

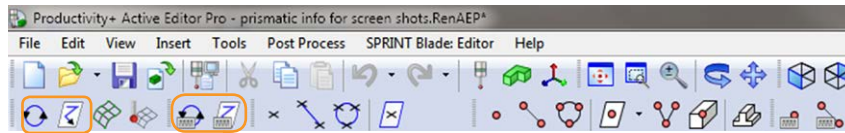
# Creating prismatic scanned feature statements

This module demonstrates how to select and program scanned circle, arc and plane features from an imported solid model.

In some cases you may find that a feature is not selectable or that you do not have a CAD model available to use for programming. If so, you can use Basic Scanned Statements instead of a CAD model to program the features.

Having completed this module you will be able to:

- Identify and use the various Scanned Statement icons
- Create Basic Scanned Statements



The Scanned Statement icons should be used with CAD models and the information from the CAD model is used to set a program.



The Basic Scanned Statements icons should be used when you do *not* have a CAD model. You will need to input all necessary information manually.

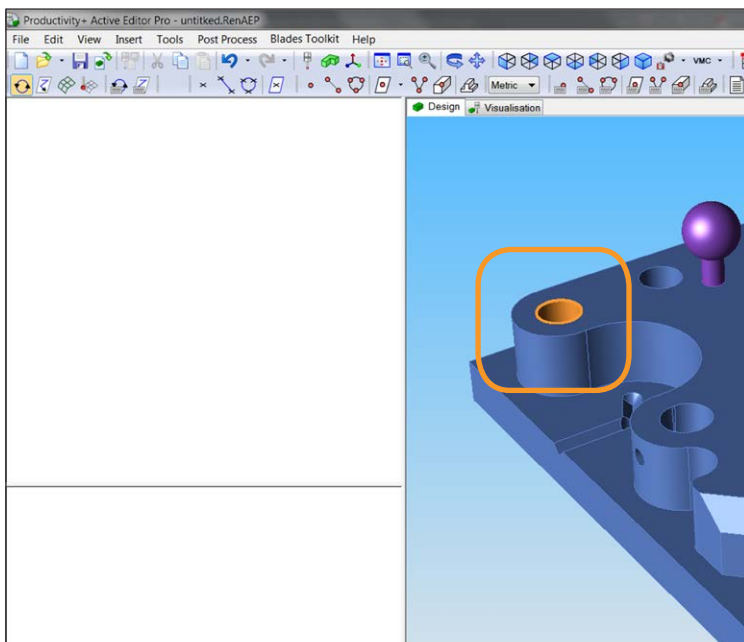


### Scanned circle feature: Circles

Ensure an OSP60 (SPRINT™) probe is defined in the database and selected.

Import and align the sample solid model, as covered in the 'Initial set-up' module.

To begin programming, you need to select the type of scanning feature you wish to perform on your component. In this example, a Scanned Circle feature has been selected.



As you move the cursor over the CAD model, circular and arc features will be highlighted (see image).

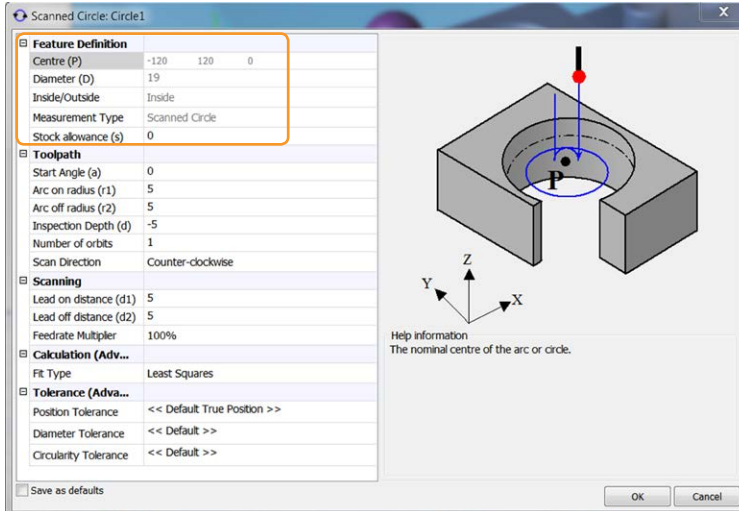
The ease of programming offered by Productivity+™ Active Editor Pro using the Scanned Circle feature allows you to select the circular feature to be measured and the edge on that circular feature to be scanned. Simply use a single mouse click then press 'Enter' or use a double mouse click.

Once the circular feature to be measured has been selected, a dialog box will appear. There are five sections in this dialog box:

**Feature Definition**, which details:

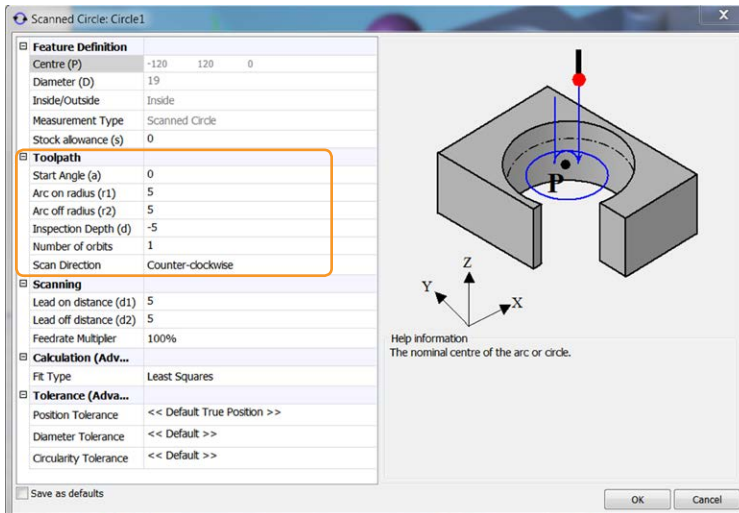
- the *Centre (P)*;
- the *Diameter (D)*;
- *Inside/Outside*, i.e. whether the circle to be scanned is an internal/bore or an external/boss feature;
- the *Measurement Type*, e.g. Scanned Circle; and
- whether any *Stock allowance (s)* is to be compensated for during scanning.

The elements shown in grey type are those taken from the CAD model and cannot be amended. Only the *Stock allowance (s)* field can be amended.



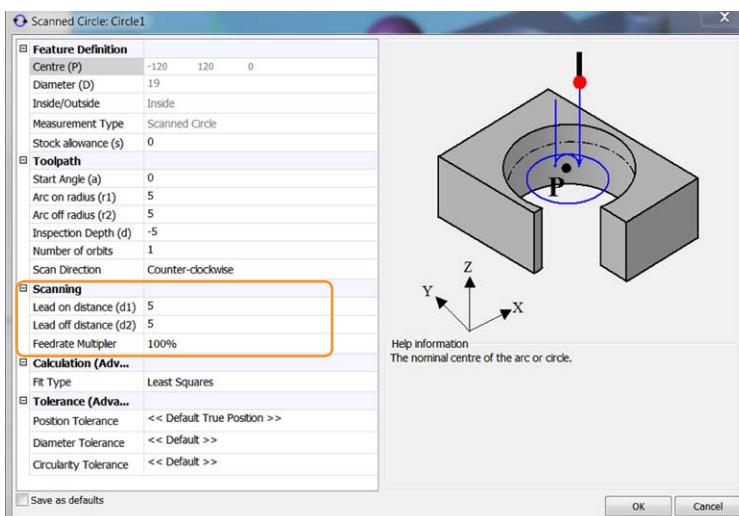
**Toolpath**, which allows you to define the following elements:

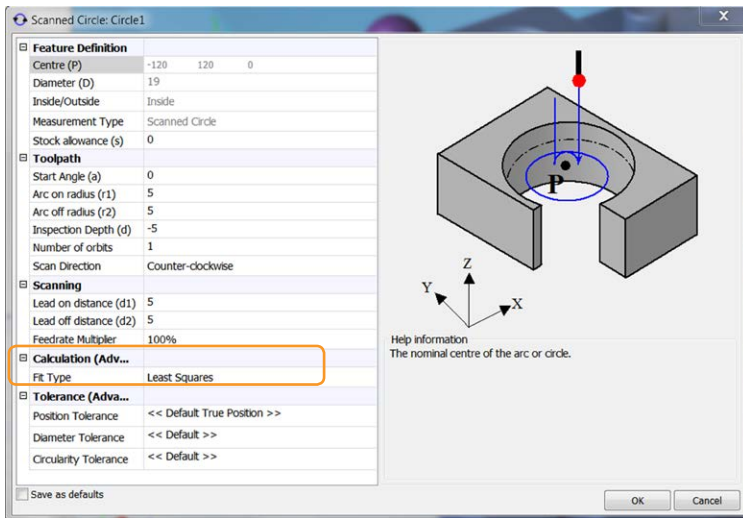
- the *Start Angle (a)*;
- the *Arc on radius (r1)* and *Arc off radius (r2)* – the default is 5 mm for each;
- the *Inspection Depth (d)* – this is the depth from the nominal surface to the contact point (equator) of the stylus ball;
- the *Number of orbits* (how many times the machine will scan the feature); and
- the *Scan Direction*, either clockwise or counterclockwise – the default is counterclockwise.



**Scanning**, which allows you to edit the *Lead on distance (d1)* and the *Lead off distance (d2)*. This is the distance to move around the circle before and after data is collected, and after the *Arc on radius (r1)* move and the *Arc off radius (r2)* move.

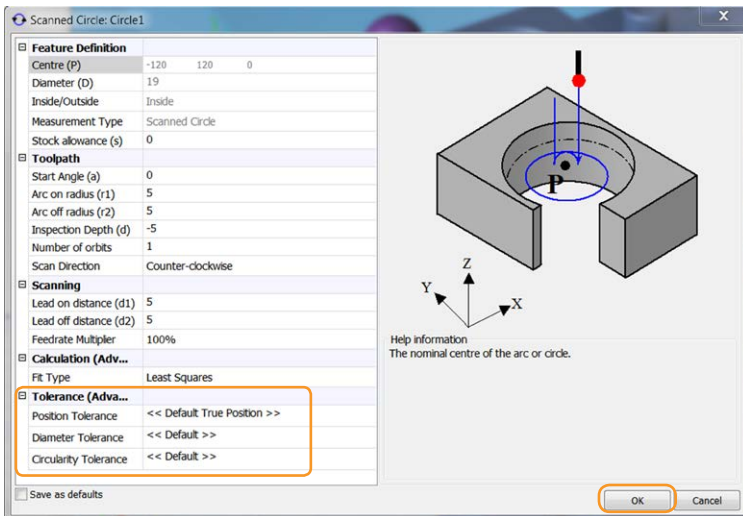
You can also edit the scanning feedrate using the *Feedrate Multiplier*. This is a percentage of the Scan Full Speed detailed in the .RENMF.





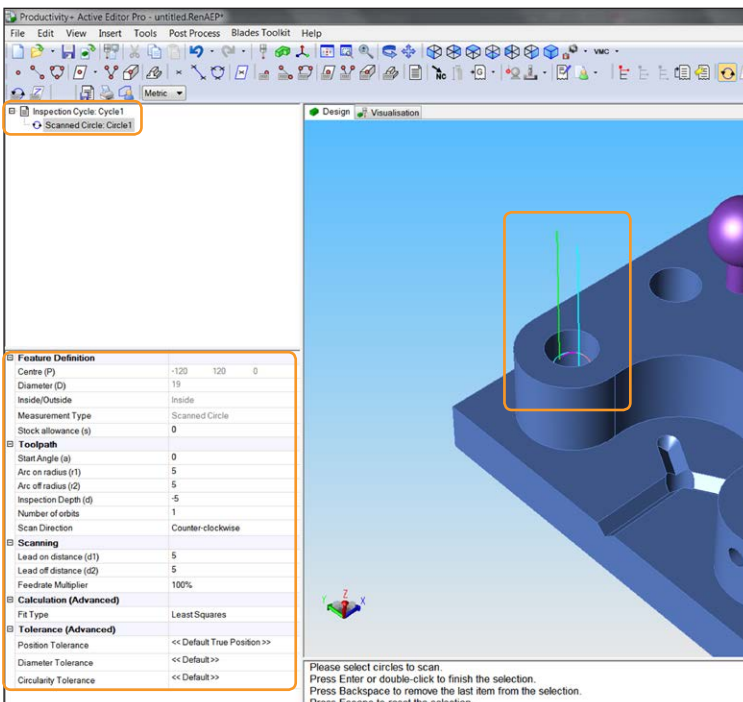
**Calculation**, in which you can change how Productivity+ Active Editor Pro calculates the accuracy of the scanning path using one of two methods:

1. *Least squares*. This will create a scan path that is the average fit of the points of the curve.
2. *Min zone*. This will calculate a curve that is the median of the maximum and minimum of the curve tolerance.



**Tolerance**, in which the *Position Tolerance*, *Diameter Tolerance* and *Circularity Tolerance* are populated by default from the inspection cycle, but can be edited to suit individual features.

Click on 'OK' once all the sections have been edited.



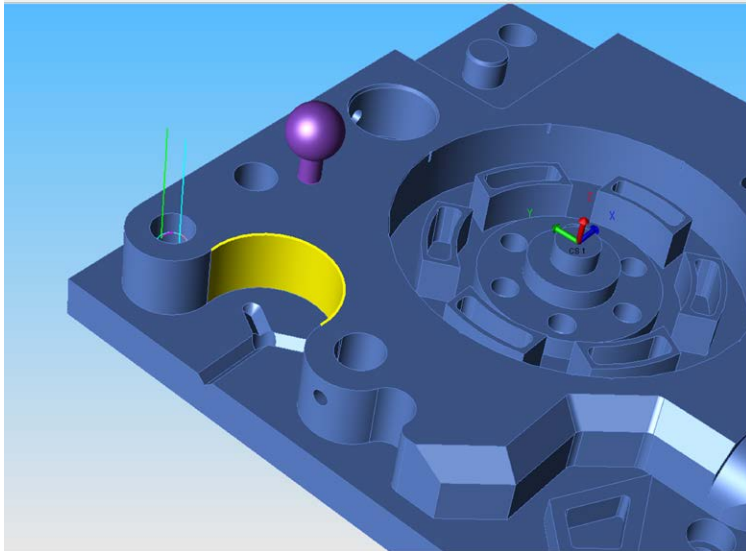
A Scanned Circle feature will be added to the process tree in the program viewer window and the probe path will be shown on the solid model.

The dialog box will appear below the process tree and you will be able to edit the Scanned Circle from here.



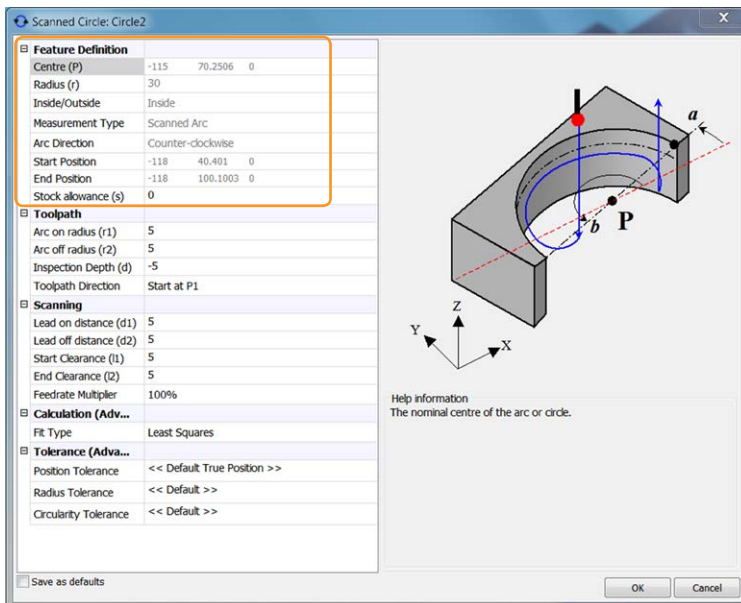
### Scanned circle feature: Arcs

Select the Scanned Circle feature on the toolbar.



Move the cursor over the solid model until an arc is highlighted.

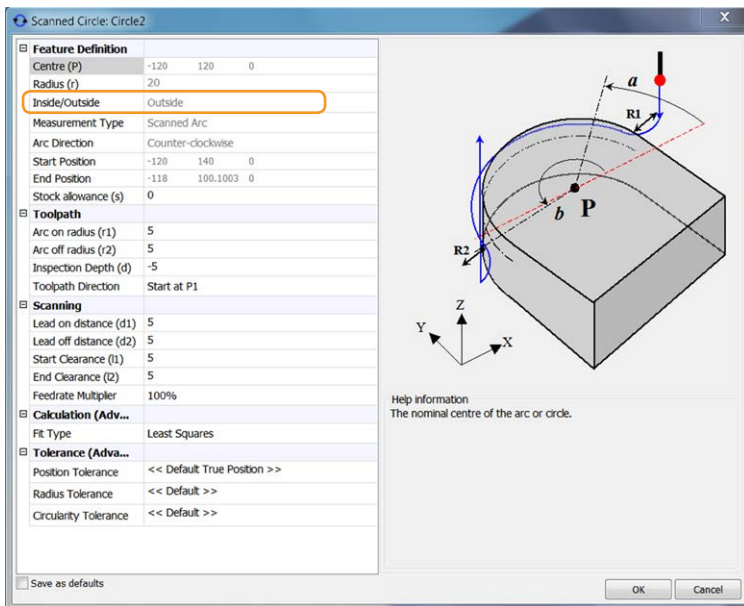
To confirm the selection of the arc to be scanned, click once on the highlighted arc then press 'Enter' or use a double click.



The Scanned Circle dialog box will appear. As you can see, this dialog box differs from the previous Scanned Circle dialog box – the graphic now details an internal arc.

This information, along with the **Feature Definition**, has been taken from the CAD model.

You can add stock to the **Feature Definition** by using the *Stock allowance (s)* section.

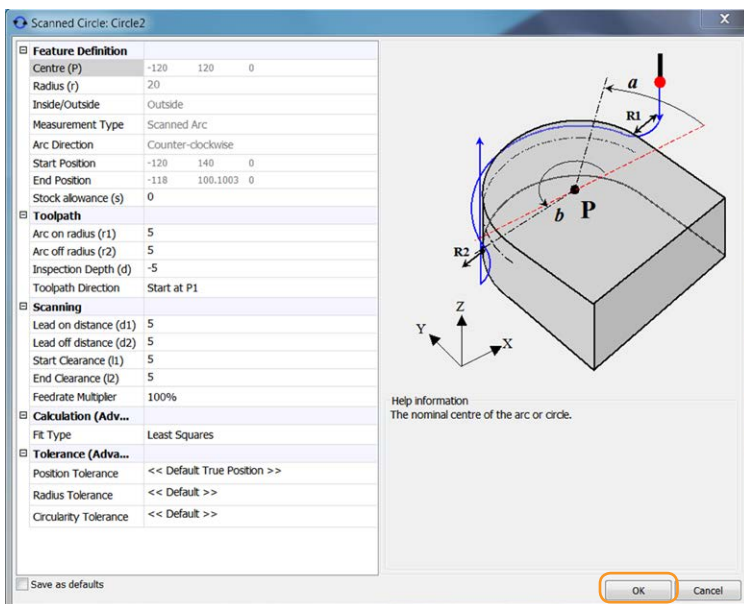


To scan an *external* arc, move the cursor over the solid model until an external arc is highlighted.

To confirm the selection of the arc to be scanned, click once on the highlighted arc then press 'Enter' or double click.

The scanned dialog box graphic will now detail an external arc.

The *Inside/Outside* field in the **Feature Definition** section will now say 'Outside'.



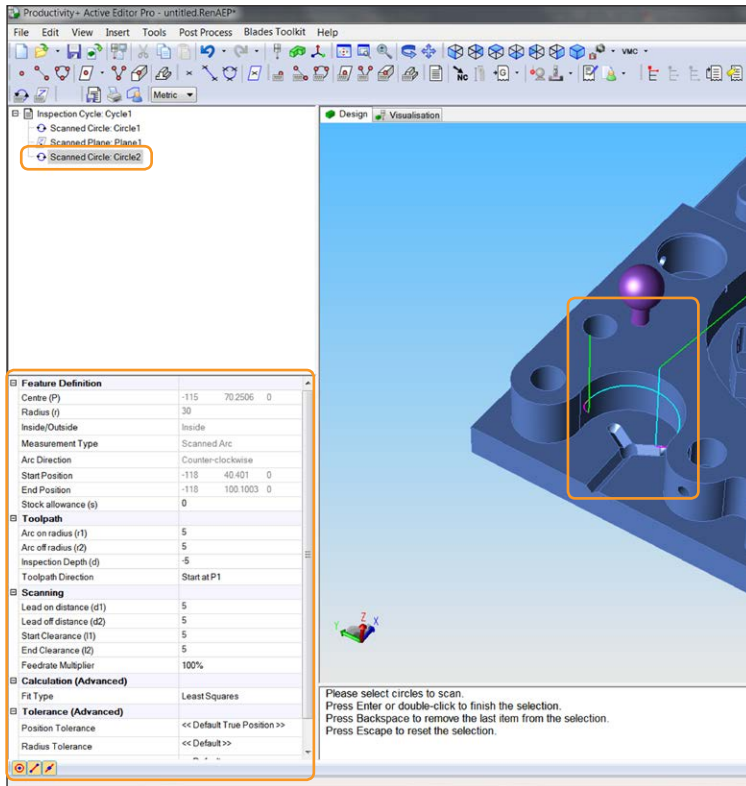
As with the previous Scanned Circle statement, the **Toolpath** and **Scanning** sections can be edited, so the *Arc on radius (r1)*, *Arc off radius (r2)* and *Inspection Depth (d)* can be changed to suit your application.

An editable *Toolpath Direction* element allows you to pick the start point of the arc.

In the **Scanning** section you will be able to edit the *Start Clearance (I1)* and the *End Clearance (I2)*. This is the distance around the arc from the start and end positions, where data is collected.

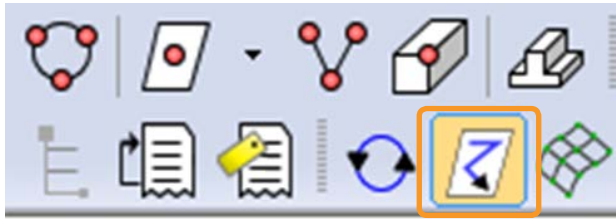
The **Calculation** and **Tolerance** sections remain the same.

Click on 'OK' once all the sections have been edited.



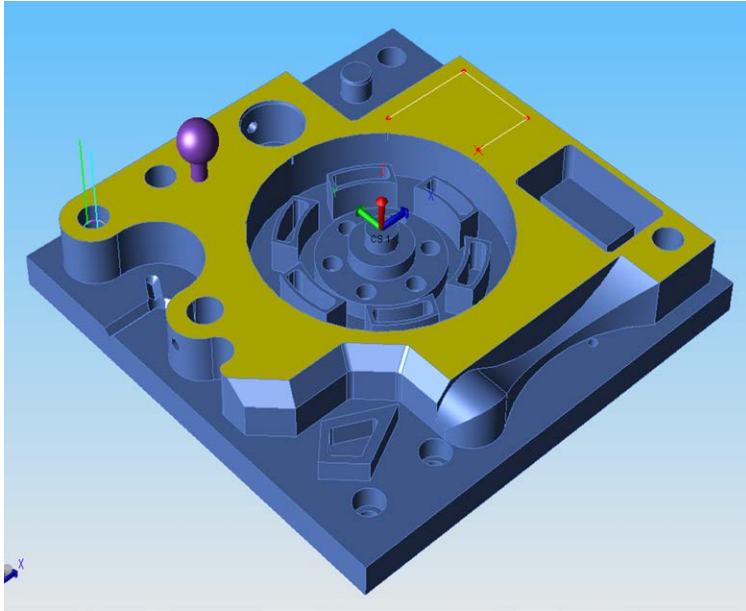
A Scanned Circle feature will have been added to the process tree in the program viewer window, and the probe path will be shown on the solid model.

Once again the dialog box will appear below the process tree and can be edited from here.

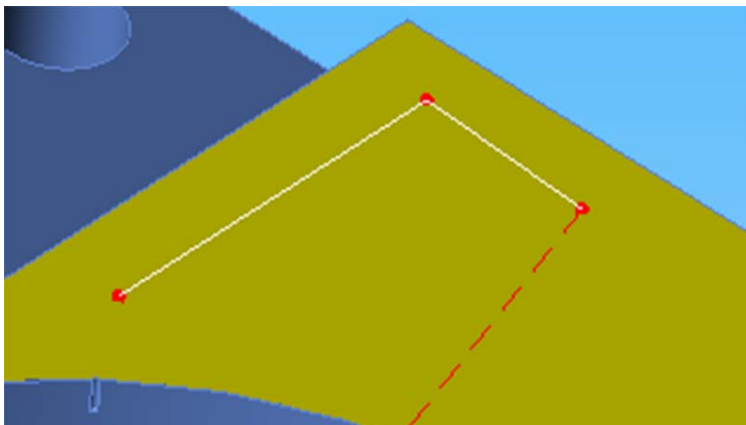


### Scanned plane feature

Select the scanned plane feature from the task bar.



As you move the cursor over the model it will highlight any plane feature which can be scanned from the model.

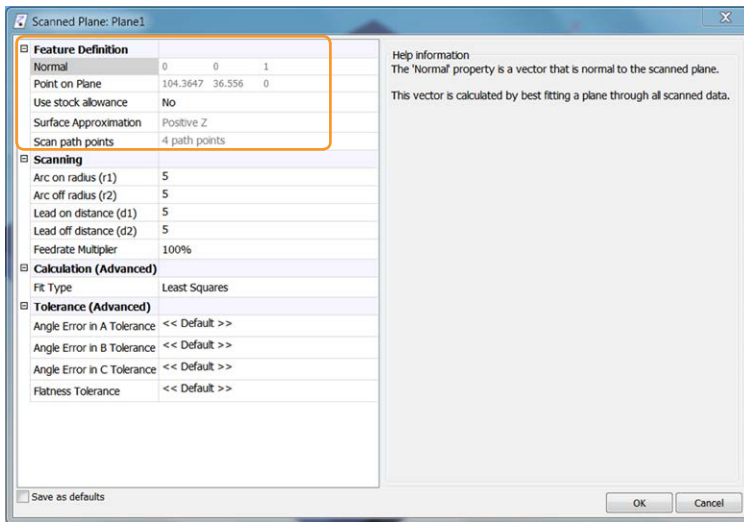


To generate a scanning toolpath, the scanned plane feature needs a minimum of three points on a surface.

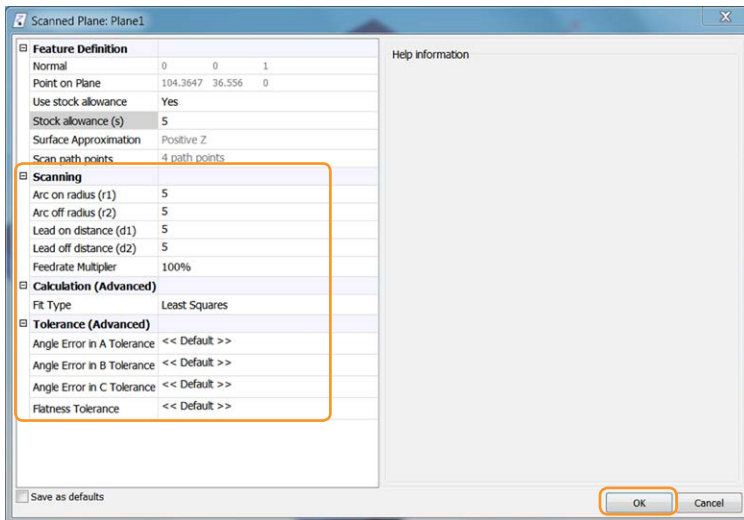
A maximum of 1,000 points can be used to define the scanned plane.

Select the required amount of points on the surface you wish to scan and press 'Enter'.

The Scanned Plane dialog box will appear.

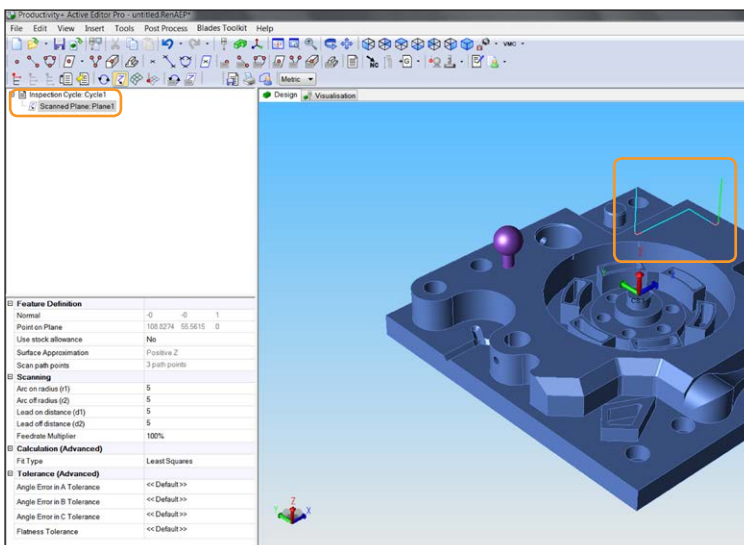


As with the Scanned Circle statement, there are elements of the **Feature Definition** section which appear in grey type. Once again, this is data from the CAD model and cannot be amended.



As with the Scanned Circle statement the **Scanning**, **Calculation** and **Tolerance** sections are also defined here.

Click on 'OK' once all the sections have been edited.



You will now see that a Scanned Plane feature has been added to the process tree in the program viewer window, and that the probe path is shown on the solid model.

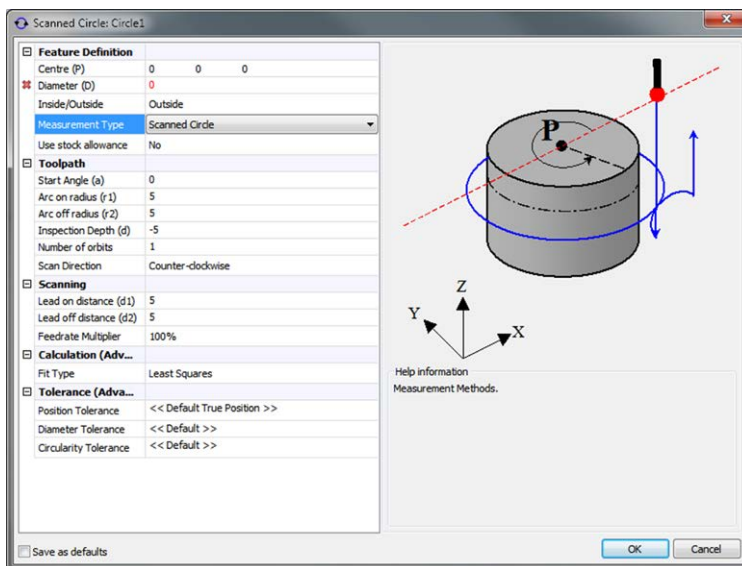
## Basic scanned circle

The following section covers the creation of Basic Scanned Statements, which are used to generate a probing program *without* a CAD model to select from. Instead the X, Y and Z co-ordinates are entered manually in order to determine where to scan a feature. Programming in this way is useful if you do not have a CAD model of your component or if you are unable use certain features on your CAD model.

You can create the same features as Scanned Statements with Basic Scanned Statements.



Select the Basic Scanned Circle feature from the toolbar. The Scanned Circle dialog box will appear.

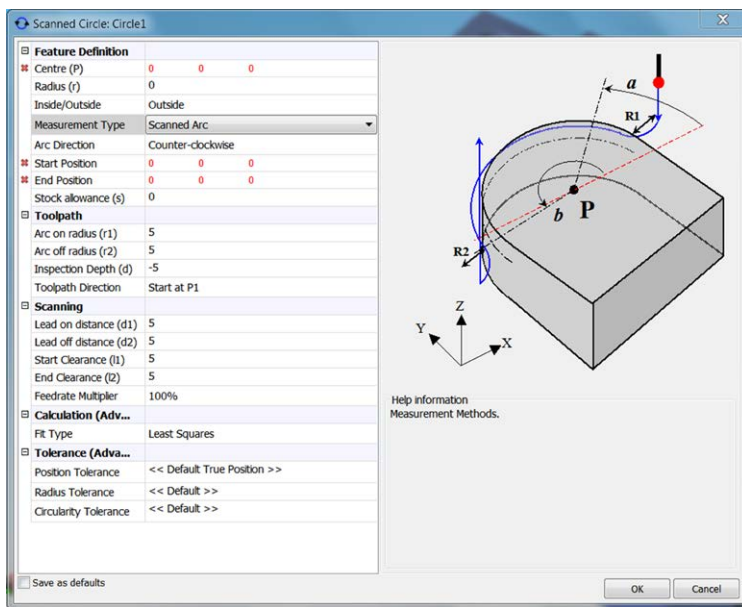


This example shows a circle feature.

The elements in the **Feature Definition** section are not greyed out as they were for the Scanned Circle Feature and will need to be added.

Enter the X, Y and Z positions into *Centre (P)* to determine the centre point of the circle.

Define the *Diameter (D)* and fill the *Inside/Outside* field to determine if the circle is a boss (*Inside*) or a bore (*Outside*). Select 'Scanned Circle' from the *Measurement Type* drop-down menu.



This example shows an arc feature which has been created using the Basic Scanned Circle icon.

In the **Feature Definition** section: add the X, Y and Z co-ordinates to *Centre (P)*; define the *Radius (r)*; specify whether the arc is internal (*Inside*) or external (*Outside*); select 'Scanned Arc' from the *Measurement Type* drop-down menu; and input the *Start Position* and *End Position* of the arc using X, Y and Z co-ordinates.

In the *Arc Direction* field you can specify whether to scan the arc in a clockwise or counterclockwise direction.

Any remaining elements in the dialog box, whether for Scanned Circle or Scanned Arc, can be dealt with in the same way as in the Scanned Circle Statement dialog box, for example, *Inspection Depth*, *Toolpath Direction* etc.

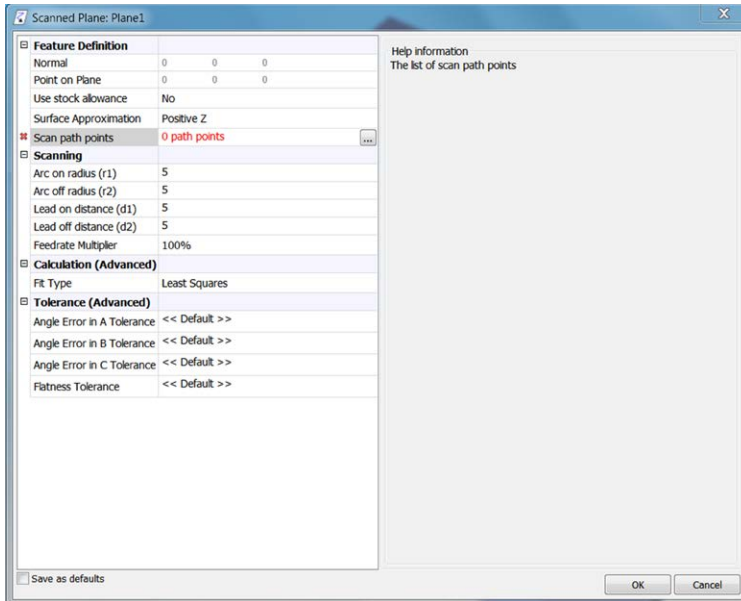
## Basic scanned plane

You can create a Scanned Plane using the Basic Scanned Plane feature.

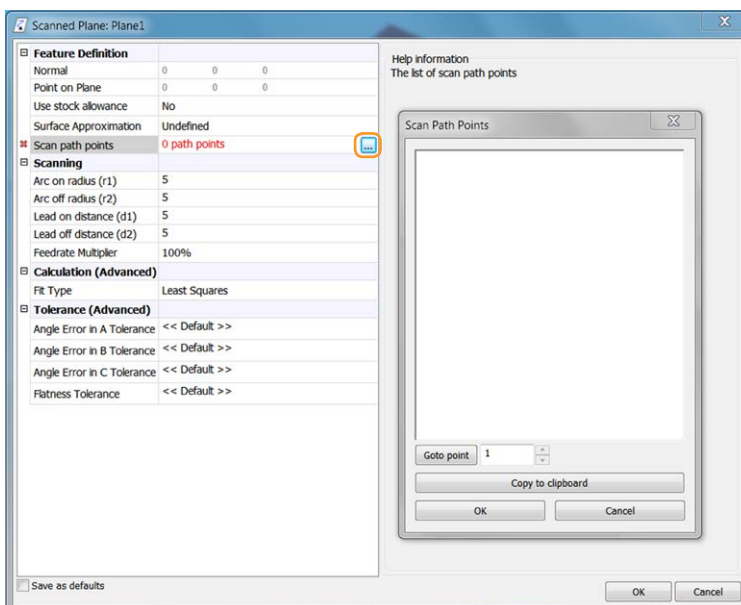
For a Basic Scanned Plane feature you need to insert X, Y and Z point data to create a Planar Scan toolpath. You will need a minimum of three path points and can have a maximum of 1,000.



Select the Basic Scanned Plane feature from the toolbar.



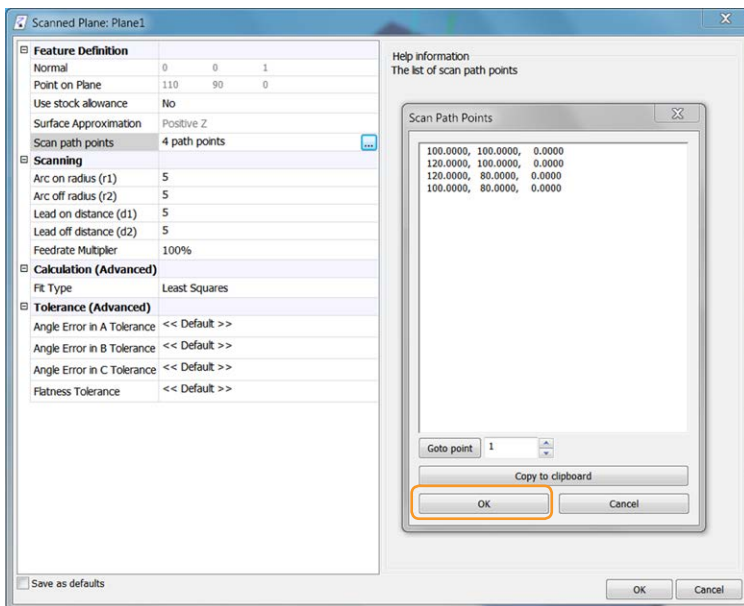
The Scanned Plane dialog box will appear.



To insert path points, click on the ellipsis button (highlighted).

You will need to enter the scan path points in the Scan Path Points dialog box as X, Y and Z positions, separated by a comma, for example: 100, 100, 0 (X100, Y100, Z0).

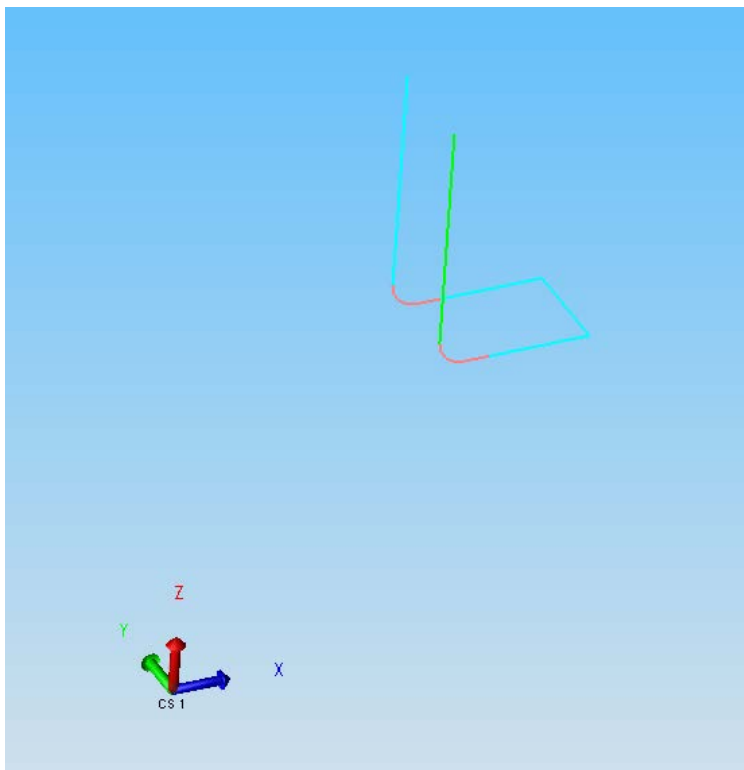
Enter a minimum of three points to create a scanned plane, as shown on the following screen shot.



Once you have entered the path points, click on 'OK' to exit the Scan Path Points dialog box.

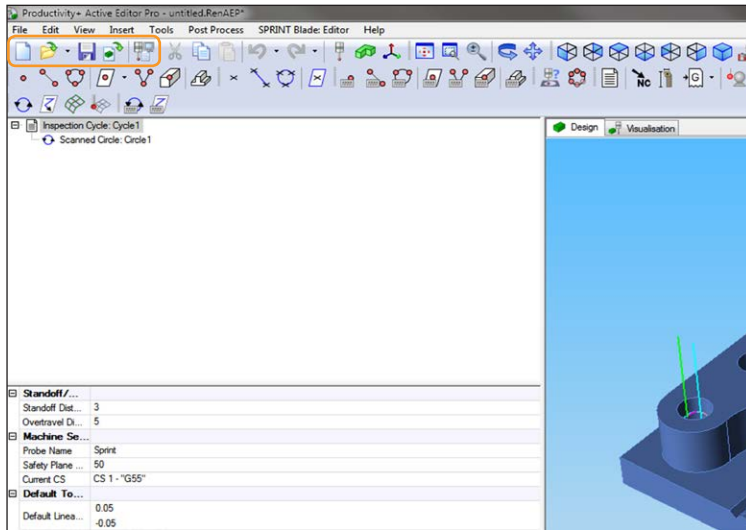
Edit the remaining fields in the same way as when using the Scanned Plane Statement icon, for example: *Arc on radius (r1)* and *Arc off radius (r2)* etc.

Once completed click 'OK' on the Scanned Plane dialog box.



A toolpath will now be visible in the Model viewer window.

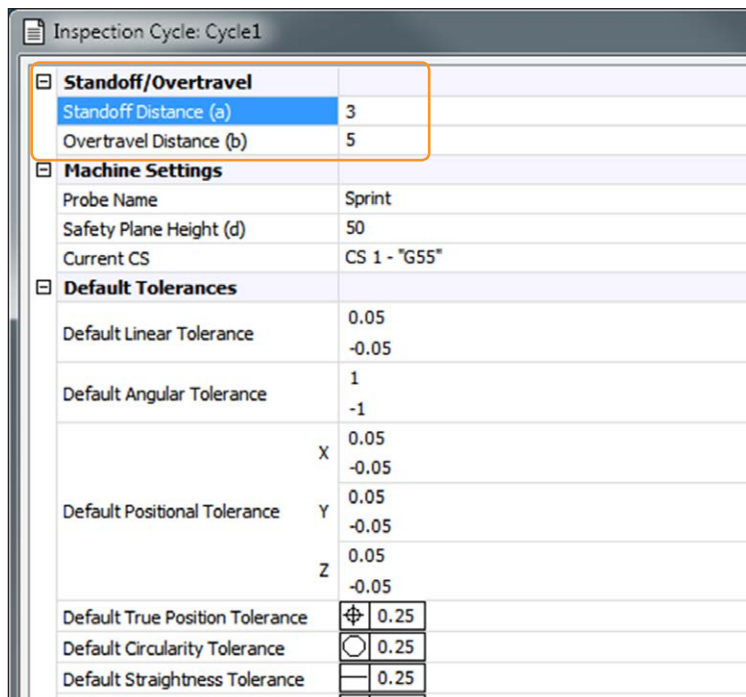
## Editing an inspection cycle



When you click on a feature on your CAD model or create a Basic Scanned Statement, an element with that feature name will be added to the process tree.

All features branch off an Inspection Cycle. By double clicking on an inspection cycle you can open the cycle for editing.

The following examples cover the fields that are available to edit.










The first two editable fields are *Standoff Distance (a)* and *Overtravel Distance (b)*. When either of these are highlighted, explanation text will appear in the 'Help information' area (lower right-hand-side of the dialog box).

The *Standoff Distance (a)* might need to be reduced if, when trying to probe a small bore/pocket, the stylus hits the back edge of the feature being measured. For example, a 9 mm hole measured with a 6 mm stylus will need a standoff distance of 3 mm or less.

The *Overtravel Distance (b)* is the distance to be searched past the surface you want to touch. This value can be increased if you want to create a bigger search distance.

In the *Probe Name* field, enter the name of probe to be used for measuring the component.

Inspection Cycle: Cycle1

<b>Standoff/Overtravel</b>		
Standoff Distance (a)	3	
Overtravel Distance (b)	5	
<b>Machine Settings</b>		
Probe Name	Sprint	
Safety Plane Height (d)	50	
Current CS	CS 1 - "G55"	
<b>Default Tolerances</b>		
Default Linear Tolerance	0.05	
	-0.05	
Default Angular Tolerance	1	
	-1	
Default Positional Tolerance	X	0.05
		-0.05
	Y	0.05
		-0.05
	Z	0.05
		-0.05
Default True Position Tolerance	 0.25	
Default Circularity Tolerance	 0.25	
Default Straightness Tolerance	 0.25	
Default Flatness Tolerance	 0.25	
Default Parallelism Tolerance	 0.25	
Default Angularity Tolerance	 0.25	
Default Perpendicularity Tolerance	 0.25	

Save as defaults

Specify the *Safety Plane Height* by entering the value required. This is the distance above the current Z zero offset which has been set in *Current CS*.

Use *Current CS* to choose the work co-ordinate system in which to measure.

Under **Default Tolerances** you can set the tolerances of the measured part. When reporting tolerances and a property is outside this tolerance, the property is reported as out of tolerance.

Click on the 'Save as defaults' check box to apply these settings to all features programmed subsequently.

**Renishaw plc**  
New Mills, Wotton-under-Edge,  
Gloucestershire GL12 8JR  
United Kingdom

**T** +44 (0) 1453 524524  
**F** +44 (0) 1453 524901  
**E** uk@renishaw.com  
[www.renishaw.com](http://www.renishaw.com)

**RENISHAW**   
**apply innovation™**



**For worldwide contact details please see our main website at**  
**[www.renishaw.com/contact](http://www.renishaw.com/contact)**

RENISHAW HAS MADE CONSIDERABLE EFFORTS TO ENSURE THE CONTENT OF THIS DOCUMENT IS CORRECT AT THE DATE OF PUBLICATION BUT MAKES NO WARRANTIES OR REPRESENTATIONS REGARDING THE CONTENT. RENISHAW EXCLUDES LIABILITY, HOWSOEVER ARISING, FOR ANY INACCURACIES IN THIS DOCUMENT.

©2014 Renishaw plc. All rights reserved.

Renishaw reserves the right to change specifications without notice

RENISHAW and the probe emblem used in the RENISHAW logo are registered trademarks of Renishaw plc in the UK and other countries.  
apply innovation is a trademark of Renishaw plc.



H - 4007 - 0045 - 01

Issued July 2014 Part no. H-4007-0045-01-A