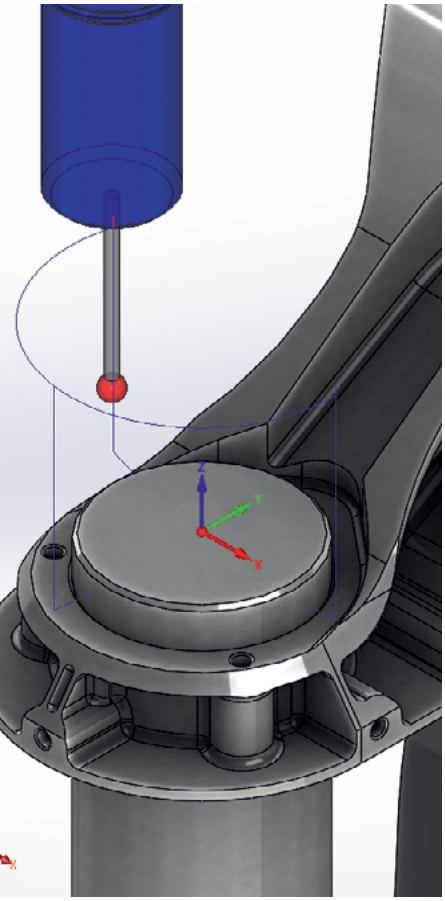
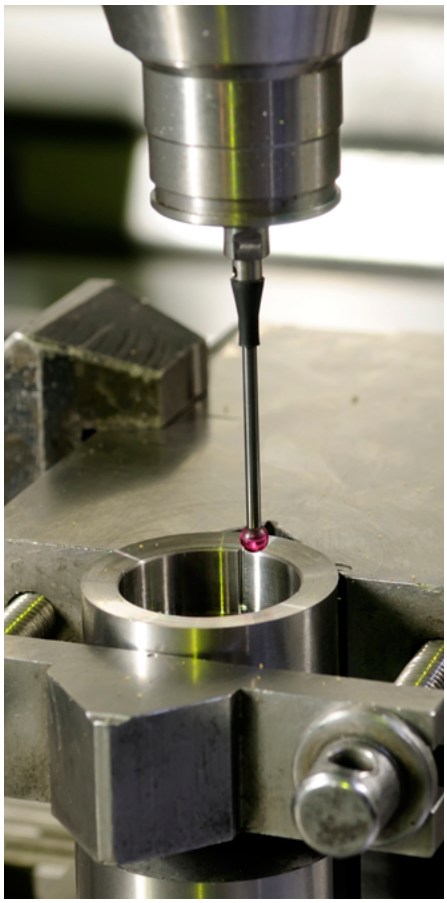
SolidCAM + SolidWorks

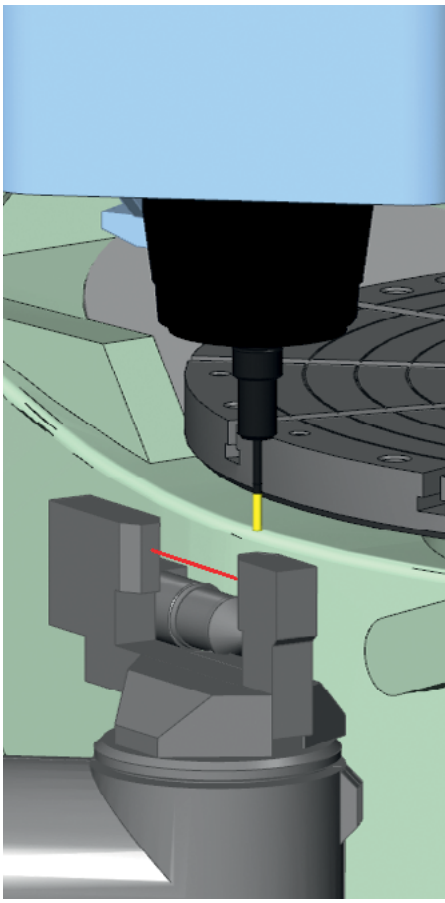
The complete integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!







SolidCAM User Guide:
Solid Probe

iMachining 2D & 3D | 2.5D Milling | HSS | HSM | Indexial Multi-Sided | Simultaneous 5-Axis | Turning & Mill-Turn | Solid Probe

www.solidcam.com

SolidCAM
iMachining – The Revolution in CAM!

SolidCAM 2013

Solid Probe User Guide

Contents

7. Process from SolidCAM to Probe

8. Measurement Results

9. Exercises

Document number: SCSPUGENG13001

Introduction 1

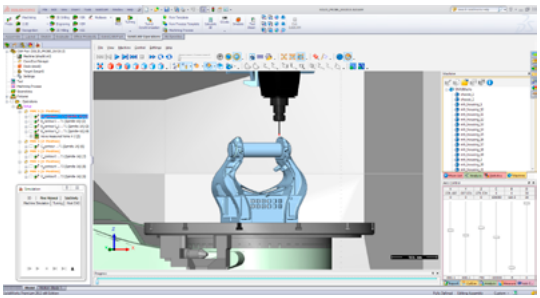
Welcome to the SolidCAM Solid Probe!

SolidCAM is bringing you Solid Probe, a new SolidCAM module that provides capabilities for Home definition and On-Machine Verification (OMV), using probes on the CNC machine, to do setup and control the quality of machined parts. Solid Probe module provides also Tool Presetter support

Full visualization of all the probe movements provided by SolidCAM Machine Simulation enables you to avoid any potential damage to the Probe tool.

Solid Probe is a must tool for every machinist using Probes:

- Easy Home definition
- On machine Verification
- Tool Presetter support
- Easy geometry selection on solid model
- Supports wide range of probe cycles
- User-defined parameters give powerful flexibility for Probe cycles programming
- Visualization of all the Probe tool movements
- Support of different Probe controllers



Home definition

Solid Probe provides an easy solution for home setting, using 16 different cycles, enabling the user to easily define home positions, replacing manual setup procedures.

Setup (Home definition) for first piece of Stock, without Solid Probe, is done by using a cycle directly on the controller, if you know the controller well. This is not the case if operator is using controller for the first time or if he has no deep machining expertise.

It is faster and safer to define the home by the Solid Probe module:

- Programming Home definition by Solid Probe is faster than manual programming on controller
- Visualization of the probe tool movements in the simulation prevents collisions and damage to the probe on the machine
- 16 different cycles of the Solid Probe are available for home definition (Each such cycle can be used only if there is support for it in the controller probe cycles)

Home definition for 2nd Stock onwards

When we put the stock of 2nd piece, we can push it against the stopper and hold it in the vice – this would be generally enough and there is no need to measure the home again (it is accurate enough).

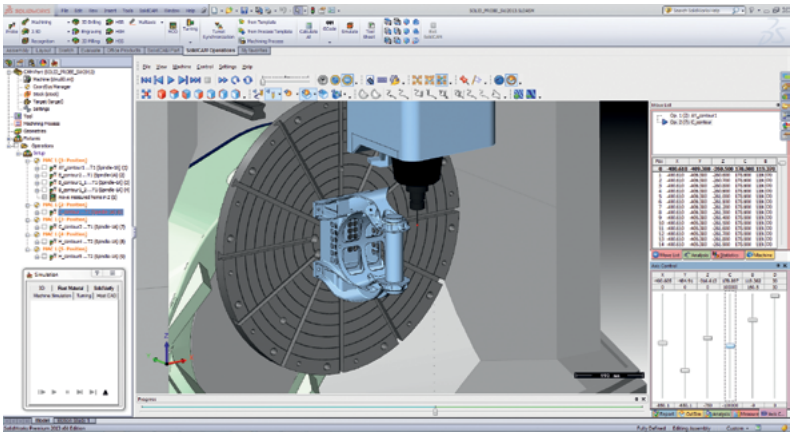
However in some cases the stock has let's say a hole in the middle and the home has to be set always at the center of this hole – in that case maybe then we have to find the correction to the home with the probe every time.

Stock is casting

When the stock is a casting (5% to 10% of parts) we need to define the home again for each new piece, because the casting stock is never exact.

On-Machine Verification

Solid Probe cycles also can be used for measuring of machined surfaces without transferring parts to any CMM machine - the part can be inspected on the machine tool itself.



- The main advantage is that any mistakes are discovered where they can be corrected – on the machine tool. The most obvious benefit of On-Machine Verification is for those companies that machine parts so large that the cost of dedicated CMM facilities is prohibitive.
- Solid Probe measurement can also improve quality control by enabling components to be monitored more frequently during the manufacturing process. This will allow any errors to be detected earlier, and so corrected more quickly and at lower cost.
- Similarly, the extent of any damage caused, for example, by a tool breakage, can be assessed accurately and a decision made immediately to determine whether the part can still be completed within tolerance or whether it will have to be scrapped.

Preview of Cycle movements

Solid Probe can use same geometry as in 2.5D milling operations.

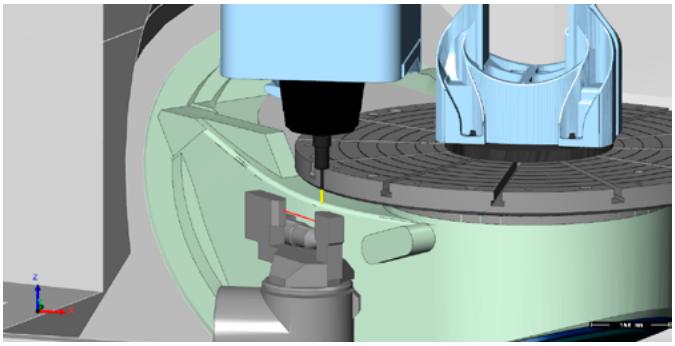
Full control over tolerances, different sorting options and direct preview of cycle movements are provided.

Tool Presetter Support

The Solid Probe module includes support of Tool Presetter.

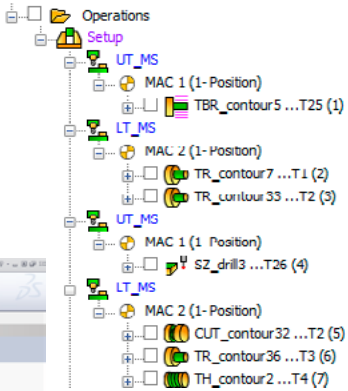
This option enables checking milling or turning tools between machining operations.

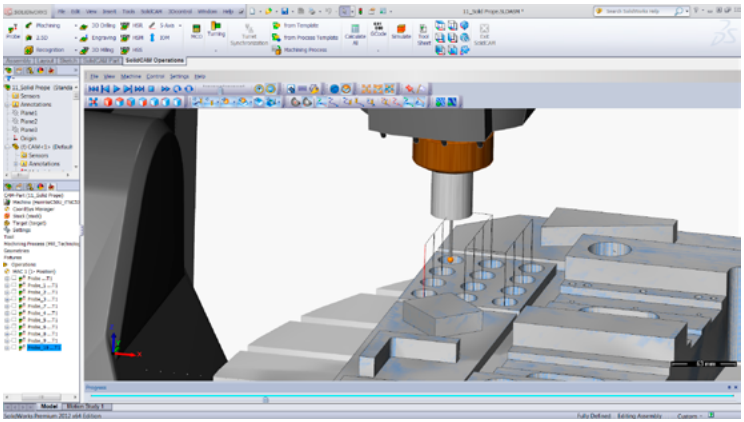
- **Tool checking:** performed after every tool change or every operation.
- **Automatic Offset Updating:** Tool measurement provides checking of tool length and radius, to correct tool offsets table in the machine controller.
- **Broken Tool Detection:** Tool breakage checking gives alarm to the user in case of a tool breakage detection, providing safe machining, by avoiding machine crashes and waste of time in case of tool damage.



Combined Probe and Machining operations

Machining operations and Probe operations are intermixed in the SolidCAM CAM-manger and can use the same geometries on the Solid CAD model. When the solid model is changed, both the machining and probe operations can be automatically synchronized to follow the updated geometry.





Post processor support

The SolidCAM post processor, for a specific CNC machine with probe, provides complete support for the machining and probing operations.

Support for subroutines can be obtained in the GCode.

About this book

This book is intended for experienced SolidCAM users. If you are not familiar with the software, start with the lessons in the **2.5 Milling Training Course** manual and then contact your reseller for information about SolidCAM training classes.

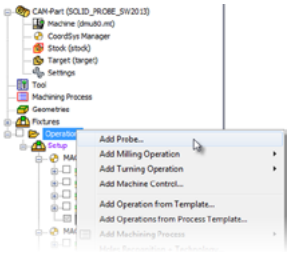
About the exercises

The CAM-Parts used for this book are attached in a ZIP archive. Extract the content of the **Examples** archive into your hard drive. The SolidWorks files used for the exercise were prepared with SolidWorks 2013.

The contents of this book and exercises can be downloaded from the SolidCAM website **http://www.solidcam.com/support/documentation**.

To learn more about Solid Probe, download the **Sold Probe presentation** in PowerPoint format.

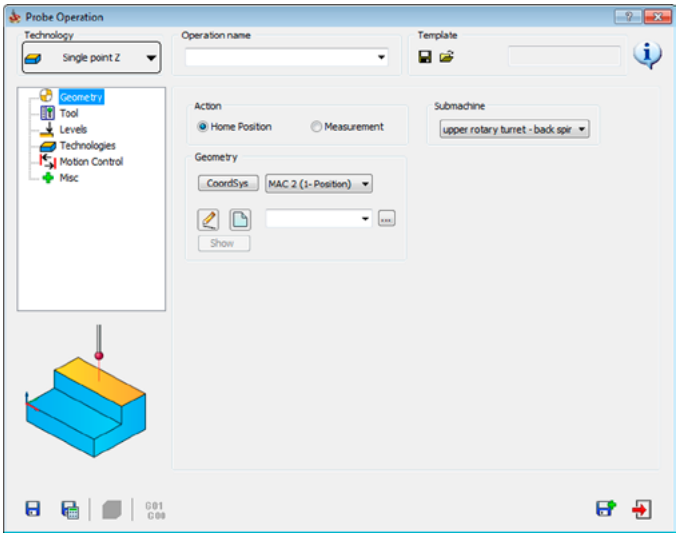
1.1 Adding a Probe Operation



To add a **Probe Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **Add Probe** command.

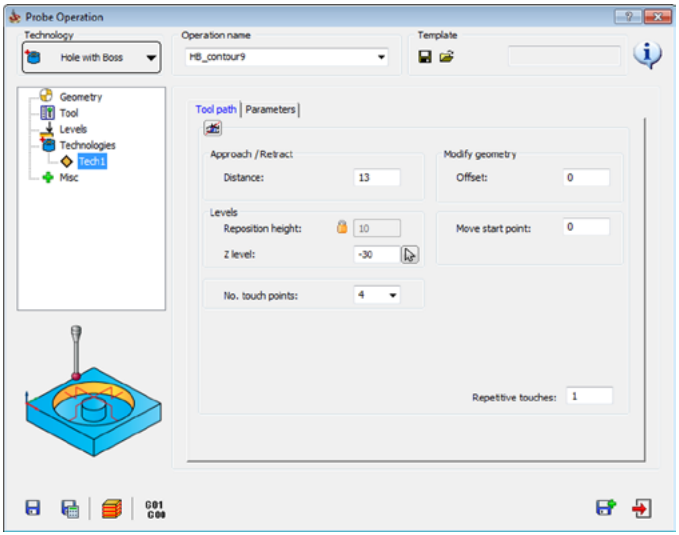
Alternatively, you can click the **SolidProbe** icon  on the **SolidCAM Operations** toolbar.

The **Probe Operation** dialog box is displayed.



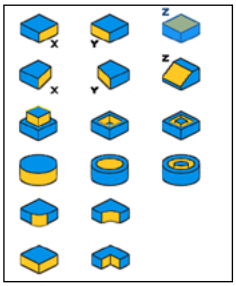
1.2 Probe Operation dialog box

The **Probe Operation** dialog box enables you to define parameters of measurements performed by a Probe tool.



Technology

This section enables you to define the type of the cycle. SolidCAM provides you with the following Probe cycles:



- **Single point X** 

  This cycle enables you to measure or correct the **Coordinate X** of the Machine Coordinate system.

- **Single point Y** 

  This cycle enables you to measure or correct the **Coordinate Y** of the Machine Coordinate system.

- **Single point Z** 

  This cycle enables you to measure or correct the **Coordinate Z** of the Machine Coordinate system.

- **Angle X** 

  This cycle enables you to correct the angular orientation of the X-axis of the Machine Coordinate System or measure the angle of an angled plane.

- **Angle Y** 

  This cycle enables you to correct the angular orientation of the Y-axis of the Machine Coordinate System or measure the angle of an angled plane.

- **Angle Z** 

  This cycle enables you to correct the angular orientation of the Z-axis of the Machine Coordinate System or measure the angle of an angled plane.

- **Boss** 

  This cycle enables you to correct either **X-** or **Y-** (or **X & Y**) coordinates of the Machine Coordinate System according to the location of a Boss, or measure its dimensions.

- **Pocket** 

  This cycle enables you to correct either **X-** or **Y-** (or **X & Y**) coordinates of the Machine Coordinate System according to the location of a Pocket, or measure its dimensions.

- **Pocket with Boss** 

  This cycle enables you to correct either **X-** or **Y-** (or **X & Y**) coordinates of the Machine Coordinate System according to the location of an Island (a Boss inside a Pocket), or measure its dimensions.

- **Cylinder** 

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located relative to the centre of a cylinder or measure its diameter.

- **Hole** 

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located relative to the centre of a Hole or measure its diameter.

- **Hole with Boss** 

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located relative to the centre of an island (a Boss inside a Hole), or measure its diameter.

- **External arc** 

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located relative to the centre of an External arc, or measure its diameter.

- **Internal arc**

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located relative to the centre of an Internal arc, or measure its diameter .

- **External corner**

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located in an External corner, or measure its angle .

- **Internal corner**

  This cycle enables you to correct **X Y** coordinates of the Machine Coordinate System located in an Internal corner, or measure its angle.

Parameter pages

The parameters of the Probe operation are divided into a number of groups. The groups are displayed in a tree format on the left side of the **Probe Operation** dialog box. When you click a group name in the tree, the parameters of the selected group appear on the right side of the dialog box.

- **Geometry**

  Define the CoordSys Position and choose a geometry of the Probe Operation.

- **Tool**

  Choose a tool for the operation and define the related parameters.

- **Levels**

  Define the Clearance area and the machining levels.

- **Technologies**

  Define the measuring parameters.

- **Motion Control**

  Optimize the calculated tool path according to the kinematics and special characteristics of your CNC-machine.

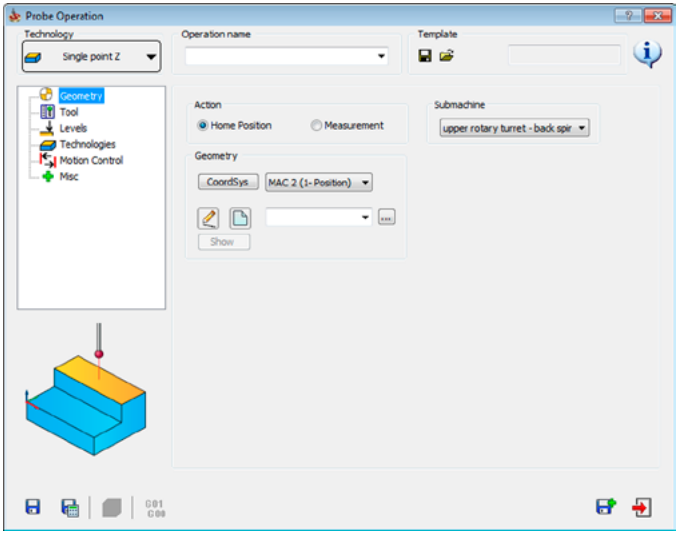
- **Miscellaneous parameters**

  Define a number of miscellaneous parameters and options related to the tool path calculation.

Geometry

2

The **Geometry** page enables you to define the geometry and its related parameters for machining.



2.1 Action

The **Probe Operation** enables you to perform two types of actions:

- defining the home position
- making measurements

When you select the **Home Position** option, you can define the home position using the available cycles. When you select the **Measurements** option, you can perform measurements on surfaces, pockets, cylinders.

2.2 Geometry

In this section, you define the Coordinate System appropriate for the operation. Choose an existing Coordinate System from the list or click the **CoordSys** button to define a new one. The **CoordSys Manager** dialog box is displayed. This dialog box enables you to define a new Coordinate System directly on the solid model.

When the Coordinate System is chosen for the operation, the model is rotated to the selected CoordSys orientation.

For more information on the Coordinate System definition, refer to the **SolidCAM Milling User Guide**.

The **Geometry** section enables you to define the geometry for the operation. The geometry consists of various elements (curves and surfaces) that must be defined depending on the machining strategy chosen from the **Technology** list.

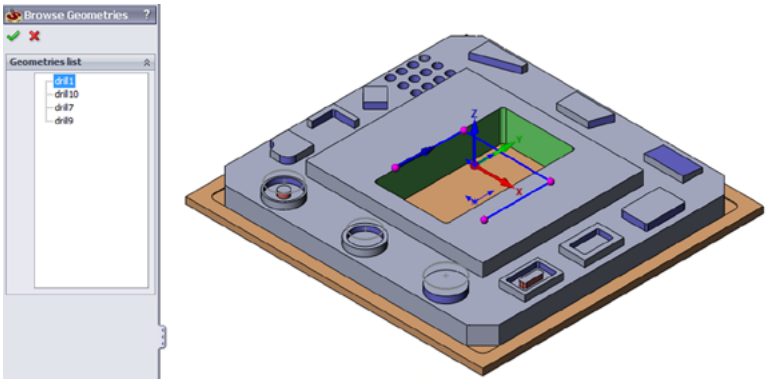
The **New** button enables you to define a new geometry for the operation.

The **Edit** button enables you to edit the defined geometries that are used for this operation.

The **Browse** button enables you to visualize the geometries during the selection process.

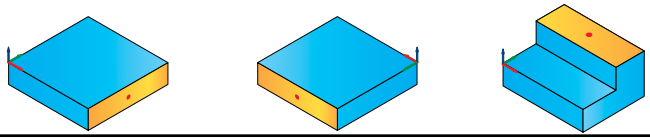
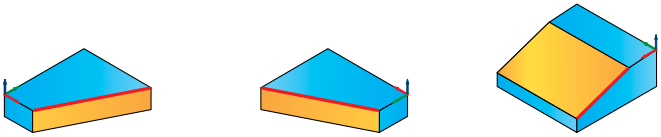
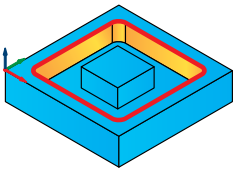
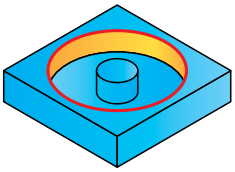
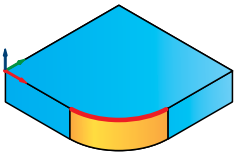
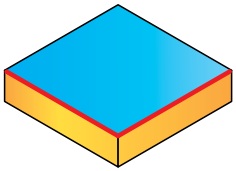
Clicking the **Browse** button displays the **Browse Geometries** dialog box that lists all the geometries of the type suitable for the chosen operation. When a geometry is chosen in this dialog box, it is highlighted on the model.

The **Show** button displays the geometry chosen for the operation directly on the solid model.



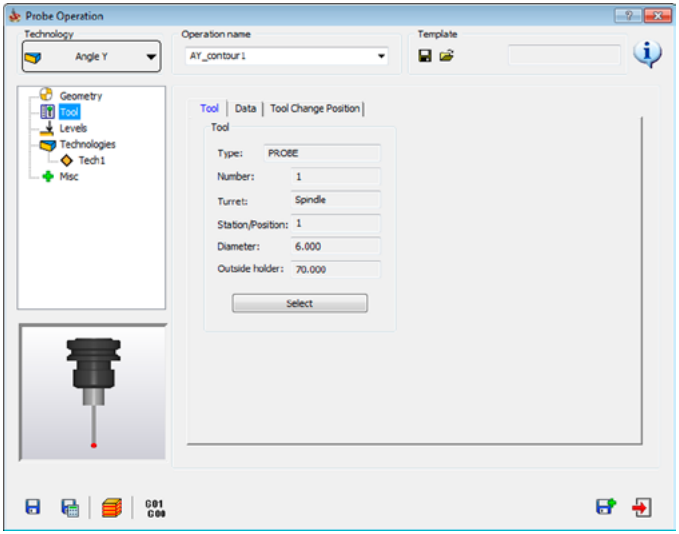
Selecting suitable geometries

To perform a certain action, each probe cycle requires a specific type of geometry as an input. The required geometries are shown in the following table:

Cycle	Used Geometry
Single point X Single point Y Single point Z	Point 
Angle X Angle Y Angle Z	Straight line 
Boss Pocket Pocket with boss	Closed chains that contain two parallel lines 
Cylinder Hole Hole with Boss	Circles (cylinders) 
External arcs Internal arcs	Arcs or circles 
External corners Internal corners	Two perpendicular entities 

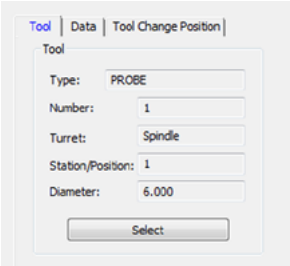
Tool 3

The **Tool** page enables you to choose a Probe tool for the operation from the **Part Tool Table**.



The **Tool** page contains three internal pages opened with tabs: **Tool**, **Data**, and **Tool Change Position**.

3.1 Tool tab



The first tab enables you to choose the Probe tool and displays the following basic parameters of the chosen tool:

The **Type** field displays the tool type, which is always Probe in Probe operations.

The **Number** field displays the selected tool number according to the current tool table.

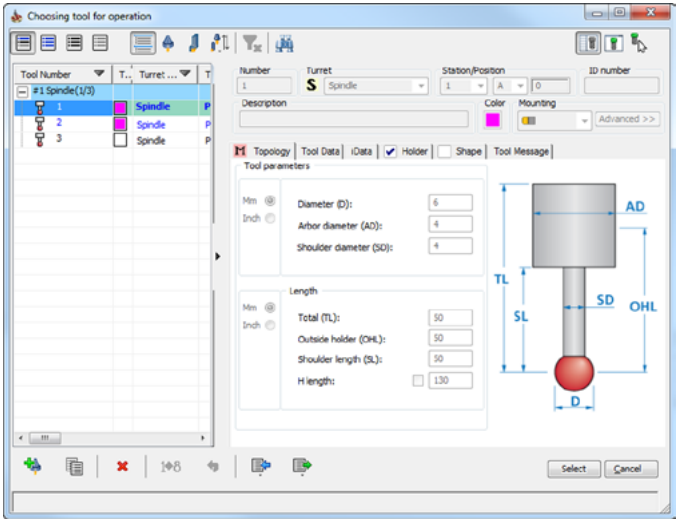
The **Turret** field displays the type of the turret on which the Probe tool is mounted.

The **Station/Position** field displays the number of the station where the Probe tool is mounted.

The **Diameter** field displays the selected Probe's diameter.

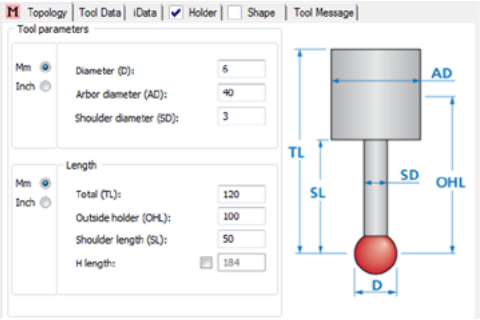
The **Outside holder** field displays the tool's outside holder length.

When you click the **Select** button, the **Part Tool Table** is displayed. The only tool you can choose for this kind of operations is the Probe tool. Click the **Add Milling Tool** button to display the Probe tool parameters.



Topology

The **Topology** tab contains the Topology data of the tool, such as diameter, length, angle, corner radius, etc.

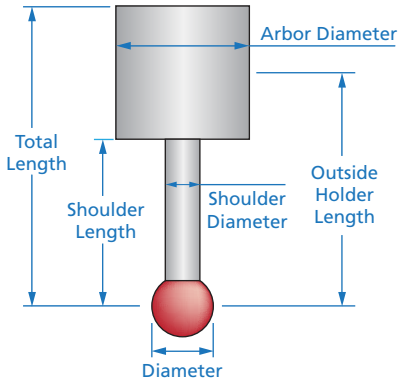


Units

The **Mm** and **Inch** buttons enable you to switch between the metric and inch system of measurement.

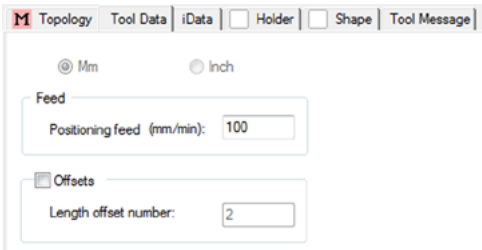
Tool Parameters

D	Defines the diameter of the probe tip ball.
AD	Defines the diameter of the tool arbor.
SD	Defines the shoulder diameter.
TL	Defines the total length as the tool arbor length
OHL	Defines the length of the tool outside the tool holder.
SL	Defines the shoulder length measured from the arbor base to the ball center.
H length	Defines the distance from the tool end to the CNC-machine spindle.



Tool Data

The **Tool Data** tab defines feeds and offsets for the Probe tool.



Feed

The **Positioning feed** field defines the default feed used for positioning movements of the probe tool.

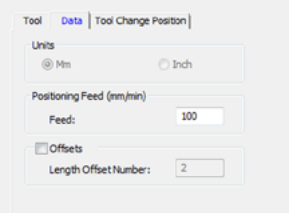
The **Length offset number** defines the number of the Length Offset Register of the current tool in the Offset table of the CNC-machine.

iData, Holder, Shape, Tool Message

For more information on these tabs, refer to SolidCAM Help.

3.2 Data tab

This tab enables you to define the feed, spin and tool offset parameters for the tool chosen in the operation. When the tool is chosen for the operation, SolidCAM fills this dialog box with the default data of the selected tool. This page enables you to edit **Units**, **Positioning Feed** and **Length offset number** for a specific operation only.



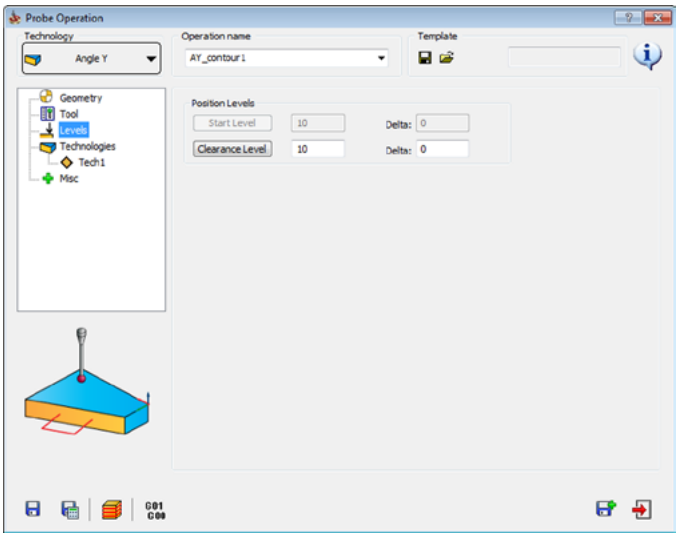
3.3 Tool Change Position tab

This tab enables you define the tool change position settings for the current CAM-Part.

The **As in Part settings** option enables you to define the tool change position according to the values set in the **Tool Change Position** section of the **Part Settings**.

Levels 4

The **Levels** page enables you to choose the machining levels of the current Probe operation.



4.1 Start Level

This parameter enables you to define the Z-level that can be optionally used for interoperational movements.

The **Start Level** button and the related edit box are available only if the **Operation Start level** option is chosen in the **Interoperational tool movement** section of the **Part Settings**.

The default **Start Level** value is equal to the **Part Clearance level** value of the Coordinate System chosen for the operation. The **Start Level** cannot be lower than the operation **Clearance Level**.

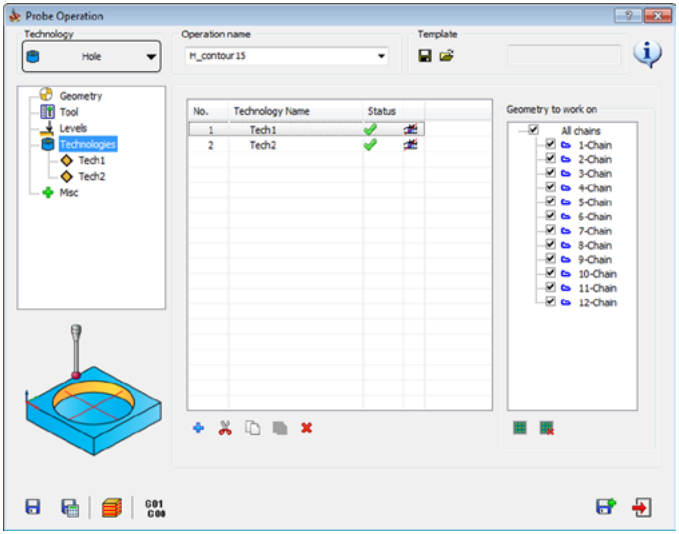
4.2 Clearance Level

This option defines the Z-level where the horizontal positioning movements are performed in order to position the probe tool before descending to the cycle start position. After the cycle completion, the probe tool ascends to the **Clearance Level** in order to move to the descending position of the next cycle.

The **Delta** value defines an additional offset from the defined **Clearance Level**. If the **Delta** value is positive, a blue arrow is displayed near its field indicating a positive offset value. If the **Delta** value is negative, a red arrow is displayed near its field indicating a negative offset value.

Technologies 5

The **Technologies** page enables you to choose the measurement cycle and define the appropriate parameters.



5.1 Managing Technological cycles

This page enables you add , cut , copy , paste and delete technological cycles.

The main table contains technology names, status and preview control.

Technology status

The status icon displays whether the cycle can be executed.

The icon means the cycle can be executed.

The icon means that some part of the cycle cannot be executed. Hover the mouse pointer over the status icon to view the explanation.

The icon means that the cycle cannot be executed. Hover the mouse pointer over the status icon to view the explanation.

Preview

The preview button enables you to toggle between the **Show** ( ) and **Hide** ( ) states to show or hide the technology geometry on the model.

Selecting chains

In the **Geometry to work on** section, you can choose certain chains by clicking in the boxes near their names.

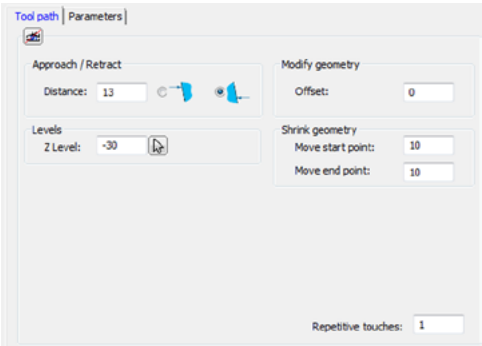
The **Sort geometry** icon ( ) enables you to define the chains sorting strategy using the **Advanced Sorting** dialog box. The **Clear sort** icon ( ) enables you to remove the advanced sorting strategy and use the default sort.

5.2 Defining Technological cycles

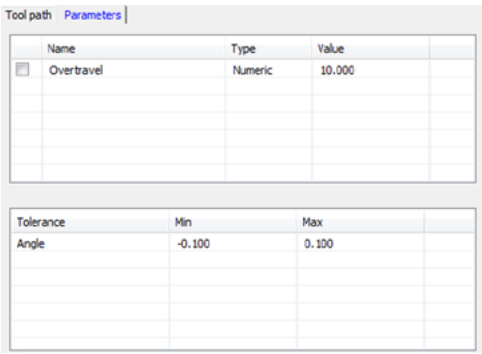
The parameters of each technological cycle have to be defined in a separate cycle parameters page. This page is displayed when you click the cycle name in the left pane of the **Probe Operation** dialog box or double-click its name in the main table.

The cycle parameters page contains the following pages:

The **Tool path** page enables you to set the cycle parameters that define the tool path.

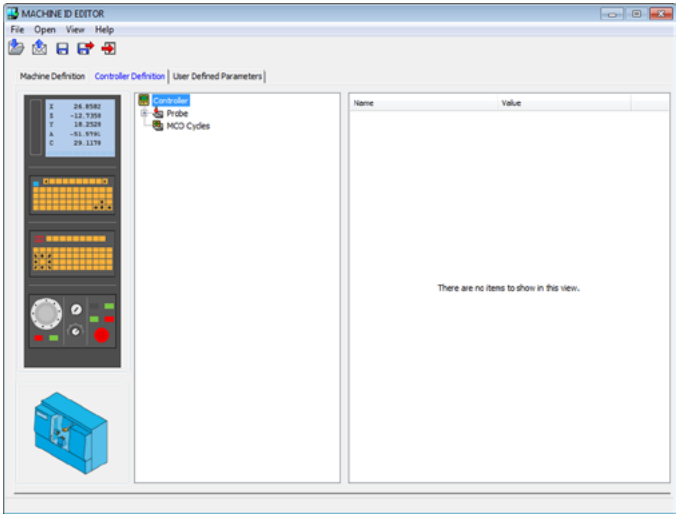


The **Parameters** page displays a parameters table defined in the Machine ID. These parameters are unique for each probe manufacturer and can be edited in the Machine settings.

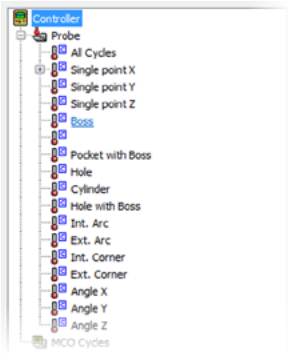


To view the Machine settings, double-click the **Machine** header in the **SolidCAM Manager** tree.

The **Machine ID Editor** dialog box is displayed. Switch to the **Controller Definition** page.

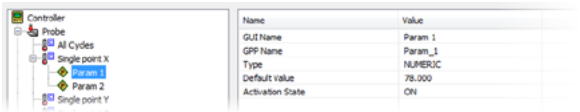


Expand the **Probe** header to display the list of cycles. Custom parameters can be assigned separately to each type of cycles or to all cycles at once.

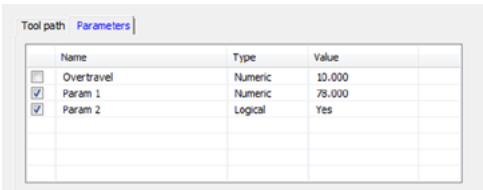


These parameters are displayed upon expanding the relevant cycle header.

In the example presented on the picture below, two custom parameters are assigned to the **Single point X**.



For the machine, these parameters appear in each **Single Point X** cycle and can be optionally enabled or disabled by the corresponding check box.



For more information on editing the Machine settings, contact your local reseller.

Tool path

The parameters displayed in this page depend on the strategy you have chosen in the **Technology** section of the **Geometry** page. The strategies are divided into the following categories:

- **Single point cycles**
- **Angle cycles**
- **Boss cycles**
- **Cylinder cycles**
- **Arc cycles**
- **Corner cycles**

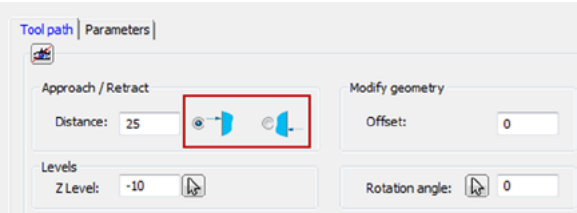
5.2.1 Single Point cycles

The **Single Point X, Y** and **Z** cycles are defined with the following parameters:

Approach/Retract

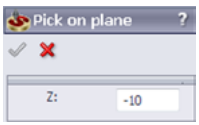
The **Distance** value defines the Probe tool position before or after performing the measurement cycle.

The direction option enables you to define the direction of the tool path relative to the plane.



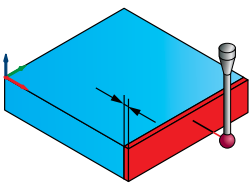
Levels

The **Z Level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

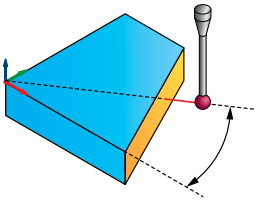


Modify geometry

The **Offset** value defines an additional distance from the contact points in the same direction as the **Approach/Retract Distance**.



Rotation angle (available for X and Y only)



This field enables you to set the angle the tool approaches the plane with.

The arrow button enables you to select the plane the angle relates to.

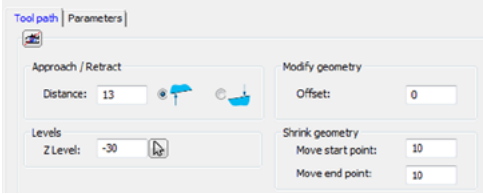
Repetitive touches

This field enables you to define the number of touches in each contact point.

5.2.2 Angle Cycles

The **Angle X, Y** and **Z** cycles are defined with the following parameters:

Approach/Retract



The **Distance** value defines the Probe tool position before or after performing the measurement cycle.

The direction option enables you to define the direction of the tool path relative to a specific plane.

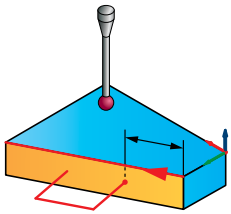
Levels

The **Z Level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

Modify geometry

The **Offset** value defines an additional distance from the contact points in the same direction as the **Approach/Retract Distance**.

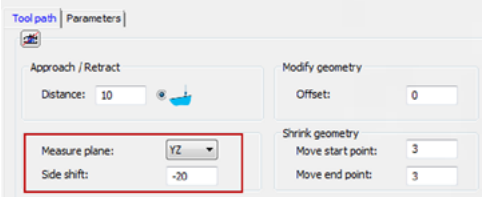
Shrink geometry



The **Move start point** field enables you to reduce the measurement distance by shifting the start point by the specified value. The point is shifted along the chain direction, parallel to the operation geometry.

The **Move end point** field enables you to reduce the measurement distance by shifting the end point by the specified value. The point is shifted in the direction opposite to the chain, parallel to the operation geometry.

Measure plane (available for Angle Z only)



The **Measure plane** option enables you to choose the plane in which the angle from Z is measured.

The **Side shift** field enables you to set the side offset from the geometry. If you choose ZX as the measure plane, the Y-coordinate is shifted. If you choose YZ plane as the measure plane, the Z-coordinate is shifted.

Repetitive touches

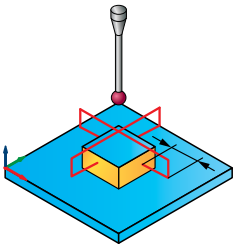
This field enables you to define the number of touches in each contact point.

5.2.3 Pocket/Boss Cycles

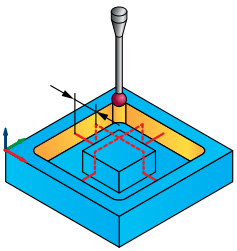
The **Boss**, **Pocket**, and **Pocket with Boss** cycles are defined with the following parameters:

Approach/Retract

In the **Boss** cycle, the **Distance** value is measured from the walls of the boss along the specified axis.



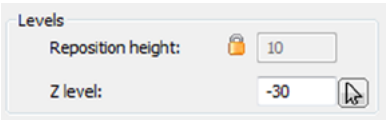
In the **Pocket/Pocket with Boss** cycles, the **Distance** value is measured from the inner walls of the pocket along the specified axis.



Modify geometry

The **Offset** value defines an additional distance from the contact points in the same direction as the **Approach/Retract Distance**. When the measurement is performed along two axes, the offset is left in both directions.

Levels



The **Reposition height** value defines an additional level for the Probe tool along the Z-axis that the tool ascends to when reaching the retract

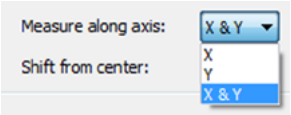
distance. This parameter enables you to avoid obstacles without retracting to the clearance level. You can click the lock icon to enable or disable the field for editing.

Note: This option is disabled for the **Pocket** cycle.

The **Z level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

Measure along axis

The **Measure along axis** option defines the axis where the measurements are performed. You can choose **X**, **Y** or **X&Y** options.



The **Shift from center** option shifts the generated tool path from the center of the part by the specified distance.

Repetitive touches

This field enables you to define the number of touches in each contact point.

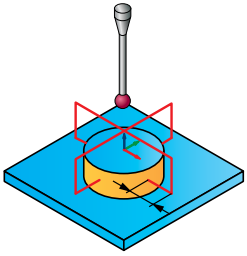
By default, all measurement are performed from the centre of the **Boss/Pocket**.

5.2.4 Hole/Cylinder Cycles

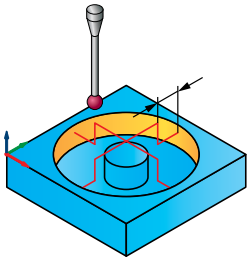
The **Cylinder**, **Hole**, and **Hole with Boss** cycles are defined with the following parameters:

Approach/Retract

In the **Cylinder** cycle, the **Distance** is measured from the perimeter of the cylinder along the radial direction.



In the **Hole** and **Hole with Boss** cycles, the **Distance** is measured from the inner walls of the hole along the radial direction.



Modify geometry

The **Offset** value defines an additional offset from the contact points in the radial direction of the cylinder.

Levels

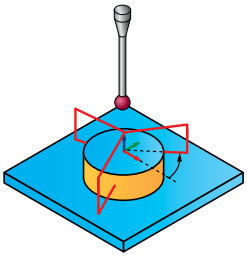
The **Reposition height** value defines an additional level for the Probe tool along the Z-axis that the tool ascends to when reaching the retract distance. This parameter enables you to avoid obstacles without retracting to the clearance level. You can click the lock icon to disable the field for editing.

Note: This option is disabled for the **Hole** cycle.

The **Z level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

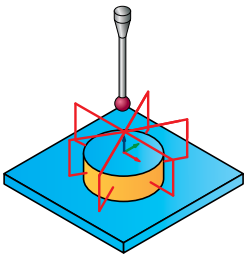
Move start point

The **Move start point** option rotates the generated tool path around the Z-axis by means of shifting the touch points along the perimeter, while it maintains the same angle between them.



No. touch points

The **Number of touch points** defines the number of touch points required to perform the **Hole/ Cylinder** measurement. These points are distributed evenly along the hole/cylinder perimeter.



Repetitive touches

This field enables you to define the number of touches in each contact point.

5.2.5 Arc Cycles

The **Internal** and **External Arc** cycles are defined with the following parameters:

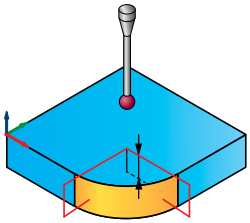
Approach/Retract

In the **Arc** cycles, the **Distance** is measured from the perimeter of the arc along the radial direction.

Modify geometry

The **Offset** value defines an additional offset from the contact points in the radial direction of the arc.

Levels



The **Reposition height** value defines an additional level for the Probe tool along the Z-axis that the tool ascends to when reaching the retract distance. This parameter enables you to avoid obstacles without retracting to the clearance level. You can click the lock icon to enable or disable the field for editing.

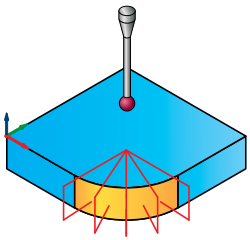
The **Z level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

Shrink geometry

The **Move start point** field enables you to reduce the measurement distance by shifting the start point by the specified value. The point is shifted along the arc perimeter, in the clockwise direction.

The **Move end point** field enables you to reduce the measurement distance by shifting the end point by the specified value. The point is shifted along the arc perimeter, in the counterclockwise direction.

No. touch points



The **Number of touch points** defines the number of touch points required to perform the internal/external arc measurement. These points are distributed evenly along the arc perimeter.

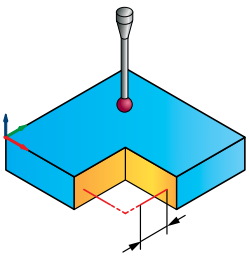
Repetitive touches

This field enables you to define the number of touches in each contact point.

5.2.6 Corner Cycles

The **Internal** and **External Corner** cycles are defined with the following parameters:

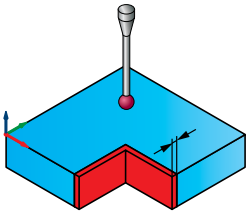
Approach/Retract



The **Distance along axis X** is measured from the contact surface along the X-axis.

The **Distance along axis Y** is measured from the contact surface along the Y-axis.

Modify geometry



The **Offset** value defines an additional offset from the contact points on both surfaces.

Levels

The **Z level** value defines the Z-coordinate of the contact point. The arrow button enables you to select the point directly on the model by using the **Pick on plane** dialog box.

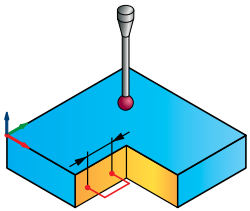
Distances

DX1 defines the distance along the X axis between the vertex of the corner and first contact point .

DY1 defines the distance along the Y axis between the vertex of the corner and first contact point.

DX2 defines the distance along the X axis between the first and the second contact point.

DY2 defines the distance along the Y axis between the first and the second contact point.



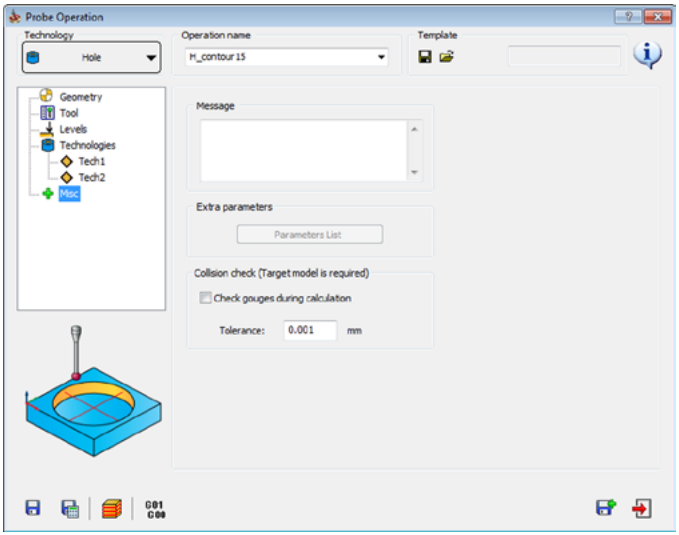
DX2 and **DY2** distances are available only on selection. This pair of distances is optional.

Repetitive touches

This field enables you to define the number of touches in each contact point.

Miscellaneous 6

The **Miscellaneous** page enables you to define a number of miscellaneous parameters for the Probe operation.



6.1 Message

In this field, you can type a message that will appear in the generated GCode file.

6.2 Extra parameters

The **Extra parameters** option enables you to use special operation options implemented in the post-processor for the current CAM-Part. The **Parameters List** button enables you to display the list of additional parameters defined in the post-processor.

6.3 Collision check

This option enables you to perform gouge checking after the calculation of 2.5D Milling operations. During this checking, SolidCAM detects all the gouges between the tool and the target model and informs about the positions where these gouges occur.

The **Check gouges during calculation** check box enables you to perform the gouge checking directly after the tool path calculation. You can also define the gouge checking tolerance in the corresponding edit box. If gouges are detected during this checking, SolidCAM informs you about their location on the model by displaying the coordinates in the **Check gouges** dialog box and dots showing the gouge positions on the model in the graphic area.

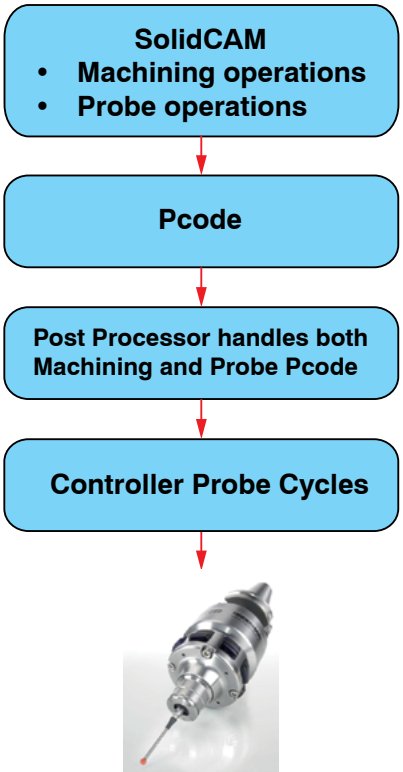
In this dialog box, the check boxes enable you to control the visibility of the gouge positions on the model. When a check box is selected, SolidCAM displays the corresponding gouge position on the model with a red dot.

When a check box is not selected, the related gouge position is not highlighted on the model. The **Show All** button enables you to display all of the detected gouge positions on the model by selecting all of the check boxes. The **Hide All** button enables you to hide all of the detected gouge positions from the model by clearing all of the check boxes.

Process from SolidCAM to Probe

7

The major stages of Solid Probe module action are the following:



7.1 Controller Probe Cycles

A cycle is a parametric program, which commands the probe device to move to a specific point and touch the part. The result of this action is used for two goals:

- ❑ Home definition (store the values in controller home offset table)
- ❑ Measurement (display the values on screen or store the result in a file)

Example of Renishaw cycle for Fanuc controller:

```
G65 P9811 Xx or Yy or Zz [ Ee Ff Hh Mm Qq Ss Tt Uu Vv Ww ]
```

Each program is written in a different language (controller language), but they all do the same actions:

- move the probe device
- touch the part
- process the result

7.2 Renishaw Inspection Plus Software

Renishaw provides the Inspection Plus software that comprises a set of macros controlling probing motions:

Inspection Plus software

Available measurement cycles include:

- X or Y or Z single surface measure
- web/pocket
- four point bore/boss
- vector and angle measure options
- print options (controller dependent)
- extended range of cycles

Renishaw cycles in different Controller

- ❑ Fanuc controller: can add **Renishaw probe cycles (Inspection Plus)**
- ❑ **HAAS** controller: built-in Renishaw cycles (Inspection Plus)
- ❑ **Mazak M32 / M-Plus / Fusion** controllers: built-in Renishaw cycles (Inspection Plus)
- ❑ **Mazatrol Matrix** controller: built-in Renishaw cycles (Inspection Plus)
- ❑ **Heidenhain** (iTNC 530) controller: built-in probe cycles
- ❑ **Siemens** (SINUMERIK 840D sl/840D/840Di sl) controller: built-in probe cycles
- ❑ **Fanuc** controller with **Blum** probe

7.3 Post processors

The SolidCAM post processor, for a specific CNC-machine with a probe, provides complete support for the machining and probing operations.

To adapt an existing post processor to support probe, perform the following changes:

1. Add to the GPP file a call to the include file that supports the probe:

```
inc "00212_P_3X_Renishaw_Fanuc_inc.gpp"
```

2. Modify @change_tool as follows:

```
@change_tool
if tool_type eq tool_probe
    call @prb_chgp ; Change Probe procedure in Include file
    exit
endif
```

Example of Change Probe procedure in Include File (@prb_chgp)

```
@prb_chgp ; change probe
           {nl'M6 T'tool_number' ( PROBE )'}
           {nl'G0  G90  G43  Z[#5043-#'(11000+tool_number)']
    H'tool_number}
    if next_tool_number ne tool_number and next_tool_number
    ne 0
           {nl'T'next_tool_number}
    endif
           call @prb_prbon ; probe on
           gn2<<1>> = 99999.999
           gn2<<2>> = 99999.999
           gn2<<3>> = 99999.999
           gn2<<4>> = 99999.999
           gn2<<5>> = 99999.999
endp
```

To see more examples, download the following files for **Makino Fanuc** machine.

Before change: gMill_Makino_Fanuc_3x_Eval.zip

Changed to support Solid Probe: gMill_Makino_Fanuc_3x_eval – with SolidProbe.zip

Include files for various probes/controllers

Sinumerik 840D: sinumerik_probe.gpp

Alzmetall post using the include file: Alzmetall_GS650_FD-Sin840D_5X.zip

Heidenhain iTNC 350: Heidenhain.iTNC530.probe.cycles.gpp

DMU post using the include file: DMU_80P_hidyn_iTNC530_5X.zip

Fanuc/Blum: 00211_P_3X_Blum_Fanuc_inc[1].gpp

Fanuc/Renishaw: 00212_P_3X_Renishaw_Fanuc_inc.gpp

Measurement Results

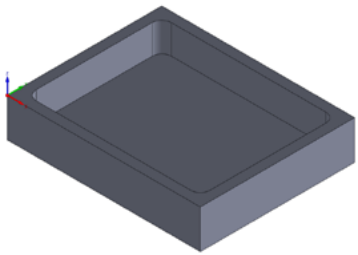
8

The results of the Probe measurements are presented as output in a **Measuring Log**. You can interpret the measuring log data according to your Probe reference manual and compare it with the data calculated by SolidCAM.

The **Examples** folder provided with this book contains a **Measurement Example** CAM-Part and CNC-machine files for the machine definition. Copy the Heidenhain.iTNC530.3X machine files (*.gpp, *.prp, *.vmid, and Heidenhain.iTNC530.probe.cycles.gpp) into your GPPTool folder (the default location is **C:\Users\Public\Documents\SolidCAM\SolidCAM2013\Gpptool**).

Analyzing Measurement Results

An example of the Probe output is presented with a simple CAM-Part having one pocket.



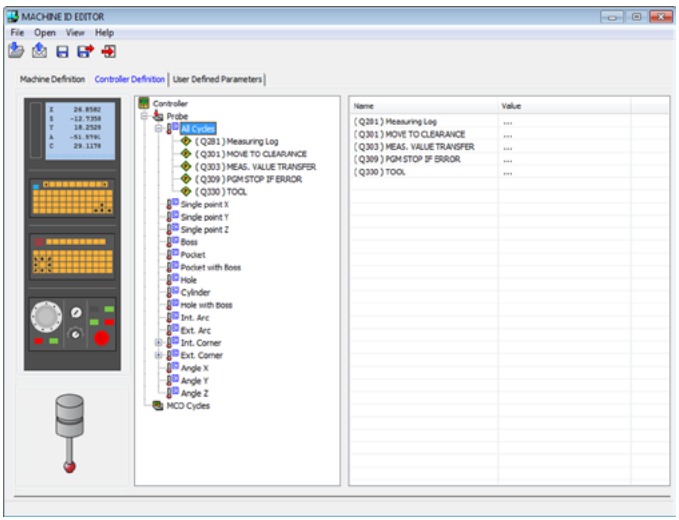
1. Open the CAM-Part

Open the **Measurement Example.prt** file. This is a simple box with dimensions 80x100. It has a 70x90 pocket measured using the Probe tool.

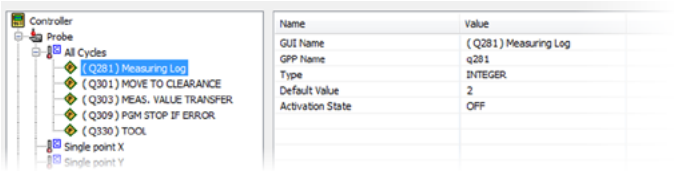
2. Check the Controller Definition

Double-click the **Machine (Heidenhain.iTNC530.3X)** header in **SolidCAM Manager**. The **Machine ID Editor** window is displayed.

Choose the **Controller Definition** tab, expand the **Probe** entry, then expand the **All Cycles** entry and click it. The parameters added for all probe cycles are displayed in the right pane of the window.



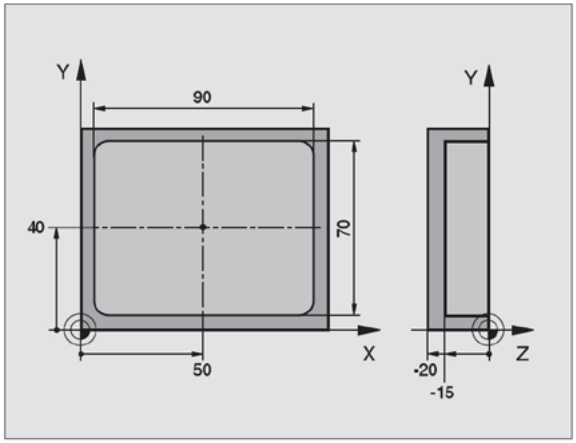
You can click each of the parameters to browse the settings defined for your Probe controller.



Close the **Machine ID Editor** without saving changes.

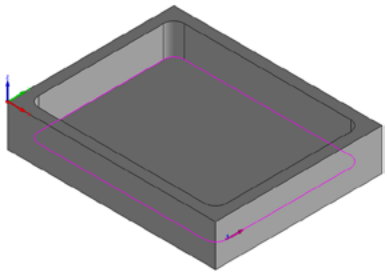
3. Review the Probe operation

The Probe operation defined for the CAM-Part enables you to measure the rectangular pocket.



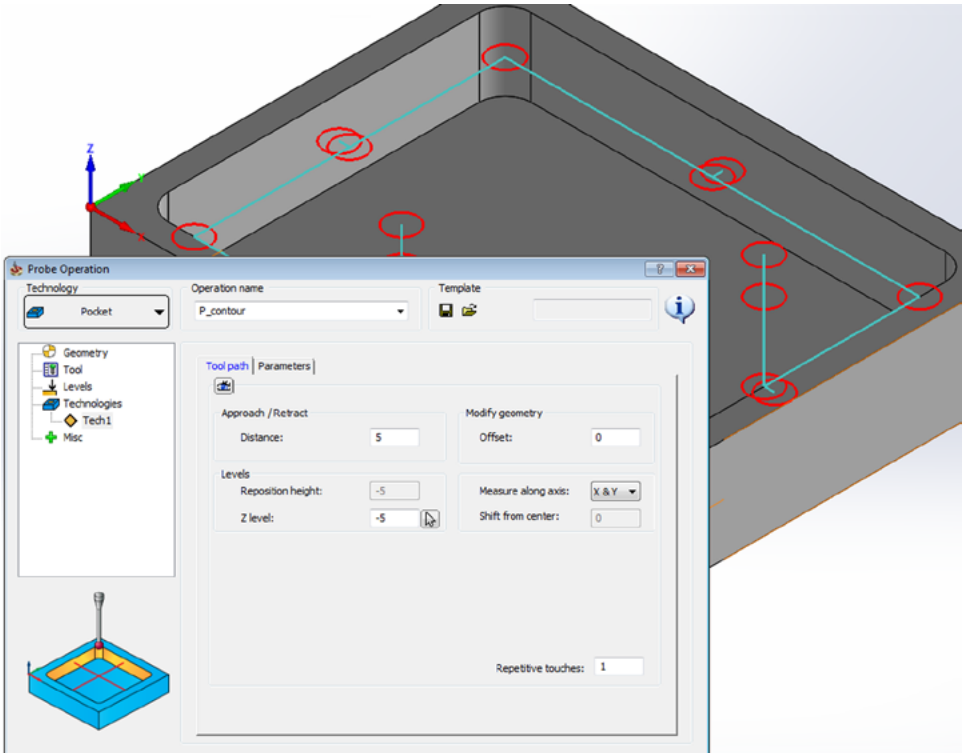
Double-click the **P_contour** operation to open it.

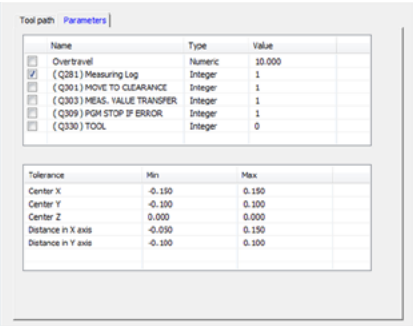
The geometry for the operation is defined by the pocket boundaries.



The tool path settings are defined as shown below.

The cycle tolerances and parameters set within the **Controller Definition** are displayed in the **Parameters** tab as shown.





4. Generate the GCode

Click the **G-Code** icon G01 G00 at the bottom of the operation dialog box. The following output file is generated:

```
0 BEGIN PGM P_CONTOUR MM
1 ; 22-SEP-2013 -- 11:18:08
2 BLK FORM 0.1 Z X0 Y0 Z0
3 BLK FORM 0.2 X0 Y0 Z0
4 L Z0 R0 FMAX M91
5 TOOL CALL 1 Z
6 ;OP1 ********** ********** **********
7 CYCL DEF 247 DATUM SETTING ~
Q339=1 ;datum number
8 TCH PROBE 423 MEAS. RECTAN. INSIDE ~
Q273=50 ;CENTER IN 1ST AXIS ~
Q274=40 ;CENTER IN 2ND AXIS ~
Q282=90 ;1ST SIDE LENGTH ~
Q283=70 ;2ND SIDE LENGTH ~
Q261=-5 ;MEASURING HEIGHT ~
Q320=0 ;SET-UP CLEARANCE ~
Q260=20 ;CLEARANCE HEIGHT ~
Q301=0 ;TRAVERSE TO CLEARANCE HEIGHT ~
Q284=90.15 ;MAX. LIMIT 1ST SIDE ~
Q285=89.95 ;MIN. LIMIT 1ST SIDE ~
Q286=70.1 ;MAX. LIMIT 2ND SIDE ~
Q287=69.9 ;MIN. LIMIT 2ND SIDE ~
Q279=0.15 ;TOLERANCE 1ST CENTER ~
Q280=0.1 ;TOLERANCE 2ND CENTER ~
Q281=1 ;MEASURING LOG ~
Q309=0 ;PGM STOP IF ERROR ~
Q330=0 ;TOOL NUMBER
9 L Z0 R0 FMAX M91
10 L X0 Y0 R0 FMAX M91
11 END PGM P_CONTOUR MM
```

5. Check the controller report

Upon completion of the measurement, the Probe controller produces a measuring log. The results can be viewed onscreen, presented in a file or have no output at all, depending on the settings.

The example of the **Heidenhain** controller report for this operation is shown below:

```
-----------------------------------------------------------------------------------------------
***************** MEASURING LOG FOR PROBING CYCLE 423: RECTANGULAR POCKET MEASURING *******************
DATE: 29-09-1997
TIME: 8:21:33
MEASURING PROGRAM: TNC:\BSMEAS\BSMES.H
-----------------------------------------------------------------------------------------------
NOML. VALUES:            CENTER IN REF. AXIS:                50.0000
                         CENTER IN MINOR AXIS:               40.0000

                         SIDE LENGTH IN REF. AXIS:           90.0000
                         SIDE LENGTH IN MINOR AXIS:          70.0000
-----------------------------------------------------------------------------------------------
GIVEN LIMIT VALUES:      MAX. FOR CENTER IN REF. AXIS:       50.1500
                         MIN. FOR CENTER IN REF. AXIS:       49.8500

                         MAX. FOR CENTER IN MINOR AXIS:      40.1000
                         MIN. FOR CENTER IN MINOR AXIS:      39.9000

                         MAX. IN REFERENCE AXIS:             90.1500
                         MINIMUM DIMENSION IN REFERENCE AXIS: 89.9500

                         MAXIMUM SIDE LENGTH IN MINOR AXIS:  70.1000
                         MINIMUM SIDE LENGTH IN MINOR AXIS: 69.9500
***********************************************************************************************
ACTUAL VALUES:           CENTER IN REF. AXIS:                50.0905
                         CENTER IN MINOR AXIS:               39.9347

                         SIDE LENGTH IN REF. AXIS:           90.1200
                         SIDE LENGTH IN MINOR AXIS:          69.9920
-----------------------------------------------------------------------------------------------
DEVIATIONS:              CENTER IN REF. AXIS:                0.0905
                         CENTER IN MINOR AXIS:               -0.0653

                         SIDE LENGTH IN REF. AXIS:           0.1200
                         SIDE LENGTH IN MINOR AXIS:          -0.0080
***********************************************************************************************
FURTHER MEASURING RESULTS: MEASURING HEIGHT: -5.0000
*************************************ENDOFMEASURINGLOG*************************************
```

The report data shows that the following deviations were found during the measurement:

`CENTER IN REF. AXIS: 0.0905`	Deviation value for rectangle center in X axis
`CENTER IN MINOR AXIS: -0.0653`	Deviation value for rectangle center in Y axis
`SIDE LENGTH IN REF. AXIS: 0.1200`	Deviation value for rectangle value in X axis
`SIDE LENGTH IN MINOR AXIS: -0.0080`	Deviation value for rectangle value in Y axis

Another example demonstrates the results of the **Fanuc** controller report:

INSPECTION REPORT

DATE 20130912 TIME 72112

MEASURING MACHINE PINNACLE DV2152

PROBING SYSTEM MARPOSS WRS

MEASURING SOFTWARE SNR-PRO

==========

FEATURE NUMBER 1

NOMINAL D 10.1300 ACTUAL 10.0980 TOL +/- .0500 DEV .0320

TRP D 10.1300 ACTUAL 10.0980 TOL TP .1000 ACT TP .0351

FEATURE NUMBER 2

NOMINAL D 10.1300 ACTUAL 10.1010 TOL +/- .0500 DEV .0290

TRP D 10.1300 ACTUAL 10.1010 TOL TP .1000 ACT TP .0361

FEATURE NUMBER 3

NOMINAL D 10.1300 ACTUAL 10.0910 TOL +/- .0500 DEV .0390

TRP D 10.1300 ACTUAL 10.0910 TOL TP .1000 ACT TP .0324

FEATURE NUMBER 4

NOMINAL D 10.1300 ACTUAL 10.0910 TOL +/- .0500 DEV .0390

TRP D 10.1300 ACTUAL 10.0910 TOL TP .1000 ACT TP .0326

For full interpretation of the details indicated in a log, refer to the User manual provided by your Probe supplier.

Click the YouTube button to watch a recording illustrating the use of the Solid Probe and tool presetter on a CNC-machine.

Exercises 9

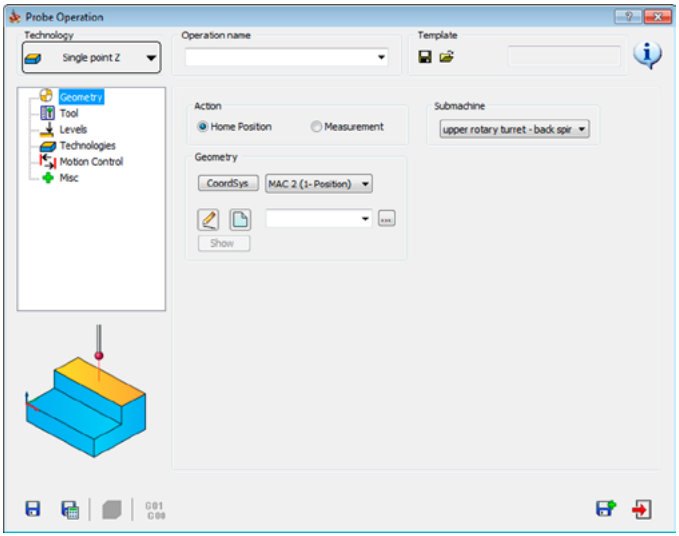
Exercise #1: Home Position

This exercise illustrates how the Coordinate System location can be corrected on the workpiece clamped in the CNC-Machine.

Open the SolidCAM part **SolidProbe.prz** located in the **Exercises** folder.

This part contains a few machining operations that were defined to reach the target model. You can add Probe operations to check these machining operations directly in SolidCAM. The final GCode you generate at the end of the work will contain both the machining and probe operations.

1. Right-click the first Face Milling Operation and choose **Add Probe** from the menu. The **Probe Operation** dialog box is displayed.

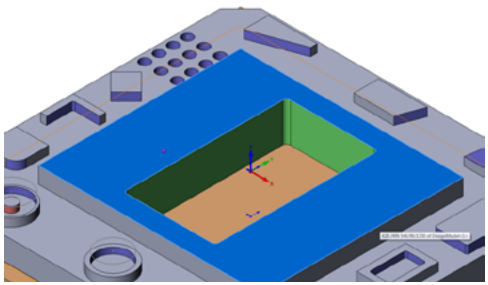


2. Make sure that the default **Single point Z** cycle  is selected in the left top corner, under **Technology**.

3. In the **Action** area, make sure that the **Home Position** option is chosen.

4. Under **Geometry**, click the **New** icon  . The **Point Geometry Selection** dialog box is displayed.

   Click on the model as shown in the picture to select the contact point where the Z-coordinate should be set to **0**.



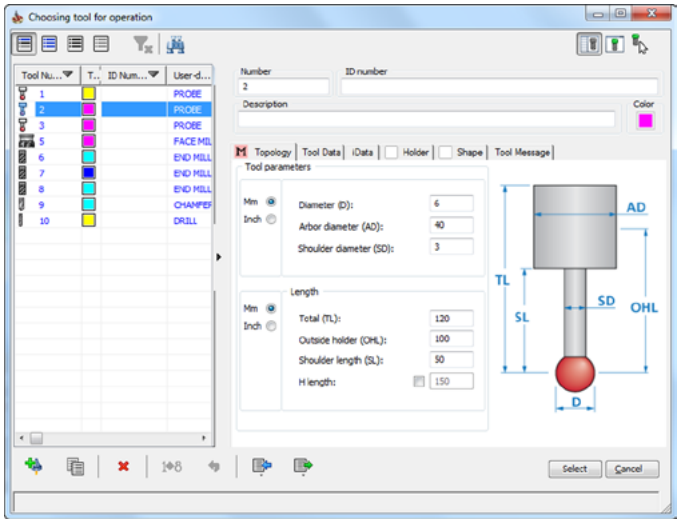
Click ✔ to confirm your selection. The **Probe Operation** dialog box is displayed again.

5. Switch to the **Tool** page. Click **Select** to choose a **Probe** tool for operation.

   The **Choosing tool for operation** dialog box is displayed.

   Click the **Add Milling Tool** icon to add a new **Probe** tool.

   The following default tool is loaded:

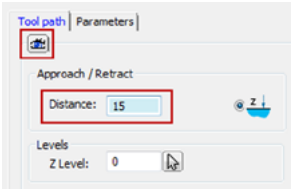


   - Set the **Diameter** to **20**.
   - Choose the **BT40ER32x60** holder.

   Click **Select** to choose the probe for the operation.

6. Switch to the **Technologies** page. Notice that the selected geometry is shown in **Geometry to work on** and the Cycle status is ✔ indicating that the geometry is chosen correctly.

7. Double-click in the **Tech1** row of the technologies table to display the operation technological parameters.



8. In the **Tool path** page, set the **Approach/Retract Distance** to **15**. Click the **Preview** icon to display the generated tool path along with the selected tool diameter.

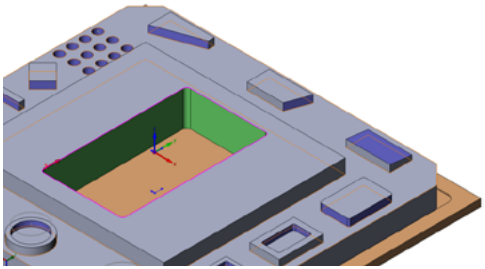
9. Click the **Save & Calculate** icon to save and calculate the GCode for this operation.

10. Click the **Save & Copy** icon to add a new Probe operation.

11. In the **Technology** section, choose the **Pocket** strategy .

12. Under **Geometry**, click the **New** icon . The **Geometry Edit** dialog box is displayed. Select the chain as shown on the picture.

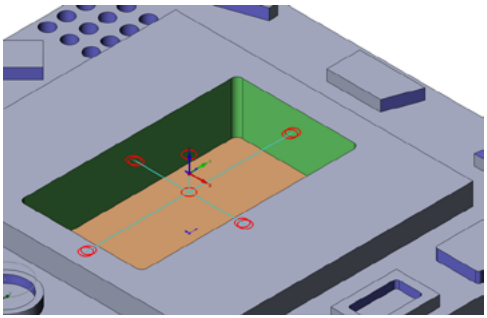
    Click to confirm your selection. The **Probe Operation** dialog box is displayed again.



13. Use the same tool as in the previous operation.

14. In the **Technologies** page, choose **Tech1** and set **Approach/Retract Distance** to **15**. Set **Z Level** to **-30**. In the **Measure along axis** list, choose **X & Y**.

15. Click the **Preview** icon to display the generated tool path.



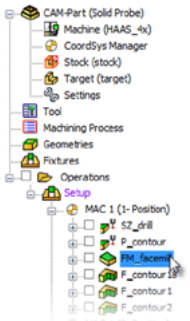
16. Click to save and calculate the GCode for this operation.

17. Click the **Simulate** icon . The **Simulation** dialog box is displayed.

    Select both operations and click the **Play** button to play the simulation.

    In the first operation, the tool comes down to the Z-level. In the second operation, it moves towards the middle and then touches all four surfaces along the X- and Y-axes defining the center of the pocket as the Home Position.

    Click **Exit** to close the dialog box.



18. In the **SolidCAM tree**, drag the Face Milling operation under the Probe operations to enable the Home Position definition at the beginning of the work.

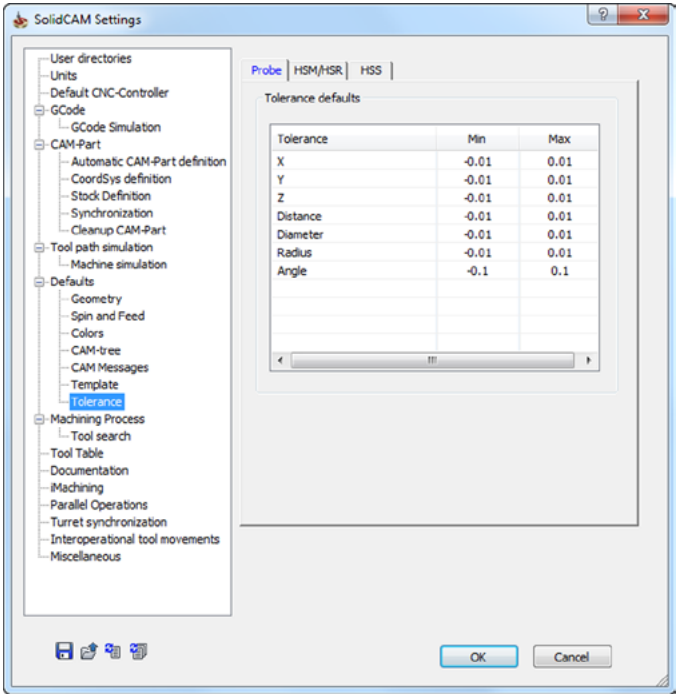
Exercise #2: Single Point Measurement

This exercise illustrates how the part measurements are performed after every stage of machining.

Use the same part **SolidProbe.prz** as in the previous exercise.

1. To set the tolerances used for measurements, in **SolidCAM Settings**, open **Defaults > Tolerance > Probe** and define the tolerances as follows:
   - For **X**, set **Min** to **-0.01** and **Max** to **0.01**.

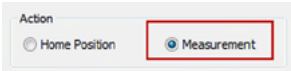
   For the rest of the parameters use the same values, except for the **Angle**.
   - For **Angle**, set **Min** to **-0.1** and **Max** to **0.1**.



   Click **OK** to save the settings.

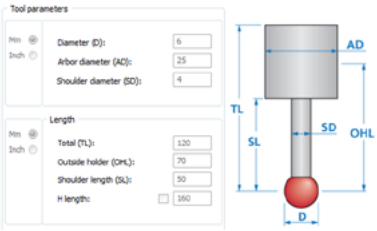
2. Right-click the Face Milling Operation and choose **Add Probe** from the menu to verify the results of the Face Milling operation.

3. In the **Geometry** page, use the default **Single point Z** strategy  . Under **Action**, choose **Measurement**.



4. Click to define the Geometry. Click on the model upper surface to define the Z-level for the operation.

5. Define a new Probe tool.

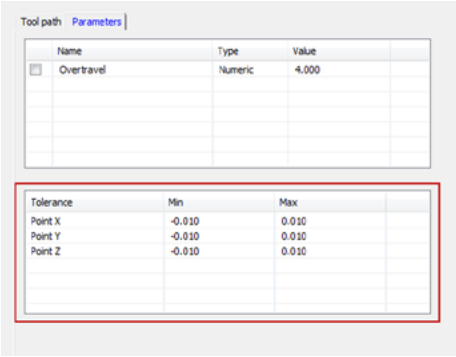


   - Set the **Arbor Diameter** to **25**.
   - Set the **Outside Holder Length** to **70**.
   - Set **H length** to **130**.

   Click **Select** to choose the probe for the operation.

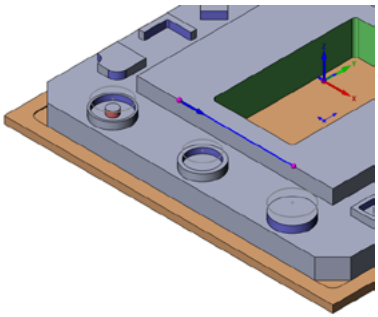
6. Switch to the **Tech1** page. In the **Tool path** area, set the **Approach/Retract Distance** to **6**. Click to display the preview.

7. In the **Parameters** area, take a note that only the relevant tolerances are chosen for the operation, in accordance with the settings that have been defined in **Step 1**.

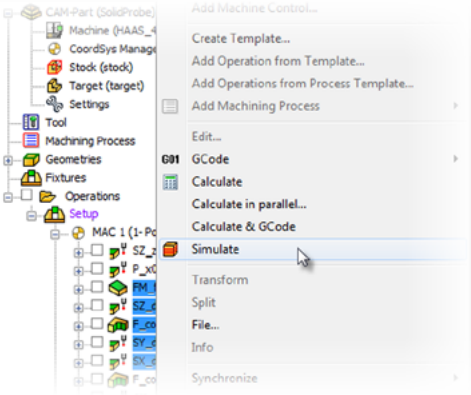


8. Click to save the operation data and calculate the tool path. Click to add a new Probe operation.

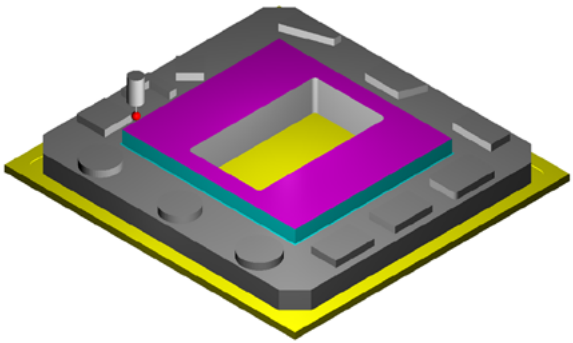
9. In the **Technology** section, choose the **Single point Y** strategy . Define a new Geometry as shown below.



10. In the **Tech1** section, enable the preview by clicking . Set the **Approach/Retract Distance** to **40**. Choose the appropriate direction for the probe tool approach.
11. Click to save the operation data and calculate the tool path. Click to add a new Probe operation.
12. In the same manner as in the previous operations, define a **Single Point X** operation.
13. Click to save the operation data and calculate the tool path. Click **Exit** ( ).
14. In the **SolidCAM tree**, drag the **Single Point Y** and **Single Point X** operation under the Profile operation **F_contour18** to set the correct order of operations.
15. Holding the **Shift** key, select the Face Milling operation, **Single Point Z**, **Profile**, **Single Point Y**, **Single Point X**, and **Profile** operations. Right-click the operations and choose **Simulate**.



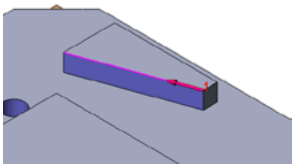
16. Play the simulation in the **SolidVerify** mode.
    The Solid Probe measurement is performed after the machining operation.



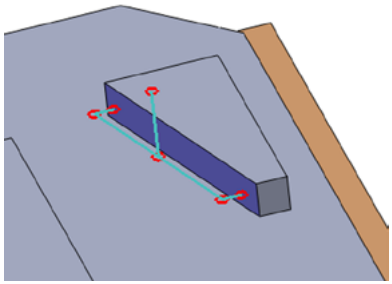
Exercise #3: Angle Measurement

Use the same part **SolidProbe.prz** as in the previous exercise.

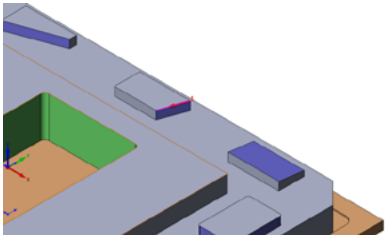
1. Right-click the **F_contour1** Profile operation and add a new Probe operation.
2. In the **Geometry** page, use the **Angle Y** strategy . Under **Action**, choose **Measurement**.
3. Choose **contour1** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.



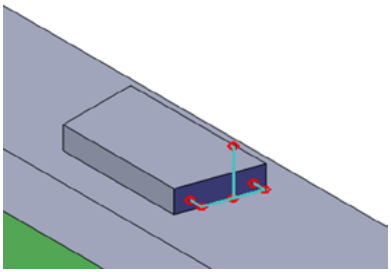
4. Use the **Ø6** Probe tool.
5. Switch to the **Tech1** page.
   - Set the **Approach/Retract Distance** to **13**.
   - Set **Z Level** to **-30**.
   - Under **Shrink Geometry**, set both the **Move start point** value and **Move end point** value to **10**.
6. Click to preview the tool path.



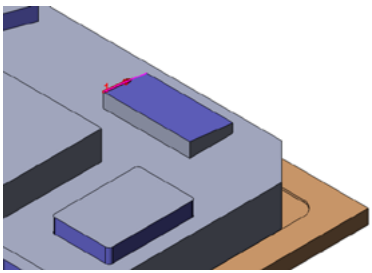
7. Click to save the operation data and calculate the tool path. Click .
8. Right-click the **F_contour2** Profile operation and add a new Probe operation.
9. In the **Geometry** page, use the **Angle X** strategy . Under **Action**, choose **Measurement**.



10. Choose **contour2** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.
11. Use the same **Ø6** Probe tool.
12. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set **Z Level** to **-30**.
    - Under **Shrink Geometry**, set both the **Move start point** value and **Move end point** value to **10**.
13. Click to preview the tool path.

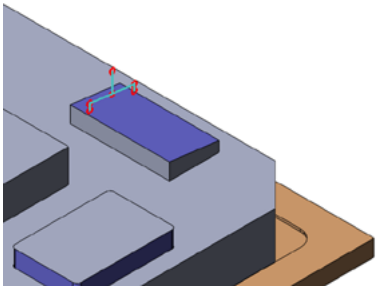


14. Click to save the operation data and calculate the tool path. Click .
15. Right-click the **FM_facemill1** Profile operation and add a new Probe operation.
16. In the **Geometry** page, use the **Angle Z** strategy . Under **Action**, choose **Measurement**.
17. Define a new geometry as shown on the picture.

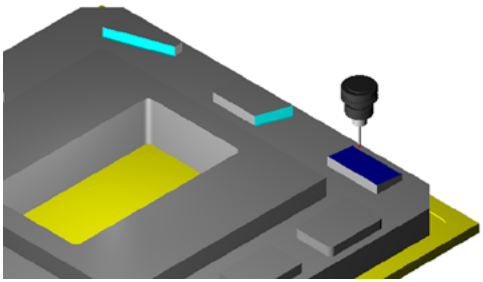


18. Use the same tool as in the previous operations.

19. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **10**.
    - Set **Measure plane** to **YZ**.
    - Set the **Side shift** value to **-20**.
    - Under **Shrink Geometry**, set both the **Move start point** value and **Move end point** value to **3**.
20. Click to preview the tool path.



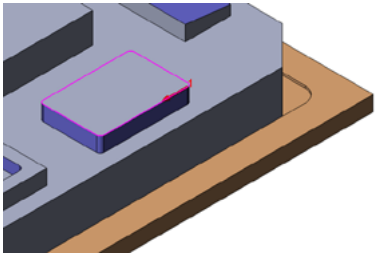
21. Click to save the operation data and calculate the tool path. Click .
22. Select the Probe operations along with the corresponding machining operations in the **SolidCAM tree** and play the simulation.



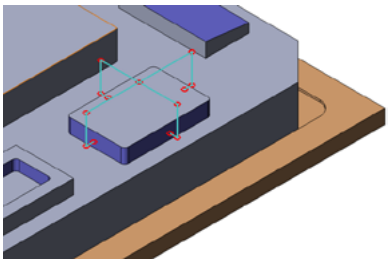
Exercise #4: Boss/Pocket Measurement

Use the same part **SolidProbe.prz** as in the previous exercises.

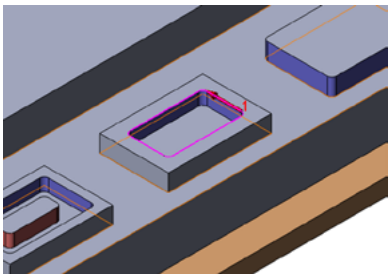
1. Right-click the **F_contour4** Profile operation and add a new Probe operation.
2. In the **Geometry** page, use the **Boss** strategy . Under **Action**, choose **Measurement**.
3. Choose **contour4** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.



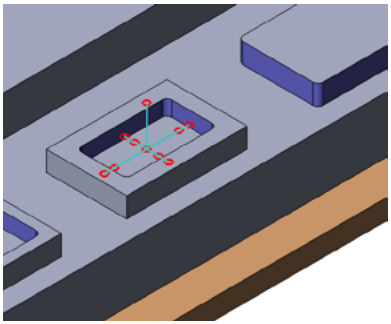
4. Use the **Ø6** Probe tool.
5. Switch to the **Tech1** page.
   - Set the **Approach/Retract Distance** to **13**.
   - Set the **Reposition height** to **10**.
   - Set **Z Level** to **-30**.
   - In the **Measure along axis** list, choose the **X & Y** option.
6. Click to preview the tool path.



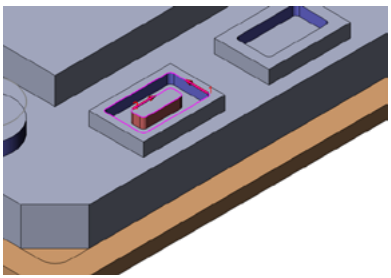
7. Click to save the operation data and calculate the tool path. Click .
8. Right-click the **P_contour5** Pocket operation and add a new Probe operation.
9. Use the **Pocket** strategy in the **Geometry** page. Under **Action**, choose **Measurement**.
10. Choose **contour5** from the list of available geometries. The same geometry was used in the corresponding Pocket machining operation.



11. Use the **Ø6** Probe tool.
12. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set **Z Level** to **-30**.
    - In the **Measure along axis** list, choose the **X & Y** option.
13. Click to preview the tool path.

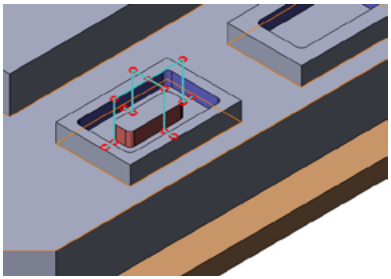


14. Click to save the operation data and calculate the tool path. Click .
15. Right-click the **P_contour6** Pocket operation and add a new Probe operation.
16. In the **Geometry** page, use the **Pocket with Boss** strategy . Under **Action**, choose **Measurement**.
17. Choose **contour6** from the list of available geometries. The same geometry was used in the corresponding Pocket machining operation.

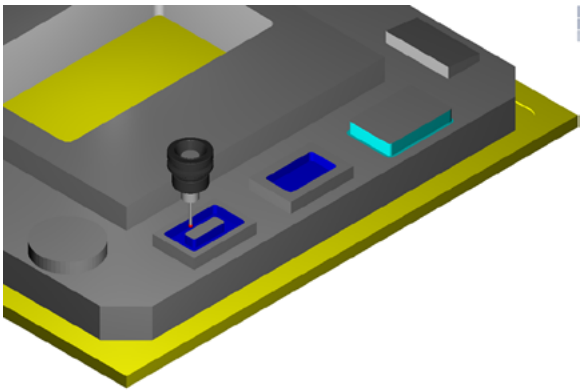


18. Use the **Ø6** Probe tool.
19. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set the **Reposition height** to **10**.
    - Set **Z Level** to **-30**.
    - In the **Measure along axis** list, choose the **X & Y** option.

20. Click to preview the tool path.



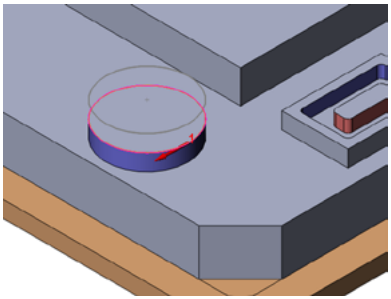
21. Click to save the operation data and calculate the tool path. Click .
22. Select the Probe operations along with the corresponding machining operations in the **SolidCAM tree** and play the simulation.



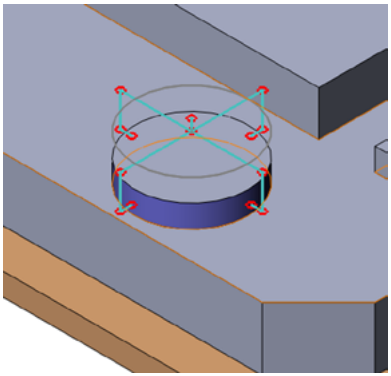
Exercise #5: Cylinder/Hole Measurement

Use the same part **SolidProbe.prz** as in the previous exercises.

1. Right-click the **F_contour7** Profile operation and add a new Probe operation.
2. In the **Geometry** page, use the **Cylinder** strategy . Under **Action**, choose **Measurement**.
3. Choose **contour7** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.

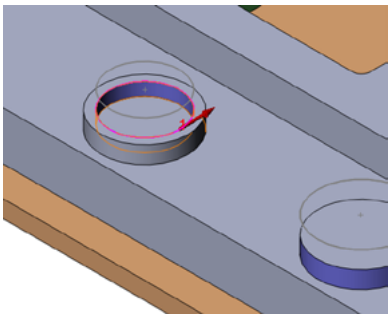


4. Use the **Ø6** Probe tool.
5. Switch to the **Tech1** page.
   - Set the **Approach/Retract Distance** to **13**.
   - Set the **Reposition height** to **10**.
   - Set **Z Level** to **-30**.
   - Set the **No. touch points** to **4**.
6. Click to preview the tool path.

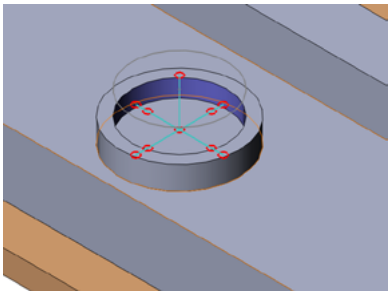


7. Click to save the operation data and calculate the tool path. Click .
8. Right-click the **P_contour8** Pocket operation and add a new Probe operation.
9. In the **Geometry** page, use the **Hole** strategy . Under **Action**, choose **Measurement**.

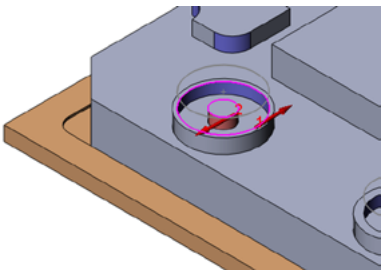
10. Choose **contour8** from the list of available geometries. The same geometry was used in the corresponding Pocket machining operation.



11. Use the **Ø6** Probe tool.
12. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set **Z Level** to **-30**.
    - Set the **No. touch points** to **4**.
13. Click to preview the tool path.

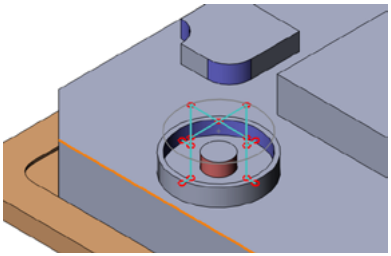


14. Click **Save & Calculate** to save the operation data and calculate the tool path. Click **Exit**.
15. Right-click the **P_contour9** Pocket operation and add a new Probe operation.
16. In the **Geometry** page, use the **Hole with Boss** strategy . Under **Action**, choose **Measurement**.
17. Choose **contour9** from the list of available geometries. The same geometry was used in the corresponding Pocket machining operation.

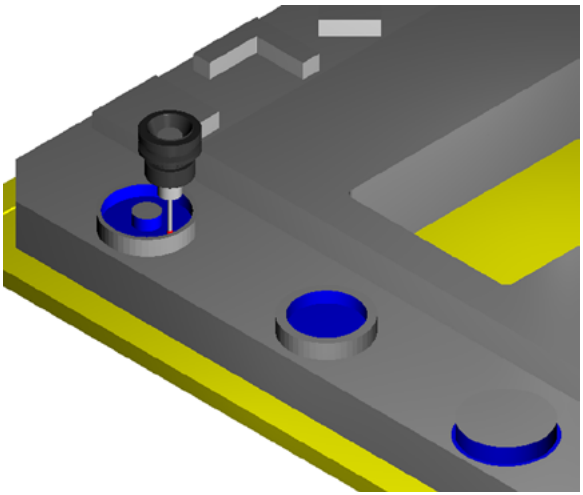


18. Use the **Ø6** Probe tool.
19. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set the **Reposition height** to **10**.
    - Set **Z Level** to **-30**.
    - Set the **No. touch points** to **4**.

20. Click to preview the tool path.



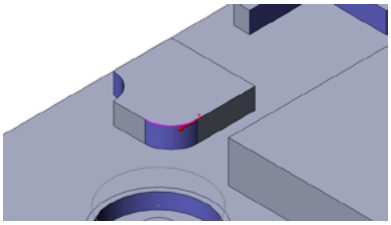
21. Click to save the operation data and calculate the tool path. Click .
22. Select the Probe operations along with the corresponding machining operations in the **SolidCAM tree** and play the simulation.



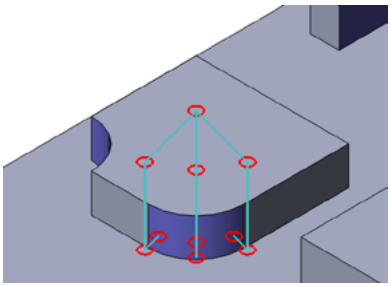
Exercise #6: Corner Measurement

Use the same part **SolidProbe.prz** as in the previous exercises.

1. Right-click the **F_contour10** Profile operation and add a new Probe operation.
2. In the **Geometry** page, use the **External arc** strategy . Under **Action**, choose **Measurement**.
3. Choose **contour10** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.

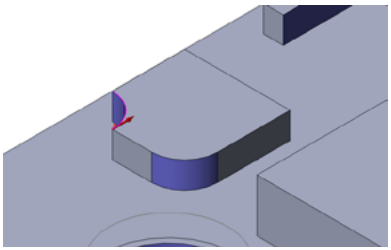


4. Use the **Ø6** Probe tool.
5. Switch to the **Tech1** page.
   - Set the **Approach/Retract Distance** to **13**.
   - Set the **Reposition height** to **10**.
   - Set **Z Level** to **-30**.
   - Set the **No. touch points** to **3**.
   - Under **Shrink Geometry**, set both the **Move start point** value and **Move end point** value to **15**.
6. Click to preview the tool path.

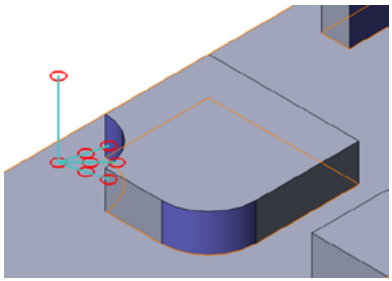


7. Click to save the operation data and calculate the tool path. Click .
8. Right-click the **F_contour11** Profile operation and add a new Probe operation.
9. In the **Geometry** page, use the **Internal arc** strategy . Under **Action**, choose **Measurement**.

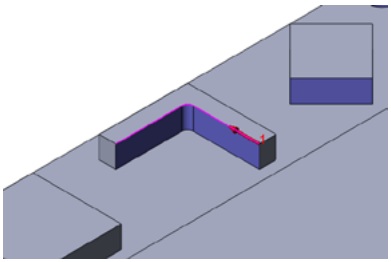
10. Choose **contour11** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.



11. Use the **Ø6** Probe tool.
12. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set **Z Level** to **-30**.
    - Set the **No. touch points** to **3**.
    - Under **Shrink Geometry**, set both the **Move start point** value and **Move end point** value to **15**.
13. Click to preview the tool path.

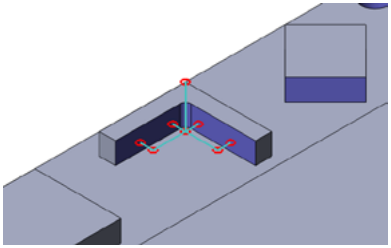


14. Click to save the operation data and calculate the tool path. Click .
15. Right-click the **F_contour12** Profile operation and add a new Probe operation.
16. In the **Geometry** page, use the **Internal corner** strategy . Under **Action**, choose **Measurement**.
17. Choose **contour12** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.

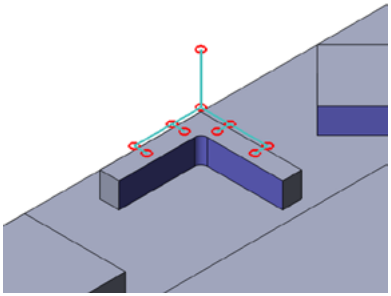


18. Use the **Ø6** Probe tool.
19. Switch to the **Tech1** page.
    - Set both **Approach/Retract Distance along axis X** and **Distance along axis Y** to **15**.
    - Set **Z Level** to **-30**.
    - Under **Distances**, click the check box and set both the **DX2** and **DY2** to **30**.

20. Click to preview the tool path.



21. Click to save the operation data and calculate the tool path. Click .
22. Right-click the **F_contour13** Profile operation and add a new Probe operation.
23. In the **Geometry** page, use the **External corner** strategy . Under **Action**, choose **Measurement**.
24. Choose **contour13** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation.
25. Use the **Ø6** Probe tool.
26. Switch to the **Tech1** page.
    - Set both **Approach/Retract Distance along axis X** and **Distance along axis Y** to **13**.
    - Set **Z Level** to **-30**.
    - Under **Distances**, set both the **DX1** and **DY1** to **10**. Click the check box and set both the **DX2** and **DY2** to **30**.
27. Click to preview the tool path.

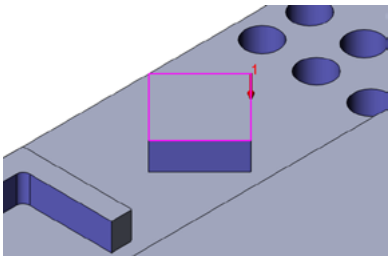


28. Click to save the operation data and calculate the tool path. Click .

29. Right-click the **F_contour14** Profile operation and add a new Probe operation.

30. In the **Geometry** page, use the **Boss** strategy . Under **Action**, choose **Measurement**.

31. Choose **contour14** from the list of available geometries. The same geometry was used in the corresponding Profile machining operation. Take a note that the boss edges are not parallel to the Coordinate System axes.

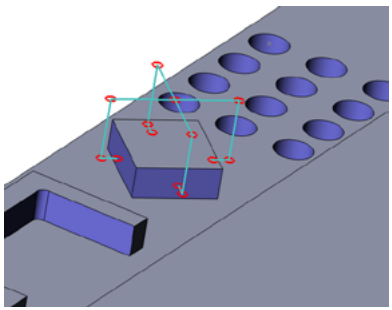


32. Use the **Ø6** Probe tool.

33. Switch to the **Tech1** page.
    - Set the **Approach/Retract Distance** to **13**.
    - Set the **Reposition height** to **10**.
    - Set **Z Level** to **-30**.
    - In the **Measure along axis** list, choose the **X & Y** option.

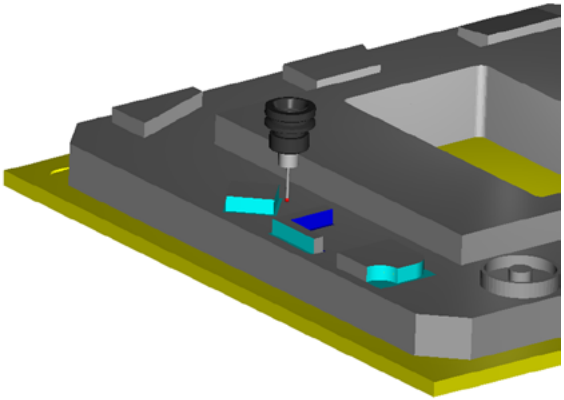
    With this strategy, you do not have to set the angle, it is defined automatically on the model.

34. Click to preview the tool path.



35. Click to save the operation data and calculate the tool path. Click .

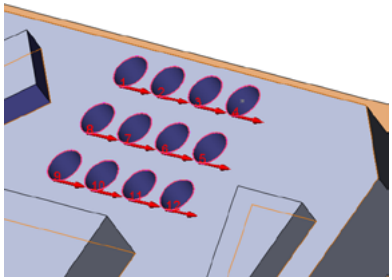
36. Select the Probe operations along with the corresponding machining operations in the **SolidCAM tree** and play the simulation.



Exercise #7: Multiple Holes Measurement

This exercise illustrates how multiple holes can be measured using different technologies.

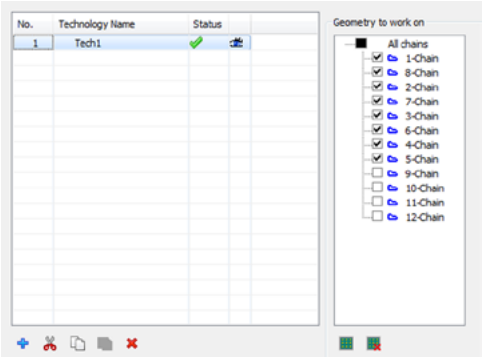
1. Right-click the **F_contour15** Profile operation and add a new Probe operation.
2. In the **Geometry** page, use the **Hole** strategy . Under **Action**, choose **Measurement**.
3. Choose **contour15** from the list of available geometries. The same geometry was used in the corresponding Profile operation.



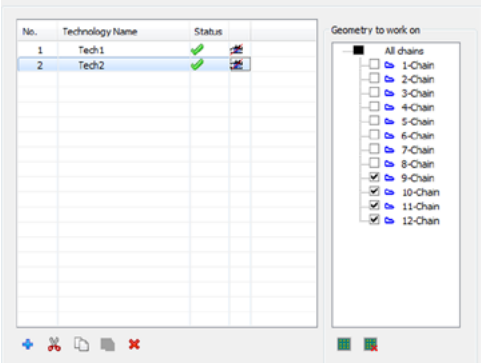
4. Use the **Ø6** Probe tool.
5. Switch to the **Technologies** page.

   You can split the geometries listed on the right side of the dialog box in order to apply different types of measurements.

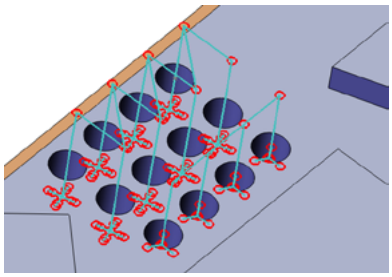
   Click the check boxes near the last four geometries to remove the selection.



6. Click the **Add Cycle** button to add a **Tech2** technology. Clear the check boxes near the first eight geometries to remove the selection.



7. Switch to the **Tech1** page. Click to preview the tool path.
   - Set the **Approach/Retract Distance** to **6**.
   - Set **Z Level** to **-70**.
   - Set the **No. touch points** to **4**.
   - Set the **Move start point** value to **15**.
8. Switch to the **Tech2** page. Click to preview the tool path.
   - Set the **Approach/Retract Distance** to **12**.
   - Set **Z Level** to **-50**.
   - Set the **No. touch points** to **3**.
9. Switch back to the **Technologies** page. You can also set the sorting order for chosen geometries. Select the **Tech1** row. Below the geometries list, click the **Sort geometry** button . In the **Advanced Sorting** dialog box, choose the first option and click **OK**.
10. Click to preview both types of technologies.
11. Click to save the operation data and calculate the tool path.



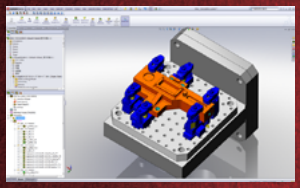
12. Simulate the operation in the **SolidVerify** mode.

SolidCAM + SolidWorks

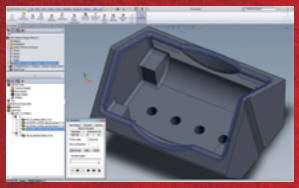
The complete integrated Manufacturing Solution

SolidCAM

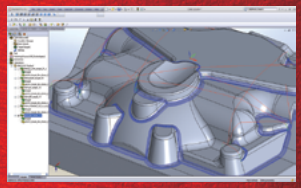
iMachining – The Revolution in CAM!



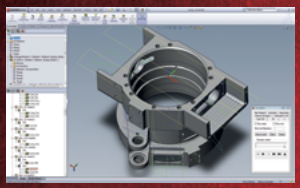
2.5D Milling



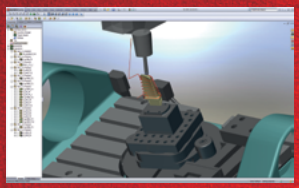
HSS (High-Speed Surface Machining)



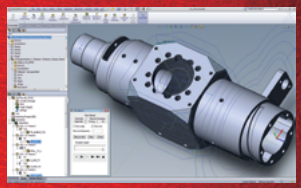
HSM (High-Speed Machining)



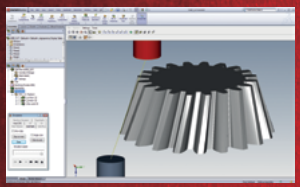
Indexed Multi-Sided Machining



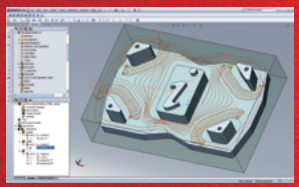
Simultaneous -5Axis Machining



Turning and Mill-Turn up to -5Axis



Wire EDM



iMachining

Service and Support

www.youtube.com/SolidCAMProfessor
www.youtube.com/SolidCAMiMachining

www.facebook.com/SolidCAM

www.solidcam.com