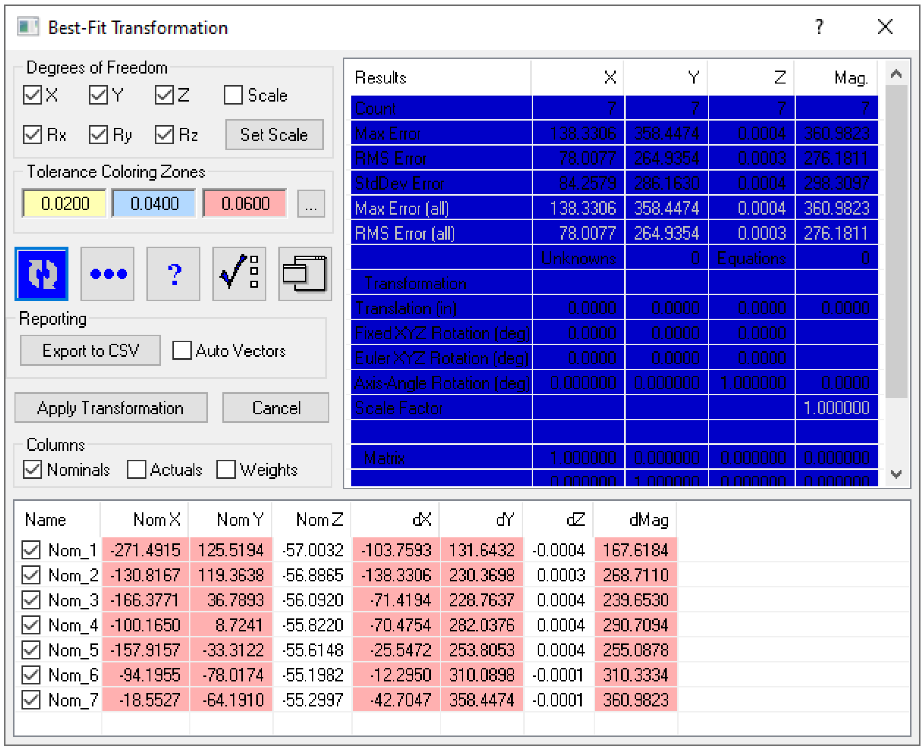


Figure 5-14. The Best-Fit Transformation dialog.

2. Click the Re-fit button to calculate the fit. Immediately, the results of the fit are displayed (Figure 5-15).

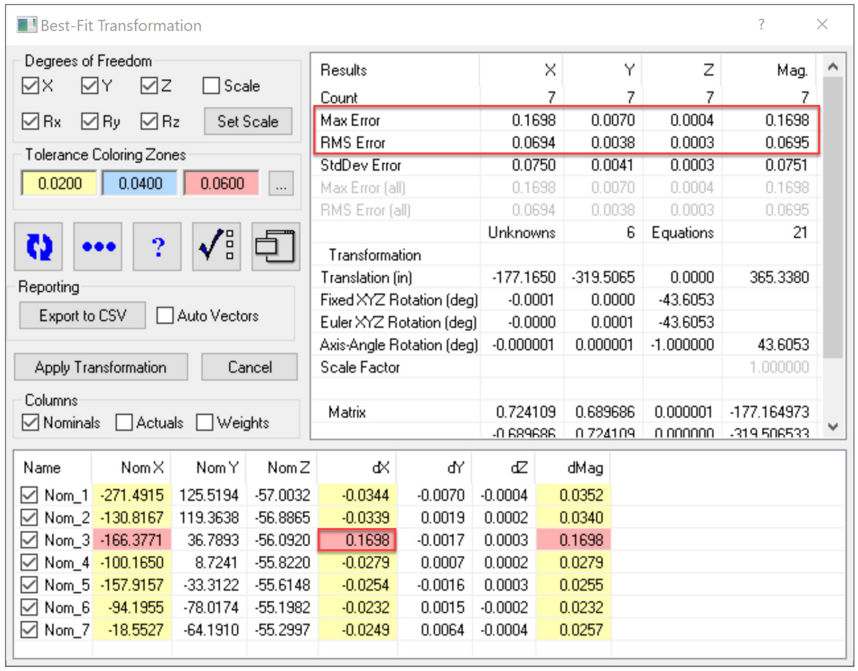


Figure 5-15. The fit results.

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 5-16), set the values to *0.005"*, *0.010"*, and *0.020"* respectively.

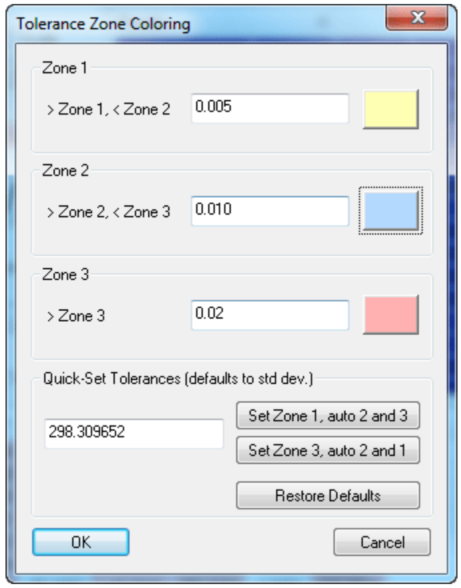


Figure 5-16. The Tolerance Zone Coloring dialog.

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 5-17), click the 0 button under the Y component to set its component weight to zero and click the OK button.

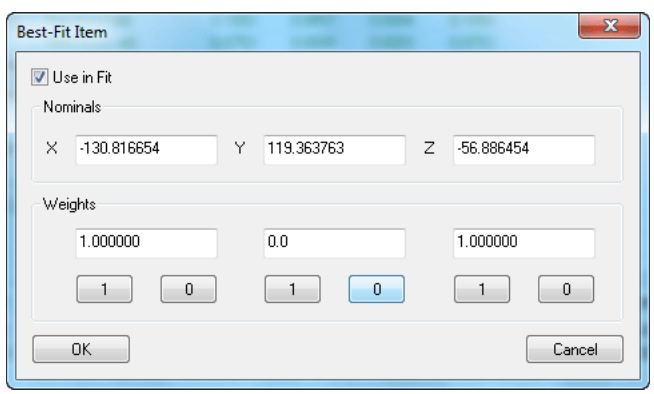


Figure 5-17. The Best-Fit Item dialog.

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 5-18). 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.

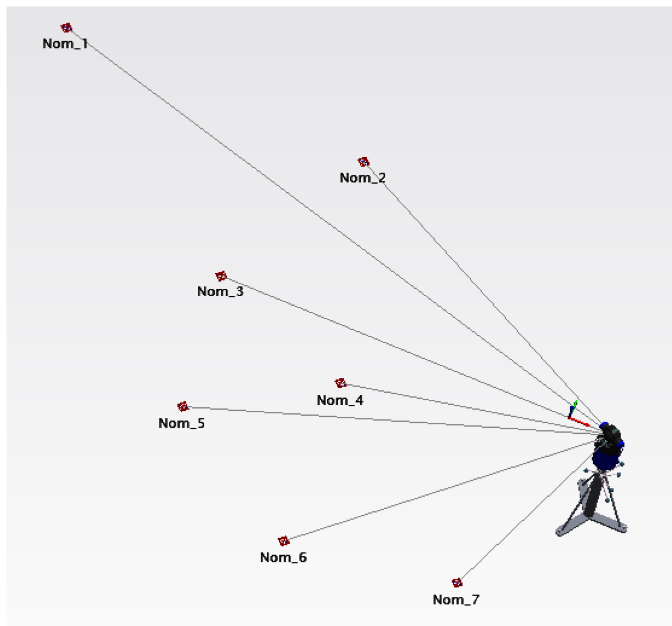


Figure 5-18. After the best fit.

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.

Quick Align to CAD

Quick Align is a tool used to align a measurement device to a nominal CAD model. Like its name suggests, it is a fast, easy process. However, some caution should be taken when selecting the points used in the Quick Align. A small error in measured data can have a large effect on the final alignment. For this reason, this section will deal with how to best select points on the CAD model for Quick Align.

First lets import a CAD model to work with

1. Open a new SA file.
2. In the Quick Access toolbar select **Auto Import** and navigate to the SA `DEMOPART_inches.CATPart` within the samles directory (C:\Program Files (x86)\New River Kinematics\SpatialAnalyzer 2020.04.09_65432\Samples) (Figure 5-19).

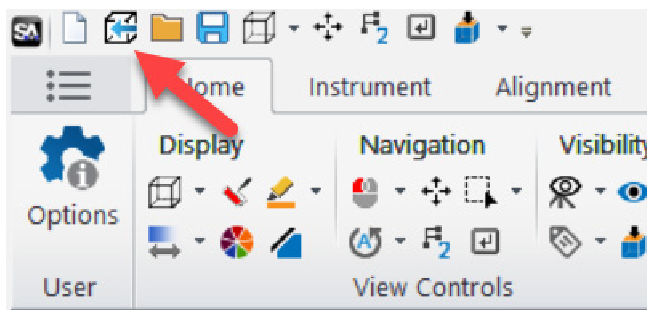


Figure 5-19. Importing CAD with the Auto Import Button

3. In the Choose CAD Features dialog, first uncheck the top box to de-select all items. Then, check the two boxes in the surfaces category as shown (Figure 5-20). This will exclude all the additional model details which we won't be needing for this exercise.

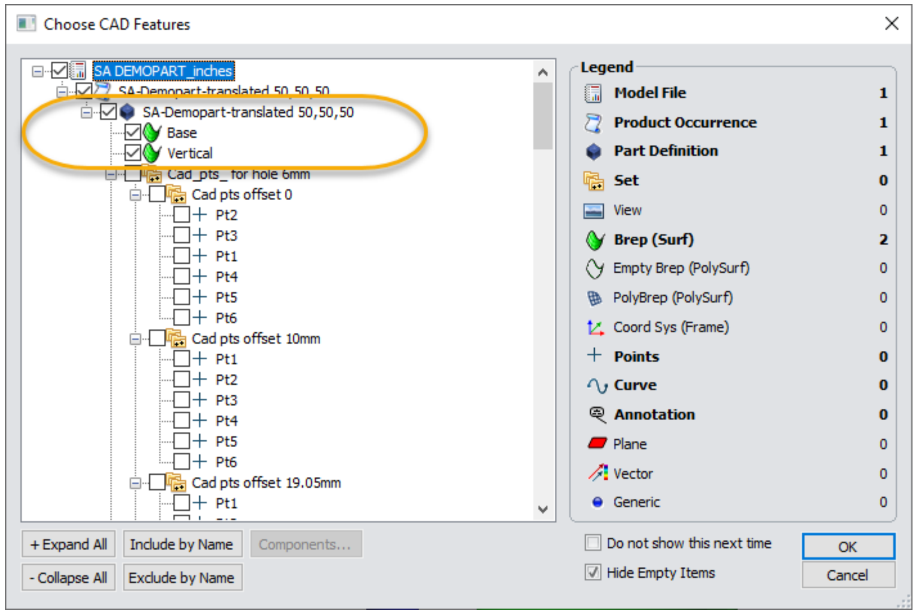


Figure 5-20. Select Surfaces to Import

4. In the home tab in the view controls section, use the drop-down arrow next to the surface rendering command (Figure 5-21).

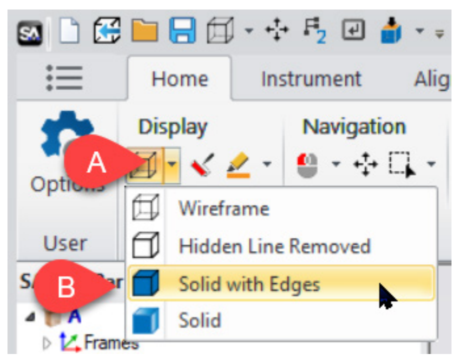


Figure 5-21. Setting the Render Mode to Solid with Edges

Your model should now appear as follows(Figure 5-22):

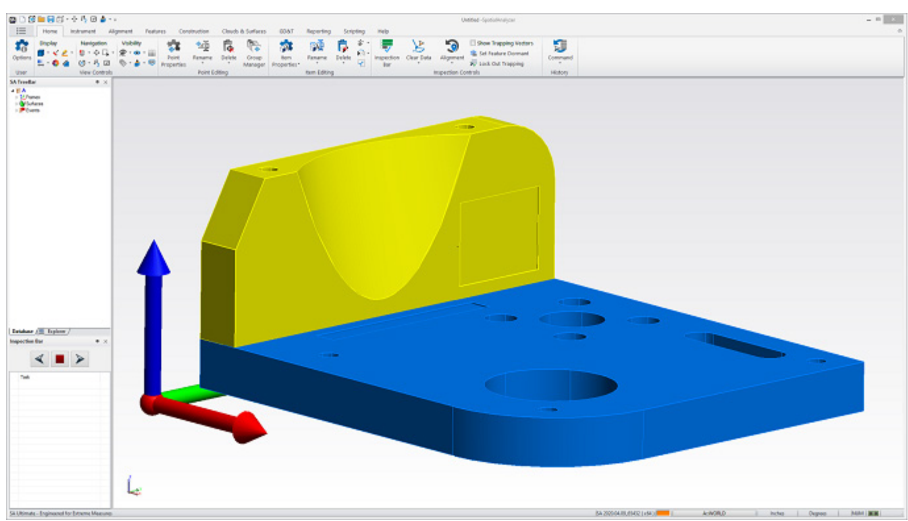


Figure 5-22. Imported NRK Part

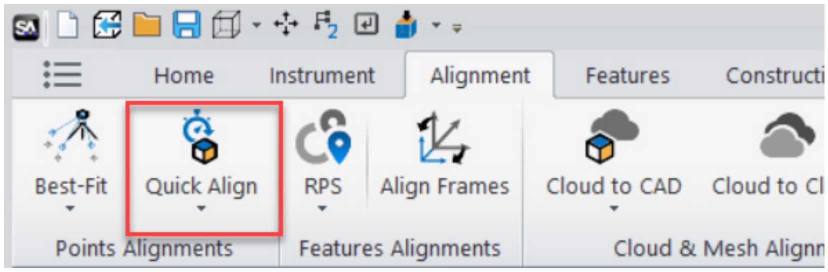
Next Add an Instrument and Connect

More details on that process can be found here "Measure Nominal Points" on page 49.

Now we can Align using Quick Align

1. From the Alignment tab, choose Quick Align (Figure 5-23). Or simply right-click on the instrument you want align and select **Locate>Quick Align to CAD.**

Figure 5-23. Quick Align to CAD button in the Ribbon Menu



When choosing points for the next dialogue, it is important to remember that these points must fully constrain your model. This is easy to envision for a model that has several flat and perpendicular sides, but can be more challenging for models with an organic shape. You should still think of this type of alignment as a 3-2-1 alignment. First, pick three points on the primary interface. Next, pick two points on the secondary interface, and finally at least one point on the tertiary interface of the CAD model. You can think of this as a plane, line, point alignment.

Another important factor to consider when choosing the alignment points is whether your instrument will be able to measure those points in its current physical location to the part. For a laser tracker, you should consider line of sight restrictions, and for a portable CMM arm, you should consider the reach of the instrument to ensure that all points you select on the surface will be able to be measured. If there are features or surfaces of the part which are critical, adjust the location or orientation of your instrument to ensure that those can be measured.

The minimum points required for a quick align is 6 points. Again this is related to the 3-2-1 method of alignment. However, there is NO maximum number of points. The more points you choose in the alignment, the more confidence you will have in the alignment accuracy.

2. When prompted, pick 3 points on the top surface of the base model (to define a plane), then two points on the front of the Vertical face (forming a line), and finally a point on the left side (Figure 5-24).

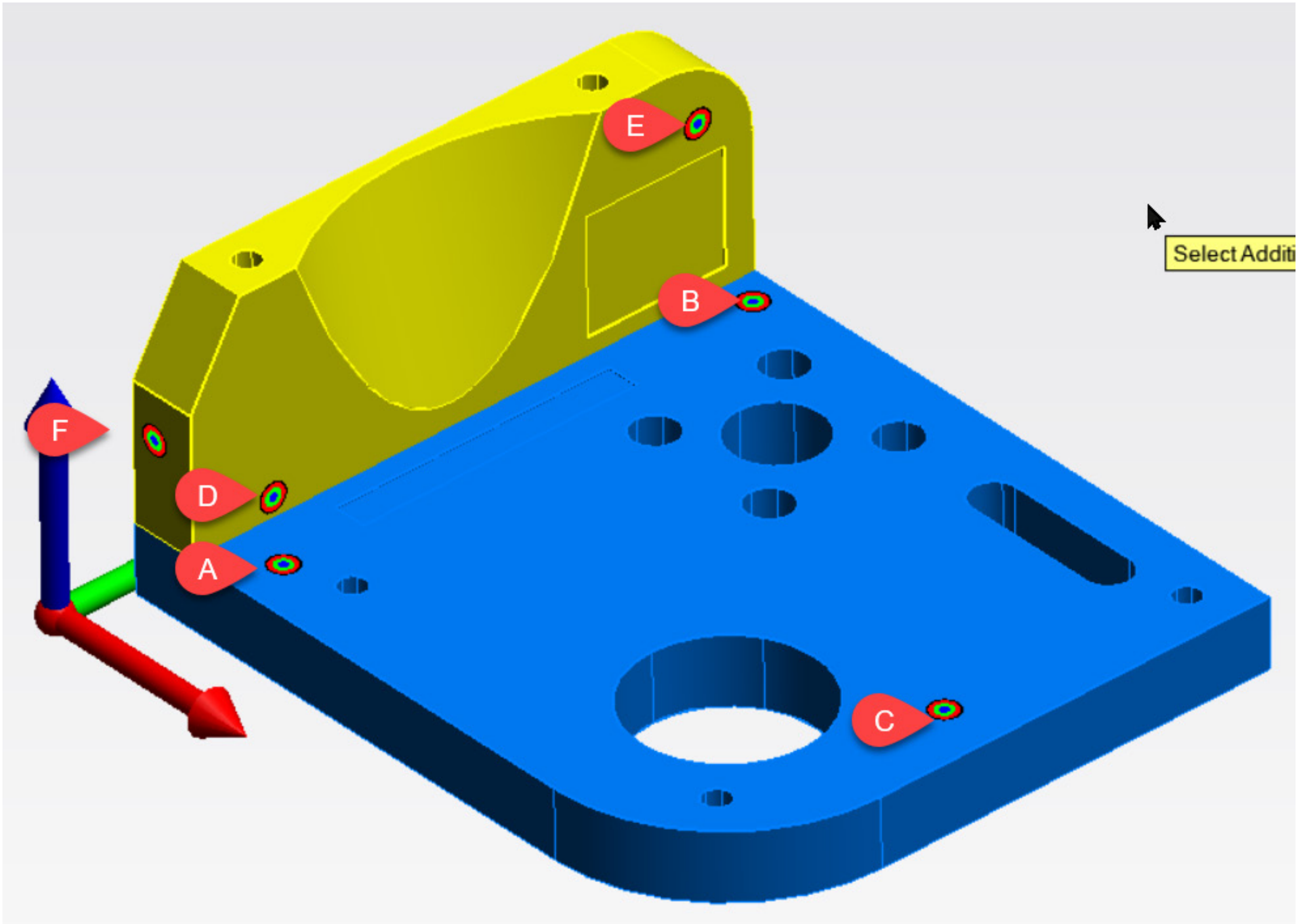


Figure 5-24. Quick Align Points

In this exercise, these are the points chosen. Note how they completely constrain translation in X, Y, and Z as well as rotation in Rx, Ry, and Rz. The order of selection is also shown. The points don't necessarily have to be measured in any particular order, but, from an inspector's standpoint, it helps to group the points together by area because the points must be measured in the order selected.

3. The next step is to directly measure these points on your cad model. If you have already measured these points you can also pick the measurements associated with each alignment point (Figure 5-25).

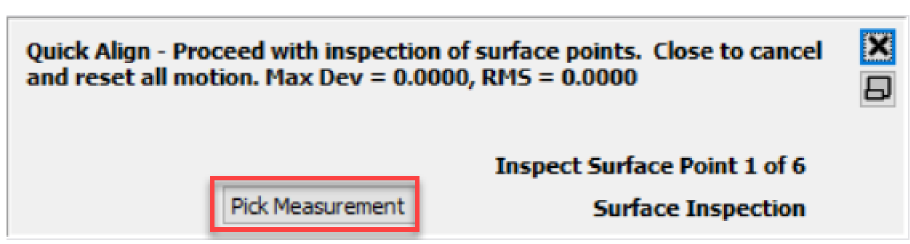
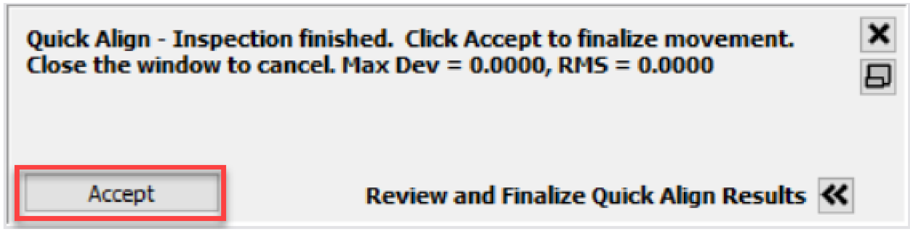


Figure 5-25. Picking Measurements that have already been taken as an alternative to direct measurement

In this case, instead of measuring the points, select the 'Pick Measurement' button and click on the inspection points in order. The points are in order from p0 to p5. Choose each one in order by double clicking on them in the tree or single clicking on them in the graphics window.

4. You should see the results below (Figure 5-26). Choose Accept to confirm alignment

Figure 5-26. Quick Align results showing a perfect fit with 6 points, the minimum for a 3-2-1 alignment.



Feature Inspection

CHAPTER 6

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Feature Inspection

Geometry Construction verses GR-Features

SA uses Geometry Relationships (GR-Features) to effectively link nominal and measured geometry with the measured points creating dynamic geometry ready to report. These features can be built in advance or after measurements are made and, once built, provide a template for future measurements. Static geometry construction is still an option but has generally been replaced by feature based measurement.

Why GR-Features verses GD&T

Geometry can also be inspected using our ASME and ISO compliant Standardized GD&T inspection process either separately or in combination with feature measurement. The primary reason these evaluation processes are separate in SA are:

1. Standardized GD&T feature checks use a built in datum alignment behind the scenes for each check. GR-Features display deviations using the current job alignment. What you see is what you get which lends its self to greater clarity.

2. GD&T feature use standardized inspection processes that are designed to establish if a part is within specification. It assumes measurements are perfect any deviations are reflections of the part. This makes the results more dependant on outliers in the data set. GR-Features use an RMS fit to help identify measurement error.

As a general rule of thumb start with GR-Features which help identify measurement errors and are more flexible. Then move to GD&T inspection for part inspection following a standard protocol.

Index

Workflow Reference Guide:

The following reference for inspection are available:

Tutorial Index:

Feature Inspection

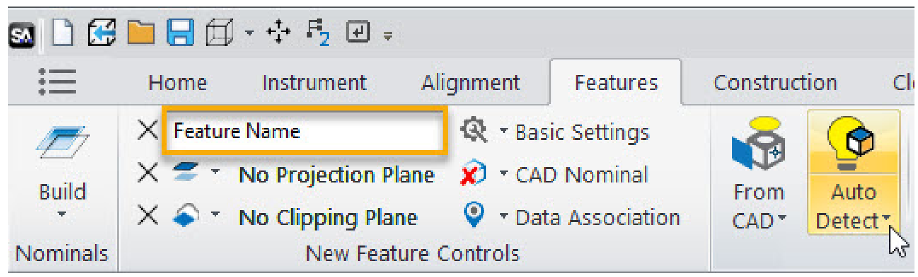
Workflow Reference Guide

How to Measure Features (Auto Detect) Using a Probing Device.

SA can automatically detect what feature type you are measuring using the points you measure. To begin:

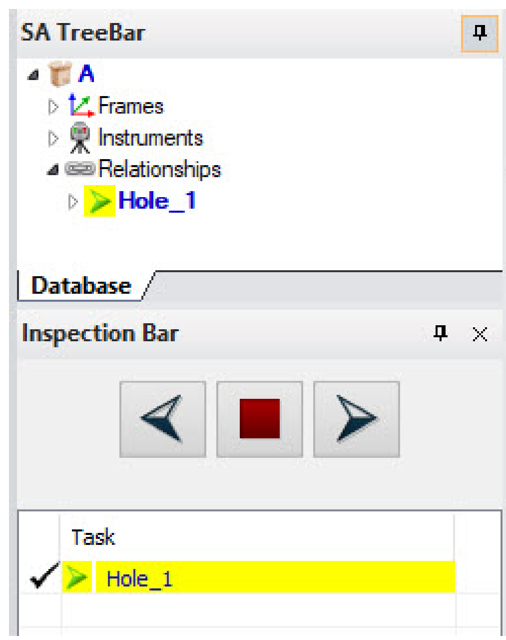
1. Connect your instrument.
2. Navigate to the Features Tab of the Ribbon, enter a name for your new feature and then add an **Auto Detect Feature** (Figure 6-1).

Figure 6-1. Auto Detect Feature



When you add the **Aut Detect** feature to the job, trapping will begin like this (Figure 6-2). You will see trapping indicated both in the SA Tree bar and in a popup Inspection Bar, which also gives you navigation controls. Settings for this behavior are on the Home tab of the ribbon under **Inspection Options**.

Figure 6-2. Trapping has begun



3. When trapping is active measurements will be captured and used by the selected feature. When you have measured all the points you want for this feature press the next button 

When you press next SA will cycle through the available fit options and settle on the simplest feature it can find that fits with an RMS (root mean squared deviation) under the Auto Fit Tolerance you have set in the **Auto Detect** feature settings(Figure 6-3).

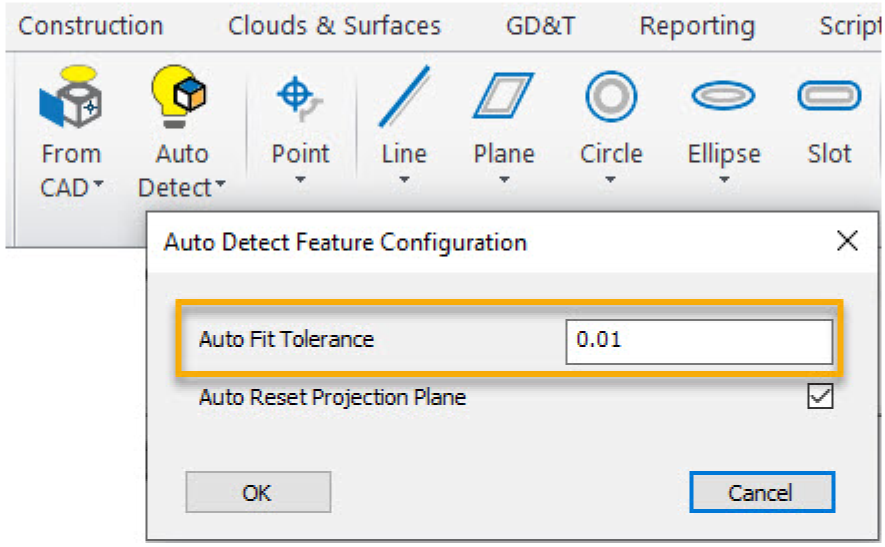


Figure 6-3. AutoDetect feature Settings

A new placeholder feature will also be added for you so that you can continue to measure (Figure 6-4). An audible beep will sound when this new feature has been added and is ready to be measured (which can be helpful when the screen is hard to see).

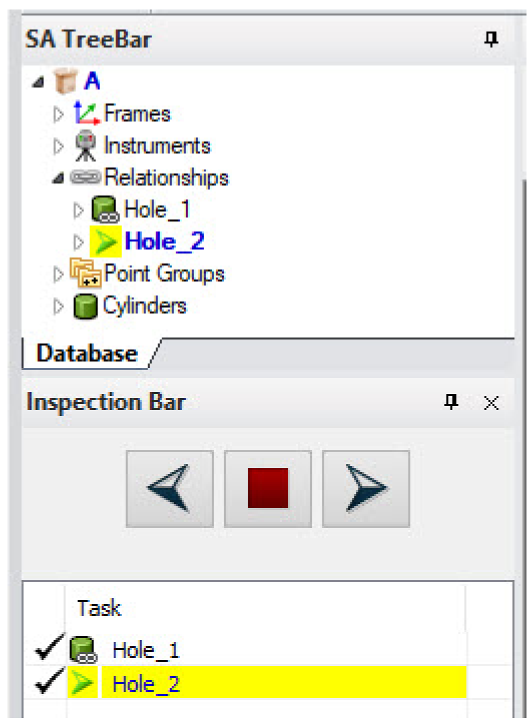


Figure 6-4. Measuring the next feature

Continue to measure pressing **Next** after each feature has been measured.

Troubleshooting/Additional Details

If you measure a plane with the **Auto Reset Projection Plane** option enabled (Figure 6-3) then the newly measured plane will become the active projection plane. This can be seen in the Features Tab and can be adjusted from there if need be (Figure 6-5).

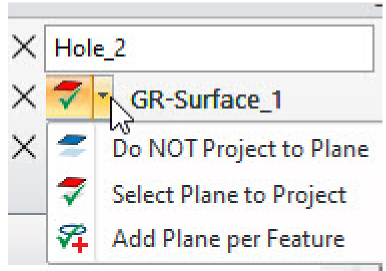


Figure 6-5. Projection Plane Control

Should the Auto Detect Feature fail to resolve a fit within the RMS tolerance you have set it can be manually assigned through the right-click menu(Figure 6-6).

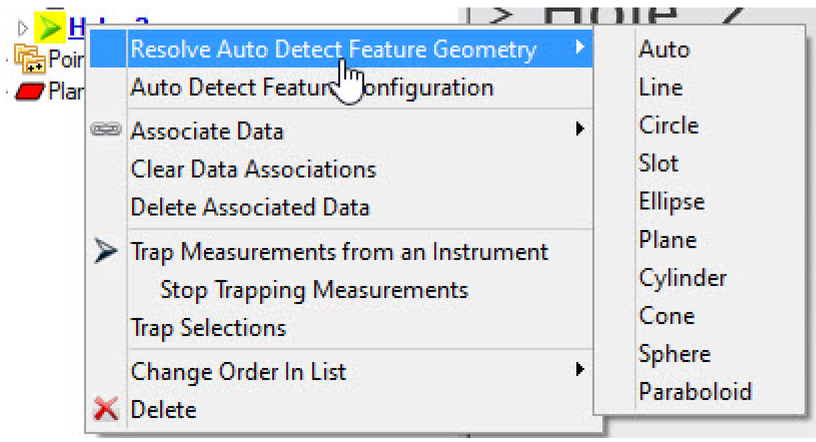


Figure 6-6. Assigning the Feature type within an Auto Detect Feature

Alternatively if a solution was found but it was not the desired geometry then the newly built geometry's type can be changed through its right-click menu .

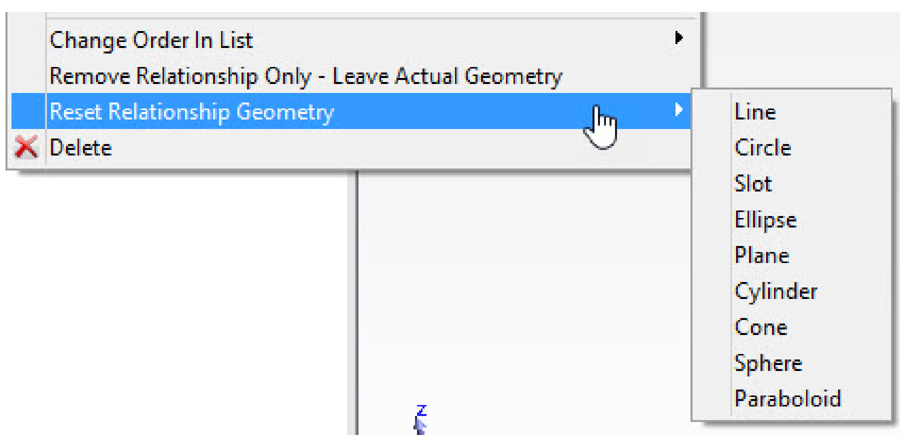


Figure 6-7. Correcting the geometry type.

Building Geometry from Existing Measurements.

1. Navigate to the Features Tab of the Ribbon and enter a name for your new feature in the Feature Name entry field.

2. Choose a selection mode (points, or groups) from the drop down list(Figure 6-8).

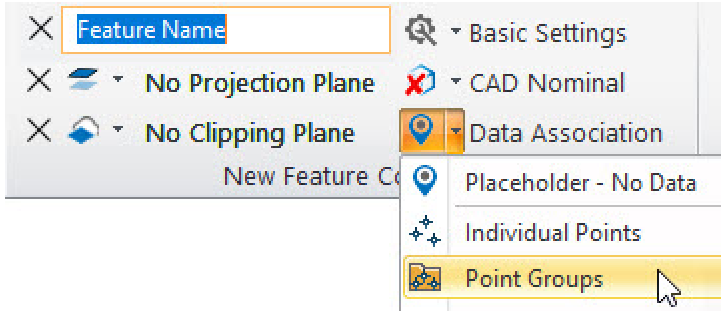


Figure 6-8. Preparing to build a feature

If you would like to build a 2D feature like a circle or slot projected to a plane, the projection plane should also be set (this is not necessary when building geometry like planes and cylinders). A projection plane can be selected from the drop down list under the feature name field, if it already exists, or you can build a new one first by selecting that option from the Plane's drop down list(Figure 6-9).

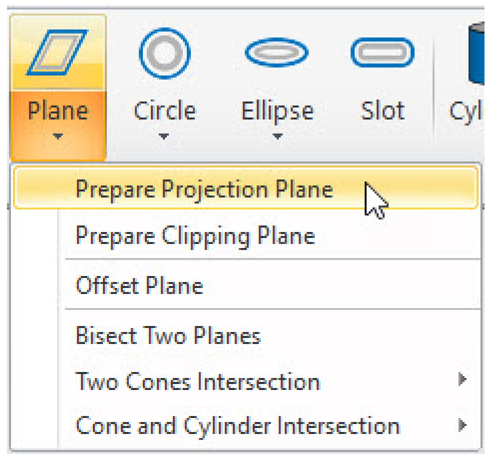


Figure 6-9. Prepare Projection Plane

3. Select a feature type to build. This new feature will take the name you enter and then prompt you to select. You can select a feature type directly or use the Auto-Detect feature to have SA figure out a desired feature type (Figure 6-10).

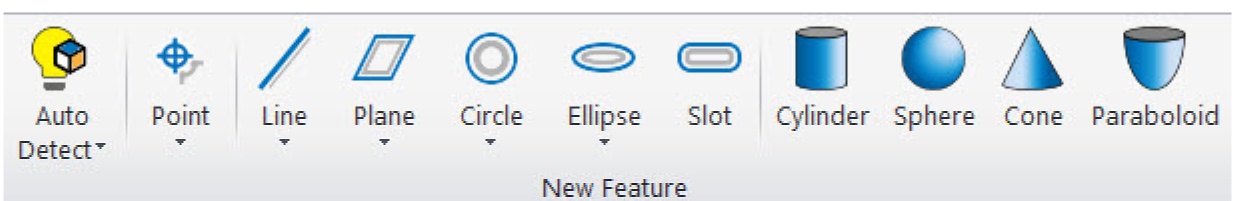


Figure 6-10. Features list you can build

SA expects you to define the feature you want to build before you begin selecting anything. When you add the feature by clicking the desired feature button, a feature of the type you select will be added to the tree using the name you entered. It will then prompt you to select either Individual Points or Point Groups (all the points in a group) based upon your selection in the drop down (Figure 6-8).

Selection can be performed in multiple ways:

- You can single click on items in the graphics
- You can hold down the Shift key + Left click and hold with the mouse button to drag a dotted rectangle around the items you want to select.
- You can also select items in the Tree Bar Select Items panel.

Troubleshooting/Additional Details

If you forget to enter a name, the feature can be renamed through its properties. These can be accessed by double clicking on it the tree or right-clicking and selecting properties. Then the Rename button.

If you forget to set the Data Association drop down menu the feature will be created but you will not be prompted to select anything. To fix this you can right-click on the placeholder geometry and select **Associate Data** (Figure 6-11).

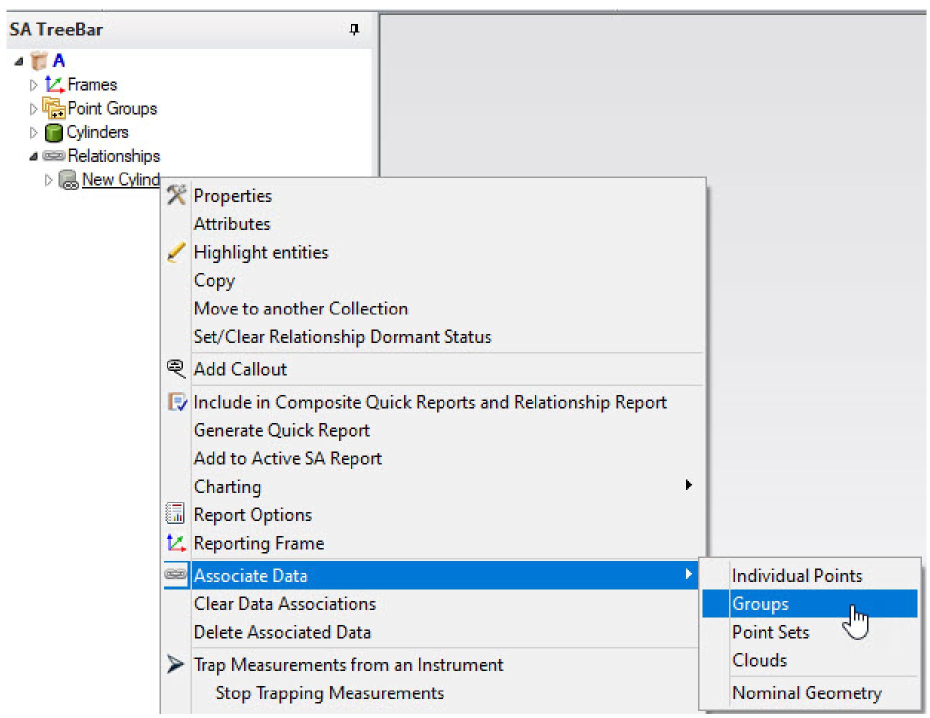


Figure 6-11. Direct Data Association.

Building Center Points and Intersections from Existing Geometry.

Lets say you have a patter of 4 holes you have already measured. You would now like to fit geometry, build the center points from each hole, and determine the diameter of a circle fit through those center points (Figure 6-12). The following are the steps to do so:

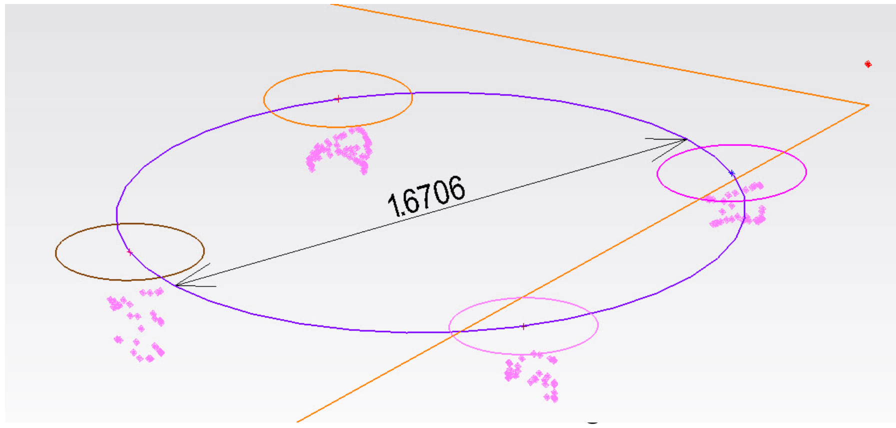


Figure 6-12. Pattern of circle with a circle fit through the center points.

1. Build a projection plane. Circles and slots are typically measured by placing a probe inside the hole and measuring the wall of the feature. Therefore the first step is to build a projection plane that will define the plane for the circles. To do so, first enter a feature name, select the data to use, and add a feature like this:

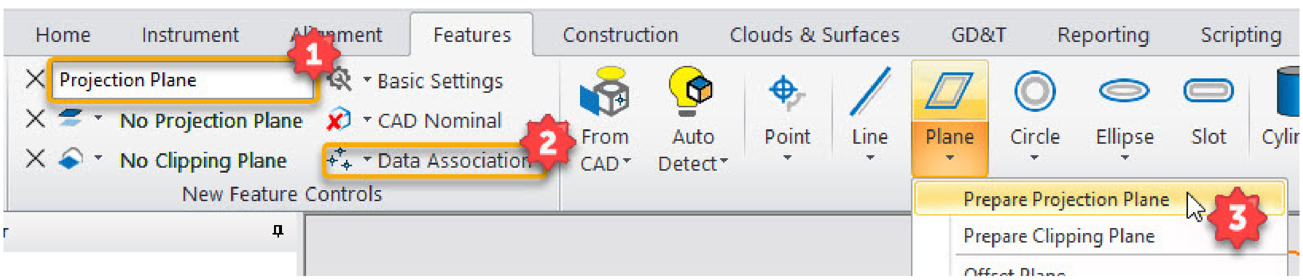


Figure 6-13. Steps to build a projection plane

2. Now you can build circle by editing the name and then selecting the circle icon, and then selecting the points to use in the circle fit.

Don't be surprised if you fit a circle as a pin rather than a hole, we can fix that later. As you can see below, the circle is projected to the plane and fit as a pin inside the measured points, compensating automatically for the point offsets (Figure 6-14).

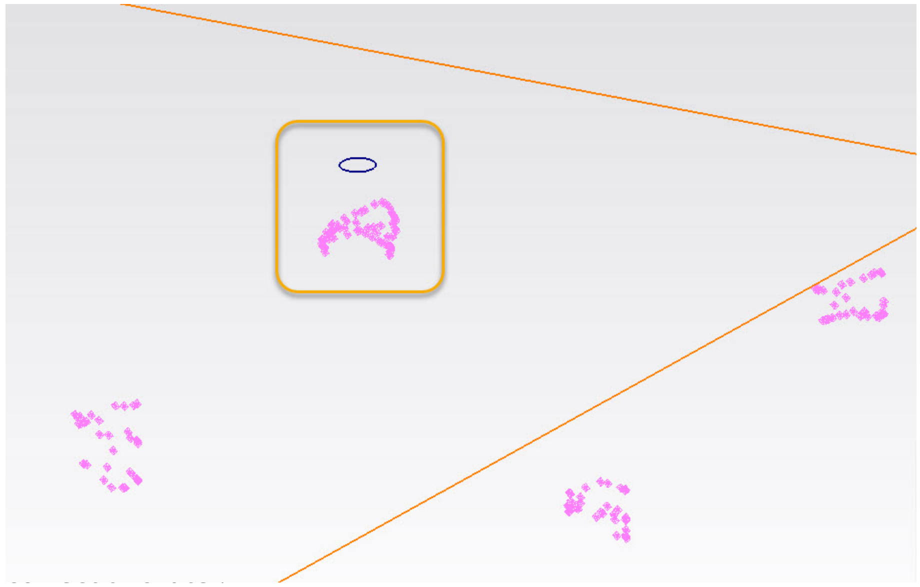


Figure 6-14. First circle

3. Go ahead and build 3 more holes.
4. Open the properties of one of the circles and fix the geometry fit settings, such that the points define a hole measurement (Figure 6-15).

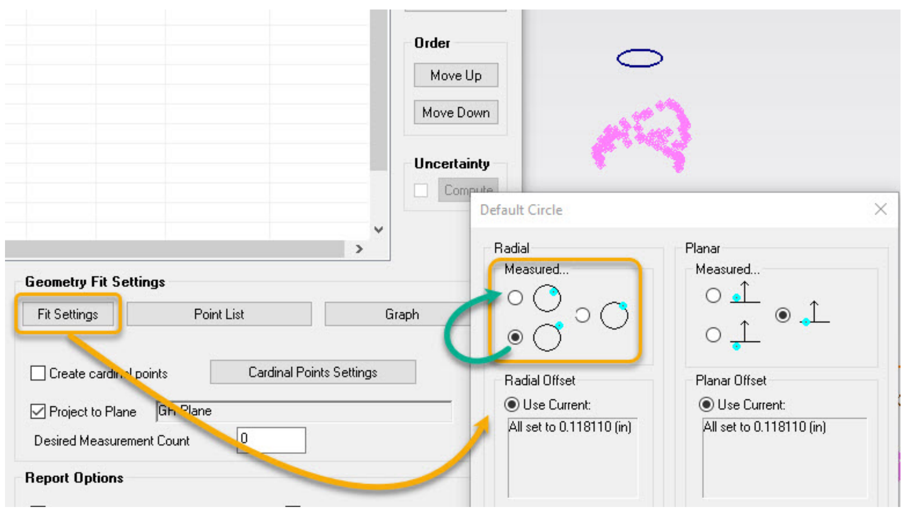


Figure 6-15. Changing the Geometry fit settings from pin to hole

5. Turn on the **Cardinal Points**. The circle center points are included as part of the cardinal points. To turn them on first edit the settings as desired (in this case you probably want to turn off the point on normal) and then check the box to turn them on(Figure 6-16).

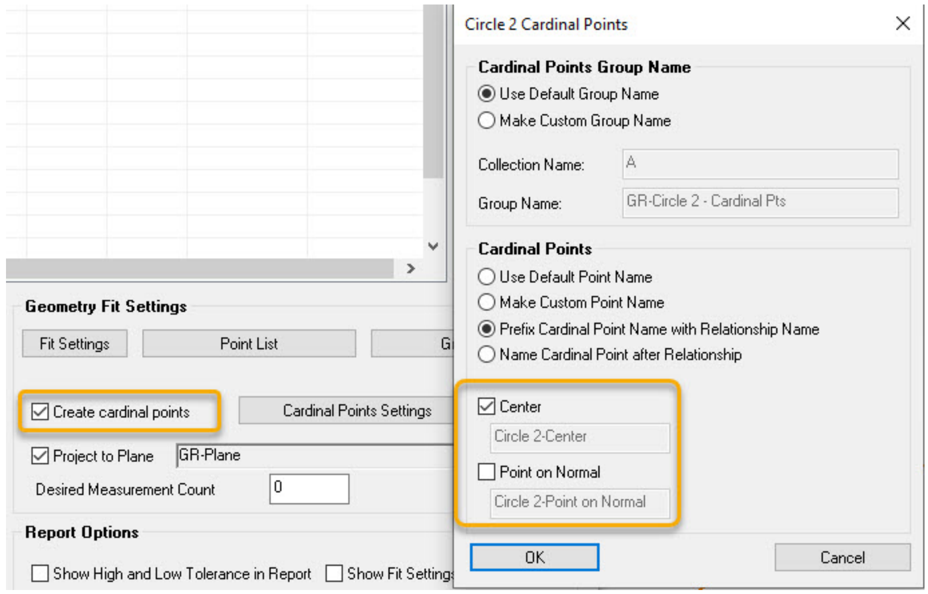


Figure 6-16. Enable the center cardinal point

6. Apply your settings to the other circles. As soon as you make changes to a circle you will see the graphics update accordingly. You can apply your settings to other circles using the Apply to Selected Relationships Button (Figure 6-17).

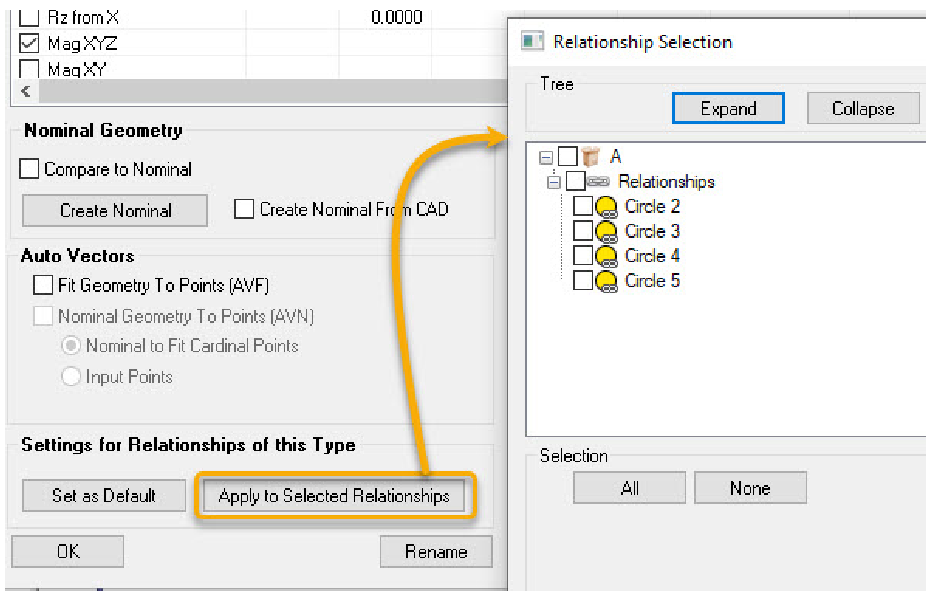


Figure 6-17. Applying your settings to the other circles.

You will see a fairly complicated dialog offering you the ability to select only particular settings to apply, but just select all and your settings will be applied to the other circles (Figure 6-18).

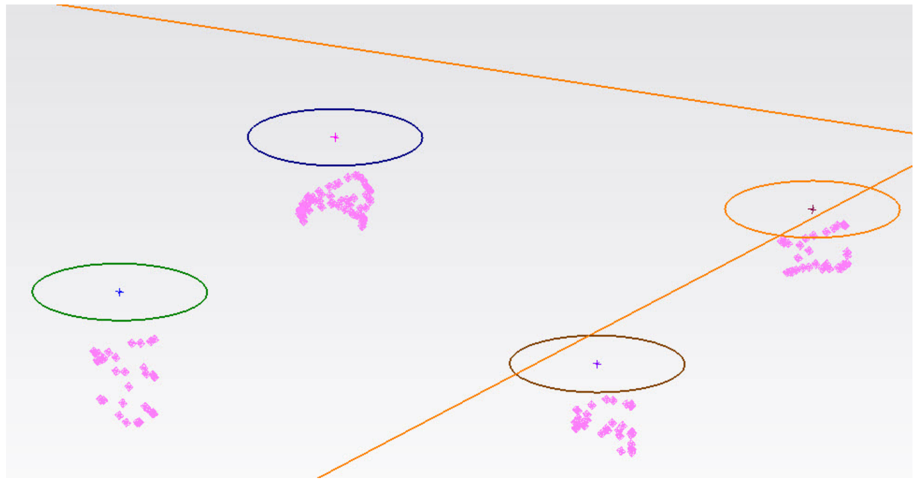


Figure 6-18. Final fit circles.

7. Build an additional circle through the existing circle centers. This can be done by selecting the center points but there is an easier option. Select **From Multiple Circle** from within the circle drop down list. Geometry intersections of all types can be found in these feature drop down lists (Figure 6-19).

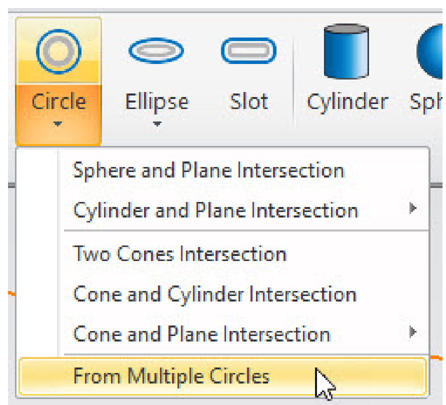


Figure 6-19. Selecting Circle from Multiple Circles

8. When prompted, select all 4 circles.

Note that from the tree you cannot select the GR-Feature under the relationship category directly, but you can expand it to select the measured circle within. GR-Features tie together all the contributing pieces so that you can easily build reports(Figure 6-20).

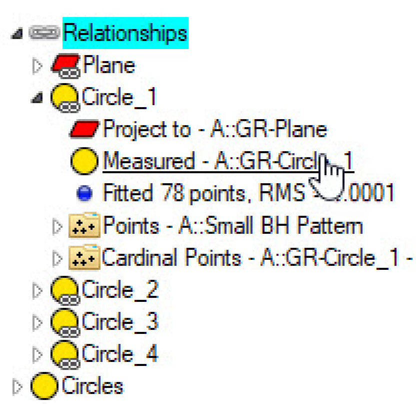


Figure 6-20. Selecting the measured circle within the GR-Circle Feature

9. Finally, add a diameter dimension to the circle you just created by navigating to the Reporting tab and selecting the diameter dimension from the dropdown list

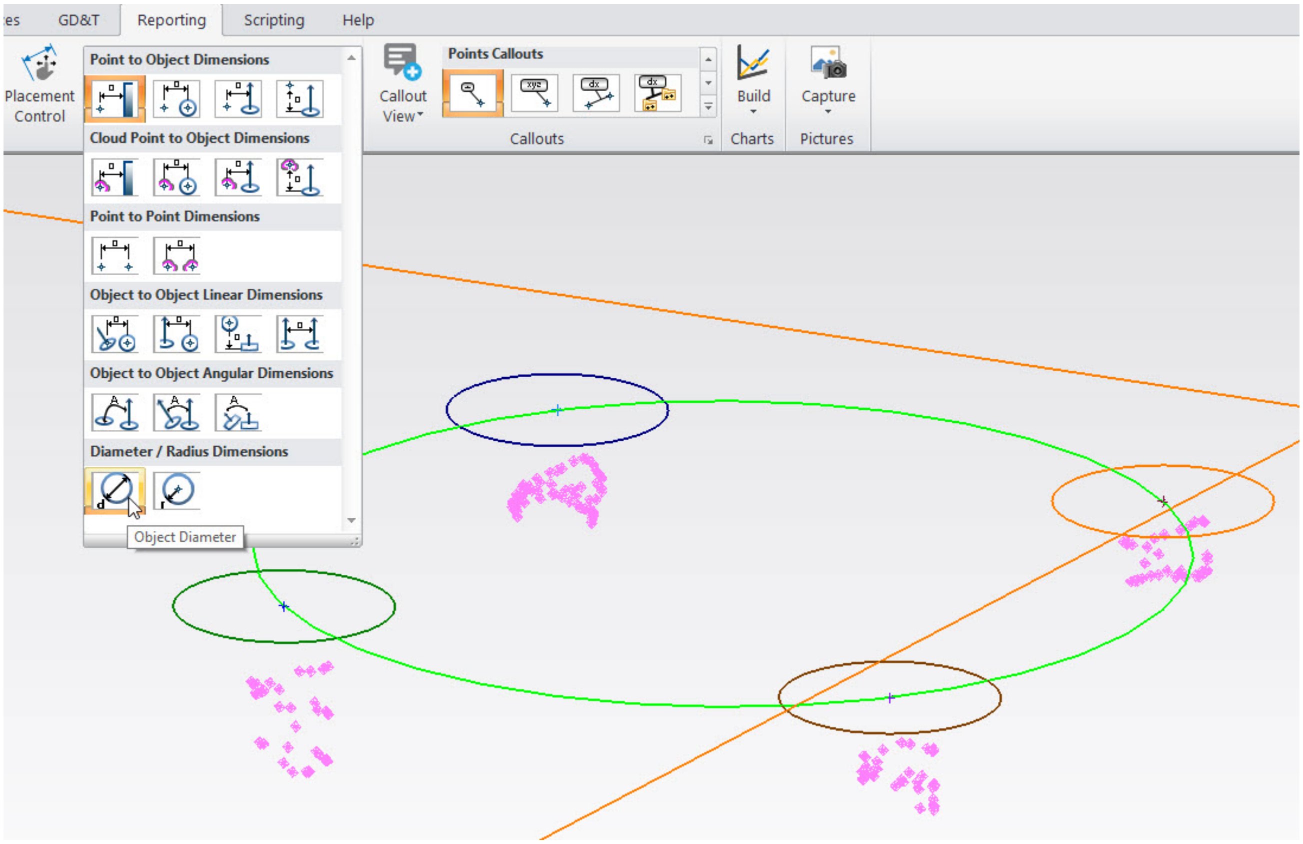


Figure 6-21. Adding a diameter dimension to the circle from multiple circles

How to build a coordinate system using measured features

Taking the prior example, lets build a new coordinate frame from the projection plane and this circle patter (Figure 6-22).

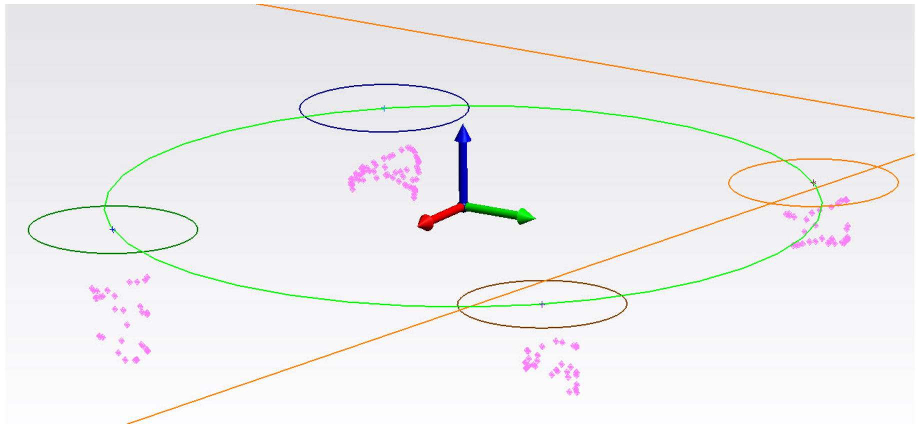


Figure 6-22. Coordinate Frame built from a bolt hole pattern

To do so, do the following:

1. Switch to the Features Tab of the ribbon, enter a name for this new frame and then press the Frame button like this (Figure 6-23).

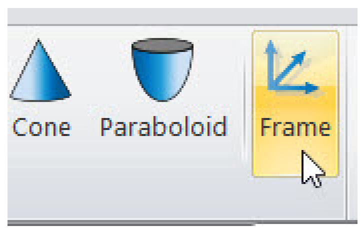


Figure 6-23. Adding a frame.

Frame are defined based upon a 3 inputs which can be entered in any order:

- **Origin.** This defines the base coordinate of the frame
- **Primary Axis.** This axis is help perfectly to the selected reference
- **Secondary Axis.** This axis is fit as well as is possible to the reference while maintaining the primary axis alignment.

2. Lets pick the origin first. In this case, we want to build a frame centered on the circle through the patter, or the center of the patter. To do so use the Select Object button and select this circle as the Origin like this(Figure 6-24).

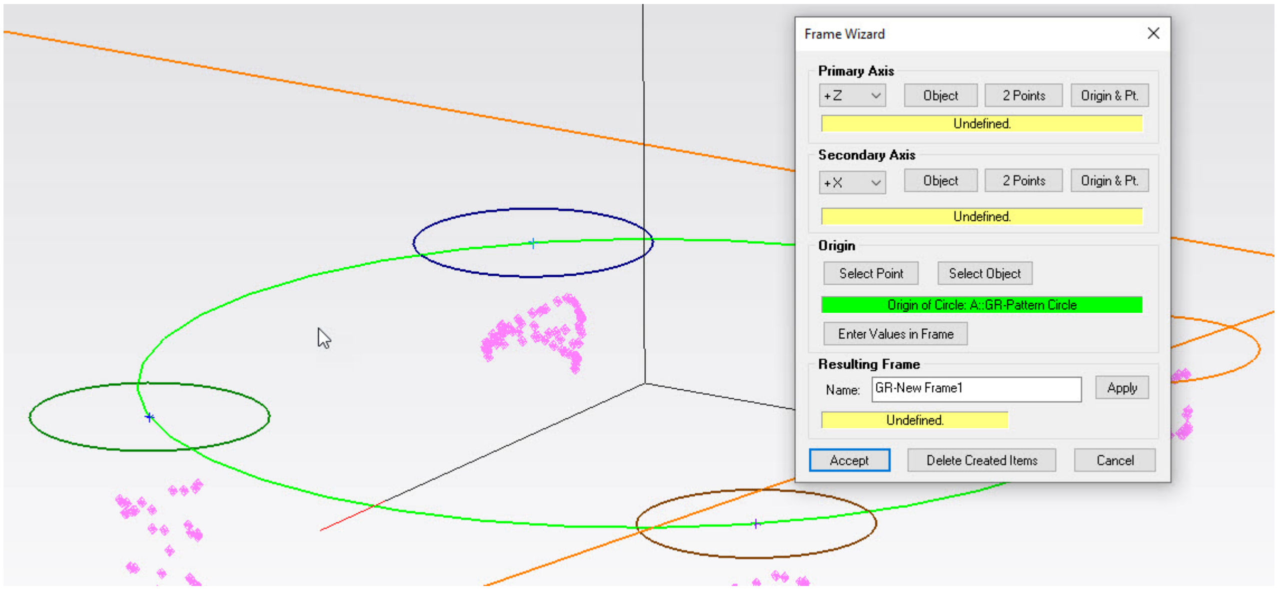


Figure 6-24. Selecting center circle as the reference for the new frame.

3. Next lets define the Primary Axis. In this case everything is being projected to the same plane, so lets use that. To do so select Object and select the projection plane.

With each new selection the wire frame placeholder frame will update give you a preview of the frame you are creating(Figure 6-25).

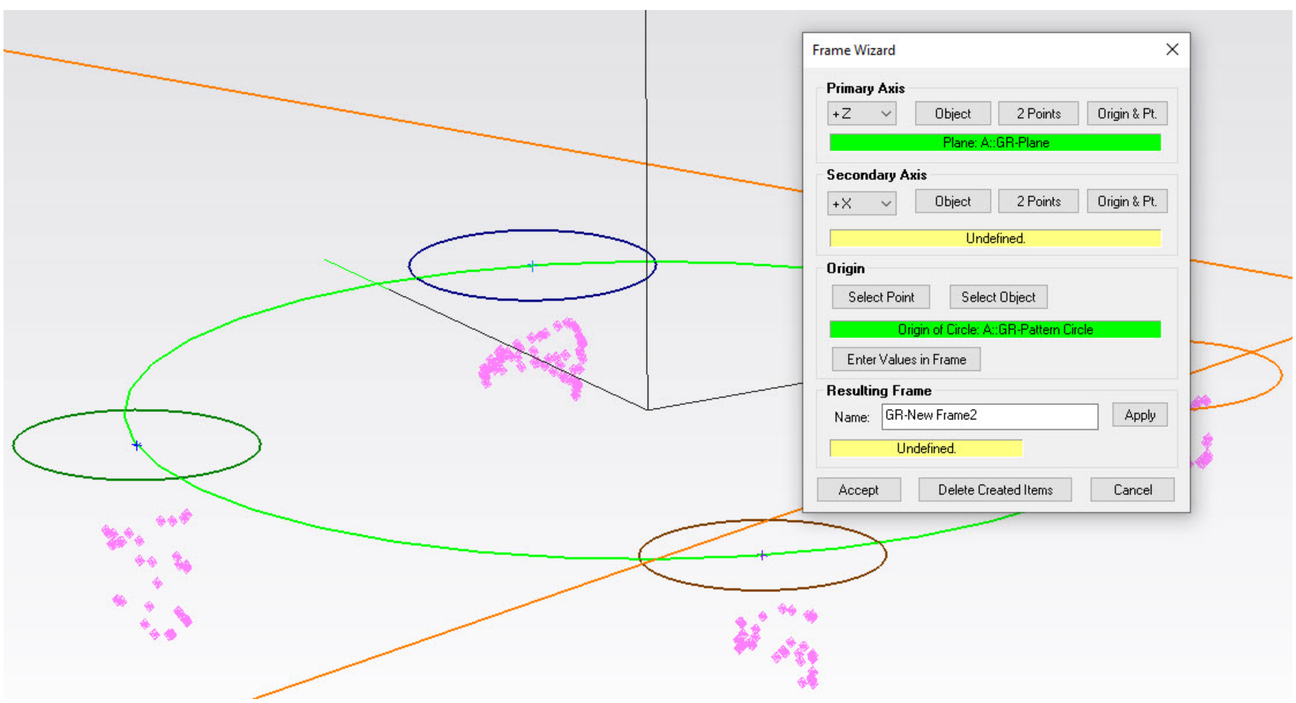


Figure 6-25. Origin and Primary Axis selection

If you notice, the definition for each axis can be changed. The primary axis is +Z by default but it can be edited as desired(Figure 6-26)

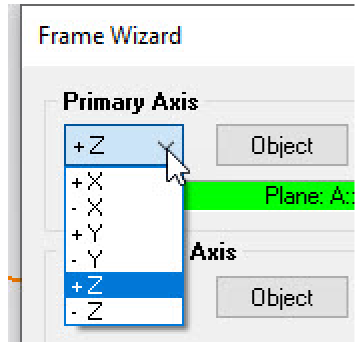


Figure 6-26. Axis definition

4. Finally we want to clock the frame such that the secondary axis points from the center point of the first circle to the 4th circle. Select the 2 Points button and select the circle points in the order you want them to be used.

If you are defining the +X axis for example select the first point as the start and the second point as the direction from that starting point.

5. Accept the frame and pick a nominal reference frame if you wish. Doing so will provide a comparison between the frame for reporting purposes.

How to measure a set of reference points.

Another common measurement task is to measure a set of reference points using a probing device. This could be any set of defined point coordinates exported from another file or saved in a text file.

To begin lets import a set of points from an ascii file.

1. Drag and drop an ascii file with your nominal points into SA.

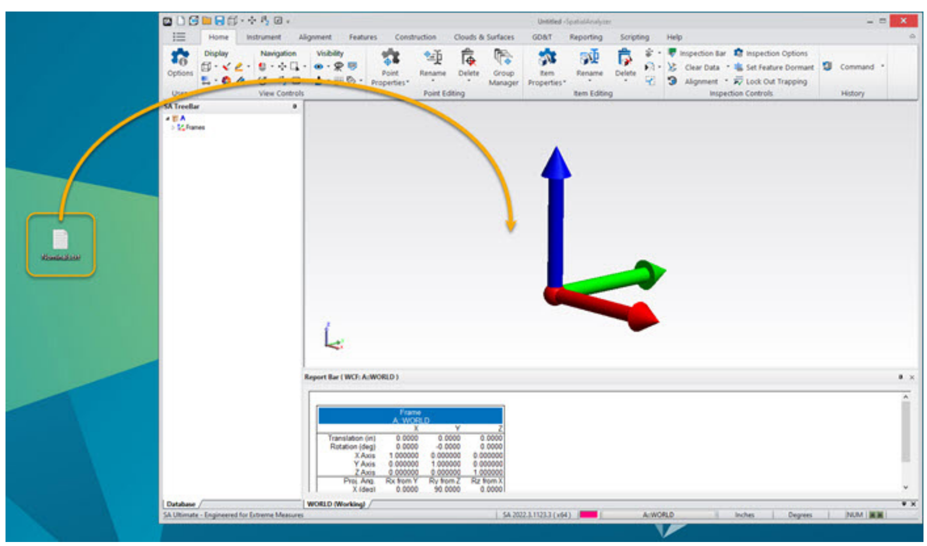


Figure 6-27. Importing a file into SA using drag and drop.

This can be any *.txt file with a standard format such as Point Name, X,Y,Z format. When you import it, our Ascii import dialog will open and you can choose the format that matches your file (Figure 6-28).

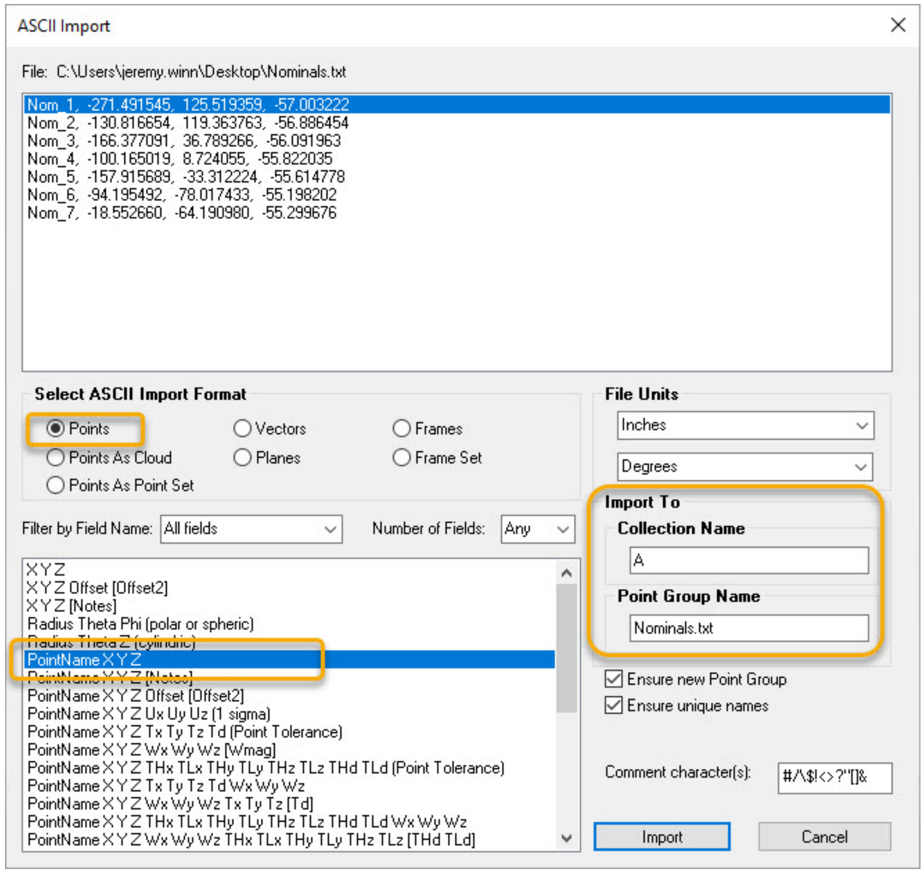


Figure 6-28. Ascii Import

2. Select the Points option for what you are importing, the format your file is in from the list of formats, and double check the collection and group names as to where you want the data to be stored.

3. Navigate to the Features tab of the ribbon and build a **Group to Nominal Group** relationship from this new point group(Figure 6-29).

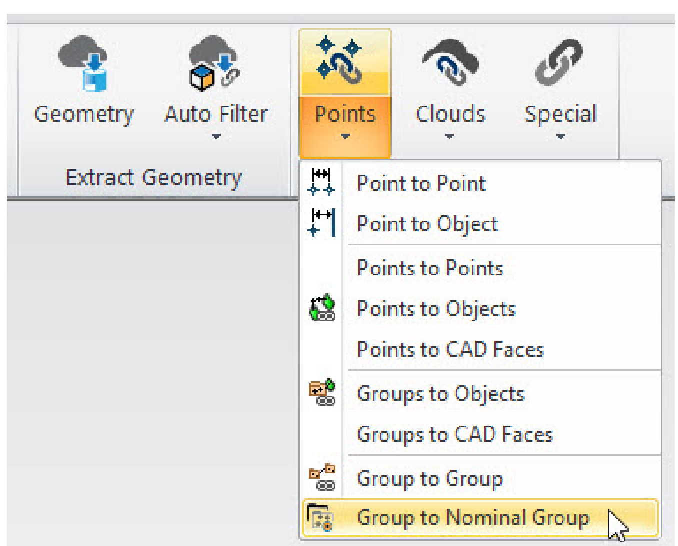


Figure 6-29. Build a Group to Nominal Group relationship

This will build a relationship or dynamic link between the imported created group you selected and a new placeholder group that is ready to be measured.

4. Double click on the relationship in the tree to open its properties, and select the options you would like to use, closing the properties dialog when satisfied.

Group to Nominal Group relationships offer a set of options which can be enabled while you measure for greater user feedback. These are optional and include:

- **Use closest point to match new measurements.** This option is on by default and will rename the measured points to match the nominal points based on proximity
- **Display closest point watch window.** A watch window will open when you start trapping that shows the distance to the closest point.
- **Use view zooming and proximity.** This option enables the view tracking and zooming as you get closer to the reference point.
- **Ignore points beyond threshold.** This option prevents you from measuring if you are further than the preset distance from the reference point.
- **Update a vector group with relationship.** This option creates auto-vectors to show you how far off your measurements are from the reference after you have taken the measurement.

5. Verify your instrument is connected and ready to measure.
6. Right-click on the relationship in the tree and select **Trap Measurements from an Instrument** (Figure 6-30).

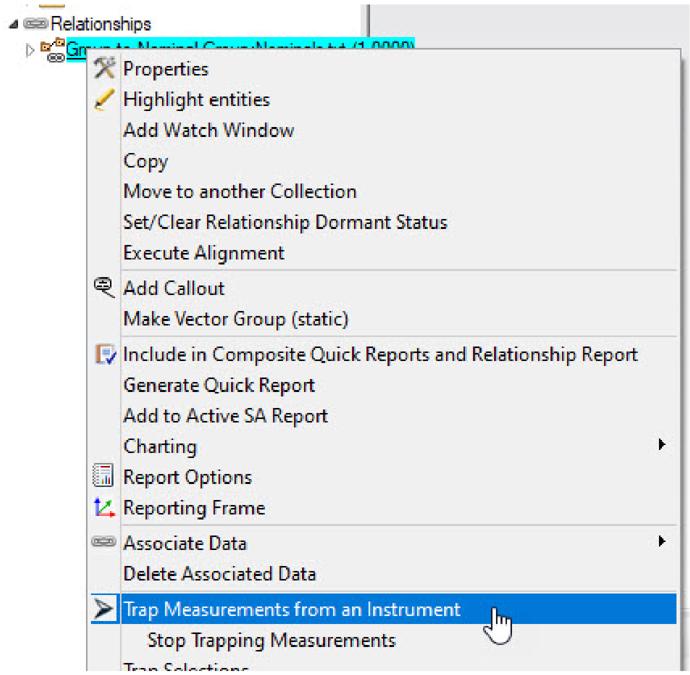


Figure 6-30. Start Trapping

Measure the first 3 points from the list to locate the instrument. If you are roughly aligned and measure the points in the same order as the reference group then you can continue measuring. But if not,

Tutorials

Creating Features from Existing Points

This tutorial is set up to be a broad overview of Feature construction possibilities using the Features tab of the Ribbon menu.

To begin by opening a sample file to work with.

1. In the Ribbon Bar, go to the **HELP** tab. In the **Documentation** section, choose **Sample Files**.
2. Open the SA file GD&T `with Multigage.xit.`
3. Go to the **HOME** tab. In the **View Controls** section, select the **Solid** render mode.
4. Go to the **Visibility** section and select **Hide Selected Items**, selecting all the *Annotations*. We will be inspecting without GD&T today.
5. Next, go to the **FEATURES** tab.

CREATING FEATURES WITH EXISTING POINTS

6. In the New Feature Controls section.
7. In the white box type "Datum A", which will be the name of the created feature.
8. Change the Data Association, by selecting , then choosing .
9. Select , then select the Datum A point group which will result in the creation of a plane.
10. Press Enter to confirm your selection. Our graphical view should look like (Figure 6-31).

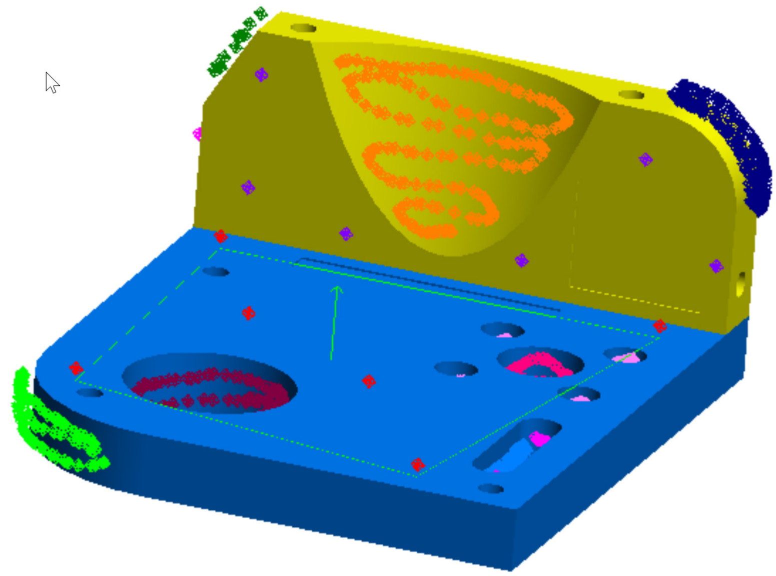


Figure 6-31. Plane Feature Creation.

11. Return to the Ribbon Bar and in the white box and type "Large BH".

12. Select , then select the Large BH point group which will result in the creation of a cylinder.

CHECKING FIT SETTINGS AND CREATING CARDINAL POINTS

13. Double-click on the Large BH relationship in the SA Treebar and follow the steps in Figure 6-32 (below):

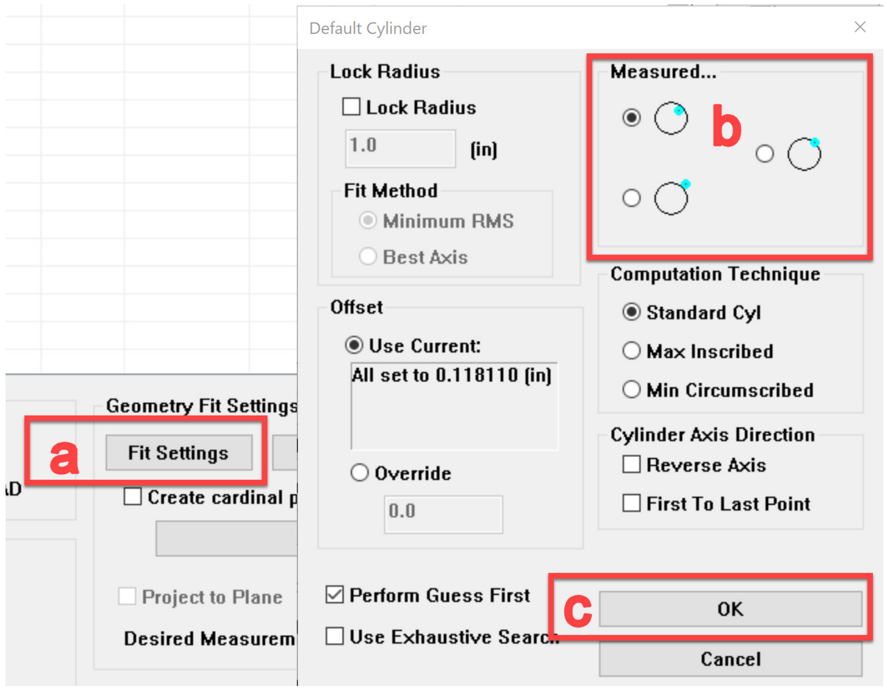


Figure 6-32. Changing the measured side of a cylinder.

a. Select Fit Settings.

b. Choose the first option in the section labeled **Measured**….

c. Click Ok.

Notice the diameter value will increase since we have just changed the measured side of the cylinder from outside to inside (Figure 6-33).

Criteria	Nomi...	Meas...	Delta	Low ...	High ...
☑ Diameter		1.5754			

Figure 6-33. The diameter value reported for the criteria.

14. Follow the steps in Figure 6-34 (below) to set the cardinal points for the Large BH cylinder.

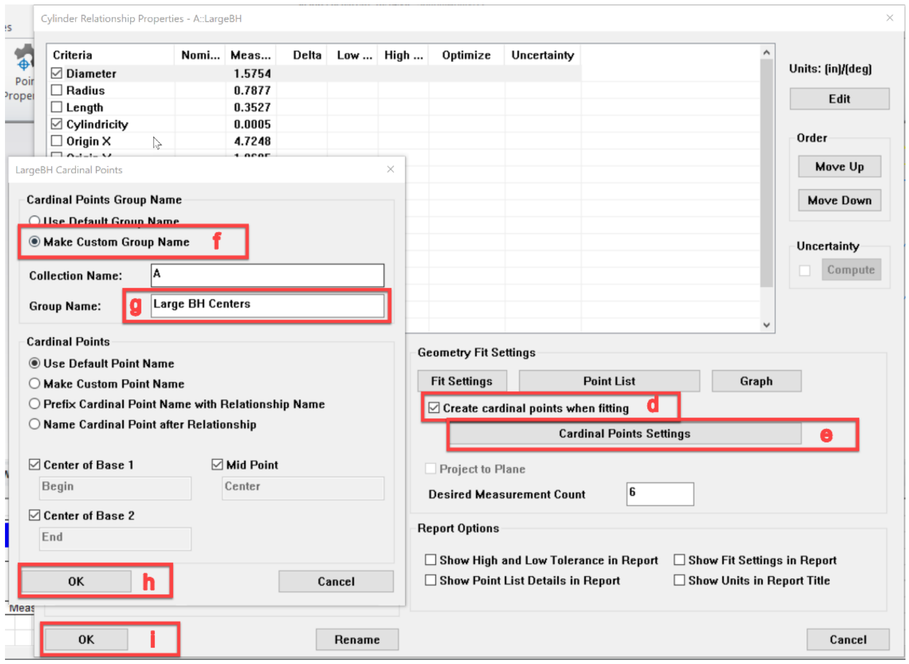


Figure 6-34. Creating Cardinal Points.

d. Select **Create cardinal points when fitting** checkbox.

e. Click Cardinal Points Settings.

f. Select the radio button next to Make Custom Group Name.

g. Then type "Large BH Centers" in the Group Name: box.

h. Click Ok.

i. Click Ok.

SETTING A PROJECTION PLANE

This will set a geometry exactly in the plane it is getting projected to. We are going to make four circles and then project them to our Datum A plane.

15. Return to the Ribbon Bar and in the white box and type "Pattern BH1".

16. Change the **Data Association**, by selecting

17. Select , then select the points designated in Figure 6-35. This can be done by holding down the shift key and simultaneously holding down the mouse left click and dragging a box around the desired points.

18. Then hit Enter.

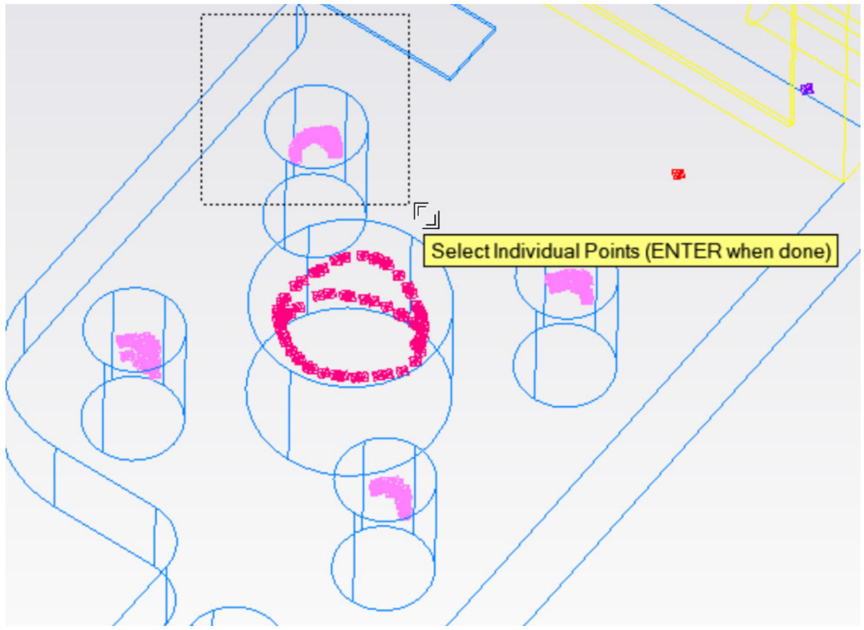


Figure 6-35. Selecting point for circle creation.

19. Repeat steps 14 – 17 while moving to the next hole "Pattern BH2", and so on until four circles are created.

(The newly created circles may be oriented incorrectly, which is why we need to project them to a plane.)

20. Double-click on the Pattern BH1 relationship in the SA Tree-bar and follow the steps in Figure 6-36 (below):

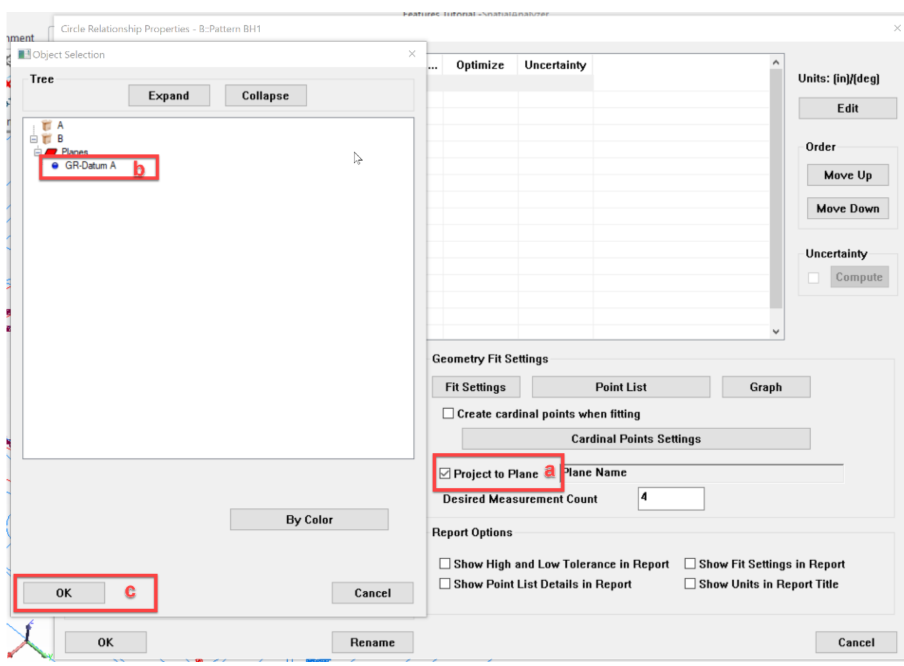


Figure 6-36. Setting a projection plane for circles.

a. Check the box next to **Project to Plane**.

b. Single left-click GR-Datum A to select the projection plane.

c. Click Ok.

APPLYING TO SELECTED RELATIONSHIPS

The settings of one geometry relationship can be applied to other relationships of the same type. We will apply the projection plane of Pattern BH1 circle to the rest of the holes in the pattern. Follow steps in Figure 6-37 and Figure 6-38.

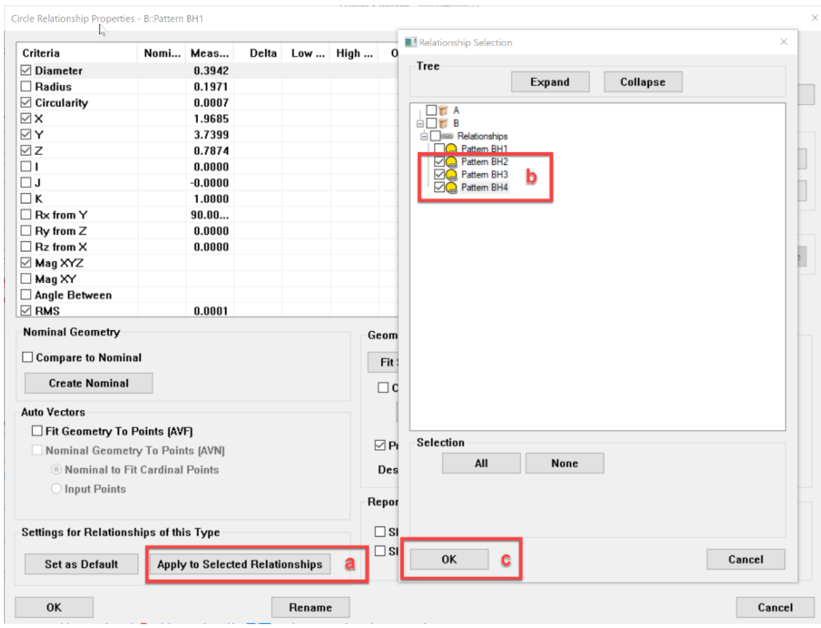


Figure 6-37. Applying settings to selected relationships.

a. Click Apply to Selected Relationships

b. Check the boxes next to the other circle relationships.

c. Click Ok.

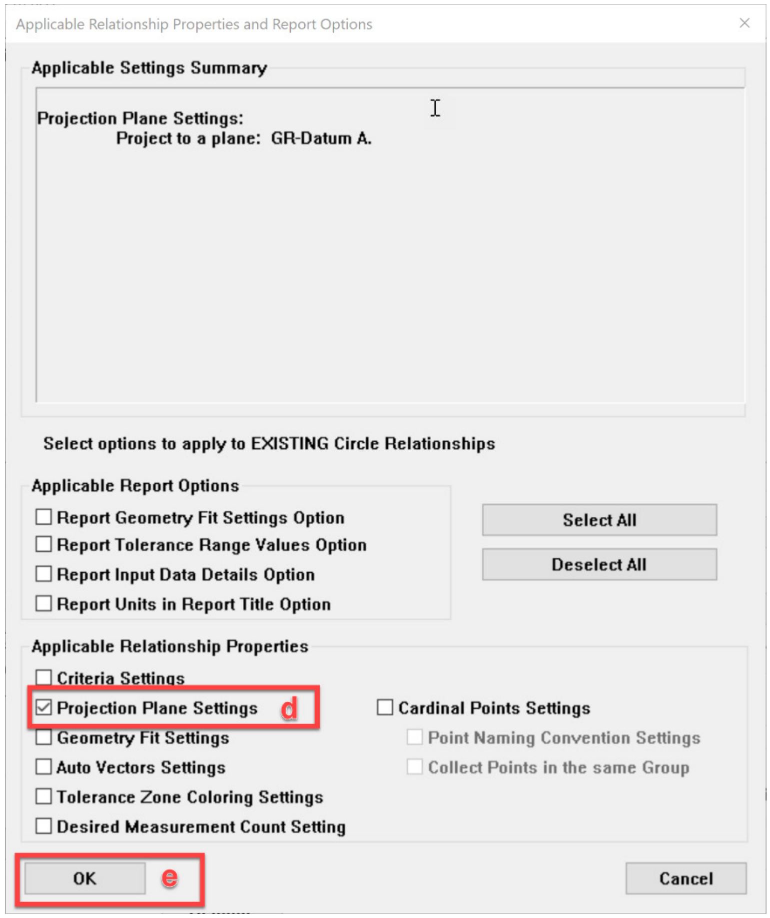


Figure 6-38. Selecting the settings to apply to other relationships.

d. Check the box next to **Projection Plane Settings**.

e. Click Ok.

Note: A projection plane can be set prior to selecting points via the Features Tab of the ribbon bar. See below.

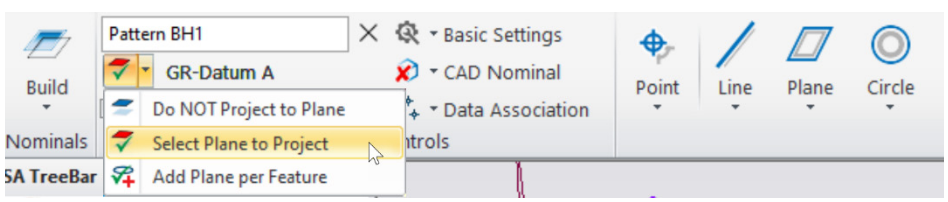


Figure 6-39. Setting a projection plane in the Features Tab.

CREATING DYNAMIC GEOMETRY

21. In the white box type "Datum B", which will be the name of the created feature.
22. Change the Data Association, by selecting .
23. Select  , then select the Datum B point group which will result in the creation of a plane.

24. Press Enter to confirm your selection.

25. Next click the drop-down form the line feature and select **Two Planes Intersection** (Figure 6-40).

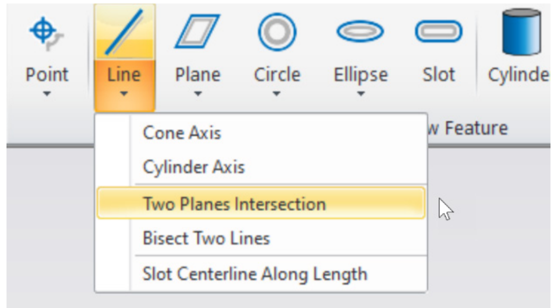


Figure 6-40. Creating an intersection line.

26. In the following selection prompts, graphically select the Datum A plane then the Datum B plane by single-clicking.

Fitting Geometry from Points

This tutorial provides a more in-depth look at feature construction and basic geometry fitting using 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. It also provides an alternative means to build features. 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. 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), 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 drop down (Figure 6-41). Be sure to check the **Make Geometry Relationship** option in order to build a feature or you will simply build a static construction.

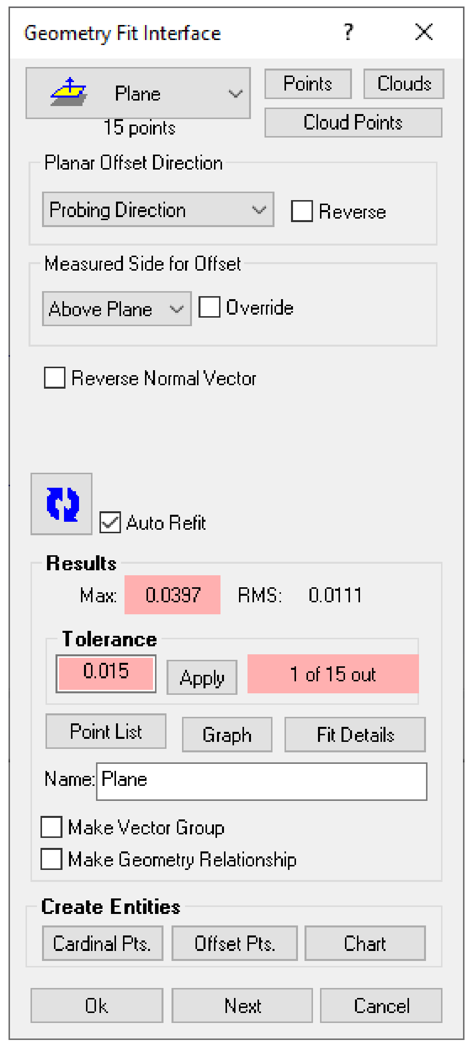


Figure 6-41. The ***Geometry Fit Interface***.

Fit Tolerance

4. Click on the Point List button. This will display the ***Geometry Fit Point Listing*** dialog (Figure 6-42) 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.

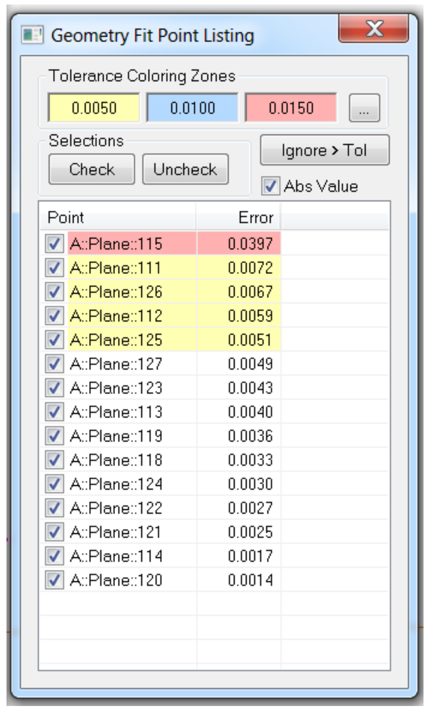


Figure 6-42. A geometry fit's point listing.

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 6-43).

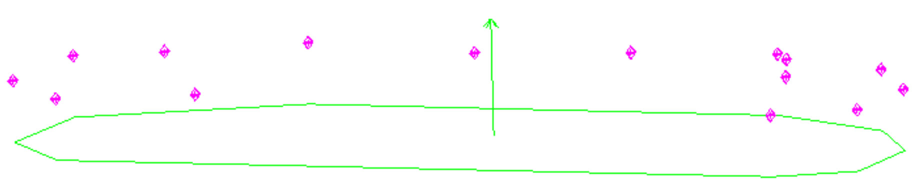


Figure 6-43. The plane is automatically offset from the measured points.

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.

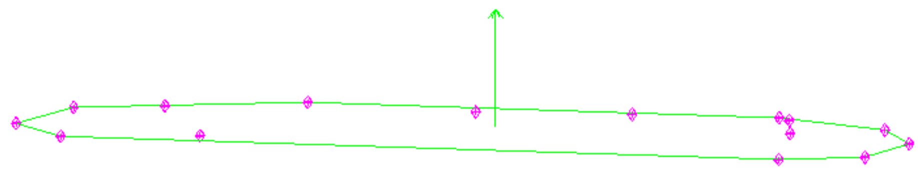


Figure 6-44. Overriding the offset to 0.0".

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 6-45). 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 6-45).

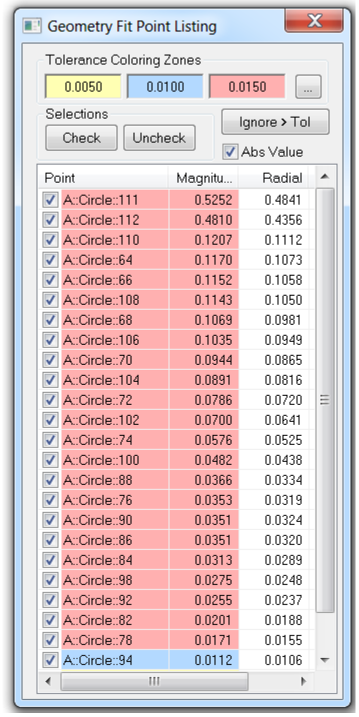


Figure 6-45. The point list for the circle, showing many out of tolerance points.

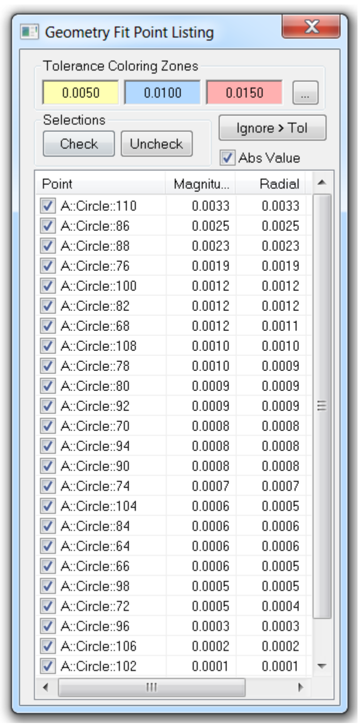


Figure 6-46. The fit improves dramatically after removing a few stray points.

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, right-click on this gorup and select **Fit geometry**.
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.

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 6-47).

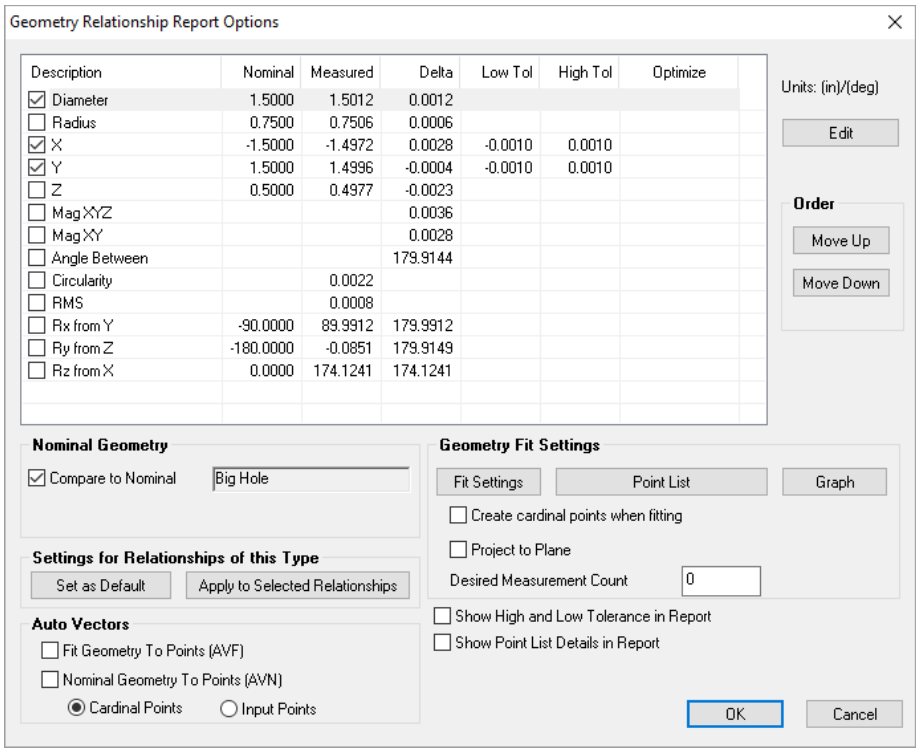


Figure 6-47. The Geometry Relationship Report options.

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 6-48).

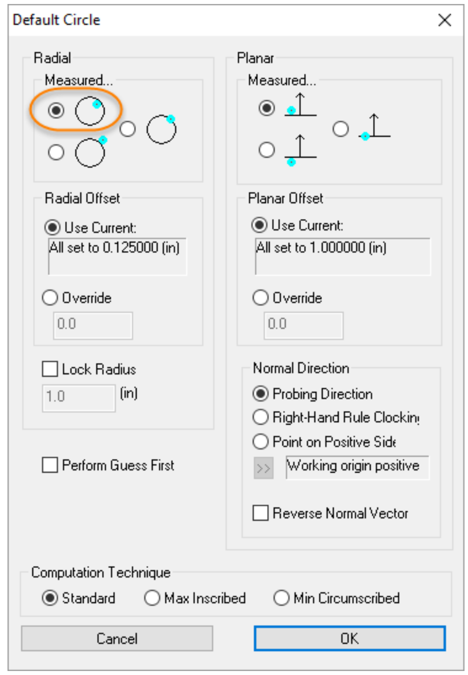


Figure 6-48. Setting the circle fit options to offset outward.

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 6-49).

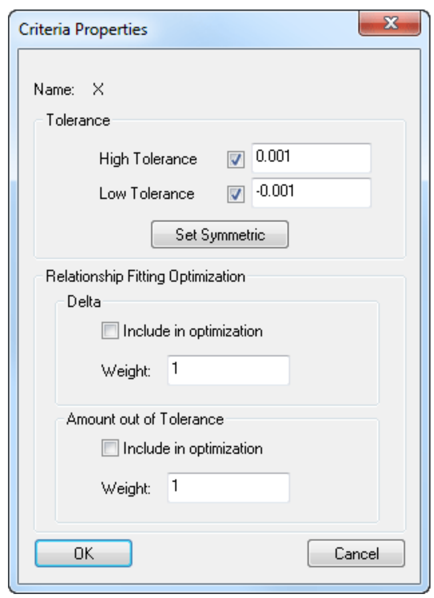


Figure 6-49. Modifying the high and low tolerances for the X attribute.

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 6-50).

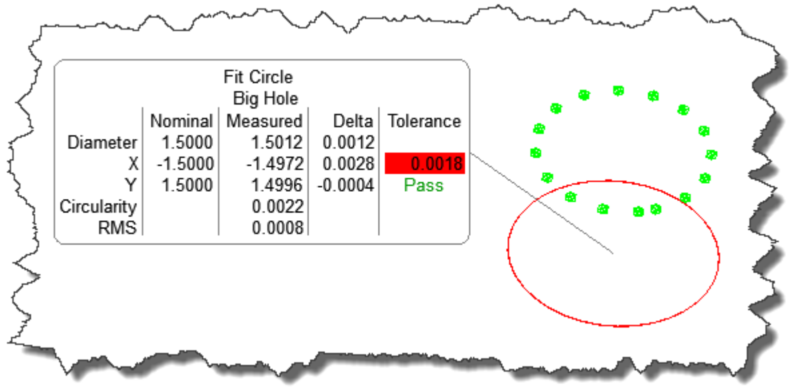


Figure 6-50. Adding the relationship callout to the circle.

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 6-51). Press **ESC** to cancel and restore the instrument position.

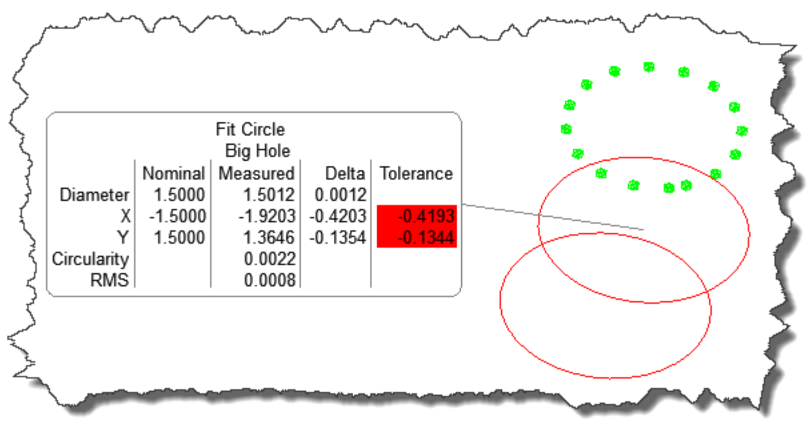


Figure 6-51. The callout automatically updates as the situation changes.

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 6-52 and Figure 6-53).

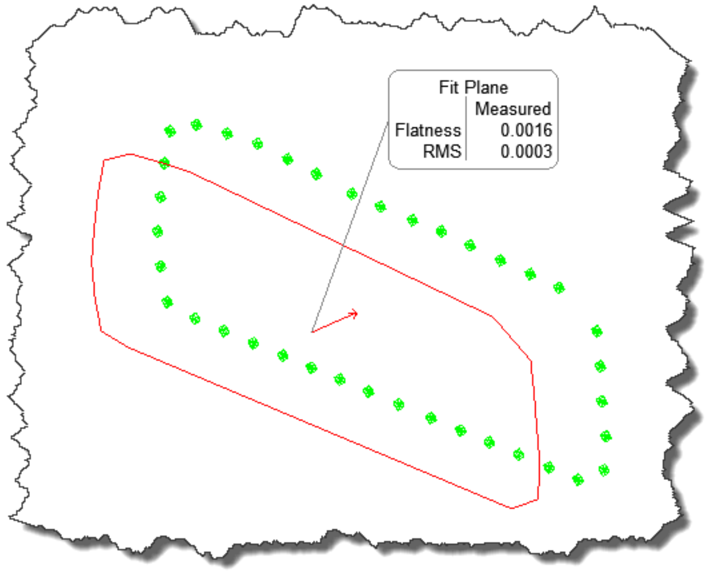


Figure 6-52. A fit-only relationship.

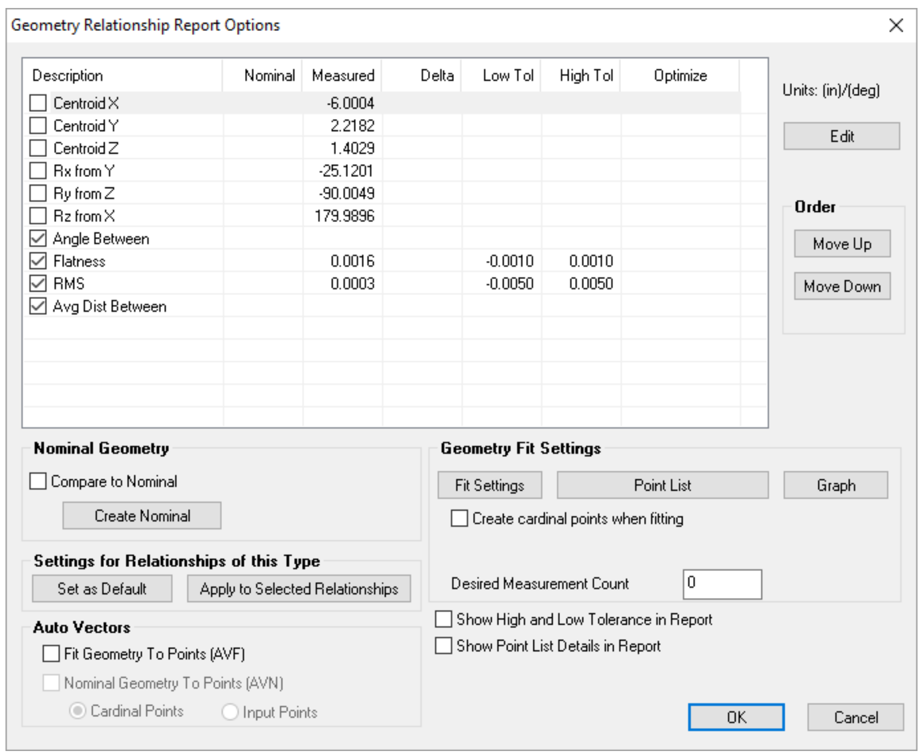
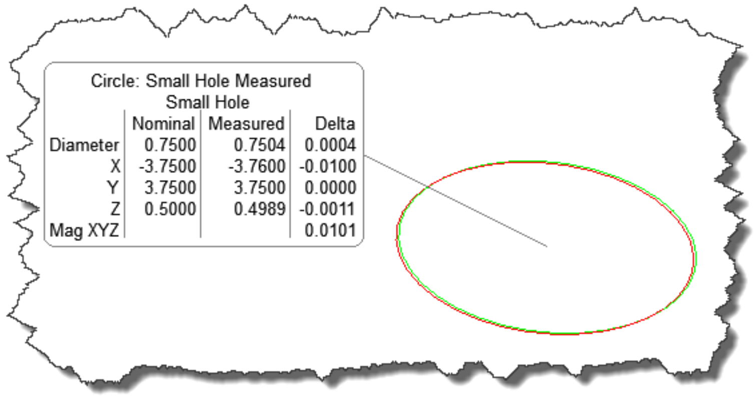


Figure 6-53. Geometry Relationship report options for fit-only relationships.

Compare Only Relationships

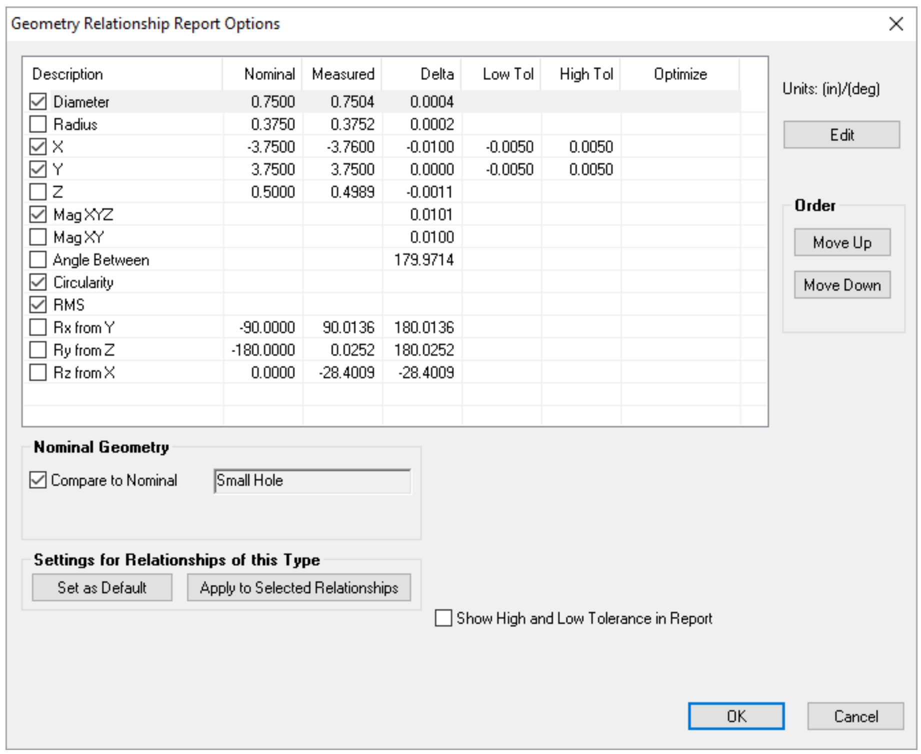
1. Now let's compare two circles. Select **Relationships>Geometry Comparison>Compare Only** and select the *Nominal::Small Hole* and *Measurements::Small Hole Measured* items from the tree and accept the default name.
2. Right-click *Circle: Small Hole Measured - Small Hole* and select **Add Callout**.
3. Compare only relationships utilize the same Geometry Rela-

tionship Options as well (Figure 6-54 and Figure 6-55).



Circle: Small Hole Measured / Small Hole	Nominal	Measured	Delta
Diameter	0.7500	0.7504	0.0004
X	-3.7500	-3.7600	-0.0100
Y	3.7500	3.7500	0.0000
Z	0.5000	0.4989	-0.0011
Mag XYZ			0.0101

Figure 6-54. Compare-only relationships.



Description	Nominal	Measured	Delta	Low Tol	High Tol	Optimize
☑ Diameter	0.7500	0.7504	0.0004			
☐ Radius	0.3750	0.3752	0.0002			
☑ X	-3.7500	-3.7600	-0.0100	-0.0050	0.0050	
☑ Y	3.7500	3.7500	0.0000	-0.0050	0.0050	
☐ Z	0.5000	0.4989	-0.0011			
☑ Mag XYZ			0.0101			
☐ Mag XY			0.0100			
☐ Angle Between			179.9714			
☑ Circularity						
☑ RMS						
☐ Rx from Y	-90.0000	90.0136	180.0136			
☐ Ry from Z	-180.0000	0.0252	180.0252			
☐ Rz from X	0.0000	-28.4009	-28.4009			

Figure 6-55. Geometry Relationship report options for compare-only relationships.

Conclusion

Geometry Relationships provide a quick and powerful method for creating and comparing geometry dynamically.

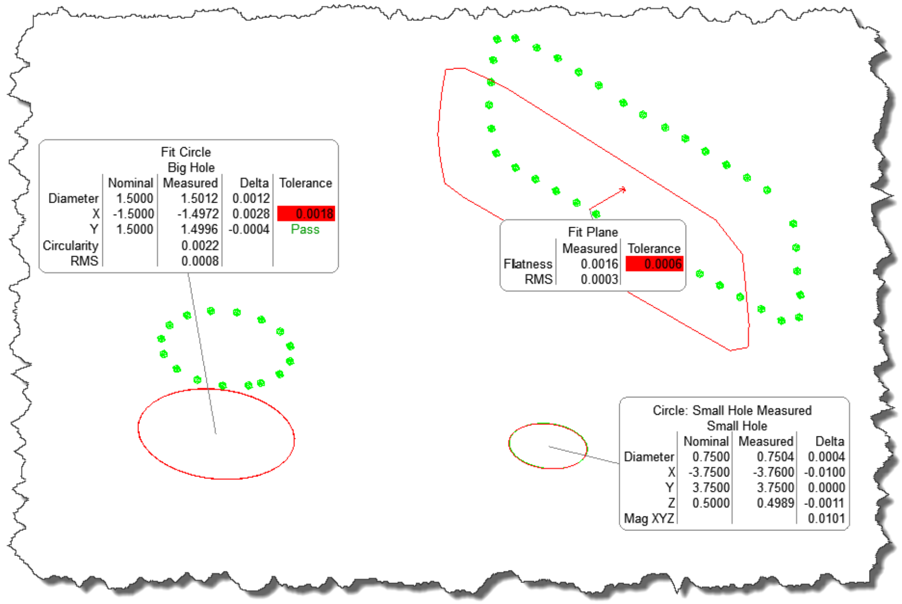


Figure 6-56. Various geometry relationships in a job.

Contoured Surface Alignment and Evaluation with 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 6-57). Since this sample file already has con-

structed 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.

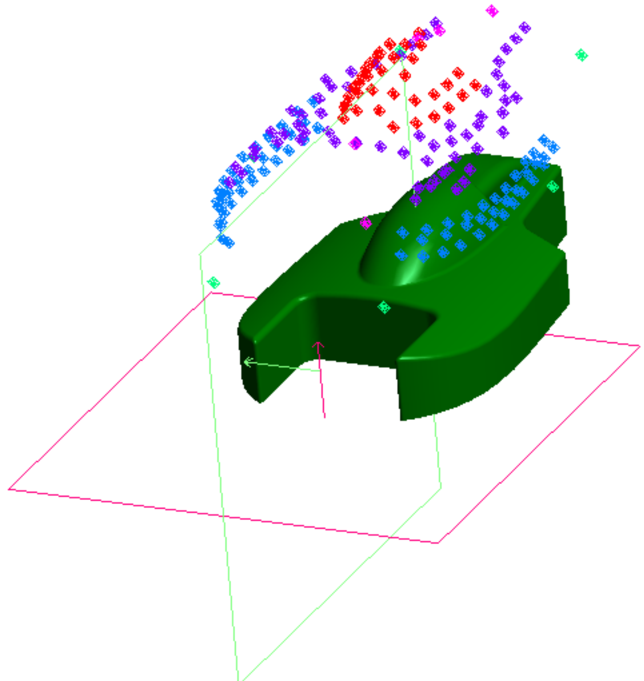


Figure 6-57. The measured points have not yet been aligned to the model.

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 6-58). 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`.

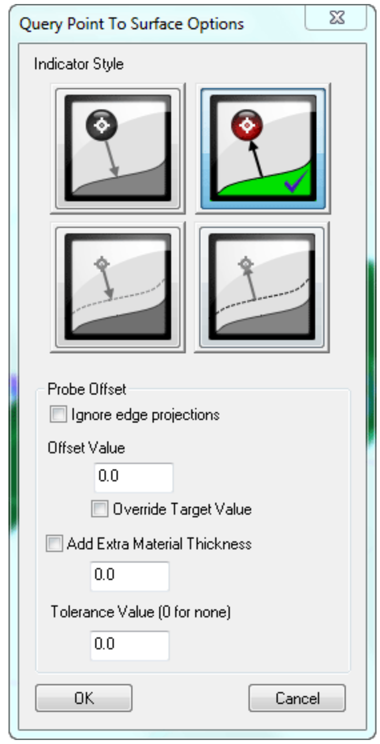


Figure 6-58. Query Point to Surface options.

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 6-59), select the **Single Line** format:

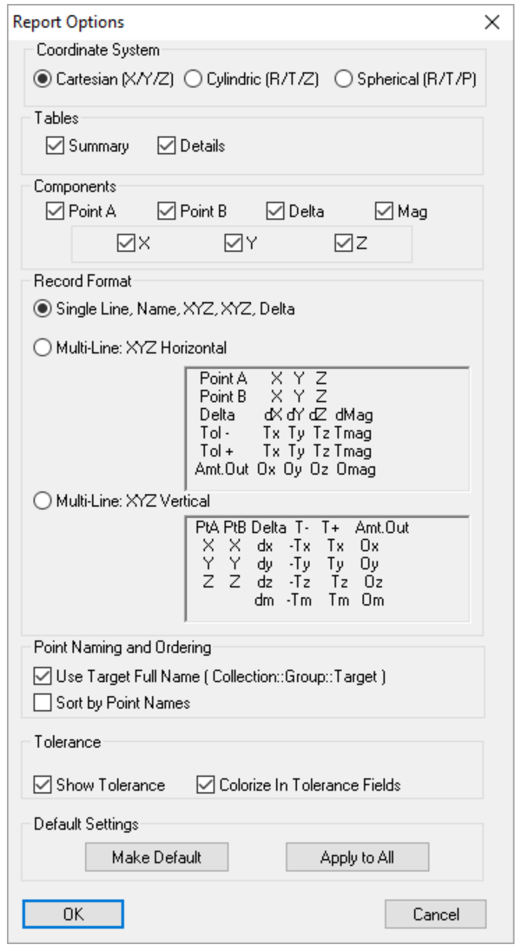


Figure 6-59. Report Options for the relationship.

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 6-60). 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.

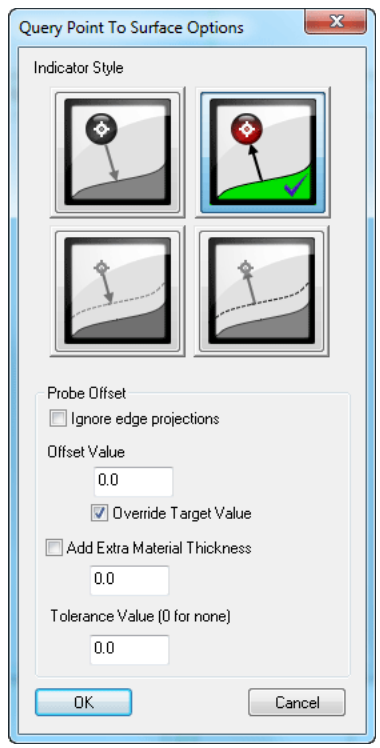


Figure 6-60. Overriding target offsets to zero.

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.

You may want to do the same with the Surface Profile Check 2 vector group created as part of this evaluation.

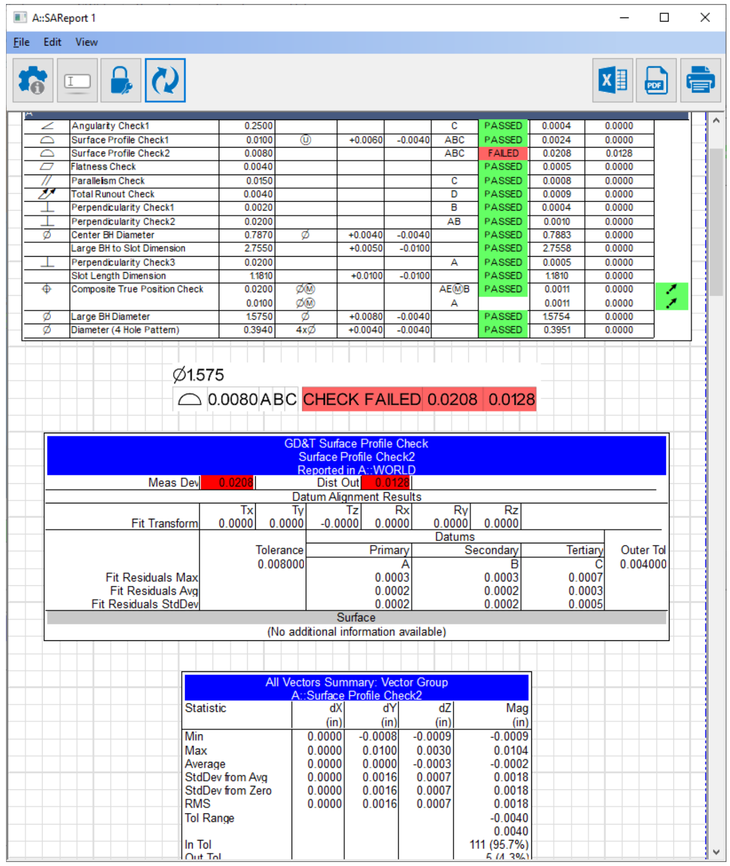


Figure 6-61. Additional details tables included for a failed check.

When a check fails, it will report two values for troubleshooting purposes: the Measured Deviation and Distance Out of Tolerance. The first number (0.0208) is the Measured Deviation or the size of the tolerance zone required for the part to pass. The second number (0.0128), is the Distance out of Tolerance or the necessary expansion of the tolerance zone in order for the part to pass. For the above profile check, the sum of the check tolerance plus the Distance Out of Tolerance will equal the *Measured Deviation* (0.008+.0128=.0208).

Conclusion

In this tutorial, we covered how to import CAD with GD&T annotations and how to create, measure and report feature checks.

This Page Intentionally Left Blank.

Cloud Inspection

CHAPTER 7

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Cloud Based Part Inspection with an Arm

- **Skill Level.** Beginner.
- **Description.** In this tutorial, we will cover how to scan a part , align to CAD and extract features.
- **Areas Covered.**
- **Time to Complete.** Approximately 15 minutes.

Start SA and Connect Arm

The first step is to connect to a PCMM Arm with a scanner. For this tutorial we will use a Hexagon Absoulte arm.

1. In SA, start a new job file by selecting **File>New File** from the menu or choosing the New File Icon from the Quick Access toolbar.
2. Go to the INSTRUMENT tab. In the INTERFACE section, select the add instrument icon, .
3. The ADD INSTRUMENTS TO SA dialog will open. Select the arm you want to connect from the list Figure 7-1 below.

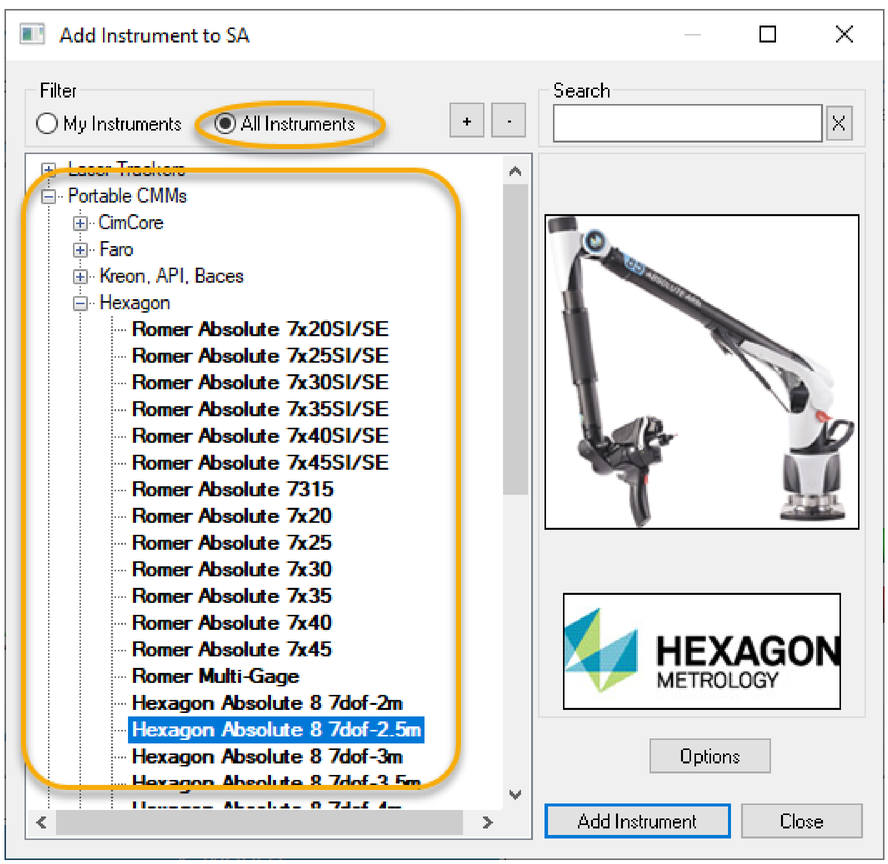


Figure 7-1. Add Instrument Options.

A. Select the **All Instruments** filter if it is not already selected.

- **B.** Single left-click the arm you would like to add to the job file. The Hexagon Absolute is selected here, but you can also make a selection from the or Faro or API sections as well.

4. In the INSTRUMENT tab In the INTERFACE section, select the Connect icon , to connect the arm. For arm driver information or connection troubleshooting refer to the arm quick-start guild for your model arm. The arm interface should appear

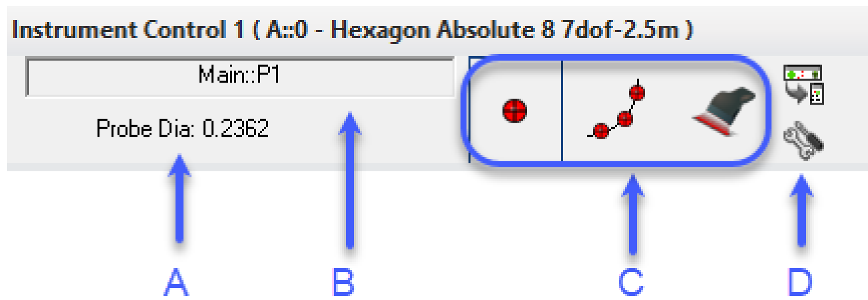


Figure 7-2. PCMM Arm Instrument Toolbar

- **A.** The probe diameter
- **B.** The Collection Group and Target Name used for the next measured point.
- **C.** The 3 primary measurement modes (Single Point, Spatial Scan, and Cloud Scan)
- **D.** Checks and Utilities.

Many scanners such as the Hexagon Absolute change measurement modes automatically between point probing and cloud scanning.

Scanning a Part on a Tabletop

Clipping planes define an exclusion zone which eliminates unwanted cloud data. Assuming you have a part on a flat surface, this surface can be measured as a plane and used for cloud data "clipping".

1. To measure a clipping plane you need to first decide if you want to measure it with points or by scanning the plane, and set the measurement mode you want, to begin we will use points.
2. From the Clouds & Surface Tab select **Prepare Clipping Plane** (Figure 7-3).

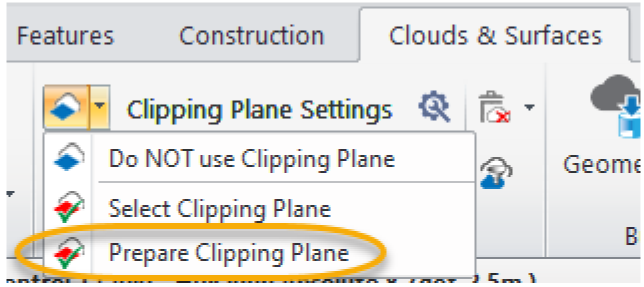


Figure 7-3. Selecting Prepare Clipping Plane.

3. Measure the table top or surface you are using to define the clipping plane (the surface your part is sitting on). The Cloud Clipping Plane we added should be actively trapping and collect the measurements, automatically fitting a plane (Figure 7-4).

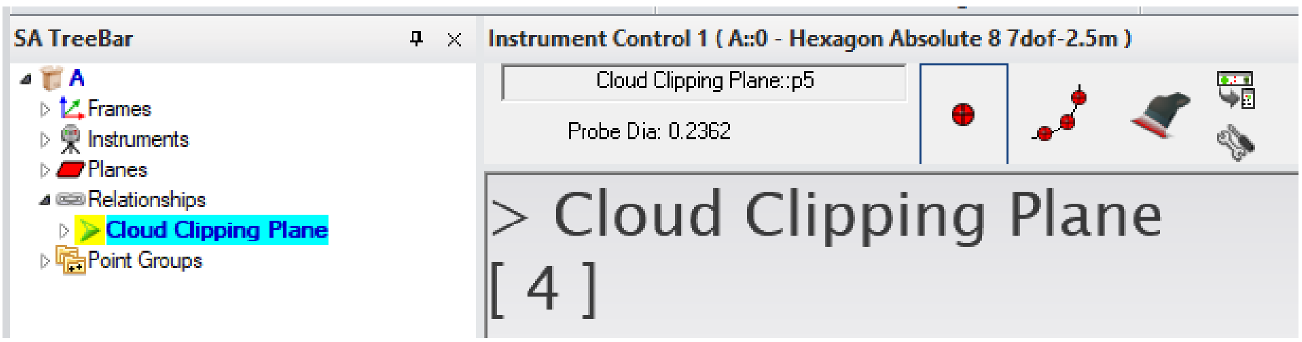


Figure 7-4. Trapping into a Clipping Plane feature

4. Stop Trapping. This can be done but pressing the Next button on the arm. It can also be done from the controls on the Inspection bar or by directly right-clicking on the Clipping Plane in the tree.

Configuring the View and Scanning your part

The most important thing when scanning is to make sure you have good scan coverage. The easiest way to see that is to enable the instrument view and zoom in tight.

1. From the **Home** tab in the *View* section select *Enable Viewpoint from Instrument Updates.*

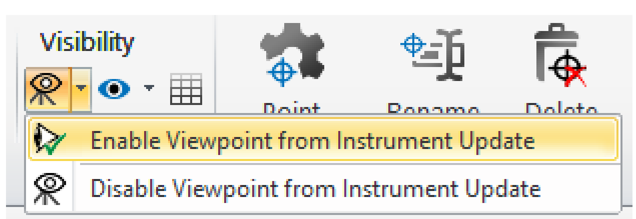


Figure 7-5. Enabling viewpoint from Instrument.

2. Change the Group name in the instrument toolbar, this group name will be used for the newly measured cloud.

3. Scan the part.

Compare to CAD

1. Import CAD by going to **File>Import>CAD Formats**. CAD import can be performed at any point in the measurement process and can be performed by simply drag-dropping a file into the graphics.
2. Align your scan to the CAD file by right-clicking on the cloud and selected **Align to CAD**. When prompted select the CAD model(s) you have scanned and press OK.

You will see an alignment report like this (Figure 7-6). If you are happy with the results press Yes to accept. But if the fit looks questionable press the Perform N-Point Alignment button.

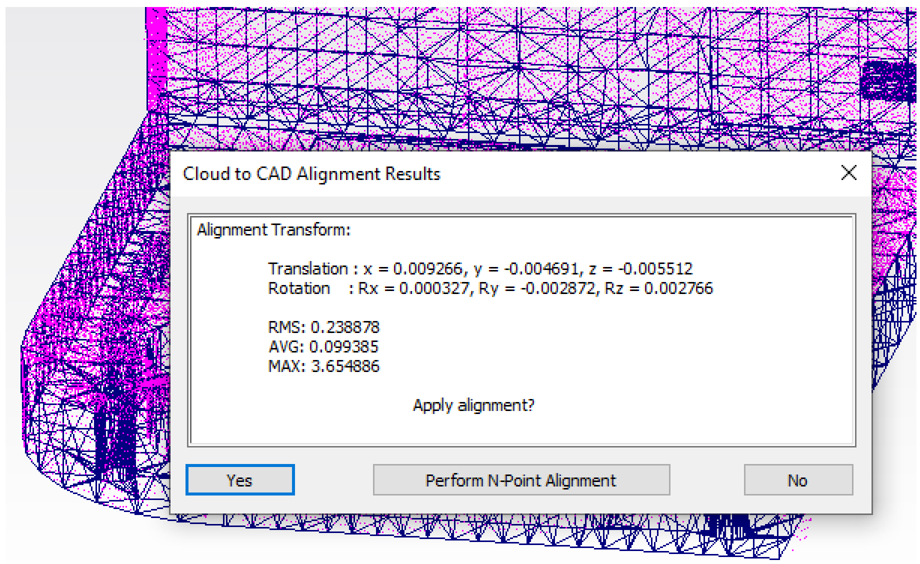


Figure 7-6. Cloud to CAD Alignment Results.

To Align using an N-Points Alignment perform the following steps:

1. Arrange the second graphic window and Align Cloud to CAD dialog so that you can see both the CAD model and cloud scan to facilitate selection of corresponding points such as the below (Figure 7-7):

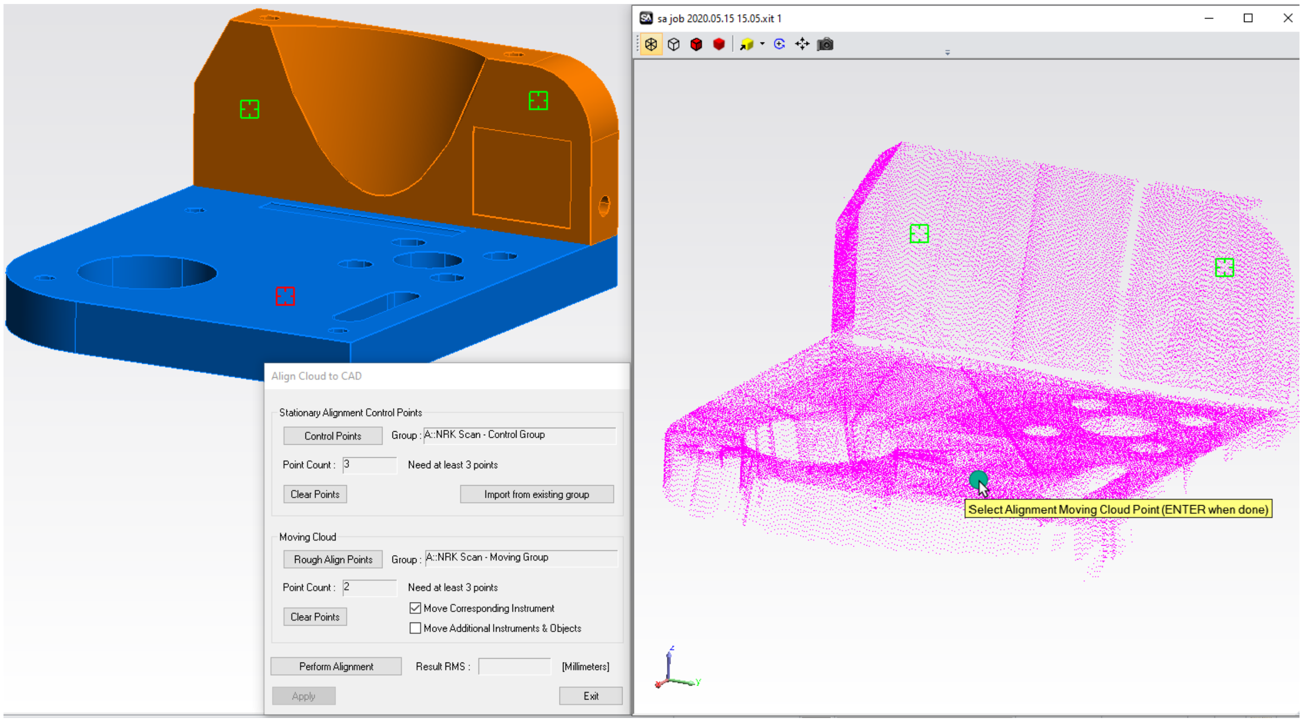


Figure 7-7. Align Cloud to CAD windows and arrangement.

2. Press the Control Points button in the **Align Cloud to CAD** dialog and select at least 3 points on the CAD model. These points should be on surfaces with good scan coverage.
3. Press the Rough Align Points button and graphically select a corresponding cloud point from the axillary view for each of the control points you placed on the CAD.
4. Press Perform Alignment and accept the alignment by pressing Exit when you are happy with the results.

Report Overall Part Acceptance with a Color Map

Rather than using vectors to report deviations you can get much nicer results with cloud data by using a voxel display.

1. Disconnect the arm so that it is no longer connected to SA.
2. Build a cloud to CAD relationship between your scan and the CAD model.
3. Adjust Voxel Settings and colorization levels to build heat map

Report Individual Features

Identify Features to extract

Extract features

Add overview heatmap to Report

Add features to report

This Page Intentionally Left Blank.

GD&T Inspection

CHAPTER 8

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Feature Measurement

Inspection with GD&T

- **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.

1. In SA, start a new job file by selecting **File>New File** from the menu or choosing the New File Icon from the Quick Access toolbar.
2. Import the native CAD model with GD&T annotations by selecting **File>Import>CAD File Formats>Direct CAD Access** or use the Auto Import button on the Quick Access toolbar.
3. Choose SA `DEMOPART_inches.CATPart` from the Samples file folder in the SpatialAnalyzer install directory (C:\Program Files (x86)\New River Kinematics\SpatialAnalyzer xxxx.xx.xx\Samples).
4. When the ***Choose CAD Features*** dialog appears (Figure 8-1), turn off the points and lines but make sure the **Annotations** are selected and press OK.

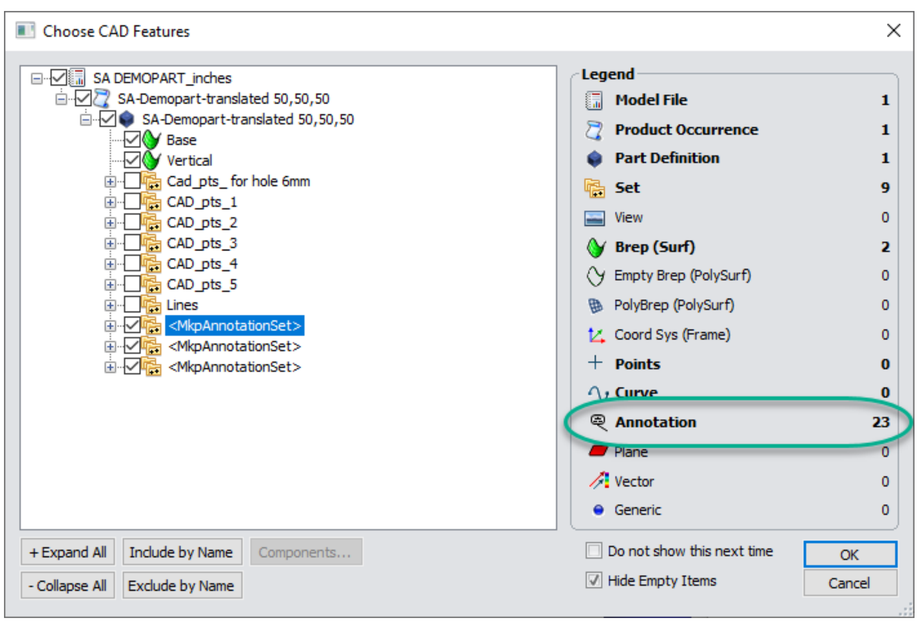


Figure 8-1. The Direct CAD Access settings with the Selective Importer.

5. Once imported, you will find the *Annotations* category is created in the tree along with the Surfaces, which contains each individual annotation for the model Figure 8-2.

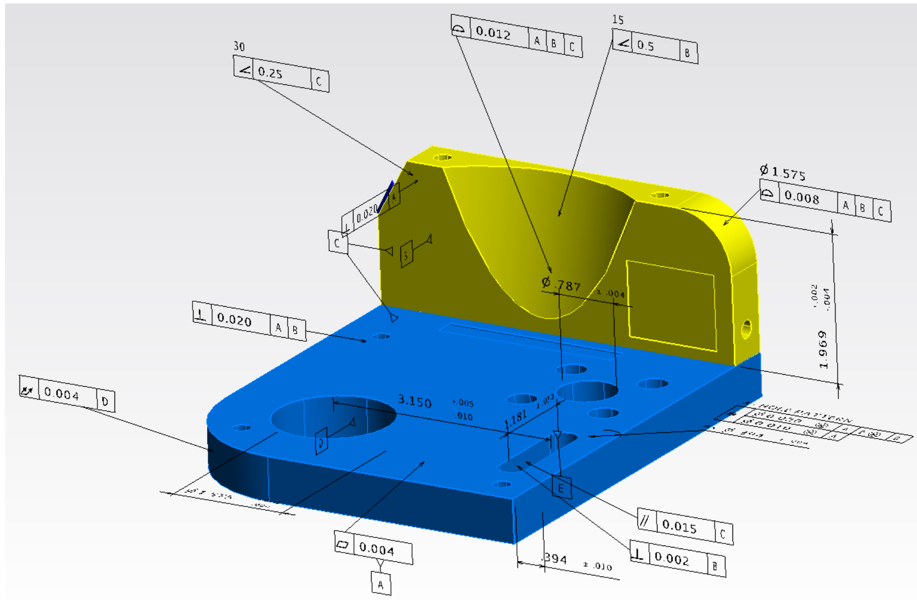


Figure 8-2. Annotations in the graphical view.

Annotations define the tolerances, datums used and all the graphical aspects of the annotation attached to the model. It also holds the link to the specific CAD faces used for analysis.

You may also see a new collection appear following the import. We will bring in the saved views from the CAD model for the annotations as callouts that you can use for reporting if desired.

Creating GD&T Datums and Feature Checks

For this tutorial we will inspect only one of the GD&T Features.

6. Create a New Collection in the SA Tree bar and name it *GD&T". Just like annotations, Feature Checks cannot be moved to another collection once created so be sure the GD&T collection you create is the **Default Collection**.

7. Right-click *Geometrical Tolerance.34* 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. Here we wanted to build a feature check for a Surface Profile annotation which referenced 3 datum features so the datum features were built for you as well automatically.

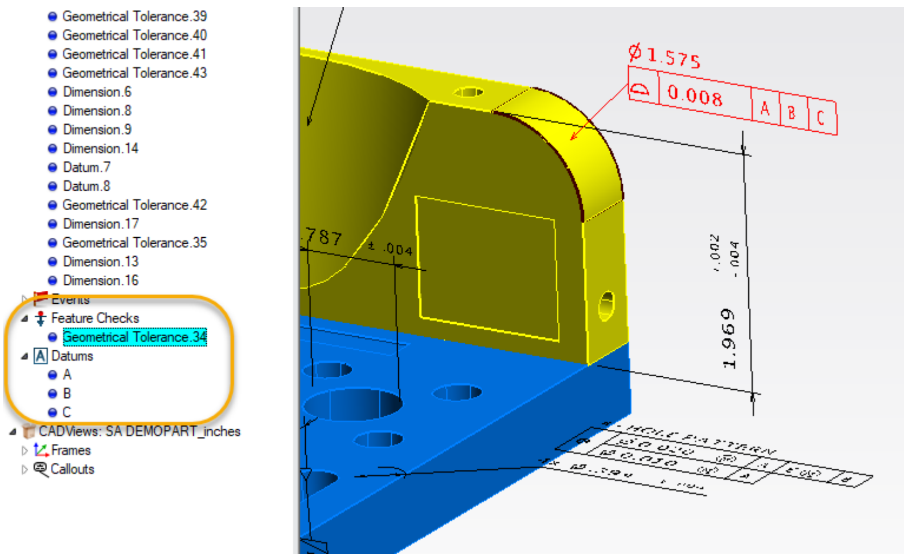


Figure 8-3. Feature Checks and Datums built in the tree

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.

- If you already have points in the job file you can right-click on the feature check and select **Associate Points.**
- For this tutorial we will assume that an instrument is present. So let's add an instrument and run the interface.

2. Now that the instrument is added and running, open the **Inspection Bar** from the **Home** tab  and double click on the feature you want to measure in order to start trapping (Figure 8-4).

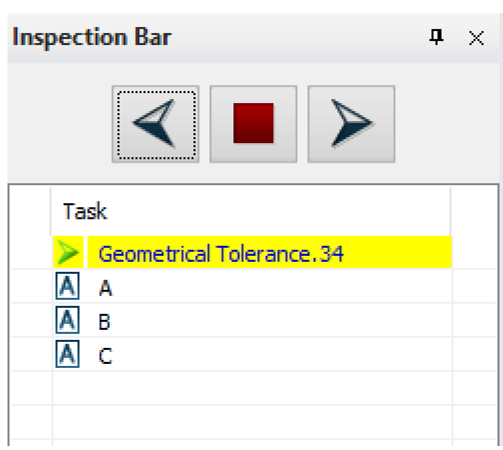


Figure 8-4. Selecting by double clicking on the feature of interest in the Inspection bar.

3. 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.

4. Since we did not take any actual measurements, lets open `GD&T with Multigage.xit` from the Samples directory under **Help>Sample SA Files**.

5. In the `GD&T with Multigage.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 **Reporting>Report Bar** . By default only a minimum pass/fail result is displayed:

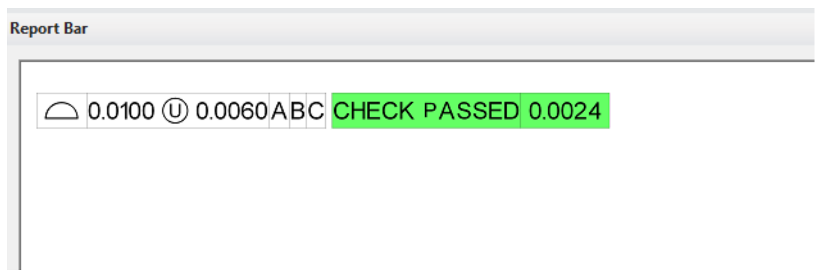
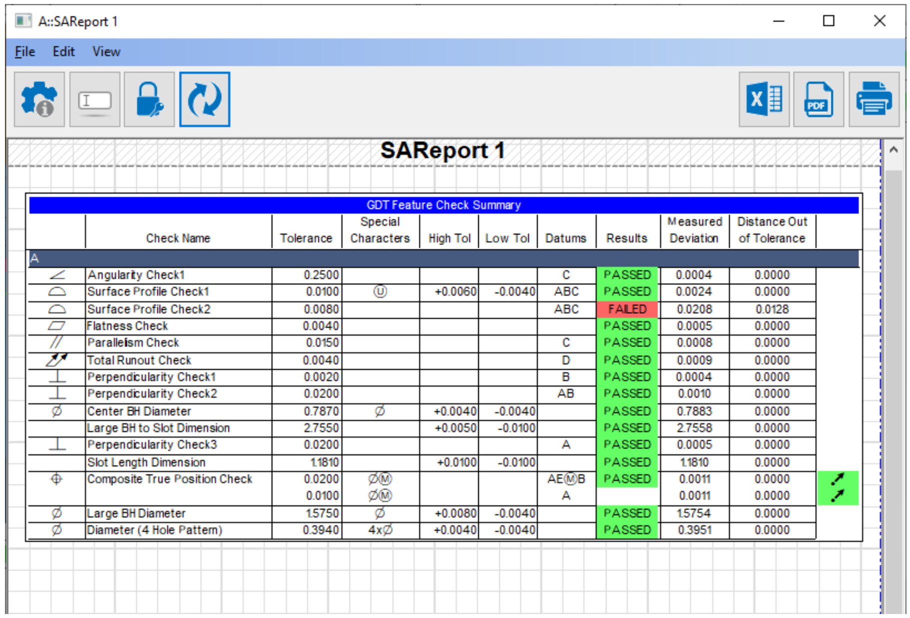


Figure 8-5. Feature check results for a surface profile check.

2. Right-click the *Feature Checks* category in the tree and select **Evaluate All Checks.**

3. To build a summary table for all, or a selection of check results, there is an additional option under the *Feature Checks* category to **Create Inspection Summary Table**. In this file, a table has already been built for you.

4. Navigate to the *GDT Feature Check Summary* within the **Custom Report Tables** section, right-click on it and select *Add to Active SA Report* (Figure 8-6).



	GDT Feature Check Summary								
	Check Name	Tolerance	Special Characters	High Tol	Low Tol	Datums	Results	Measured Deviation	Distance Out of Tolerance
A									
	Angularity Check1	0.2500				C	PASSED	0.0004	0.0000
	Surface Profile Check1	0.0100	Ⓤ	+0.0060	-0.0040	ABC	PASSED	0.0024	0.0000
	Surface Profile Check2	0.0080				ABC	FAILED	0.0208	0.0128
	Flatness Check	0.0040					PASSED	0.0005	0.0000
	Parallelism Check	0.0150				C	PASSED	0.0008	0.0000
	Total Runout Check	0.0040				D	PASSED	0.0009	0.0000
	Perpendicularity Check1	0.0020				B	PASSED	0.0004	0.0000
	Perpendicularity Check2	0.0200				AB	PASSED	0.0010	0.0000
⌀	Center BH Diameter	0.7870	⌀	+0.0040	-0.0040		PASSED	0.7883	0.0000
	Large BH to Slot Dimension	2.7550		+0.0050	-0.0100		PASSED	2.7558	0.0000
	Perpendicularity Check3	0.0200				A	PASSED	0.0005	0.0000
	Slot Length Dimension	1.1810		+0.0100	-0.0100		PASSED	1.1810	0.0000
	Composite True Position Check	0.0200	⌀Ⓜ			AEⓂB	PASSED	0.0011	0.0000
		0.0100	⌀Ⓜ			A		0.0011	0.0000
⌀	Large BH Diameter	1.5750	⌀	+0.0080	-0.0040		PASSED	1.5754	0.0000
⌀	Diameter (4 Hole Pattern)	0.3940	4x⌀	+0.0040	-0.0040		PASSED	0.3951	0.0000

Figure 8-6. A Custom Summary Table with GD&T feature check results.

Note that one of the checks in this file failed. Lets add additional details on this particular check to the report.

5. Right-Click on the Surface Profile Check 2 Feature Check in the tree and select Reporting Options.
6. Enable additional reporting tables for this check as follows (Figure 8-7), then press OK.

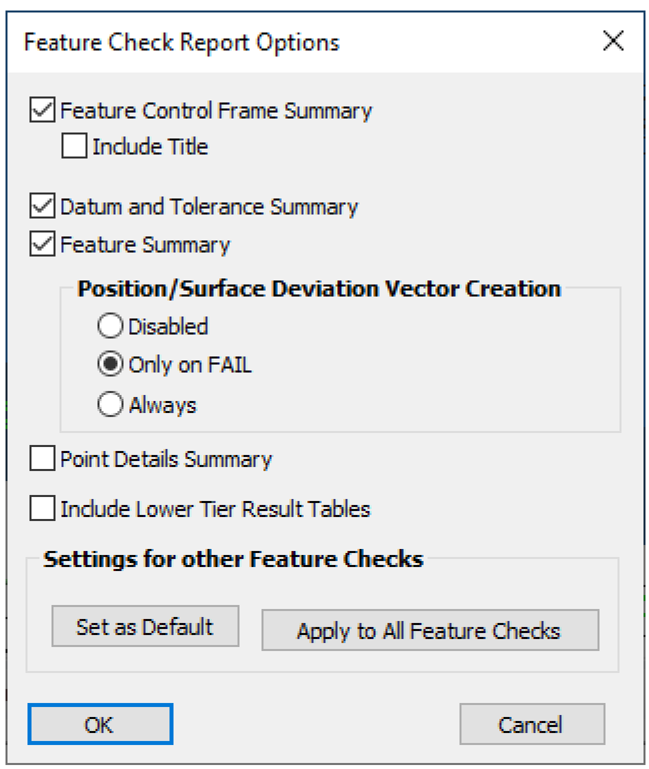


Figure 8-7. Feature Check Reporting Options

7. Add these additional tables to the report by right-clicking on the Feature Check and selecting **Add to Active SA Report**.

8. You may want to do the same with the Surface Profile Check 2 vector group created as part of this evaluation.

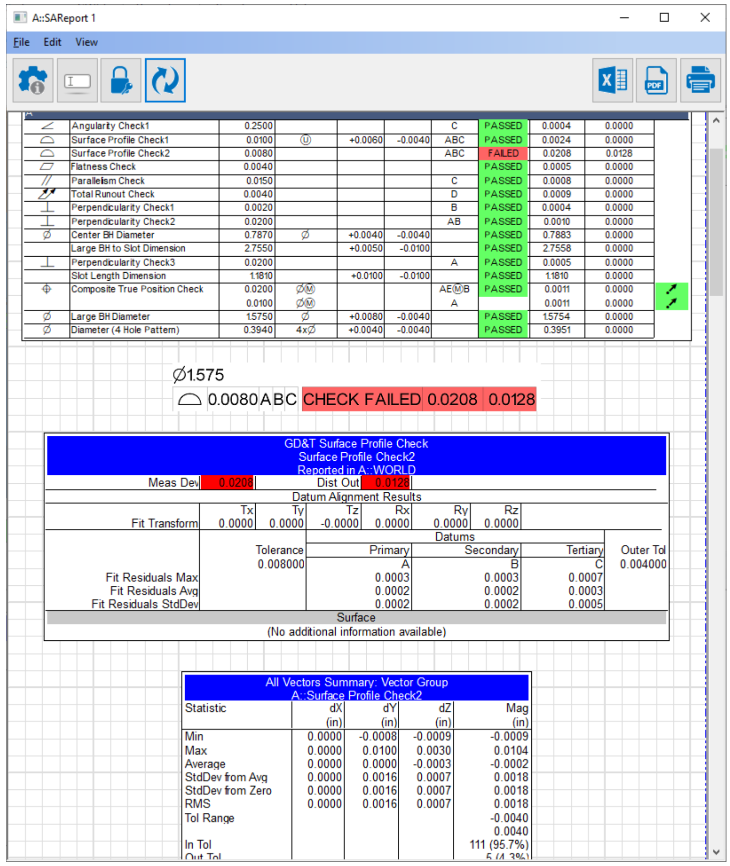


Figure 8-8. Additional details tables included for a failed check.

When a check fails, it will report two values for troubleshooting purposes: the Measured Deviation and Distance Out of Tolerance. The first number (0.0208) is the Measured Deviation or the size of the tolerance zone required for the part to pass. The second number (0.0128), is the Distance out of Tolerance or the necessary expansion of the tolerance zone in order for the part to pass. For the above profile check, the sum of the check tolerance plus the Distance Out of Tolerance will equal the *Measured Deviation* (0.008+.0128=.0208).

Conclusion

In this tutorial, we covered how to import CAD with GD&T annotations and how to create, measure and report feature checks.

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 8-9).

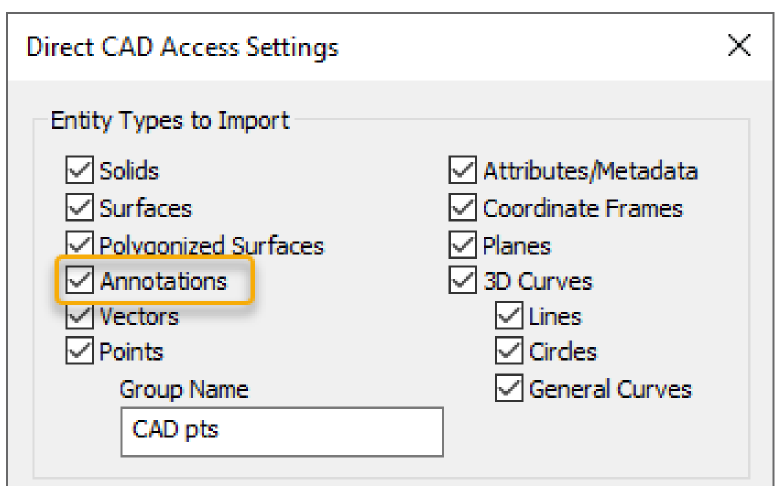


Figure 8-9. Ensuring that the Annotations option is selected in the Direct CAD Access import settings.

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 8-10).

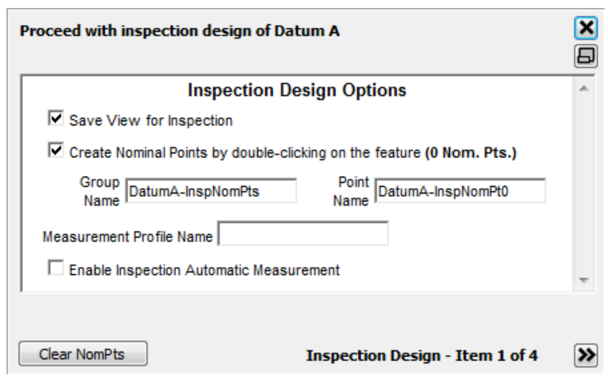


Figure 8-10. Options for designing an inspection.

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 8-11).

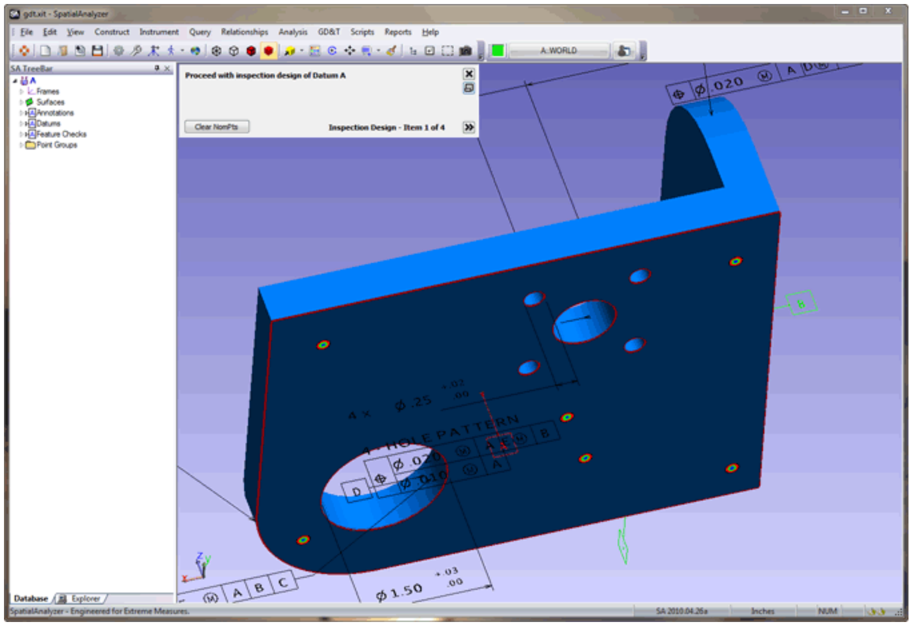


Figure 8-11. The options are now minimized.

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 8-12). During this rehearsal mode, the mouse acts as an instrument. So with each mouse click, a measurement is simulated with some random error.

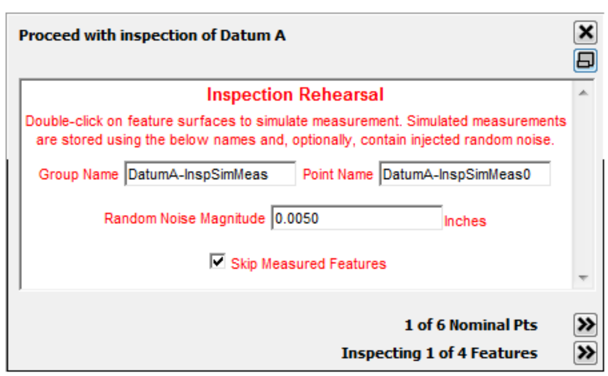


Figure 8-12. Rehearsing the inspection.

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 [X] .

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**.

This Page Intentionally Left Blank.

Reporting

CHAPTER 9

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Reporting

Analysis and Reporting

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 9-1). 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.

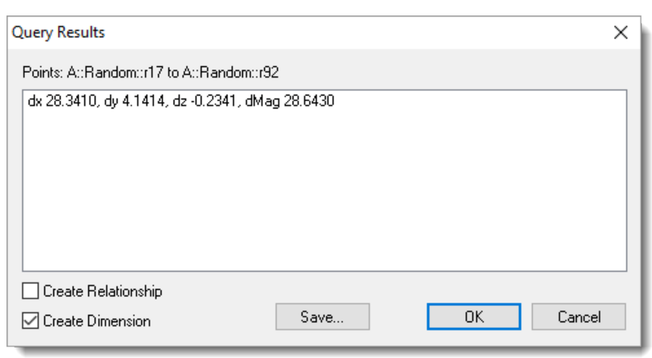


Figure 9-1. The query results dialog.

3. To record the point to point check the *Create Dimension* option check box in the bottom left corner. This will create a dimension stored in the SA tree as well as display the deviation graphically (Figure 9-2).

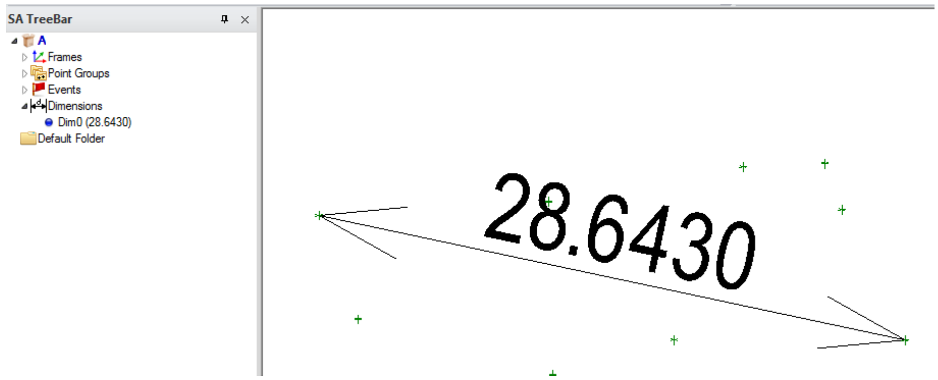


Figure 9-2. The query as a Dimension.

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 9-3). 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).

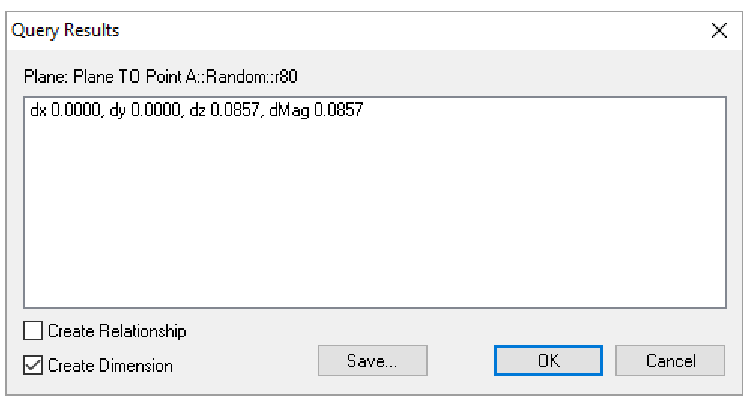


Figure 9-3. Querying a point to an object.

2. Like the Point to Point query, the results are stored in the tree as an event (Figure 9-4).

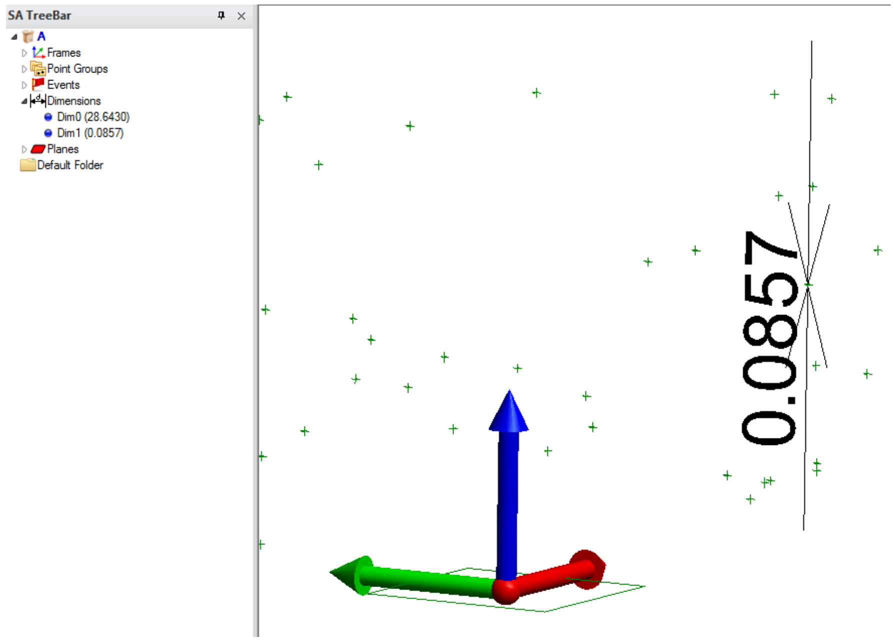


Figure 9-4. The Point to Object query as a dimension.

Dimensions of different types, both linear and angular, can be added through the toolkit as well using the selection of icons on the Reporting tab. Once added dimension can also be formatted through their properties in any way that you need (Figure 9-5).

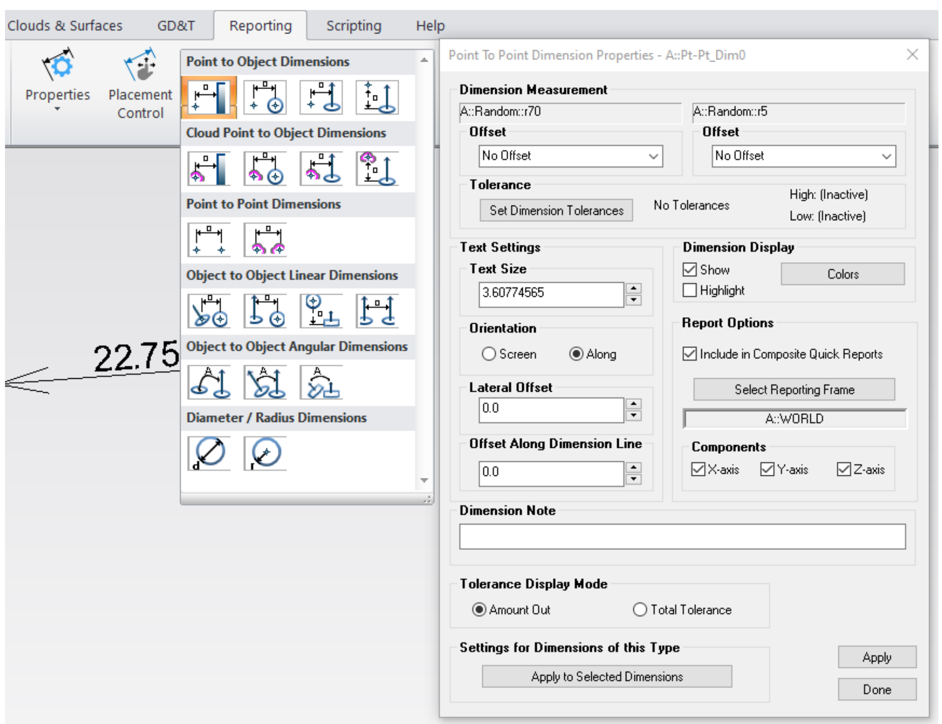


Figure 9-5. Dimensions and their properties

Calculating Multiple Points to Object Distances

During analysis operations it is often desirable 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 9-6). 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.

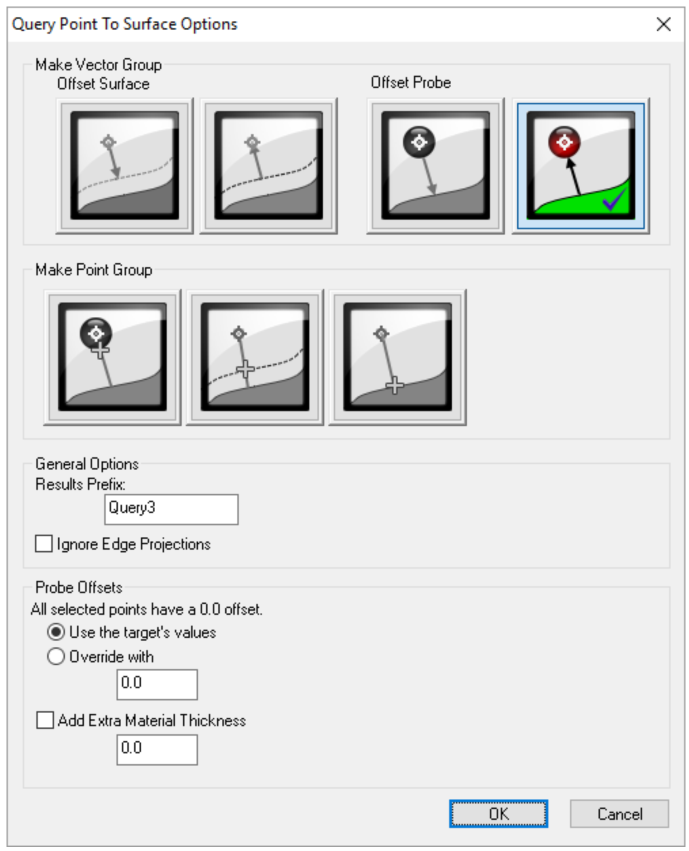


Figure 9-6. The options for querying points to objects.

2. A Vector Group will be created and the properties dialog will appear (Figure 9-7).

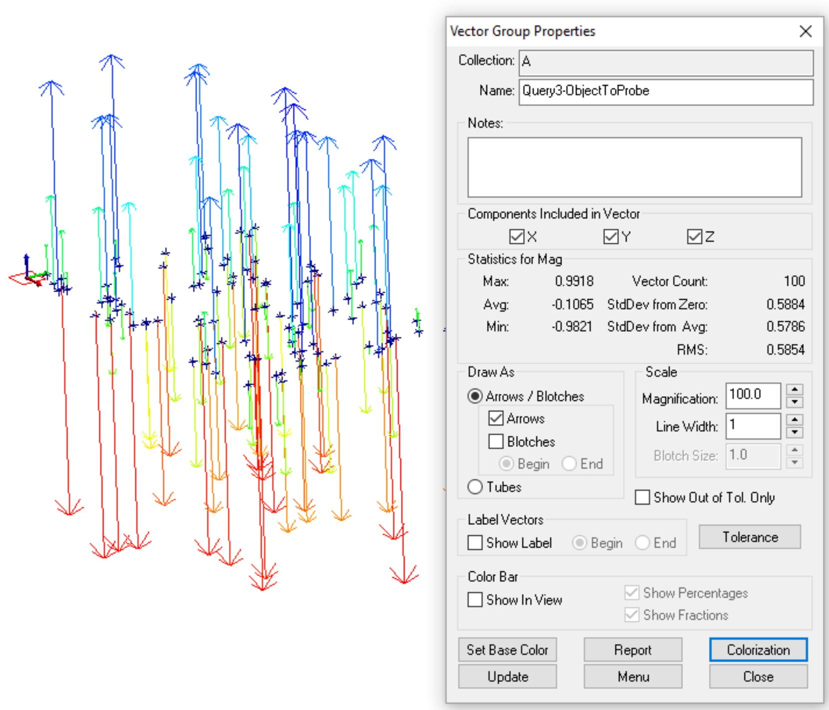


Figure 9-7. The vector group properties dialog.

3. To change the appearance of the vectors change the magnification, size, and style of the vector. Change the magnification, vector style and blotch size to match Figure 9-8.

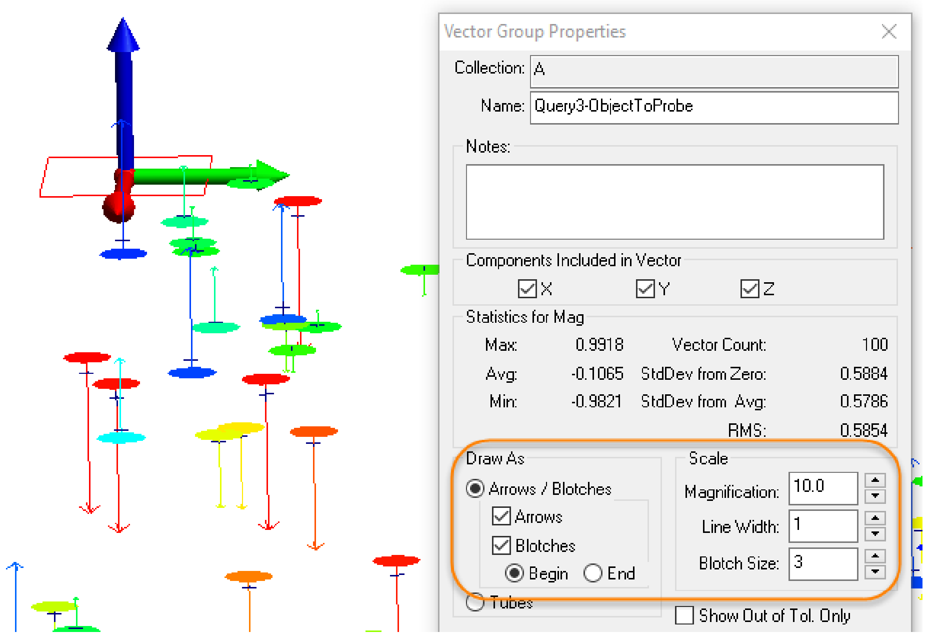


Figure 9-8. The Vector Group Display Options.

4. Now change the colorization of the vectors and apply a tolerance for colorization. There are four different colorization methods (Figure 9-9). Choose **Toleranced (Go/No-Go)**.

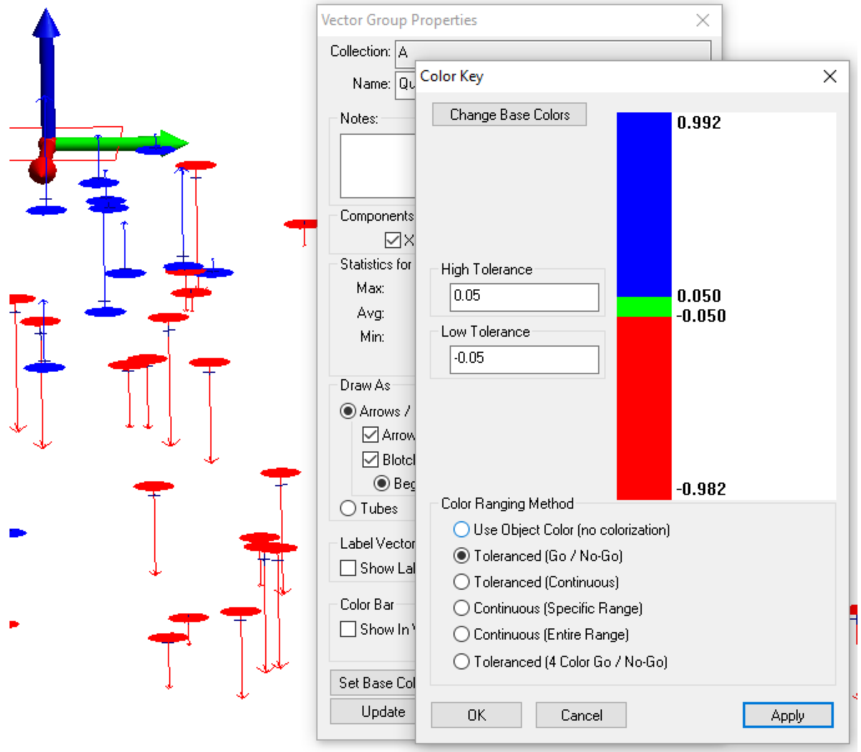


Figure 9-9. Colorization options.

Now the vector group reflects the display and colorization changes (Figure 9-10).

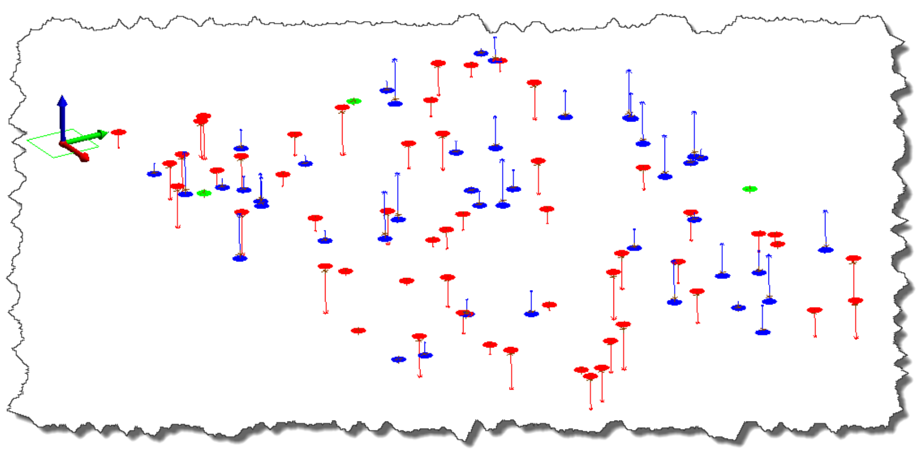


Figure 9-10. The result.

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 9-11).

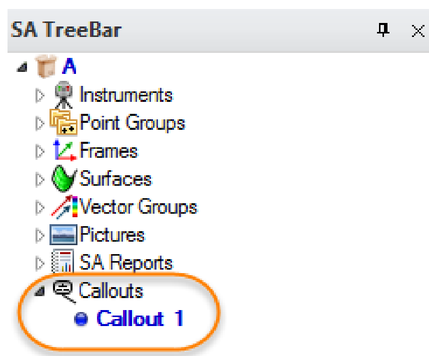


Figure 9-11. The newly-added callout view.

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 9-12).

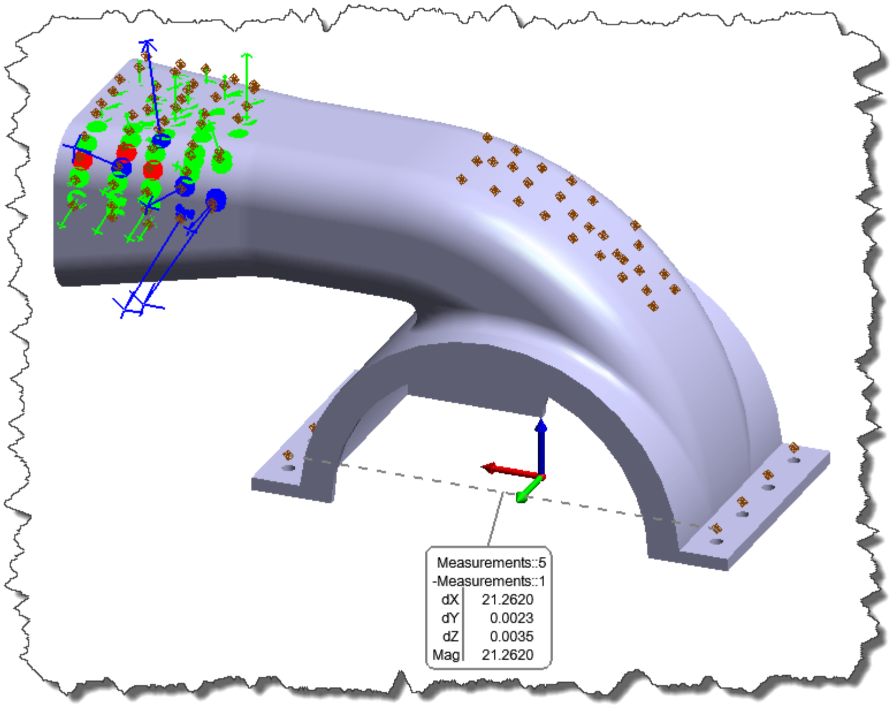


Figure 9-12. The point comparison callout.

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 9-13). 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 9-14).

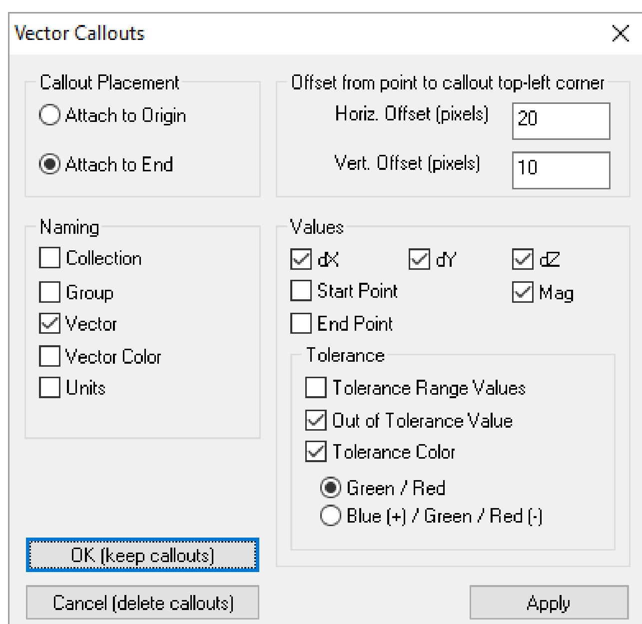


Figure 9-13. The Vector Callouts dialog.

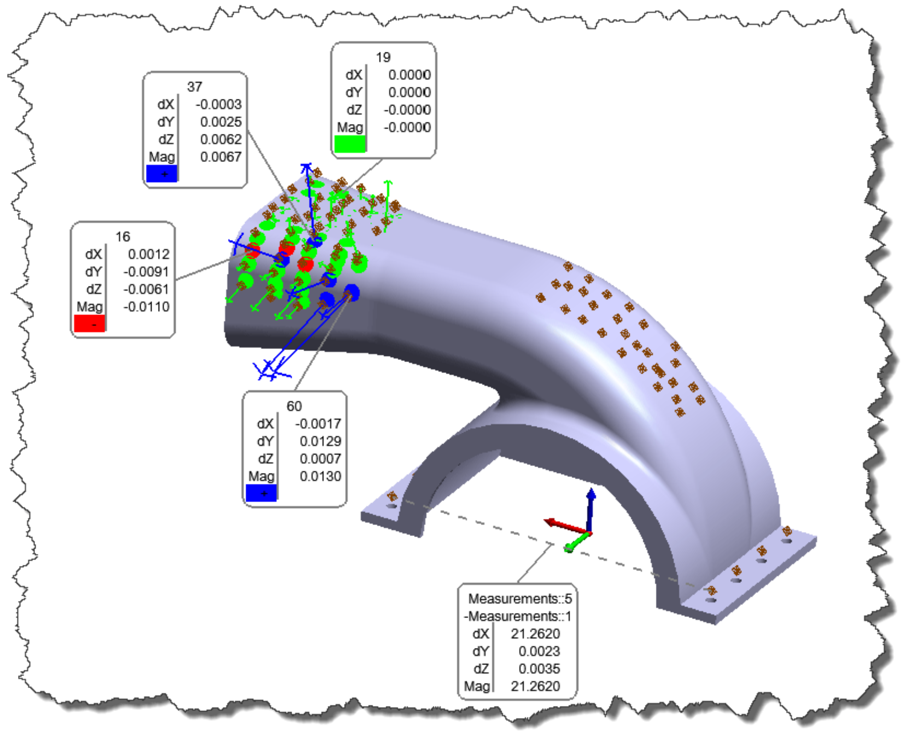


Figure 9-14. Vector callouts have been added.

Locking the Callout view

1. Right-click on *Callout 1* in the tree and select **Properties** (Figure 9-15).

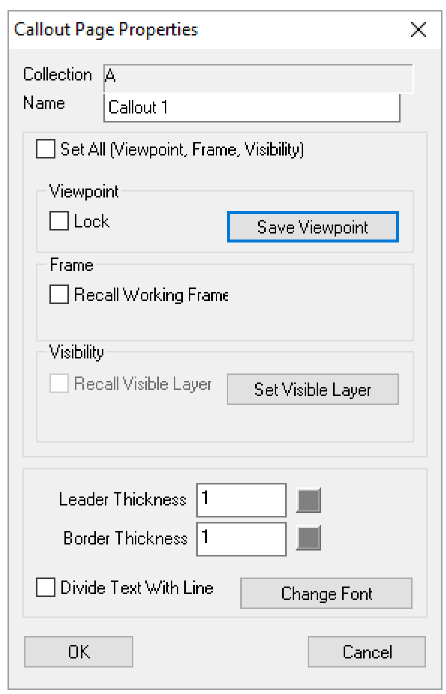


Figure 9-15. The Callout Page properties.

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 call-

out.

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.

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 9-16).

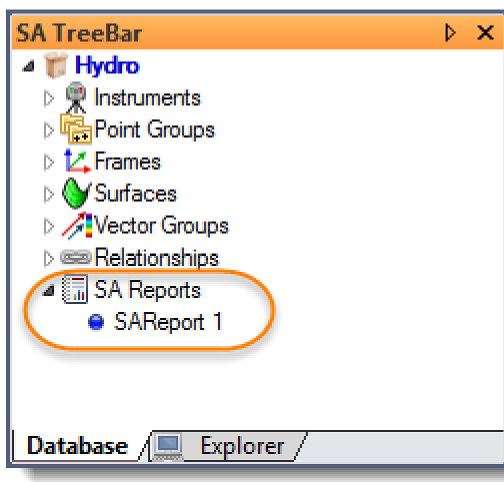


Figure 9-16. The newly-added report.

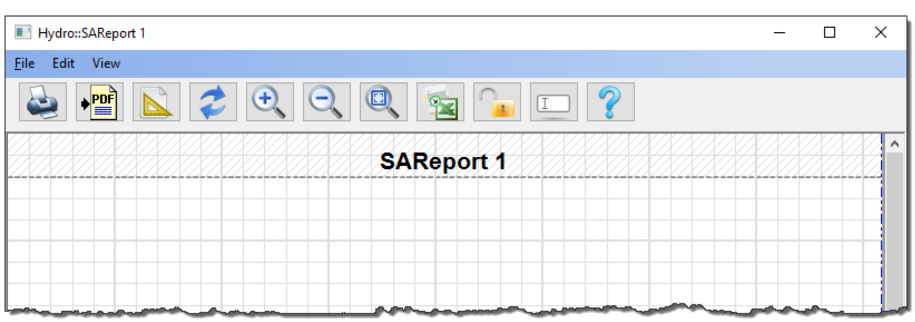
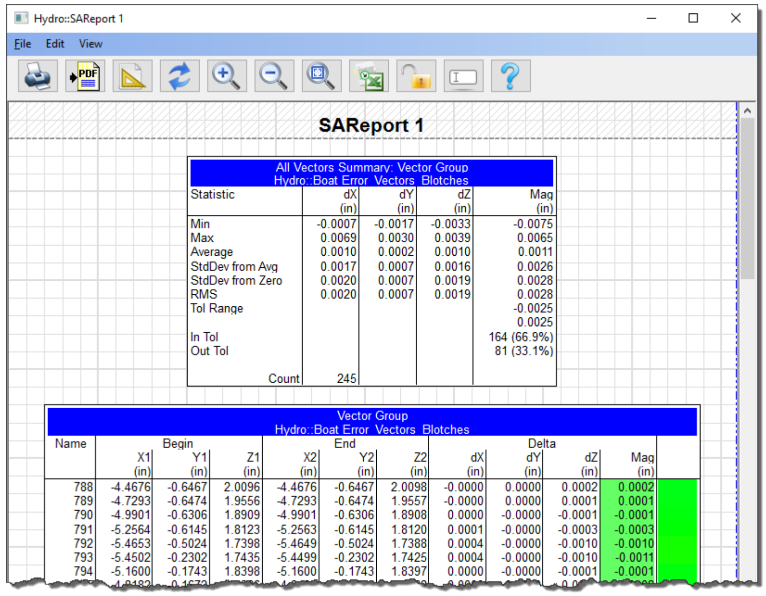


Figure 9-17. The Report Designer.

4. Let's add an item to the SA Report. Click and drag the *Boat Error_Vectors_Blotches* vector group into the report (Figure

9-18).



All Vectors Summary: Vector Group Hydro::Boat Error Vectors Blotches				
Statistic	dX (in)	dY (in)	dZ (in)	Mag (in)
Min	-0.0007	-0.0017	-0.0033	-0.0075
Max	0.0069	0.0030	0.0039	0.0065
Average	0.0010	0.0002	0.0010	0.0011
StdDev from Avg	0.0017	0.0007	0.0016	0.0026
StdDev from Zero	0.0020	0.0007	0.0019	0.0028
RMS	0.0020	0.0007	0.0019	0.0028
Tol Range				-0.0025
				0.0025
In Tol				164 (66.9%)
Out Tol				81 (33.1%)
Count	245			

Vector Group Hydro::Boat Error Vectors Blotches										
Name	Begin			End			Delta			
	X1 (in)	Y1 (in)	Z1 (in)	X2 (in)	Y2 (in)	Z2 (in)	dX (in)	dY (in)	dZ (in)	Mag (in)
788	-4.4676	-0.6467	2.0096	-4.4676	-0.6467	2.0098	-0.0000	0.0000	0.0002	0.0002
789	-4.7293	-0.6474	1.9556	-4.7293	-0.6474	1.9557	-0.0000	0.0000	0.0001	0.0001
790	-4.9901	-0.6306	1.8909	-4.9901	-0.6306	1.8908	0.0000	-0.0000	-0.0001	-0.0001
791	-5.2564	-0.6145	1.8123	-5.2563	-0.6145	1.8120	0.0001	-0.0000	-0.0003	-0.0003
792	-5.4653	-0.5024	1.7398	-5.4649	-0.5024	1.7388	0.0004	-0.0000	-0.0010	-0.0010
793	-5.4502	-0.2302	1.7435	-5.4499	-0.2302	1.7425	0.0004	-0.0000	-0.0010	-0.0011
794	-5.1600	-0.1743	1.8398	-5.1600	-0.1743	1.8397	0.0000	-0.0000	-0.0001	-0.0001

Figure 9-18. 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 9-19.

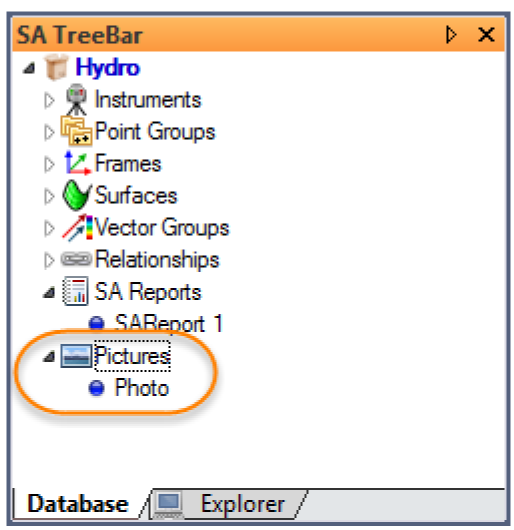


Figure 9-19. The screenshot has been added to the tree.

8. Click the photo, drag it over a table in the SA Report, and release the mouse button to add the photo to the report.

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 9-20).

10. Items in a report can be moved by dragging them manually and positioning them as you would like. To dynamically adjust the placement of items in the report right-click and select *Auto-Arrange> Set Auto-Arrange Start.* This will place a start tag in your report for where you want selection to start.

11. Right-click again and select Execute Auto-Arrange to remove overlap in tables below your start tag . If you noticed there is also an option to set a stop point for more control or to select all the entities so you can manually move them as a group.

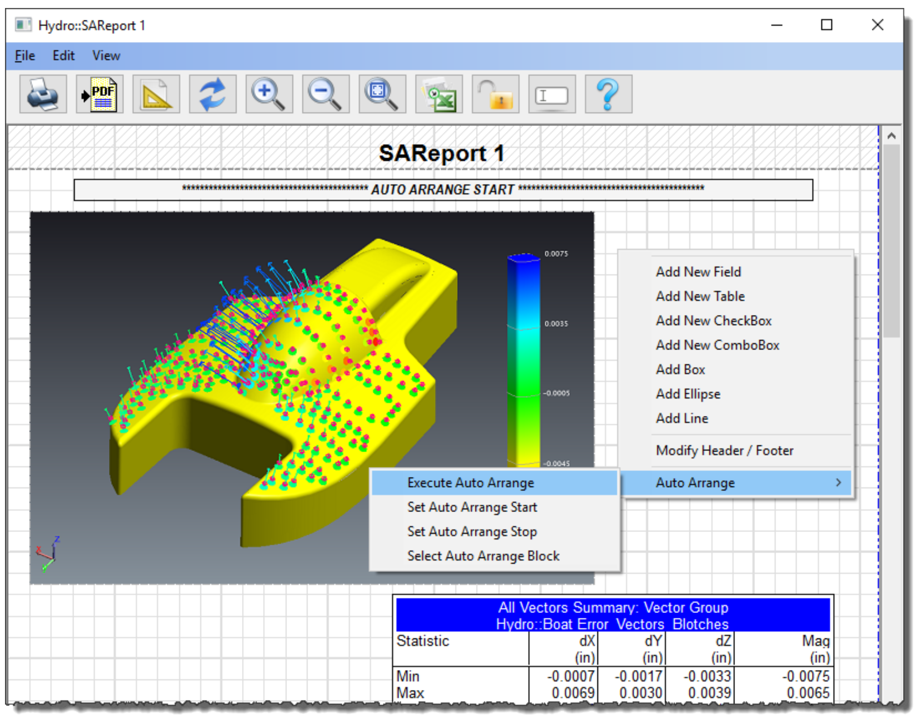


Figure 9-20. Auto-Arrange Controls

12. 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.

Advanced Instrument Operations

CHAPTER 10

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

Still Under Construction...

Instrument Operations

How to Get Accurate Time Stamps with Points in SA

As you may have noticed, measurement data from a tracker is only reported to the nearest second within SA. Read on to find out why, plus how to measure at precise increments and how to obtain sub-second timestamps in SA.

The first thing to keep in mind when attempting to record time with your measurements is that the vast majority of measurements are averages, not individual samples. If you are measuring "discrete" or average points over a period of one half to two seconds, time stamps may not be meaningful. Precise time stamps mainly have relevance when taken from a single sample obtained from a stream of points where the time and sample correspond.

Obtaining measurements at exact increments:

Standard point measurements are not a fast thing. They include a lot of measurement data and are passed to SA in packets to ensure there is no slow downs. The typical time stamp recorded with all points is used as a reference only and reflects the time to the second when the point was created in the tree, not when it was measured.

In order to record an accurate time with each measurement you will want to use either a Spatial (distance base) or Temporal (time based) measurement. Using a custom measurement profile will allow you to set a specific rate that suits your needs if you want a temporal measurement.

A basic temporal scan profile that will take a single sample at an exact time period, is simple to create. Start by opening a "Watch Update" profile. You will see the Acquisition is set to "Temporal Scan" and the Operation is set to "Send Updates to SA." If you change the Operation to "Send Points to SA," you will have created a Temporal Scan profile. The Sampling Frequency can be used to adjust the rate at which a point is recorded. Just use the "Save As" button to define and save that new profile so that you can use it again later (Figure 10-1).

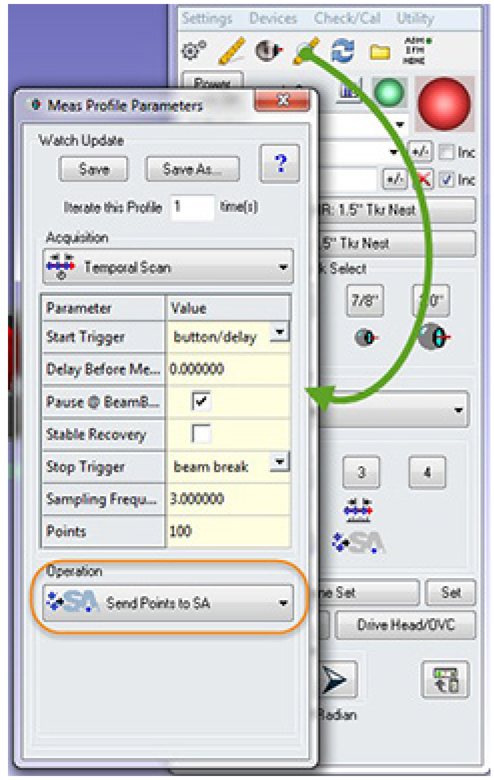


Figure 10-1. Modifying a Watch Update profile

Spatial (distance based) measurement profiles can also be used by choosing any measurement profile with an Acquisition set to "Spatial Scan". In a spatial scan the tracker will still be measuring at a constant sample rate but only measurements captured beyond pre-set distance or "Increment" will be recorded. This will capture points with an even distribution and the controller timestamp will still be available for that point.

Measurement profiles can be edited and adjusted as needed. When you save measurement profiles, they are saved in the persistence file. However if you would like them to be saved more permanently, or if you would like to move them to another machine, you can use the folder icon to manage and export your saved measurement profiles in a *.msp file.

Saving data in Point Sets

Starting with the 2020.04.09 version of SA, a new Point Sets data format is available. Point Sets provide a merged point format that allows many individual points to be saved as part of a single object, much in the same way point clouds record the point information. Using Point Sets provides the advantage of keeping the tree manageable and allowing many more points to be measured in a single job file at faster rates than are available with points.

To record data in a point set, right-click on the instrument in the tree and select "Enable Point Set Scan Mode". This will record points measured with a scan measurement profile as part of a Point Set, much like recording into a cloud (Figure 10-2). The difference is that each points includes a name, XYZ, and timestamp is recorded with each entry. This time is recorded directly from the instrument and is only adjusted such that the time is zero'ed when the scan is initiated. The meta data typically stored within each point's observation is stored once with the Point Set properties and can be passed to individual entries if you need to build individual points from the Point Set later.

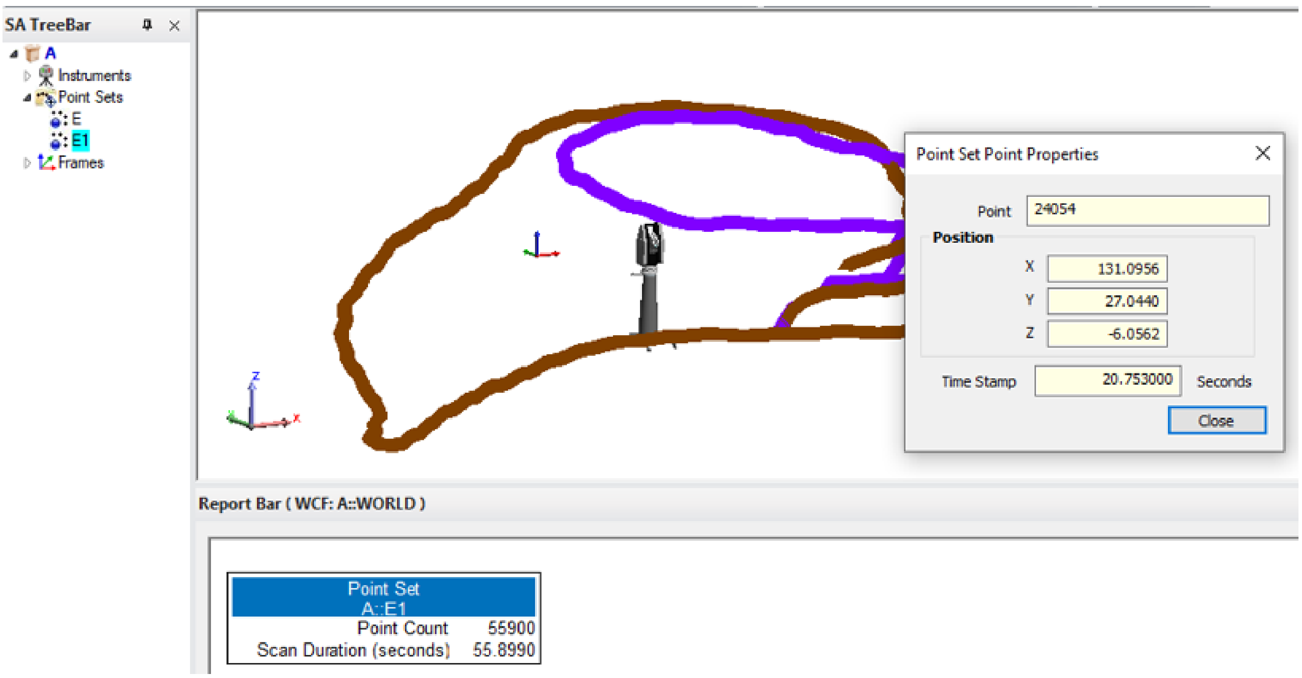


Figure 10-2. Example Point Set Data.

Recording Frame Sets with a 6D probe

If you have a 6D probe and can record measurements as Frames within SA, then starting in SA Release 2018.05.01 these frames can be collected as a "Frame Set". Much like a 6D point cloud each frame is a separate piece of a single item within a the tree. In addition to position and rotation information, a frame also has the accurate controller time saved with it.

By default frame measurements are recorded as individual frames stored separately within the tree. However, if you right-click on the frame category you will find an option to "Enable Frame Set Scan Mode" (Figure 10-3). This can be very helpful if you plan to measure many frames because recording each frame separately can make navigating your tree much more difficult. If you measure into a Frame Set then each scan can be separately saved and contain thousands of individual frames:

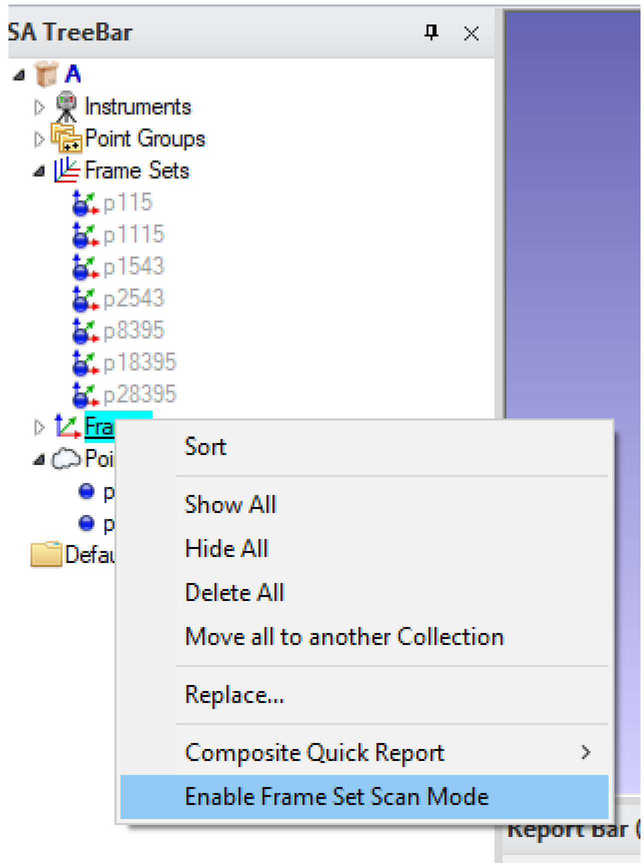


Figure 10-3. Enable Frame Set Scan Mode.

A frame set, much like a point cloud, will provide the frame count and duration of the scan. It also provides the visual control over all the frames saved within it, which can be used to visualize the 6D movement of a probe through space. To access the transform and time information from any individual frame saved with a frame set, it can be accessed by double clicking on it from within the graphic view (Figure 10-4):

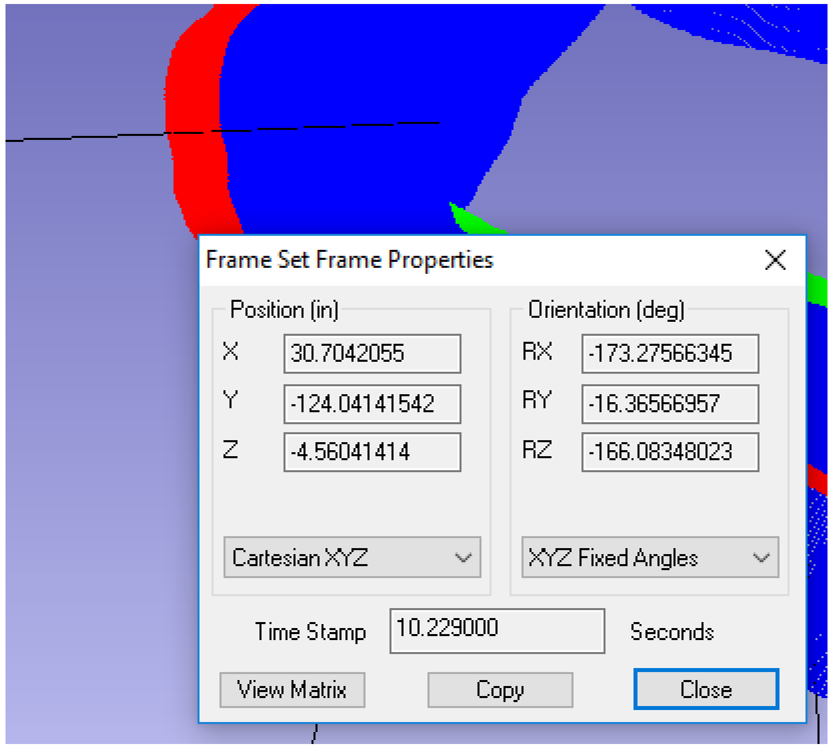


Figure 10-4. Frame Set Properties.

Obtaining tracker time with each sample:

There are applications that require obtaining the exact measurement time with each sample sent from the tracker. When data is sent to SA, it is sent as part of a network packet. Therefore, the time received in SA is still rough even when you break it down per sample. The internal clock on the tracker versus the Windows system clock is extremely accurate, so you need to be able to access that time. Using UDP will allow you to obtain the actual measurement time used by the tracker when each sample was taken. To capture UDP packets you must first tell the tracker interface to transmit the data. Within the tracker interface, go under Utility>UDP Data Stream and turn on 3D updates and Instrument Time Stamps (Figure 10-5):

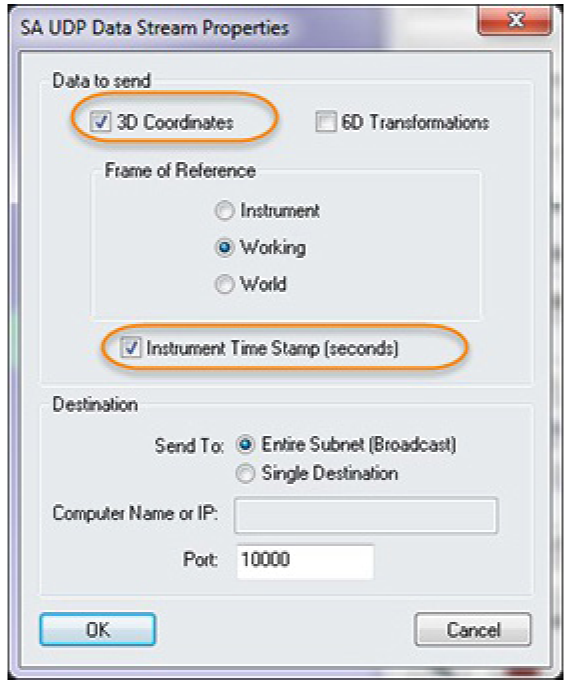


Figure 10-5. Enabling UDP Data Broadcast.

The next step is to record the data. Open the UDP Monitor directly from the **Help** tab in the **Diagnostics** section (Figure 10-6):

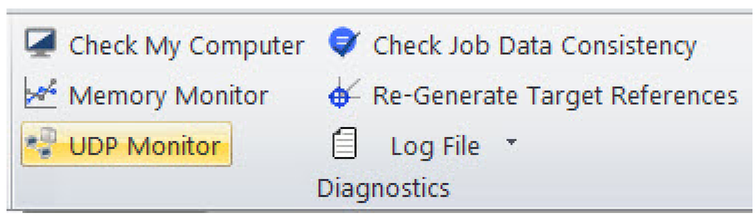


Figure 10-6. Open the UDP Monitor from the Help tab.

This utility will display line per line the coordinates and the exact controller time stamp with each sample passed across the UDP network in an asci format. This data can be saved and processed in a spreadsheet program like Excel (Figure 10-7).

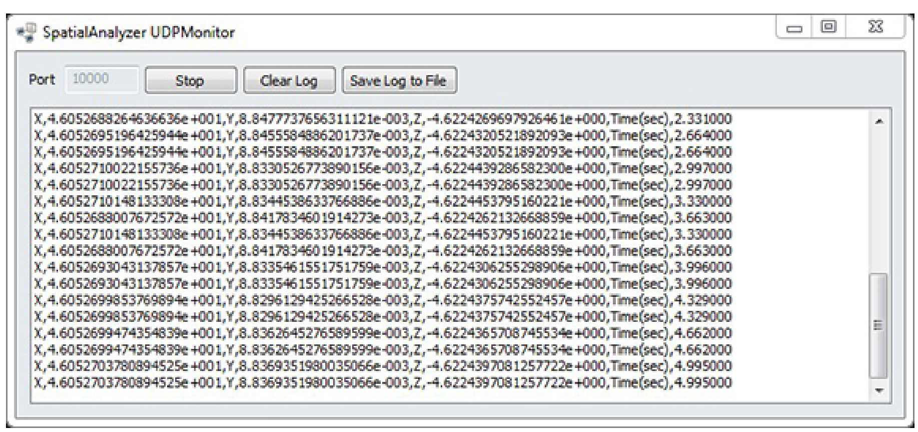


Figure 10-7. Recorded Data in the UDP

Unlike TCP/IP communication, UDP broadcasting does not require "handshaking" for data transmission. There is no guarantee that transmitted data is received, so some packets can be lost depending on network traffic. This means you are not guaranteed a recording of every sample, but each sample and the time captured with it will be as accurate as the tracker can record.

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 operations, or you can write your own custom watch window applications for near real monitoring processes.

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 10-8). 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.

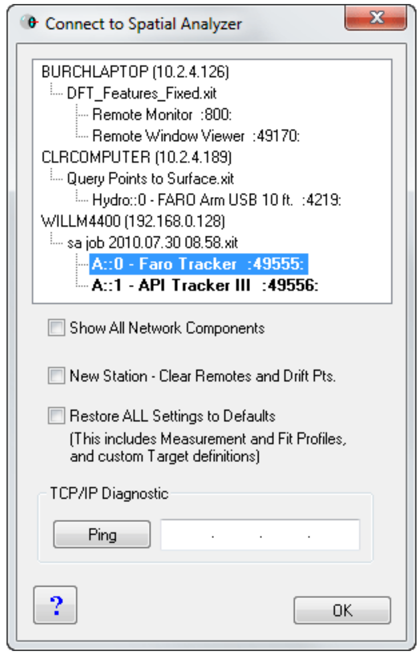


Figure 10-8. Selecting the appropriate instrument to connect to.

4. Repeat the above step for the second laser tracker (Figure 10-9).

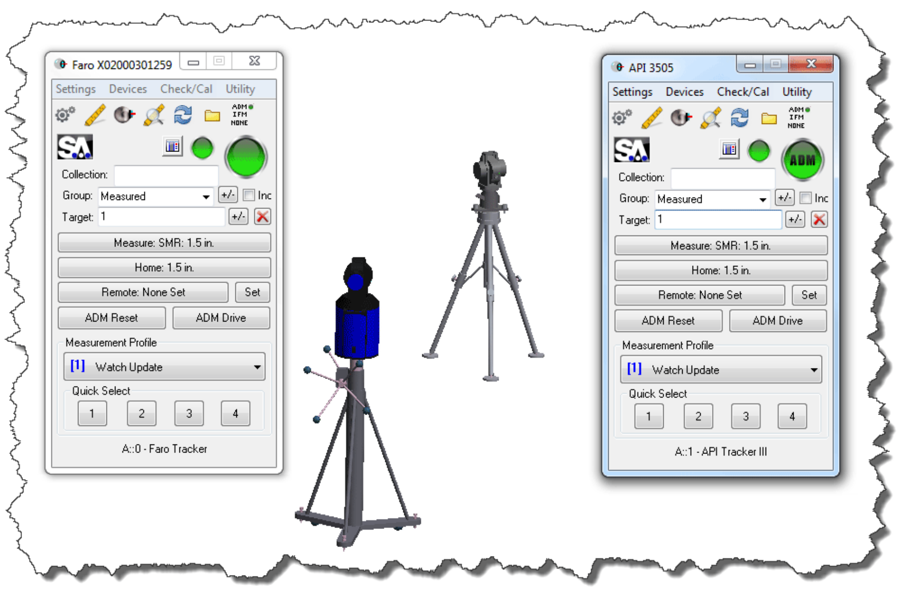


Figure 10-9. Starting a second interface.

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 10-10).

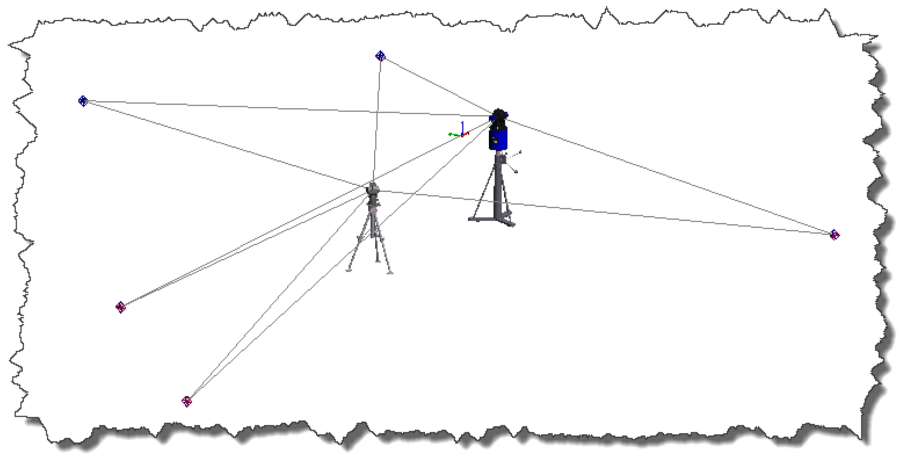


Figure 10-10. Fitting the two instruments together.

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.

This Page Intentionally Left Blank.

Advanced Alignments

CHAPTER 11

The following tutorials are designed to guide you through various topics in SA and get you up and running as quickly as possible. They are specific topic based step by step guides to accomplishing specific tasks but as a whole they should help directly though the software and help to understand the workflow and approach to operations within SA.

For greater depth and theory consider one of our courses and contact training@kinematics.com for more details.

Tutorial Index:

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 11-1).

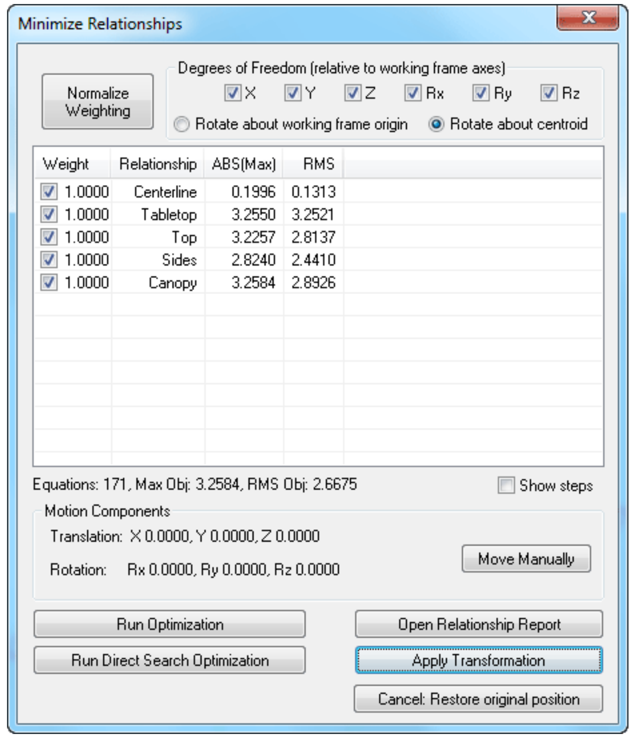


Figure 11-1. The Minimize Relationships dialog.

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 11-2), then click the Run Optimization button. This is the "standard" method of minimizing relationships.

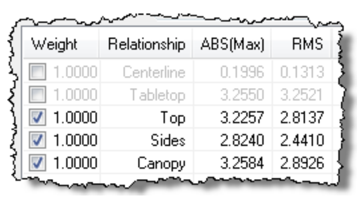


Figure 11-2. Excluding the Centerline and Tabletop relationships from the solution.

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 11-3).

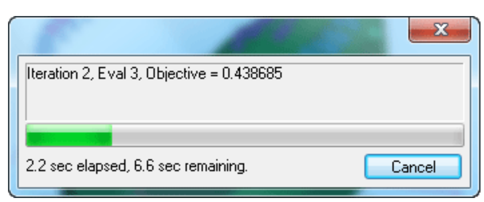


Figure 11-3. The objective progresses toward zero as the relationship solves.

6. The resulting absolute maximum and RMS errors are also displayed for each individual relationship in the ***Minimize Rela-***

tionships 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 11-4).

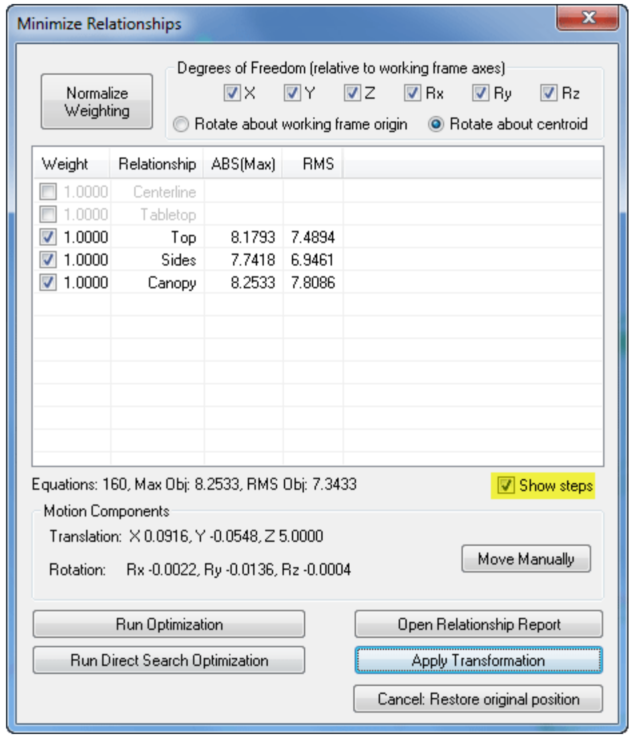


Figure 11-4. Turning on the Show Steps option.

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 Transforma-

tion 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 11-5).

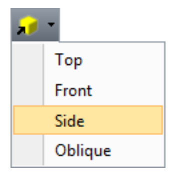


Figure 11-5. Selecting the predefined Side view.

4. From the menu, select **Instrument>Drag Graphically**. Hold down the right mouse button to rotate the instrument approximately 180 degrees (Figure 11-6).

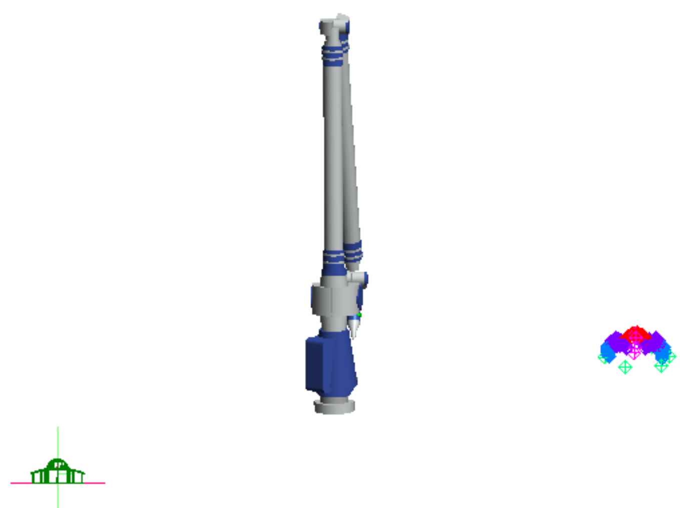


Figure 11-6. The instrument has been rotated about 180 degrees out.

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 11-7).

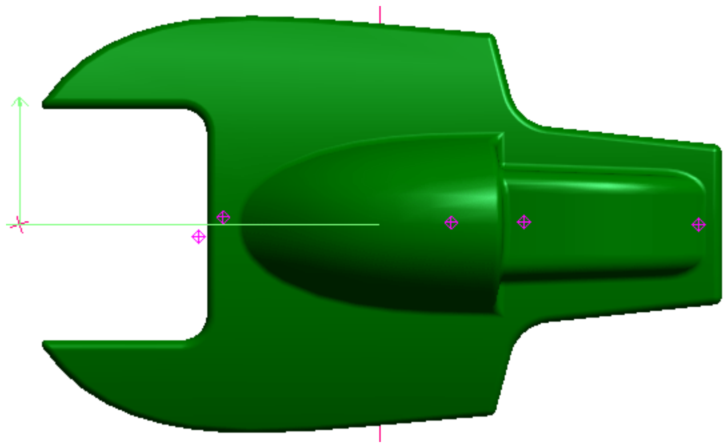


Figure 11-7. Showing the Centerline point group.

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 11-8*)*. 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 11-8. 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.

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 11-9). 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.

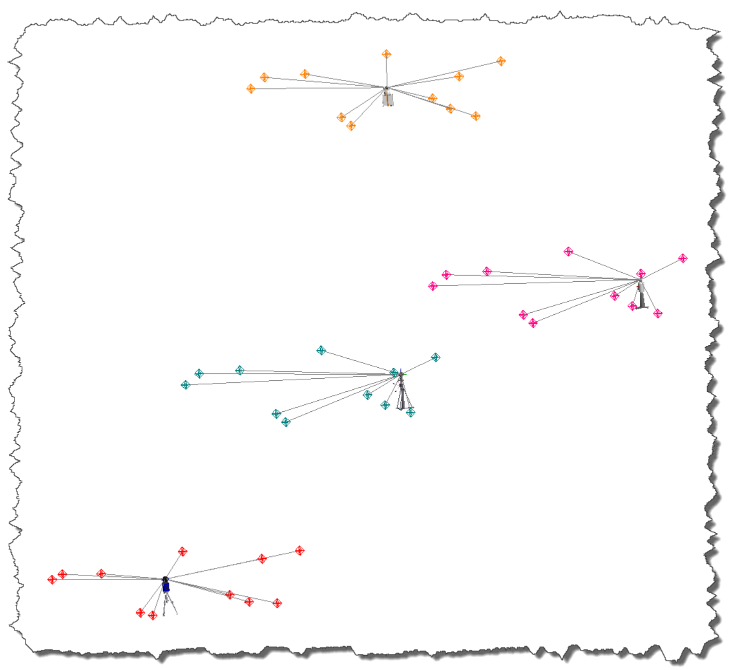


Figure 11-9. Prior to the USMN bundle.

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 11-10), and you will have the controls and tools to create and analyze the measurement network. The next few steps will walk through this process.

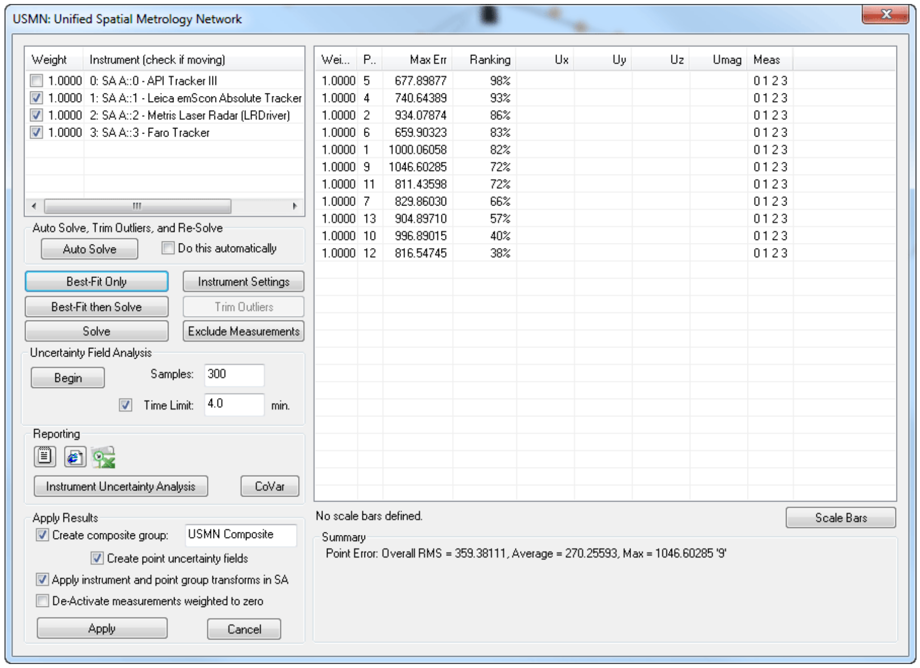
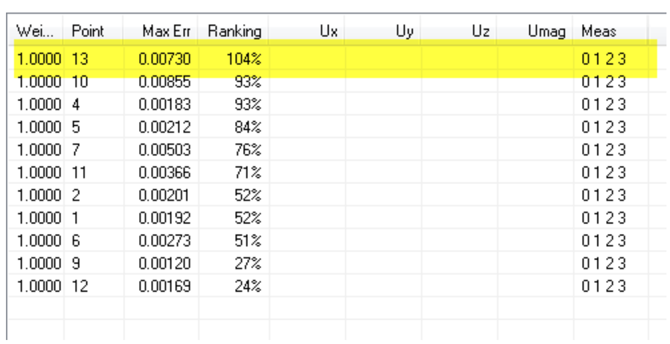


Figure 11-10. The USMN dialog.

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 11-11). 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 11-11. Point details, including Max Error and Ranking.

5. Press the Trim Outliers button to remove points that exceed a specified ranking (Figure 11-12). In our case, we will remove any points over 100%.

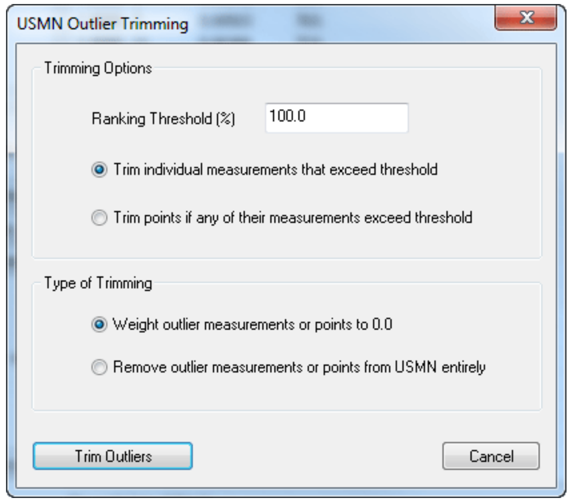


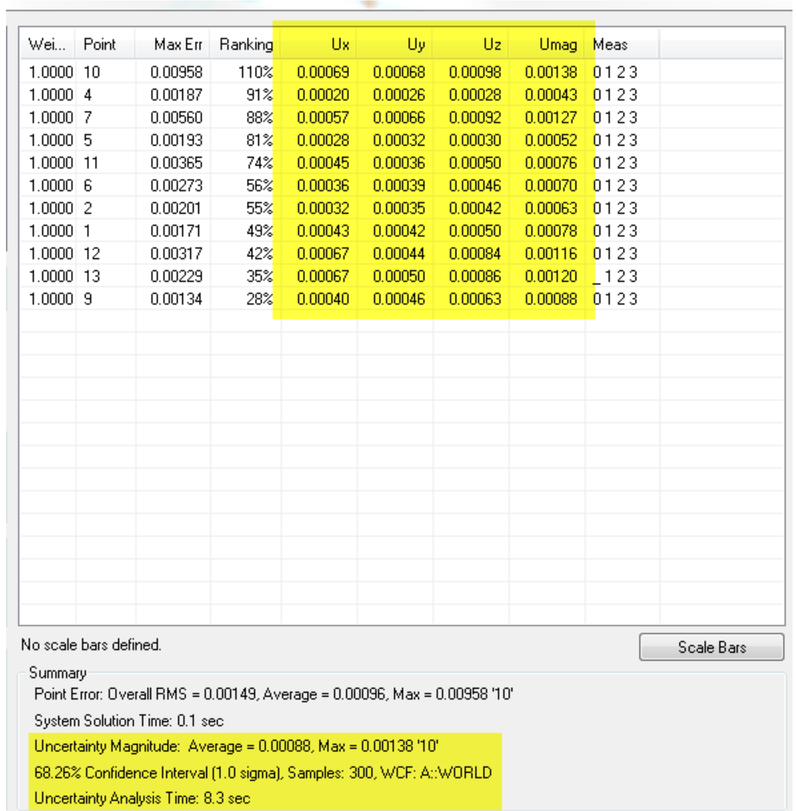
Figure 11-12. Trimming outliers.

6. Now press the Solve button to resolve the network for the remaining points. Now note the *Max error* and *Ranking* values (Figure 11-14). 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 11-13. 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 11-14).



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 11-14. Calculating composite point uncertainties.

8. Now apply the results of the USMN. Each of the four instruments should now be aligned (Figure 11-15).

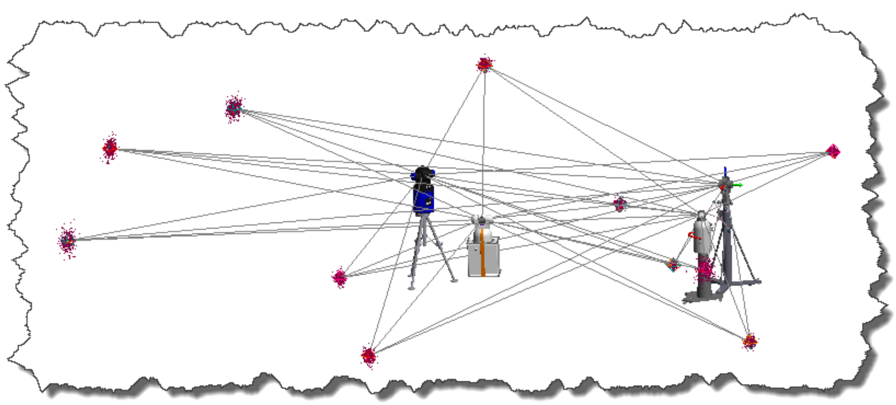


Figure 11-15. After the USMN bundle.

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 11-16 is a close up look at a composite point versus the measured points.

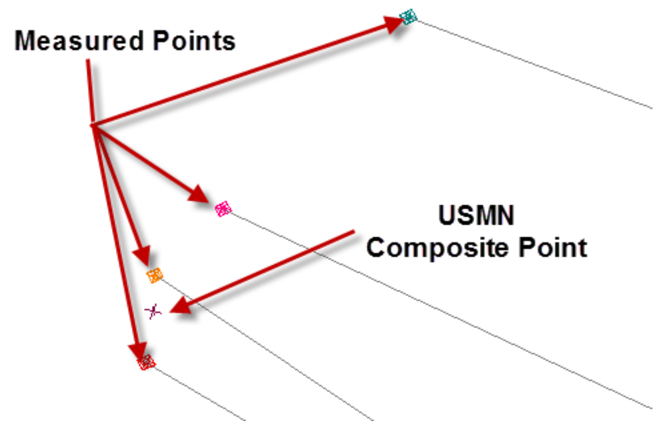


Figure 11-16. The position of the USMN composite point relative to the measured points.

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.

This Page Intentionally Left Blank.