

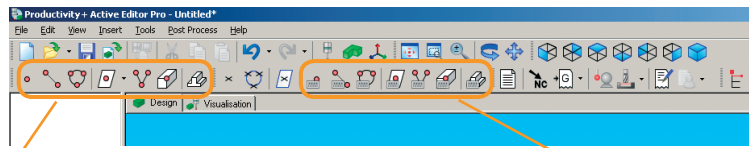
Creating feature statements

This module demonstrates how to select and program various features from an imported solid model.

In some cases you may find that a feature is not selectable or you do not have a CAD model available to use for programming. In such instances you can use Basic Statements to program the same features that you would using a CAD model.

Having completed this module you will be able to:

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



The Measured Feature icons should be used with CAD models. By using these, the information from the CAD model is used to program.



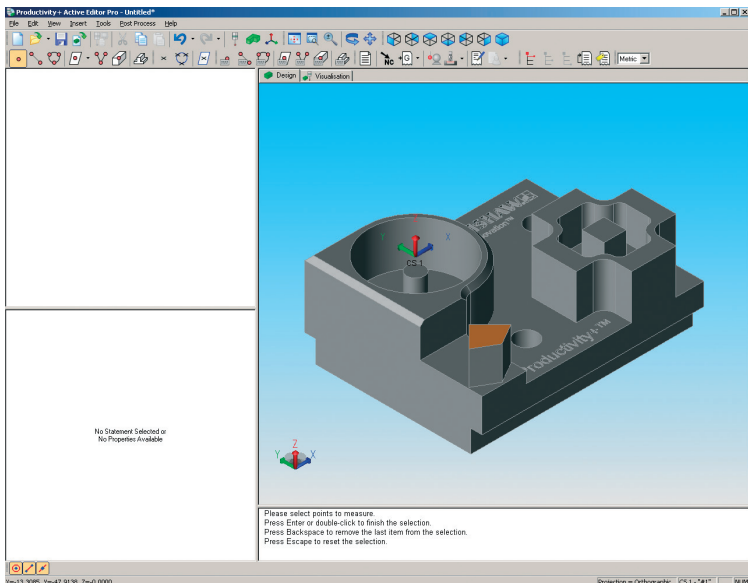
The Basic Statements should be used when you do not have a CAD model. You will need to input all necessary information manually.



Measured Point feature

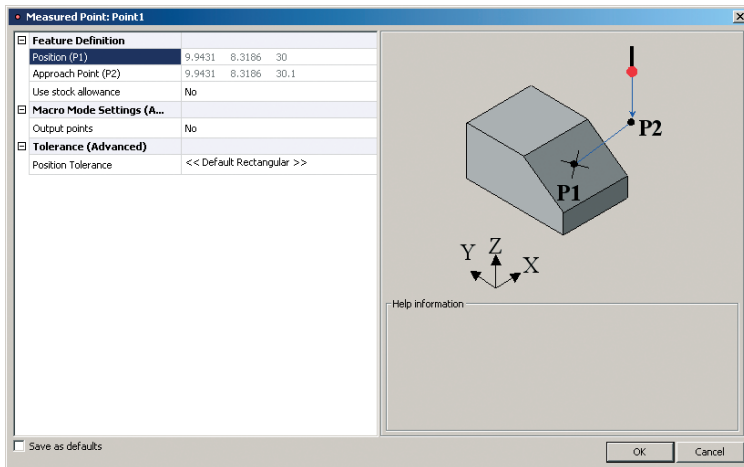
Import and align the sample solid model as covered in the 'Initial set-up' module.

To begin programming you need to select the type of probing feature you wish to perform on your component. In this case a Point feature has been selected. As you work through this module you will use the same process for additional feature types.



As the cursor is moved over the CAD model, individual faces are highlighted.

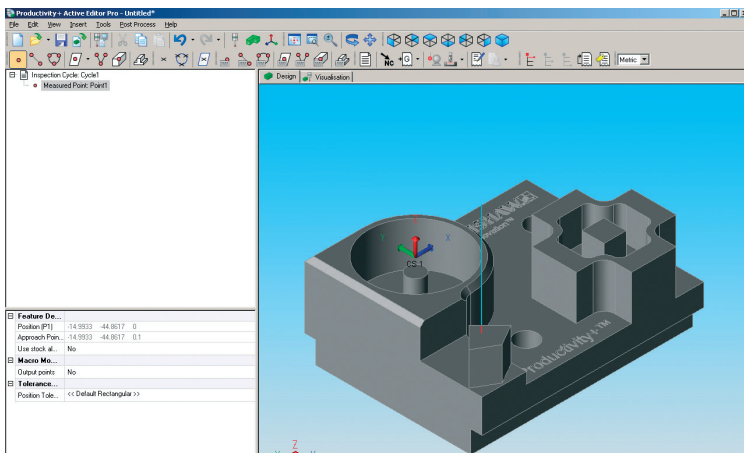
The ease of programming offered by Productivity+™ Active Editor Pro allows you to select the face on which you would like to measure the point (the highlighted face), and the position on that face to probe the point (either using double mouse click, or a single mouse click and Enter).



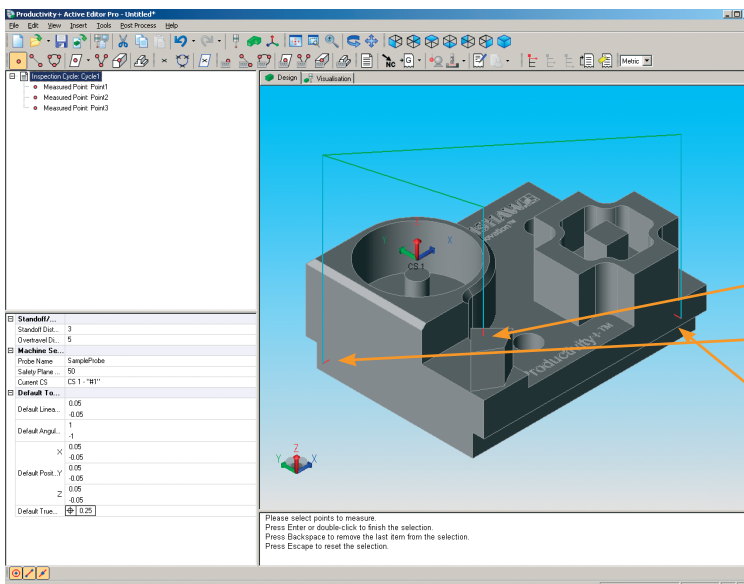
Having selected a position where the machine will probe, a dialog box will appear showing the position of the point in X, Y and Z. These dimensions are from your datum position, e.g. G54.

Note: The probe point you have just created is in the Z direction as a top face was highlighted. If you want to probe on a side face, simply select that face with the cursor and double click (or single click and Enter) on the CAD model where you want to measure the point.

The face selected dictates the probing direction.
 Now select OK.



You will now see that a Point feature has been added to the process tree in the Program Viewer window, and that the probe path is shown on the solid model.



Now use the Measured Point feature to select a probe point on an X face, followed by a Y face.

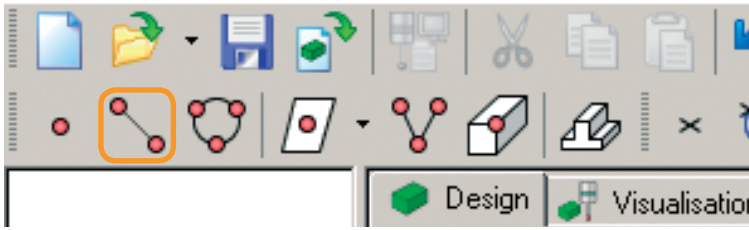
The screenshot on the left shows the Point feature used twice more on different faces.

Point 1 is in the Z- direction

Point 2 is in the X+ direction

Point 3 is in the Y+ direction

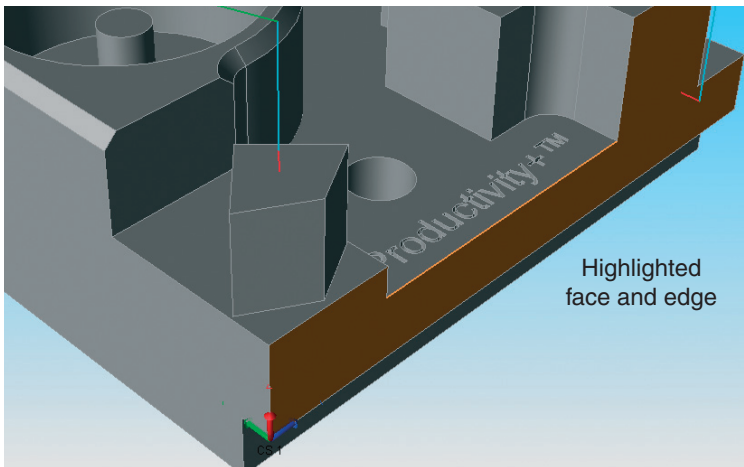
Please select points to measure.
 Press Enter or double click to finish the selection.
 Press Backspace to remove the last item from the selection.
 Press Escape to reset the selection.



Measured Line feature

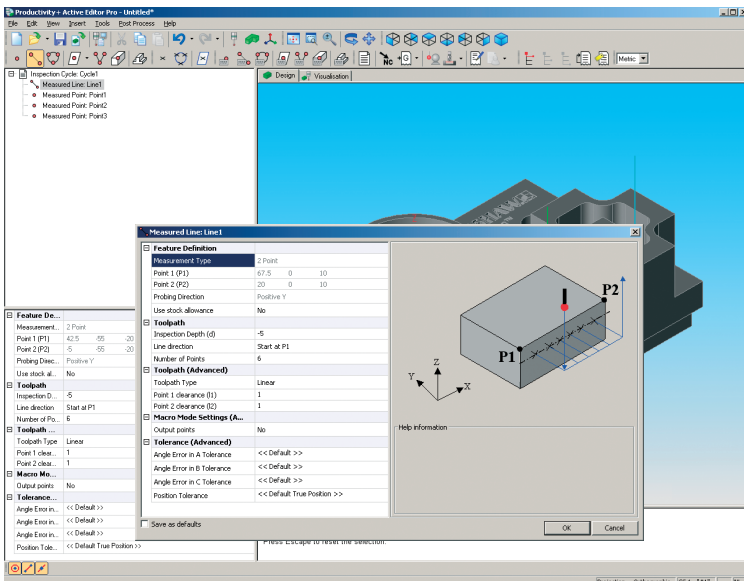
Select the Measured Line icon. This feature will be used in the same way as the Measured Point feature.

As with the Point feature, when you move the cursor over the solid model the various faces are highlighted; in addition the edge of the model is also highlighted. This is used as a reference on the model for probing depths.



With the face and edge highlighted as shown left, double click using the mouse (or single click followed by Enter).

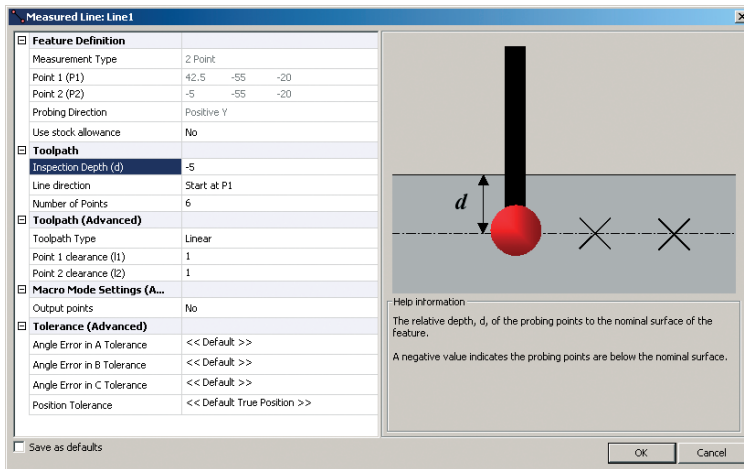
The face will change colour when successfully selected.



As with the previous Point features, the CAD model now shows a probe path for the newly created Line feature and a Measured Line element has been added to the process tree.

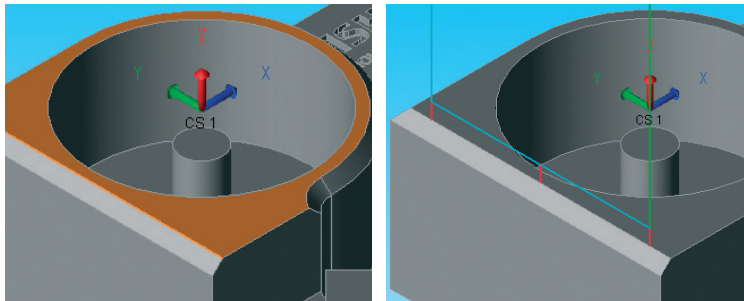
Once the line feature has been created, a dialog box is opened which allows adjustments to be made if necessary. (This dialog box can also be opened by double clicking on the Measured Line element in the process tree.)

When creating a Measured Line feature it is important to remember that both a face and an edge selection are required. An edge selection is required to get X, Y and Z information such that when the probing program is created, the probe is moved to the correct position ready to begin probing the line.



The dialog box, shown left, has Inspection Depth (d) currently highlighted with its default value of -5 (mm). This can now be edited to display a different value. Once you have made all the necessary changes, select OK.

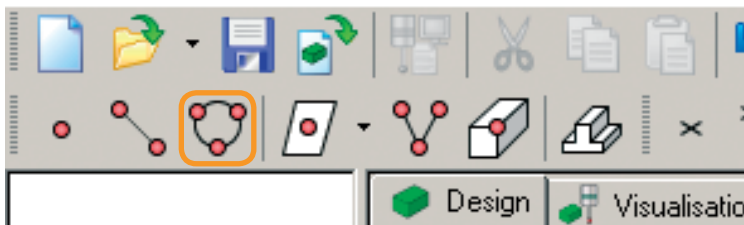
As with other dialog boxes within the software, Help information on the currently selected option is shown to the lower right of the screen, with a relevant image shown above it. (As you move up and down through the dialog box you will see the image and help information change accordingly.) There is also a series of call out boxes (tool tips) that appear as the cursor hovers over fields that are available to edit.



Now create a second Measured Line feature (the Measured Line icon at the top of the screen is still selected).

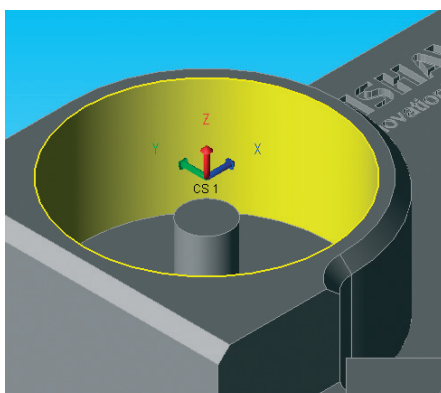
In the screenshot to the left, the top face (which determines the probing direction) and left edge of that face (which determines the orientation of the measured line) are highlighted.

Once selected you can see that this new line is in the Y direction and is measured in the Z axis.

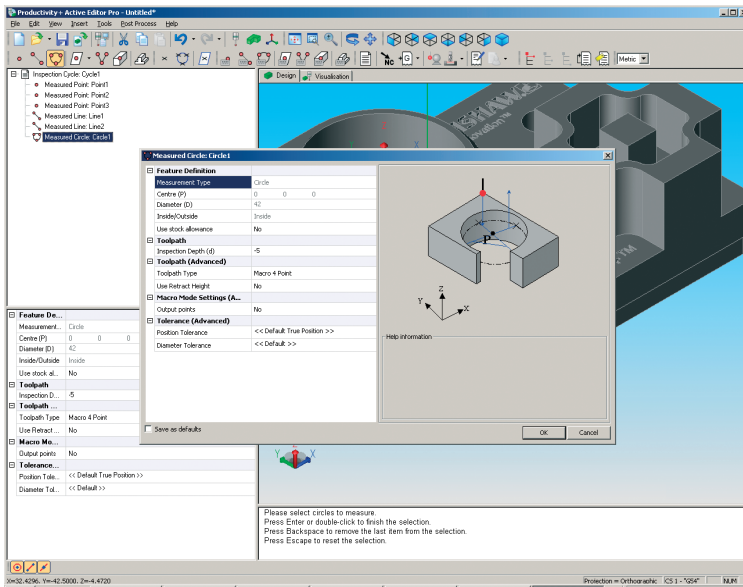


Measured Circle feature

Select the Measured Circle icon and move the cursor across the CAD model.



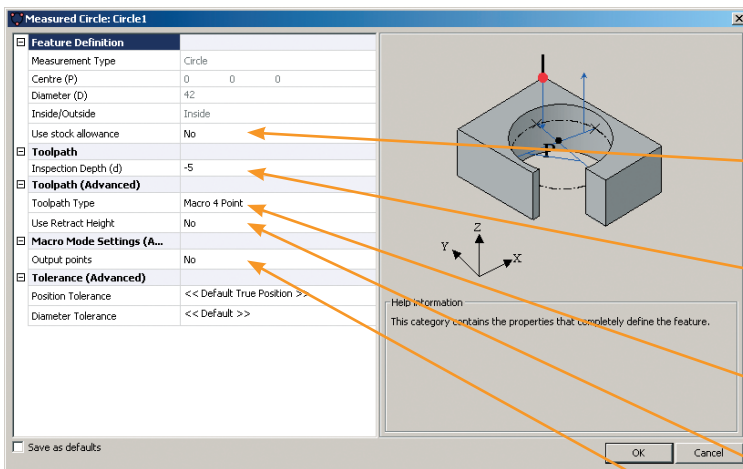
Select the large bore with the island in the centre (top edge) using either a double mouse click or single click and Enter.



In this screenshot the circle has been selected and the dialog box is open allowing values to be amended if necessary.

The X, Y and Z position of the circle are all zero as the centre point of the circle is in the same position as the datum CS1 (G54).

Note: If you have accessed this training module directly rather than working through from previous modules, this may not be true.



The first four lines of information are greyed out as they are populated with information taken directly from the CAD model.

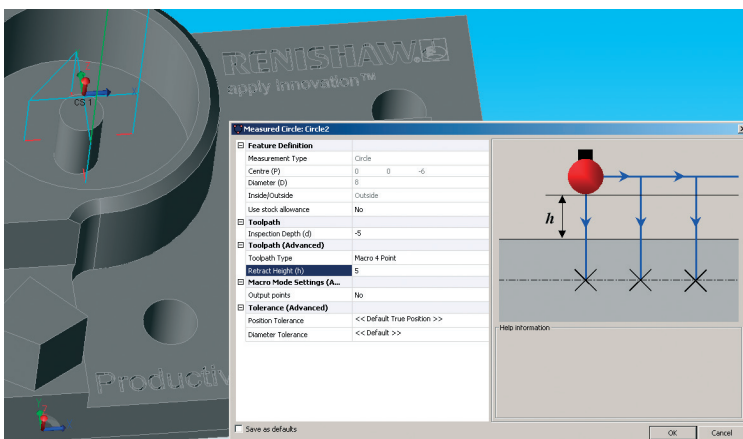
Use stock allowance: if you are probing a casting or a hole/bore with material left on.

Inspection Depth (d): the depth from the edge to the centreline of the ruby ball.

Toolpath Type: the tool path you wish to use.

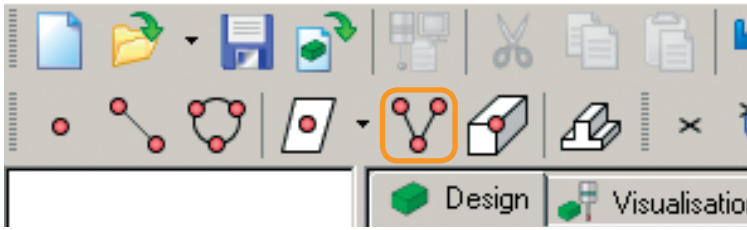
Use Retract Height: if you want to retract in between probing points.

Output points: enable output of measured points to a printer or file. The data can then be analysed externally.



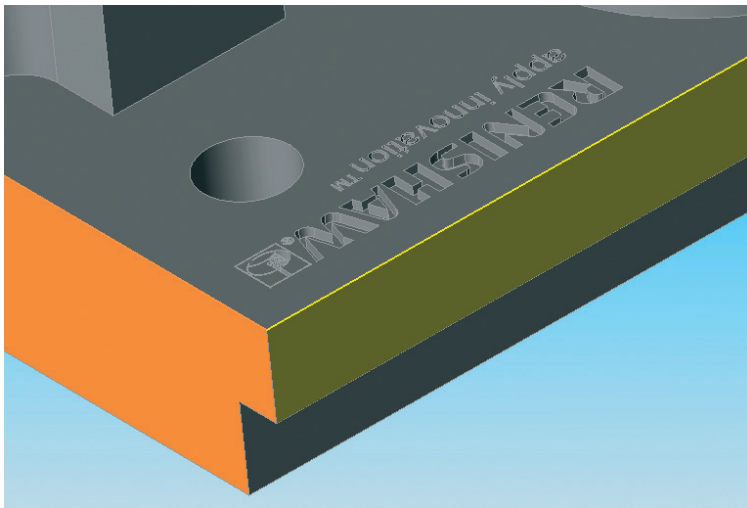
Using a second circle feature, inspect the boss.

To do this successfully the probe must retract or it would hit the boss between probe points. Set Use Retract Height to Yes, and add a Retract Height (h) of 5 (mm). This retract height is measured from the top edge of the boss as indicated by 'h' in the Help information.



Measured 2D Corner feature

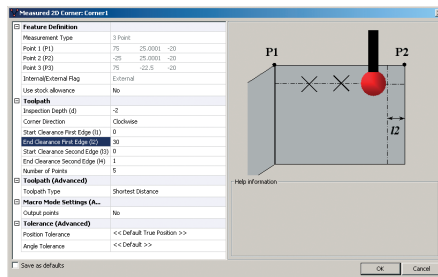
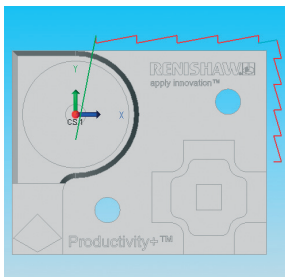
This next section will guide you through use of the Measured 2D Corner feature and how to make the selection in the example below. 2D Corner features can be either internal or external.



Select the Measured 2D Corner icon and the first of the two lines where you would like to form the 2D Corner. Now select the second line. The screen will highlight all available lines as the cursor is passed over the model.

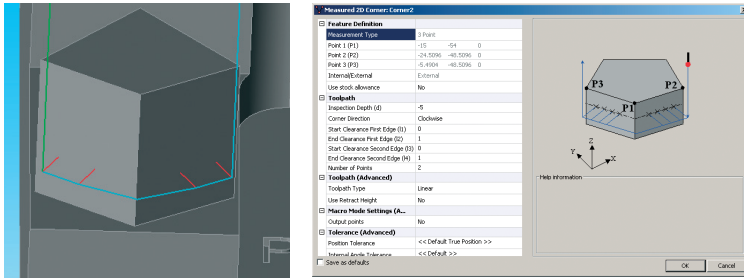
To ensure the correct probing direction is selected, the correct reference face must be highlighted when the line is selected. As soon as the second line has been selected, the corner feature will be created.

Note: The two selected lines do not need to meet.



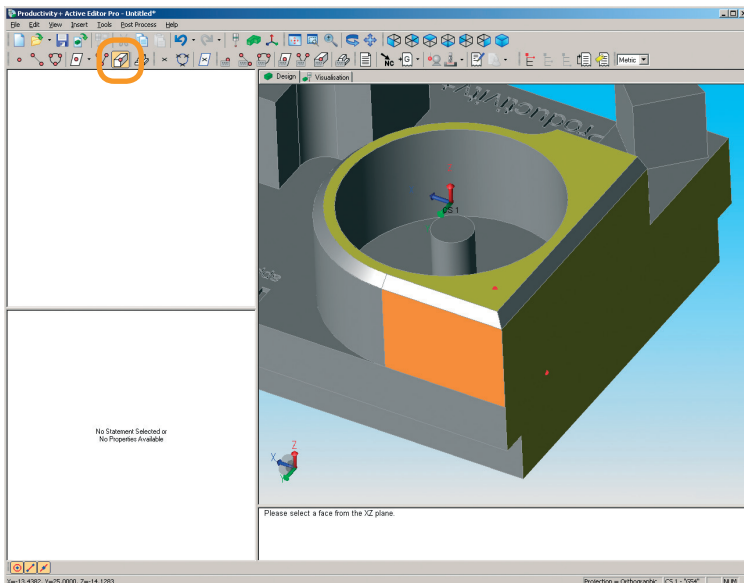
The tool path for probing this feature is shown left as a plan view. As with the previous features, values such as Inspection Depth (d), Corner Direction, and Start and End Clearances can be edited if necessary.

In this example the End Clearance First Edge (I2) has been edited to 30. This restricts measurement to a certain area. Edit some of the remaining clearance values and see how the tool path changes.



This example creates a second 2D Corner, one which is not at 90°.

Alter the Toolpath type and look at the picture in the Help information to see the change it makes. The Shortest Distance option can be good on big machines with very long probing paths.

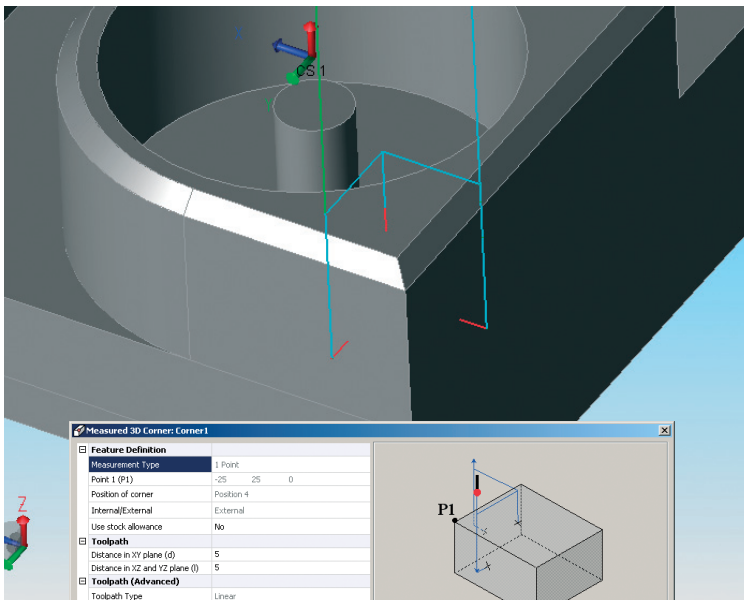


Measured 3D Corner

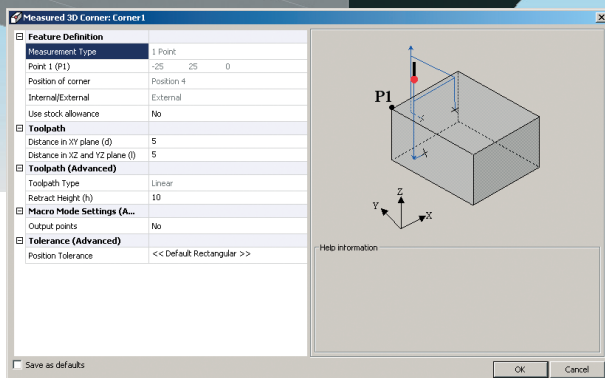
This section takes you through programming of a Measured 3D Corner feature as in the example to the left.

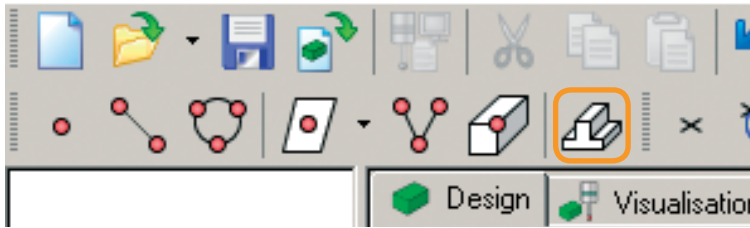
Select the Measured 3D Corner icon as for previous features.

Select the three faces of the corner that you want to inspect. As you select the third face the Measured 3D Corner dialog box opens.



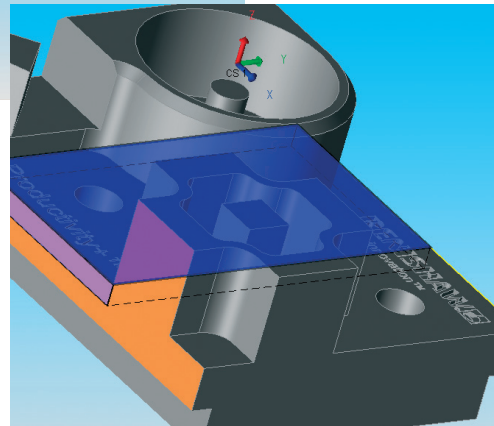
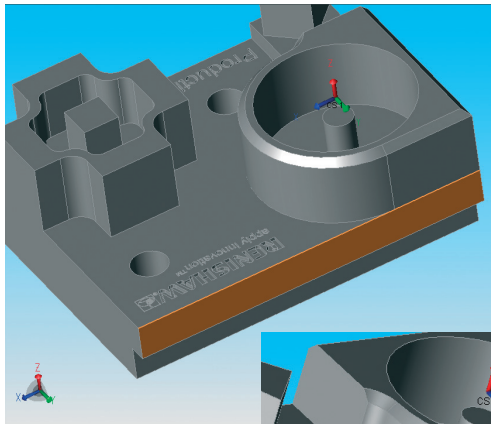
By now you will have noted that the dialog boxes for the different features have a number of common fields that you can select or amend. This is one of the features that makes programming with Productivity+™ Active Editor Pro quick and simple.





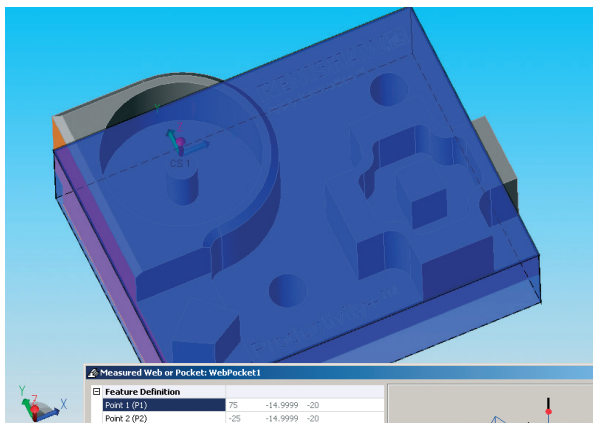
Measured Web/Pocket

When using the Measured Web/Pocket feature, Productivity+™ Active Editor Pro intelligently recognises whether the selection you make on the CAD model is a web or a pocket feature. Having selected the icon, move the cursor over the CAD model ready to make your selection.

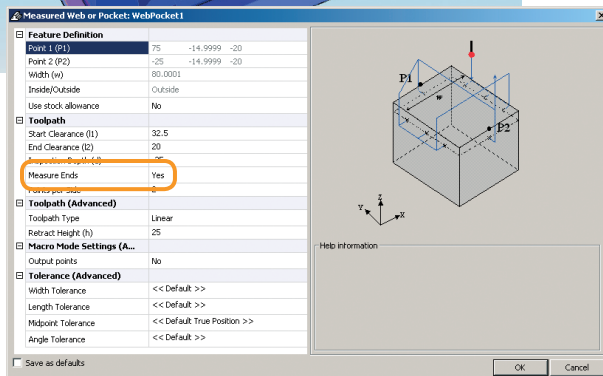


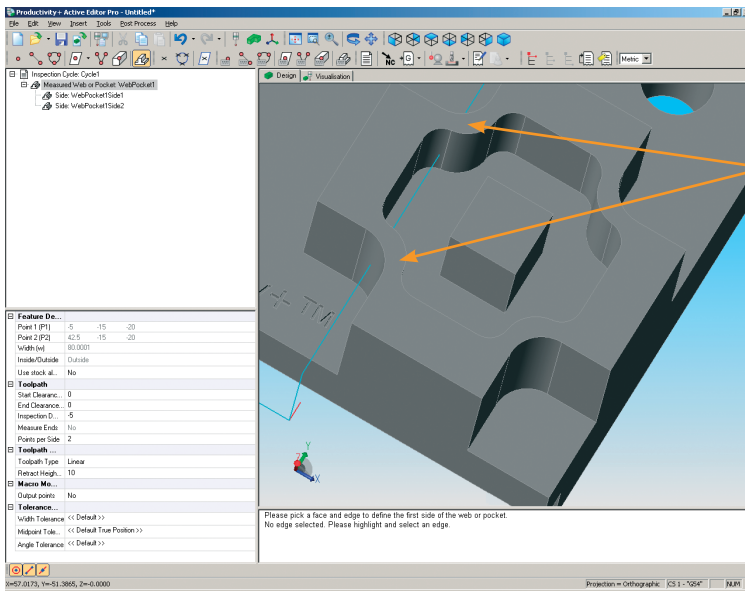
The first example covers a web feature. Select the face and wire frame as shown and double click (or click and Enter). Rotate the model and select (double click, or single click and Enter) the second side of the web feature (as the mouse moves across the model, available options are shown as a blue rectangle).

Note: Selected sides must be parallel to one another.

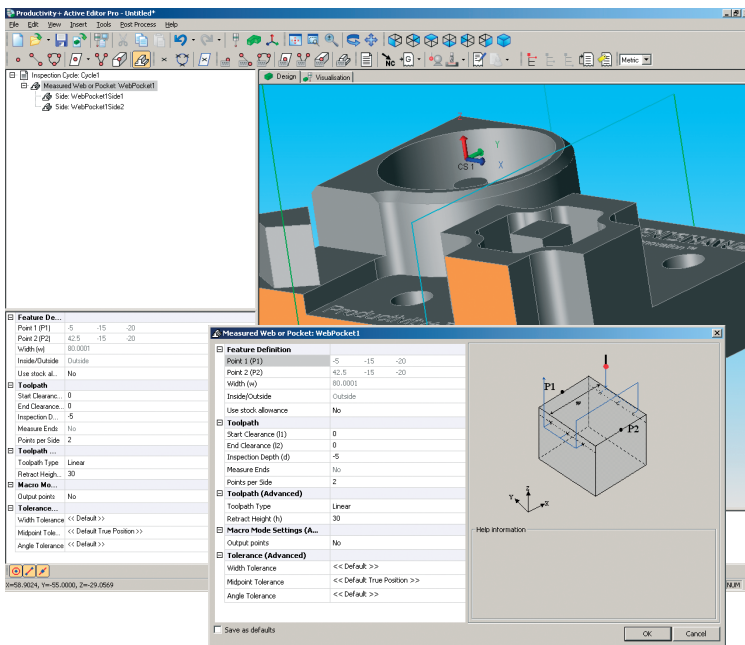


Selecting two sides is all that is required to create a web feature, but if required you can select additional sides to enable the 'Measure Ends' option within the dialog.

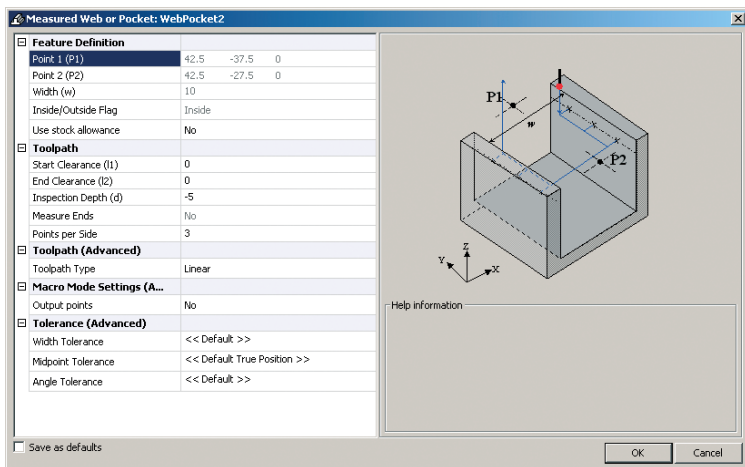




Looking at the tool path for this feature you can see that it has passed through the CAD model. This is because the Retract Height (h) is not high enough to clear the top of the component. Open the corresponding dialog box and increase the Retract Height value.



In this example, the retract height has been increased to 30 mm to ensure clearance over the top of the component.



To program a Measured Pocket feature double click on one of the faces of the pocket: as the mouse is moved across the model, the software will again show valid, available options for selection as the second face.

The Measured Ends field in the dialog box is no longer accessible as there are no ends to measure in a pocket feature. Other than this, the dialog boxes for Web and Pocket features are the same.

As your knowledge and experience of Productivity+™ Active Editor Pro develops you are likely to use common values for a number of dialog box fields, for example, Inspection Depth (d).

To further simplify programming, every dialog box contains a Save as defaults check box. Selecting this box will save the current values in all fields. Those values will then be used the next time you use that feature. If you wish to use this functionality, you will need to set values for each feature e.g. Line, Circle, 2D Corner etc. separately.

Having looked at the available features in these examples, why not try some of them on a CAD model of your own? Try experimenting with editing the different fields within the dialog boxes to see the effect on the probing paths.

A significant benefit of this software is the capability to visualise the program you have created before you run it on your machine. The Productivity+™ Active Editor Pro software has collision detection, so if you make a mistake it will become apparent when running the visualisation option. This functionality also provides a high level of confidence that if it runs correctly in visualisation mode it will run on your machine. This saves significant time when running your program for the first time: you have already proved-out the probing within the software.

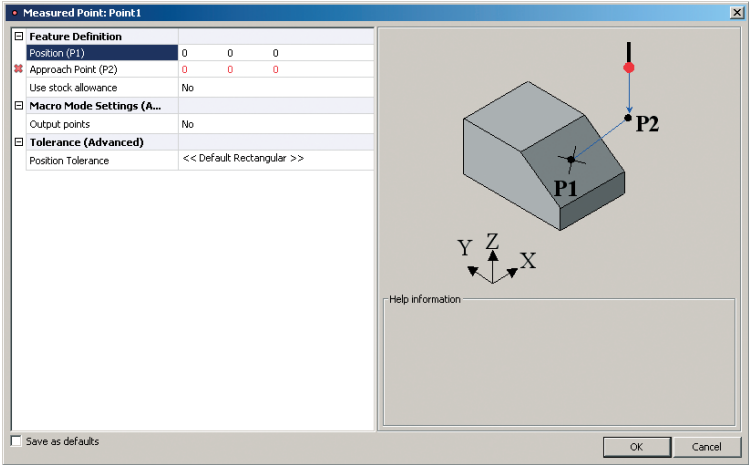
The following section covers the creation of Basic Statements. Basic Statements are used to generate a probing program without a CAD model to select from. Instead you must manually enter the X, Y and Z co-ordinates to determine where to probe a feature. Programming in this way can be useful if you do not have a CAD model of your component or if certain features on your CAD model can not be used.



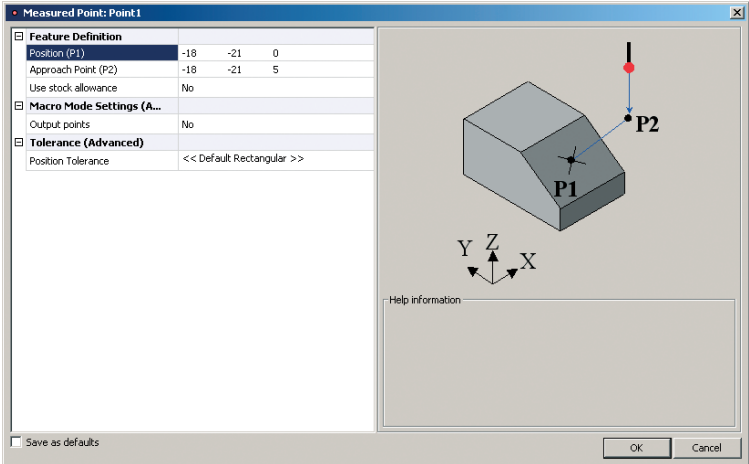
Creating a Basic Statement

You can create the same features using Basic Statements as you can by using the Measured Statements icons.

For a Basic Point feature you are required to insert X, Y and Z co-ordinates in two fields. The first field, Position (P1), is the position to measure on the surface. The second field, Approach Point (P2), defines the measurement direction, which should always be perpendicular to the surface. See the following for more detailed information.



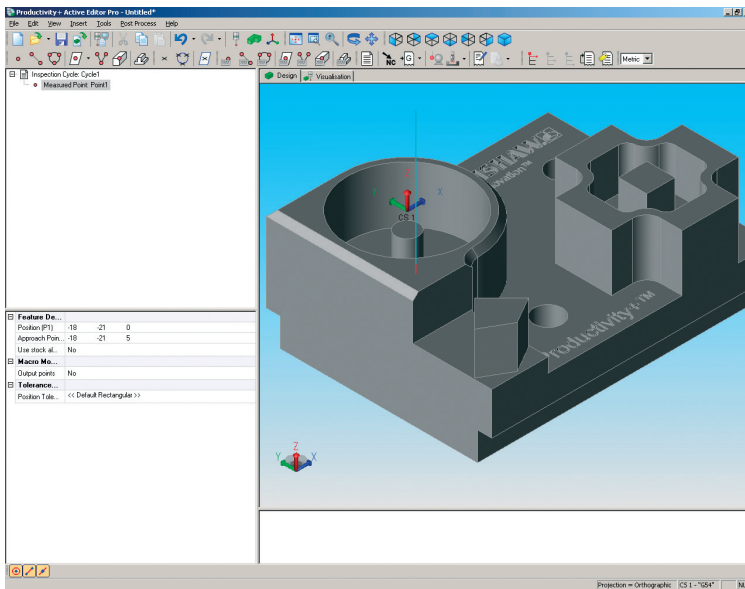
In this example a point feature has been created using the Basic Point icon. The resulting dialog box contains exactly the same fields as the one generated with the Measured Point icon when programming from a CAD model. The only difference is that you have to manually input the position in X, Y and Z (from the datum) of the point where you wish to probe your component.



The first field, Position (P1) contains the co-ordinates of the point where you want to measure on the component surface.

The second field, Approach Point (P2) contains the co-ordinates for the approach to the P1 point.

In this example the X and Y co-ordinates are the same for both P1 and P2, while Z has a co-ordinate of 0 (zero) for P1 and 5 for P2. The zero value is where the probe will touch the component, and 5 is the approach distance.



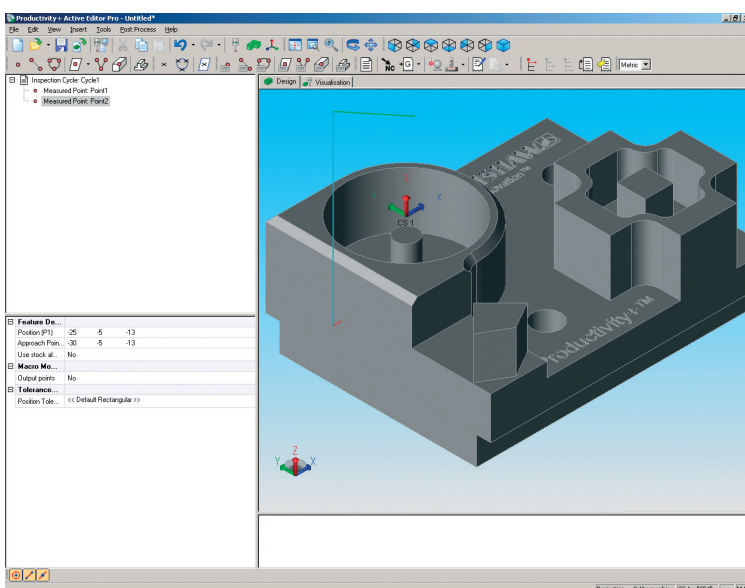
The following screenshot shows the same point programmed on the CAD model using Basic Statements.

You are still able to simulate the point with collision detection: the only difference is the manual input of X, Y and Z positions.

The Measured Point feature earlier in this module demonstrated how to click on different faces to probe in different directions. You can do exactly the same using Basic Statements. The next screenshot shows a point measured in the X+ direction.

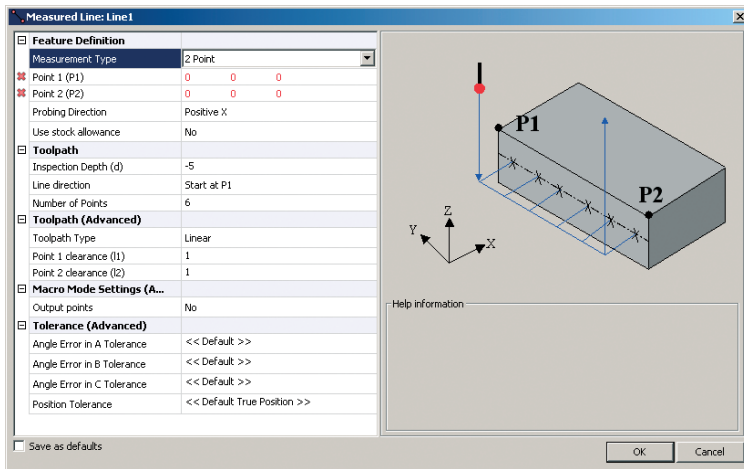
Measured Point: Point1			
Feature Definition			
Position (P1)	-25	-5	-13
Approach Point (P2)	-30	-5	-13
Use stock allowance	No		
Macro Mode Settings (A...			
Output points	No		
Tolerance (Advanced)			
Position Tolerance	<< Default Rectangular >>		

Looking at the values for X, Y and Z you can see that the -25 value in the X axis is where it will touch (P1) and the -30 is where it will approach from (P2). The Y and Z values are the same so the move is perpendicular to the side face.



Here is the probing move in the X+ direction using the values as above.

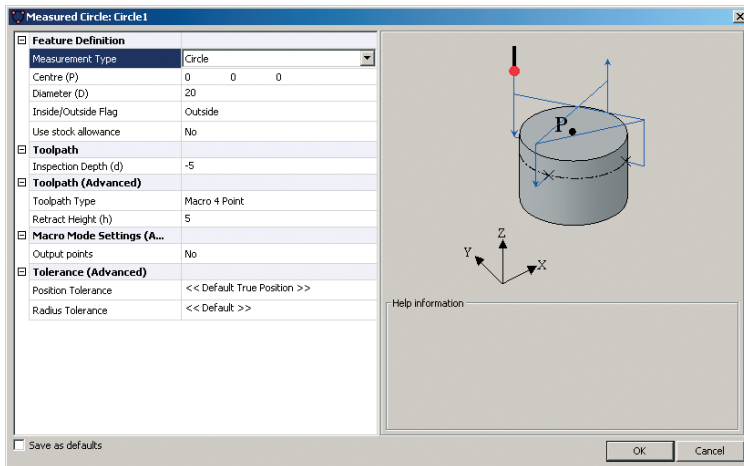
Basic statements are sometimes used when a point needs to be measured at a specific X, Y and Z position e.g. gauge height. This can not be done by just clicking on the CAD model as you can not accurately position the cursor in the correct place, and the X, Y and Z fields can not be edited when using the CAD buttons.



This example comprises a line feature created with the Basic Line icon. The dialog box is the same as you would see when using a CAD model to create a line feature. The values in red need to be amended (note the red crosses to the left of these fields).

X, Y and Z co-ordinates for positions P1 and P2 (the start and end of your probe path) need to be added.

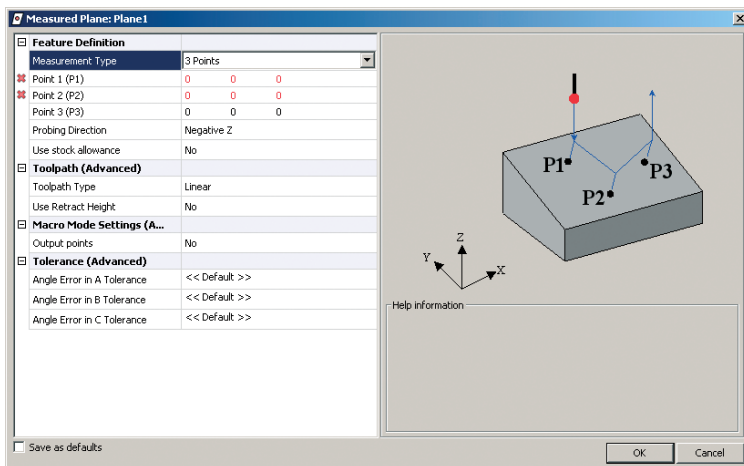
You also need to select a probing direction for your line.



This example shows a circle feature created using the Basic Circle icon.

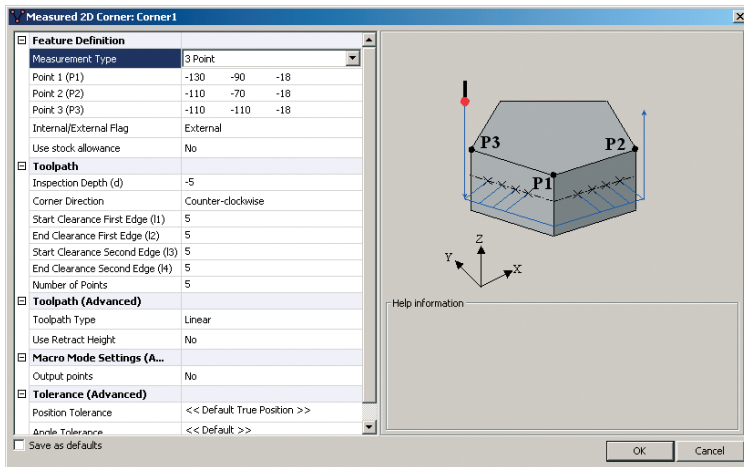
X, Y and Z co-ordinates need to be added for the centre point of the circle, the diameter must be defined, and the Inside/Outside Flag should be used to determine if the circle is a boss (Inside) or bore (Outside).

Remaining fields are the same as those that were available to edit when using the Measured Statement icon, such as Inspection Depth, Retract Height etc.



This example shows a 3 point plane feature created using the Basic Plane icon.

As with the previous examples, the resulting dialog box requires X, Y and Z co-ordinates of where you wish to probe the feature. The dialog graphic and Help information provide assistance.

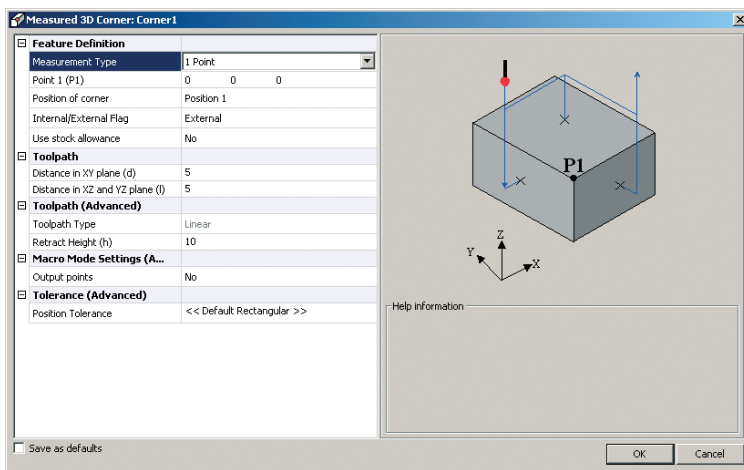


This example uses a 2D Corner feature which is defined by three points. (The Measurement Type drop down box determines how this feature is defined, which in turn determines the values you must provide.)

In this instance values are required for P1, P2 and P3, and you must let the software know if this is an internal or an external corner.

Remember, the co-ordinates you are defining are from your current datum e.g. G54.

The Save as defaults check box is also available if you are using common values in some fields.



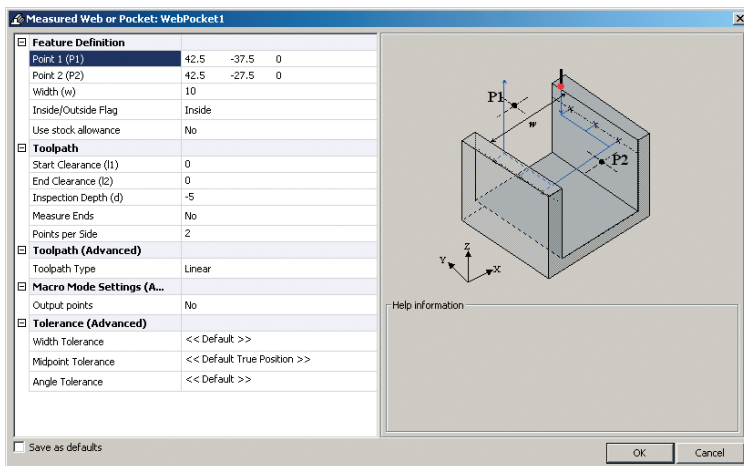
This example shows a 3D Corner feature.

This feature only requires one position, the apex of the corner, to be defined.

Add the X, Y and Z co-ordinates of your corner and use the drop down box from the Position of corner field to select the position of the corner you have defined.

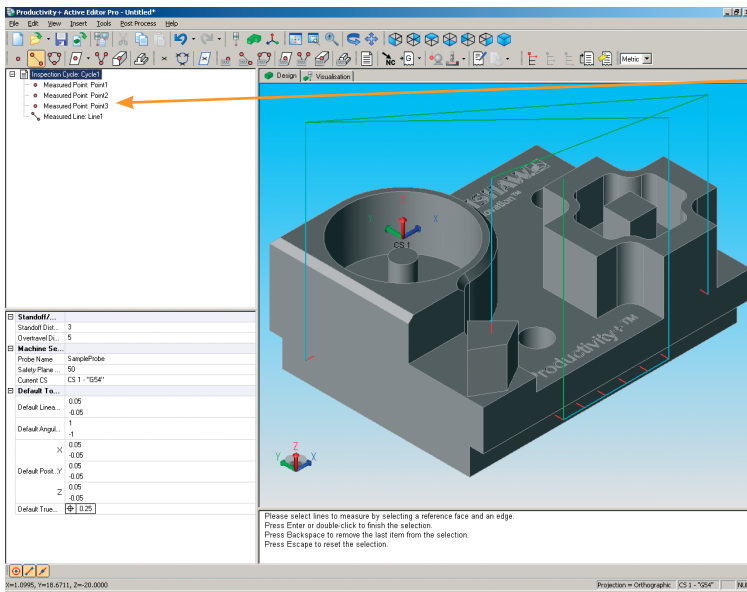
Options under the Toolpath heading allow you to edit the distance from the corner's edge where the probe will touch.

You must also identify if this is an internal or an external corner.



This example shows a Pocket feature.

Fields are the same as those within the dialog box created by the Measured Pocket icon. As with all Basic Statements you must define co-ordinates for various points. P1 is the nominal point at one end of the feature, and is positioned along the centre line between the two sides of the feature. P2 is similar but at the other end of the feature.

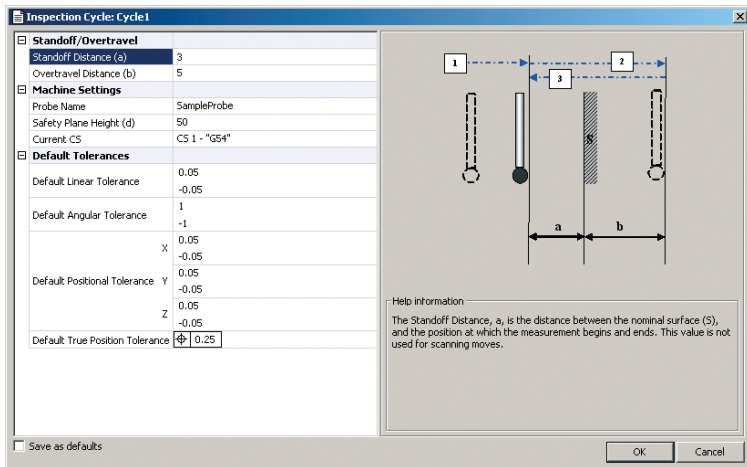


Editing an inspection cycle

As you select a feature on your CAD model by clicking on it, or when you create a Basic Statement, you will notice that an element with that feature name is added to your process tree.

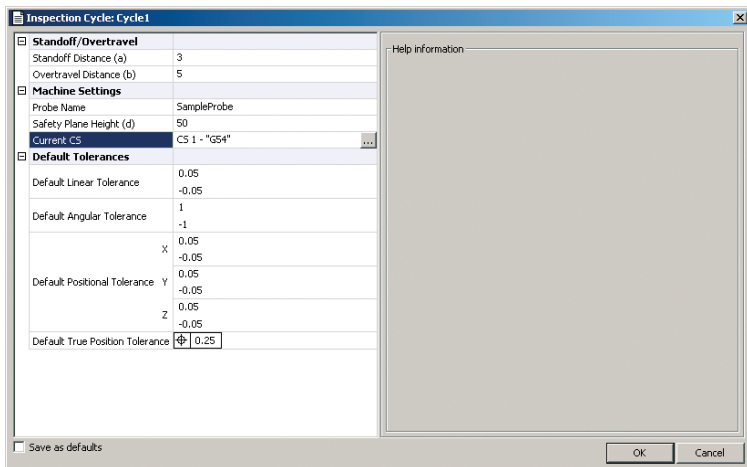
All features branch off an Inspection Cycle, known as a 'Parent Cycle'. By double clicking on the inspection cycle title, you can open the cycle for editing.

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



The first two fields you can edit are Standoff Distance (a) and Overtravel Distance (b). By highlighting either of these you will see accompanying explanation text to the right in the Help information area.

Sometimes when you are trying to probe a small bore/pocket you will need to reduce the Standoff: instances where the ruby ball hits on the back edge of the feature you are measuring. E.g. a 9 mm hole measured with a 6 mm stylus and a Standoff Distance of 3 mm.



Overtravel Distance 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.

Probe Name is where you select the probe to be used for component measurement.

Current CS lets you choose the work co-ordinate system in which measurement occurs. The area entitled **Default Tolerances** allows you to set all your tolerances. These are the values that will be used when reporting and for the 'Out of tolerance flag' if you use it when programming.

Use **Save as defaults** to apply these settings to all features programmed subsequently.

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