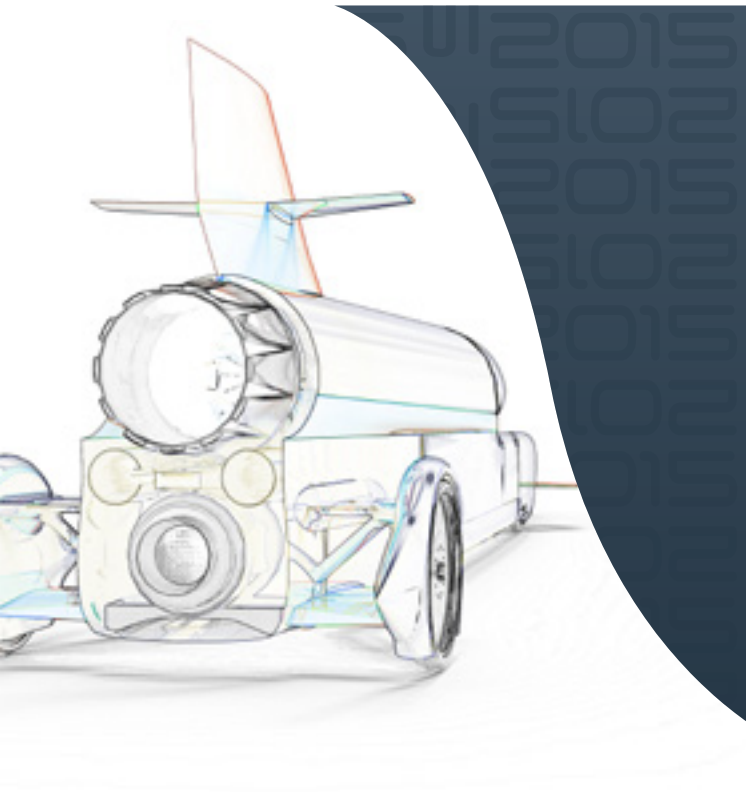


Delcam Advancec Manufacturing Solutions

PartMaker 2015

Advanced Surface Machining

Version 2015 or Higher

User Guide

PartMaker 2015

User Manual

User Guide/PartMaker Advanced Surface Machining

Important User Notices

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

Acknowledgements

Patents

PartMaker software is subject to the following patents:
Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces
Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

PartMaker 2015. Published on 08 December 2014

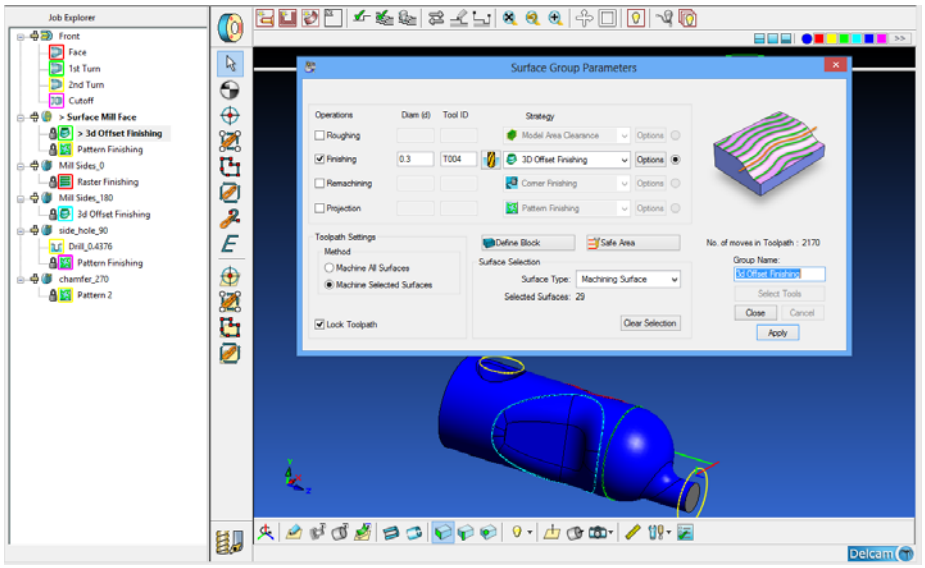
Contents

Introduction to PartMaker/Advanced Surface Machining (ASM) Module

Introduction

PartMaker/Advanced Surface Machining Module (ASM) includes toolpath-generation machining strategies for milling free-shape surfaces using Machining Centers (PartMaker/Mill), Turn-Mill Centers (PartMaker/Turn-Mill) and Swiss-type Lathes (PartMaker/SwissCAM) equipped with Live Tooling.

To use ASM, you need the ASM cost option.



Key issues in milling of complex shapes are:

- Accuracy and finish of the part.
- Convenience in generation of desired toolpaths.
- The time spent preparing and validating toolpaths prior to machining.
- The machine-time required to complete the operation.

It is also important that the toolpaths are correct and that the tool does not gouge the part or leave material in unexpected portions. ASM addresses all these key issues. It implements algorithms that dramatically cut down the time required for manufacturing of parts. Inputs required by ASM can be provided as an imported Solid Model. The user has the ability to machine the entire model or he or she can select certain sections using the mouse. Subsequent to this, ASM operates without any intervention and produces toolpaths as specified. ASM ensures that the toolpaths are mathematically accurate and gouge-free to the given tolerance. Desired toolpath geometry and the surface finish are obtained with appropriate choice of settings used to generate the toolpaths.

The machining strategies in ASM are fast, safe and reliable. They cover different phases of milling, namely, Roughing, Finishing, Remachining and Projection Machining. For each phase, toolpaths in various machining styles are possible, also known as strategies. For each machining strategy, further variation in the toolpath can be introduced by using different boundary conditions, transitions, entry or exit conditions and other toolpath parameters.

Major Features

Input Options

- Imported Solid Models: Parasolid Files (*.x_t, *.x_b), SolidWorks Files (*.sldprt), Inventor Files (*.ipt), STEP (*.STEP, *.STP) and STL files (*.stl).
- Boundary curves and Projection Curves are constructed as standard PartMaker Profiles

Surface Machining Options

Roughing

- Model Area Clearance – Includes Raster and Offset roughing

Finishing

- Raster Finishing
- Raster Flat Finishing
- 3D Offset Finishing
- Offset Flat Finishing
- Constant Z Finishing
- Steep and Shallow Finishing
- Radial Finishing
- Spiral Finishing
- Rotary Finishing

Remachining

- Corner Finishing
- Corner Pencil

Projection

- Pattern Finishing

Cutting Limits and Constraints Options

- Machining of the entire model
- Machining of mouse-selected sections
- Usage of Boundary Curves as defined by Limits
- User-Defined and Selected Surface Limits

Milling Tool Types Options

- Ball Nose Tool
- Bull Nose Tool
- Flat End Tool
- Taper End Tool
- Lollipop Tool
- Slotting Cutters

Surface and Tool Orientation Options

ASM for PartMaker / Mill supports:

- 3-Axis Milling with Z-oriented Tools

ASM for PartMaker / SwissCAM supports:

- 3-Axis Milling with Z-oriented Tools (Face Type: Mill XY Plane) on either stationary or moving stock
- 3-Axis Milling with X-Oriented Tools (Face Type: Mill ZY Plane) on either stationary or moving stock
- 3-Axis Milling with Arbitrarily-Oriented Tools (Face Type: Mill 5-Axis Plane) on either stationary or moving stock
- Polar Milling with Z-oriented Tools (Face Type: Mill End Polar) on either stationary or moving stock
- Polar Milling with X-oriented Tools (Face Type: Mill Diameter Polar) on either stationary or moving stock
- Cylindrical Milling with X-oriented Tools (Face Type: Mill Diameter Polar) on either stationary or moving stock

ASM for PartMaker / Turn-Mill supports :

- 3-Axis Milling with Z-oriented Tools (Face Type: Mill XY Plane)
- 3-Axis Milling with X-Oriented Tools (Face Type: Mill ZY Plane)
- Polar Milling with Z-oriented Tools (Face Type: Mill End Polar)
- Polar Milling with X-oriented Tools (Face Type: Mill Diameter Polar)
- Cylindrical Milling with X-oriented Tools (Face Type: Mill Diameter Polar) on either stationary or moving stock

What Makes ASM Unique?

The ASM module provides PartMaker users a powerful, yet easy to use tool for programming parts with complex geometry or those requiring advanced machining strategies. PartMaker's ASM module is totally unique to many other 3D surfacing packages on the market today for the following reasons:

Developed In House, Not Licensed – Unlike other CAM vendors who license the algorithms that underlie their surfacing software, Delcam develops all of its software in house through its industry leading development team numbering over 200 strong, the largest CAM development team in the industry. By providing you with technology we develop rather than license, we are able to provide you more intelligent, better supported software which will give you the competitive edge for less cost. The surfacing machining algorithms found in ASM are the same as those in PowerMILL, Delcam's industry leading CAM system for the manufacture of complex shapes such as those found in the mold making industry.

Wide Variety of Machining Strategies and Parameters – ASM includes a wide variety of surfacing strategies and parameters. ASM gives you a solution to any complex machining challenge you may run across, assuring total control of the tool and surface finish each step along the way.

The Power You Need, the Ease of you Demand – While ASM provides you the power of the CAM industry's most powerful machining algorithms, it is also very easy to use, so you can program even the most complex jobs very quickly. ASM has a highly graphical user interface, making it quick to learn and easy to use. The software has a very easy to follow and logical work flow.

Fast Calculation Times – Using Delcam's unique surface machining technology, ASM tool paths calculate very quickly, which means you get from model to finished part faster without having to wait for your tool paths to calculate.

Make Any Tool Path High Speed – All ASM tool paths can be turned into "high speed" tool paths using technology like trochoidal milling, profile smoothing and raceline smoothing to allow you machine parts faster, which in turns lets you get your parts out the door faster and more profitably.

Powerful Undercut Machining – ASM features a unique facility for performing undercut machining automatically. This unique functionality allows you to program features other CAM systems cannot and can save you time by allowing you machine parts in one fixturing that might otherwise require multiple set-ups.

Tuned to Rotational Parts – ASM for Turn-Mill and SwissCAM has been specifically developed to support the programming of parts that are cylindrical in nature. Programming 3D surface machining on a Swiss or Turn-Mill machine can be very different than doing so on a mill. ASM includes a unique Rotary Finishing strategy for machining 3D features on parts with rotational surfaces as well as unique technology for programming 3D cylindrical milling.

Stock Model Recognition – ASM is able to know the condition of the stock at each step in the machining process. Doing so means ASM can help you spend more time cutting chips and less time cutting air.

Chapter 1: Creating a Sample Part using PartMaker/Advanced Surface Machining (ASM) for Milling

Introduction

This tutorial is designed to help you learn **PartMaker® Mill Advanced Surface Machining** (**ASM**) commands and features.

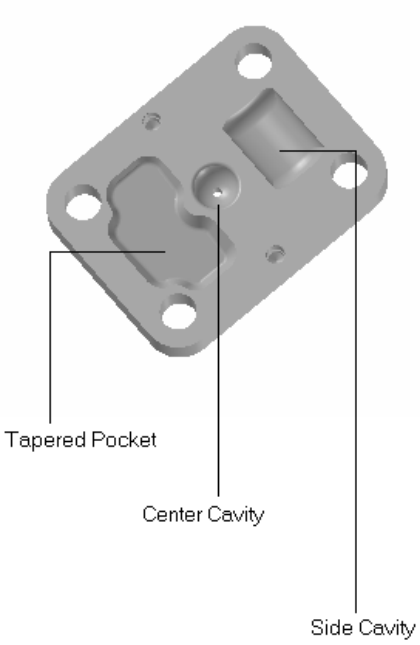
Before performing the steps in this tutorial, it is recommend you have a familiarity with PartMaker Mill and have completed the tutorial in Chapter 3 of the **PartMaker Mill User's Guide**.

Note: *You must install software from PartMaker® versions 2013 R1 or higher to have all support files and control functions referenced here.*

How you will Create the Sample Part

Here are the major steps you'll follow to create this part:

- Start PartMaker® Mill
- Import a Solid Model
- Specify part boundaries
- Set Defaults
- Open Tools, Cycles and Material files
- Define holes for a boring cycle
- Define Advanced Surface Machining Strategy for Tapered Pocket
- Define Advanced Surface Machining Strategy for Side Cavity
- Define a drilled hole in the Center Cavity
- Define Advanced Surface Machining Strategy for Center Cavity
- Define Profiles for Contour Milling
- Select a post processor
- Generate a process table containing machining data
- Simulate the cutting process
- Generate an NC program for the part



Start PartMaker® Mill

1 Click the PartMaker® Mill icon on the desktop.

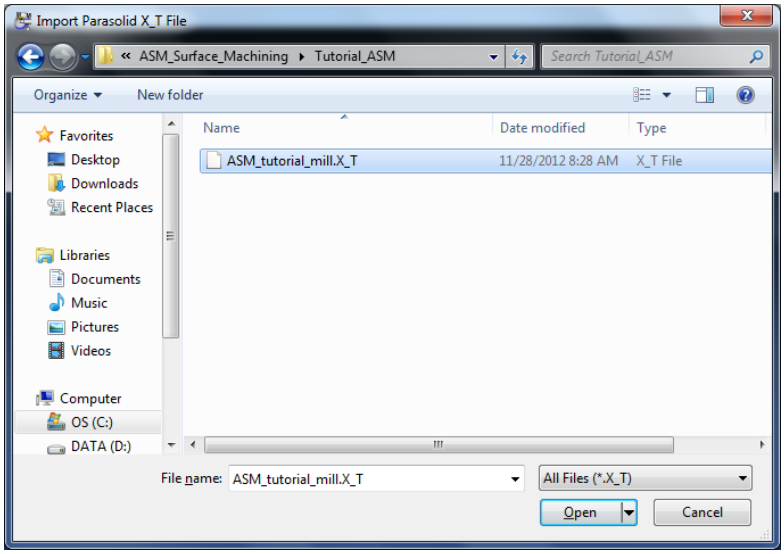
2 When the **Setup** dialog appears click the **<Close>** button. (The automatic display of the **Setup** dialog is controlled by **Preferences**).

Import a Solid Model

1 From the **File** menu choose the **Import** submenu.

2 Choose the **X_T Parasolid Transmit Text File...** command. Open the X_T Parasolid part file in directory path shown here and as shown in the dialog below:

C:\PartMaker_2014\pm-mill\ASM_Surface_Machining\Tutorial_ASM



3 Select the file **ASM_tutorial_mill.X_T** and click on the **<Open>** button to load the solid model into PartMaker.

4 Maximize the solids window by clicking the **Maximize Solids Window** button in upper right hand corner of the screen.

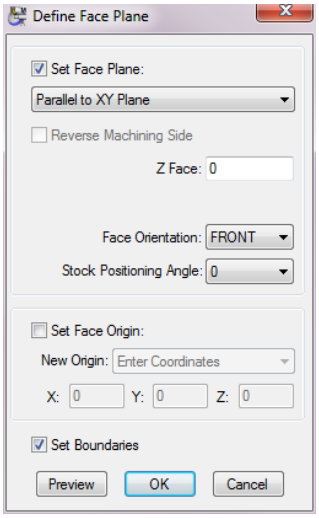
In this tutorial, you will only be working directly on the solid model and not in the 2D window at all.

Specifying the Part Boundaries

When importing a solid model into PartMaker, you can use the model to automatically set the part boundaries. You can do so as follows:

1 Click the **Define Face Plane** button from Solids Window Toolbar.

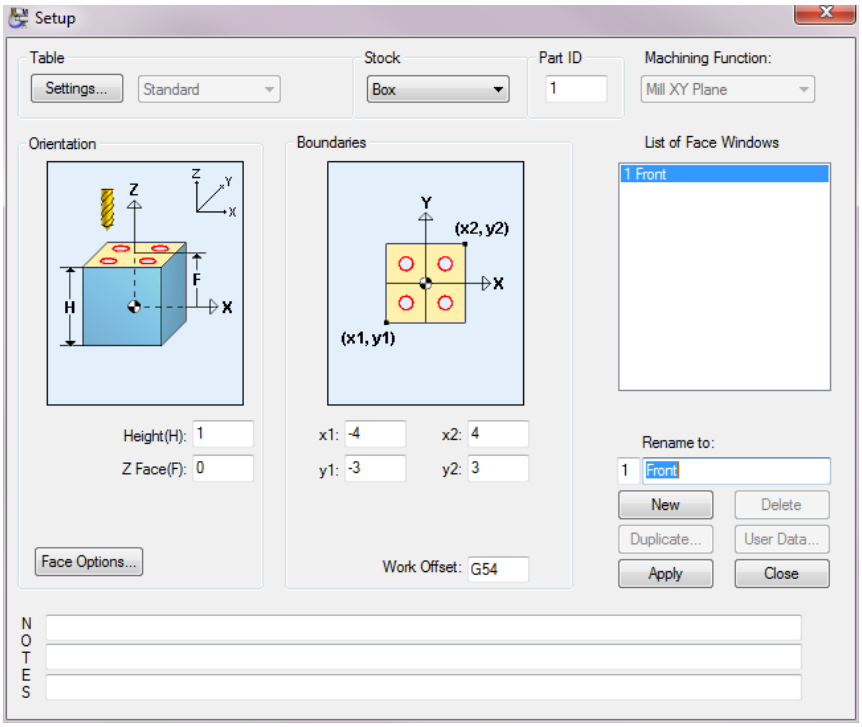
2 Check the **Set Boundaries** box as shown below:



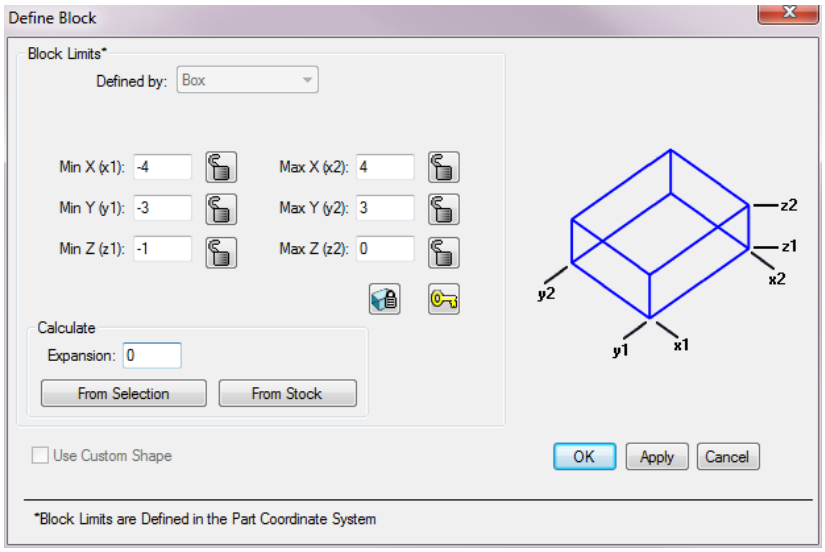
3 Click the **<OK>** button

Doing so will have the dual purpose of both setting the part boundaries in the **Setup** dialog and the block limits in the **Define Block** dialog which can be accessed from the **Surface Group Parameters** dialog for each surfacing part feature. The **Define Block** dialog is important because it is used to specify the maximum and minimum limits within which machining can occur for a given ASM group.

The Setup dialog in shown below:



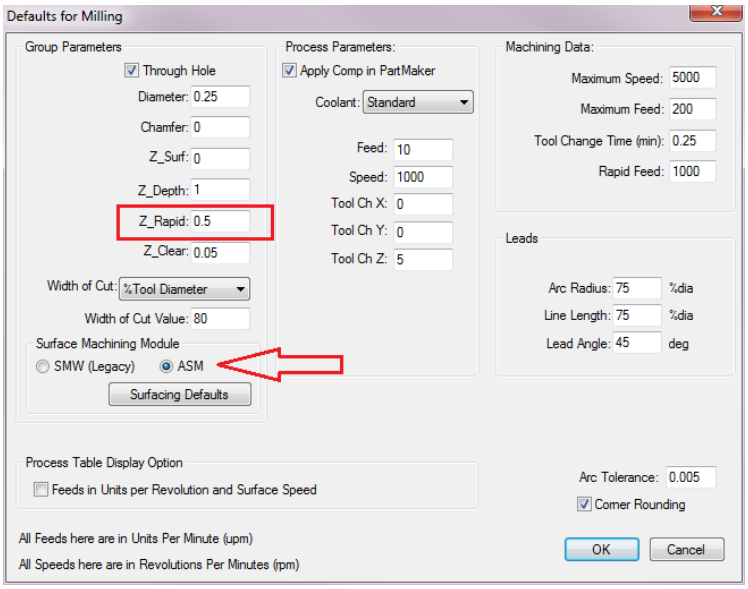
The Define Block dialog is also shown here:



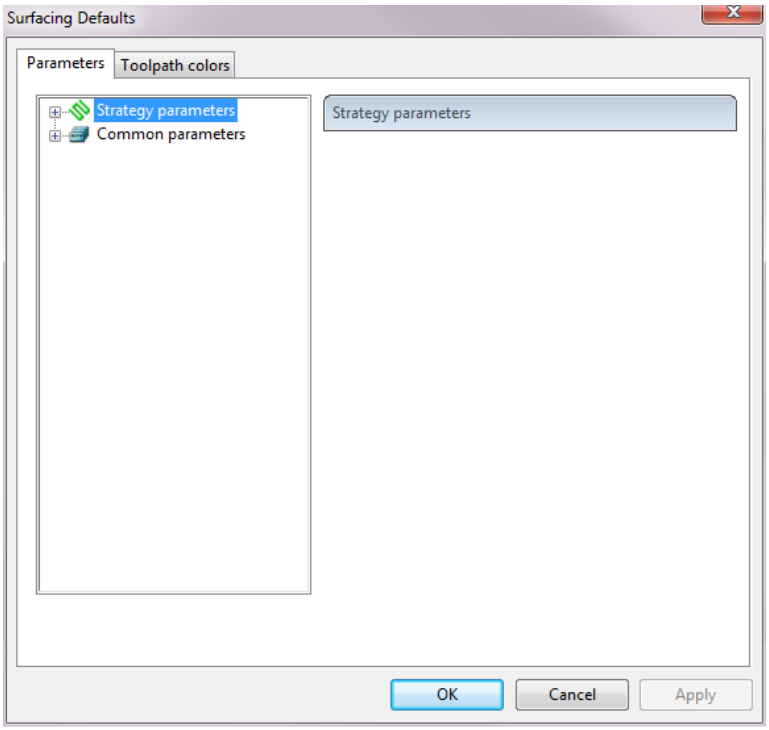
Set Defaults

1. From the **Job Optimizer** menu choose the **Defaults** command
2. Enter 0.5 (12.7 mm) in the **Z_rapid** field as shown below

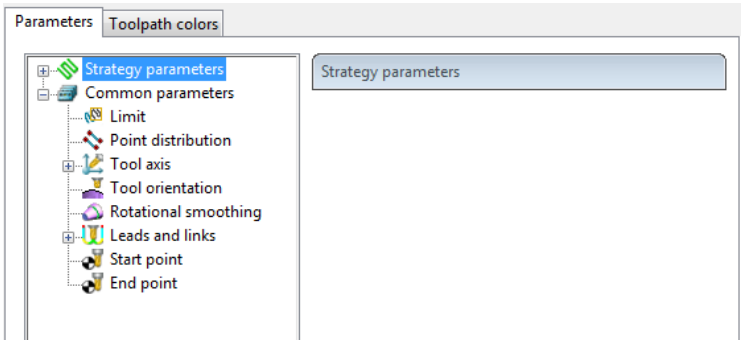
   Using such a large number for your Rapid Plane may not be required for machining this part but it will help you see how the tool path is generated in the reference to the Rapid Plane.



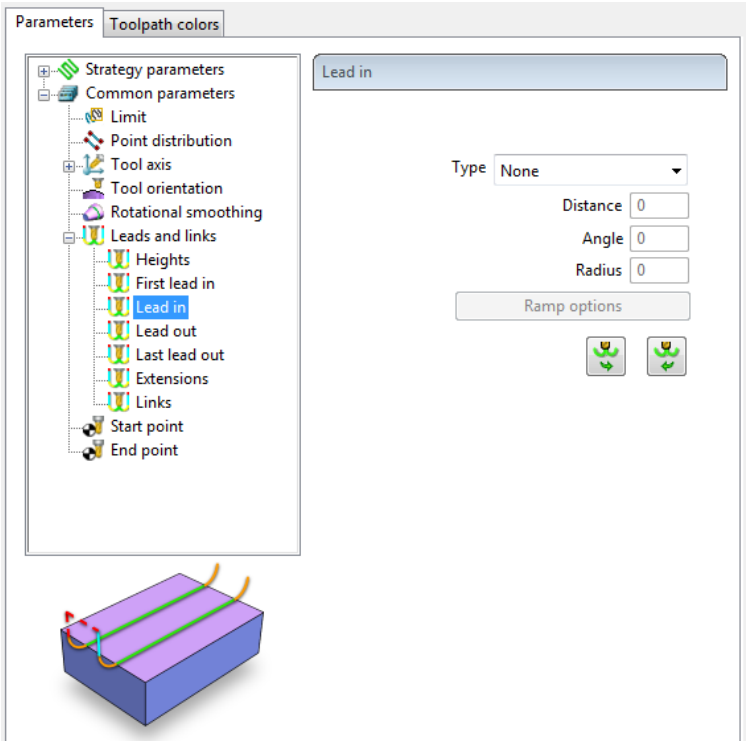
3. Under **Surface Machining Module**, make sure the **ASM** button is selected. Click the **<Surfacing Defaults>** button in the **Defaults for Milling** dialog. The Surfacing Defaults dialog will be displayed as shown below.



4 Expand the **Common parameters** item by clicking on the **plus** icon. The contents of the Common parameters should look as shown below.



5 Expand the **Leads and links** item under **Common Parameters** and select **Lead in** as shown below.



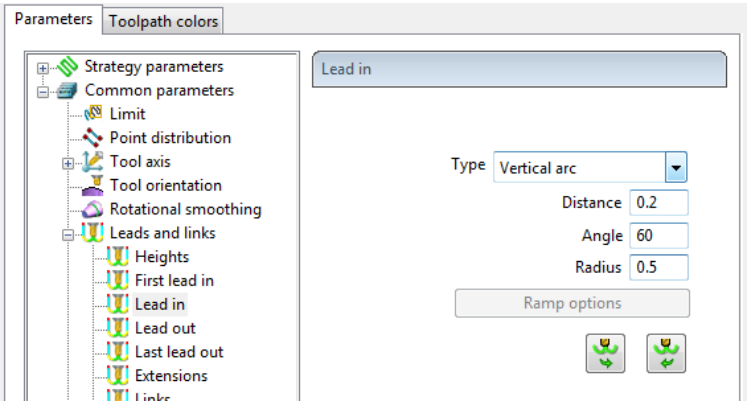
6 On the right side pane, choose **Vertical arc** from **Type** drop down menu. This should activate Distance, Angle, and Radius edit fields.

7 Set the value for **Distance** as 0.2 (5.08).

8 Set the **Angle** to 60 degrees.

9 Enter 0.5 (12.7) for **Radius**.

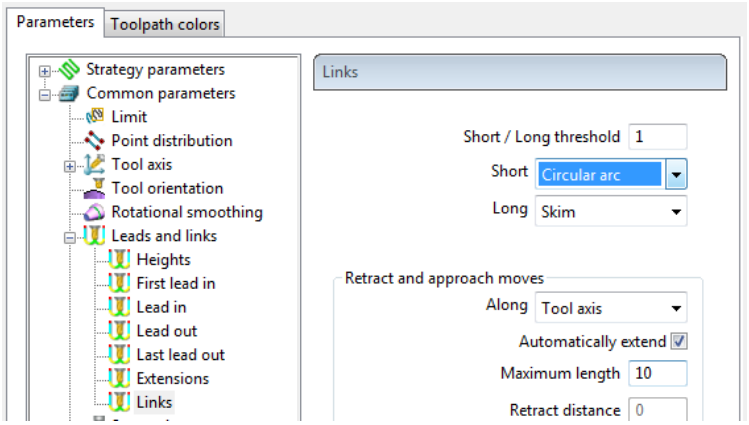
Your dialog box will display as shown below:



10 Copy the **Lead in** values to **Lead out** by clicking on the button as shown below.

11 In the Common parameters tree, click on **Links** under **Leads and links**.

12 Set the **Short** link type to **Circular arc** as shown below.



13 Click **<OK>** to close the Surfacing Defaults dialog and return to Defaults for Milling dialog.

14 Click <**OK**> to close Defaults **for Milling** dialog.

Open Tools, Cycles and Material Files

From **File** menu choose **Open Tools File** and open

C:\PartMaker_2014\pm-mill\ASM_Surface_Machining\Tutorial_ASM \ASM_tutorial_mill.tdb

From **File** menu choose **Open Cycles File** and open

C:\PartMaker_2014\pm-mill\ASM_Surface_Machining\Tutorial_ASM \ASM_tutorial_mill.cdb

From **File** menu choose **Open Material File** and open

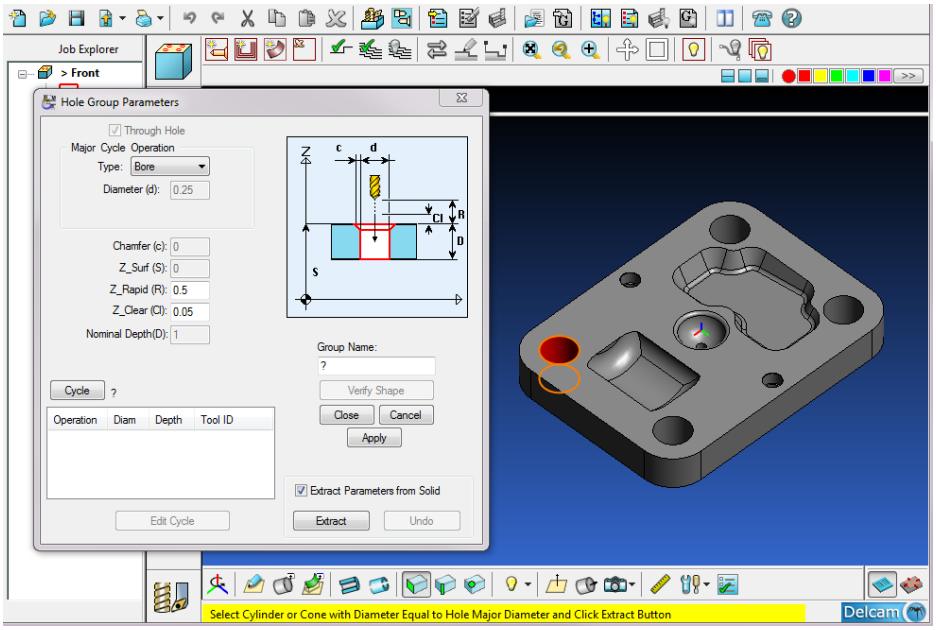
C:\PartMaker\pm-mill\Material\Alu_allw.mdb

Define Holes for the Boring Cycle

The first group of holes utilizes a boring cycle. In the **Hole Group Parameters** dialog, you must specify a major diameter for the holes and select a cycle type. PartMaker searches the Cycles Database for a cycle that matches the parameters you have specified in the **Hole Group Parameters** dialog. To create the bored holes:

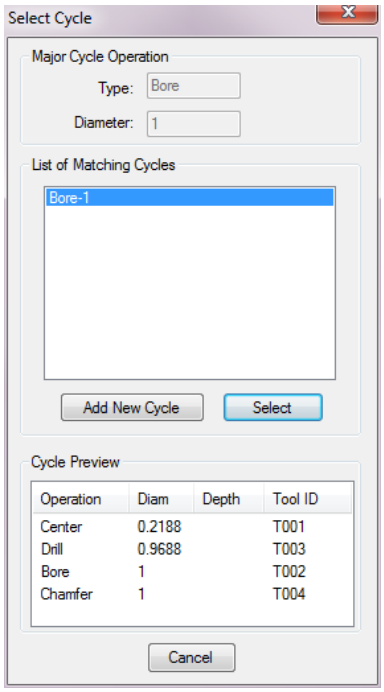
1. Click anywhere in the **Face Window** to make it active.

2. Choose New Hole Group from the Part Features menu to display the Hole Group Parameters dialog.
3. Here you will be making a through hole, so make sure to leave the **Through Hole** box checked.
4. Choose **Bore** from the **Type** drop down menu.
5. Click the Extract Parameters from Solid box.
6. Click on the bored hole in the upper left hand corner of the part as shown below. It will be highlighted in red:

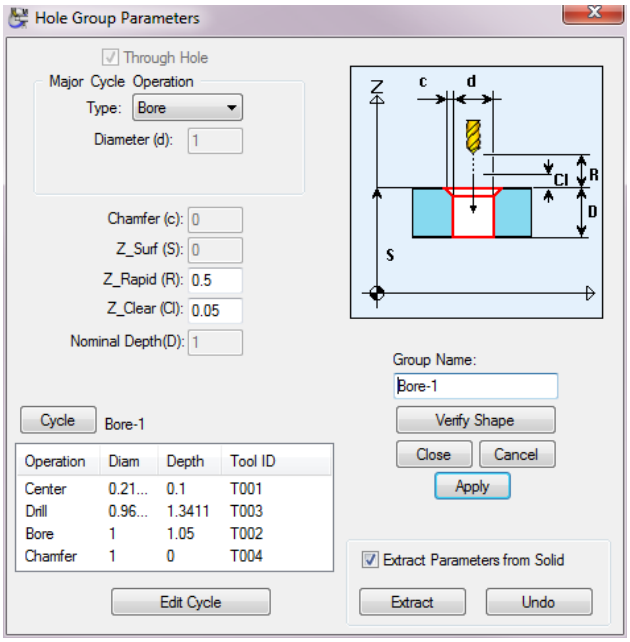


7. Click the **Extract** button to automatically extract all the geometric information about the hole into the **Hole Group Parameters** dialog.

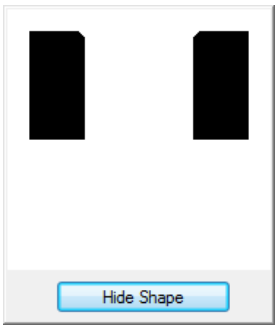
Click **<Select>** to select the Bore-1 cycle



8 **PartMaker** finds the appropriate cycle in the database, Bore-1. Calculated depths for each tool are displayed automatically at the bottom of the dialog when the appropriate cycle is found. The dialog should appear as shown below:

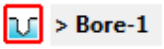


9 To verify the hole shape, click the **<Verify Shape>** button; click the **<Hide Shape>** button under the picture to hide the hole shape and return to the **Hole Group Parameters** dialog.



10 Click **<Close>** to return to the Face Window.

An icon displaying the cycle name BORE-1 appears in the **Job Explorer** window as shown below. The icon is a *group symbol.* Each part you create in **PartMaker** will display one or more group symbol icons in the **Face Window** to indicate the various machining cycles for a part.



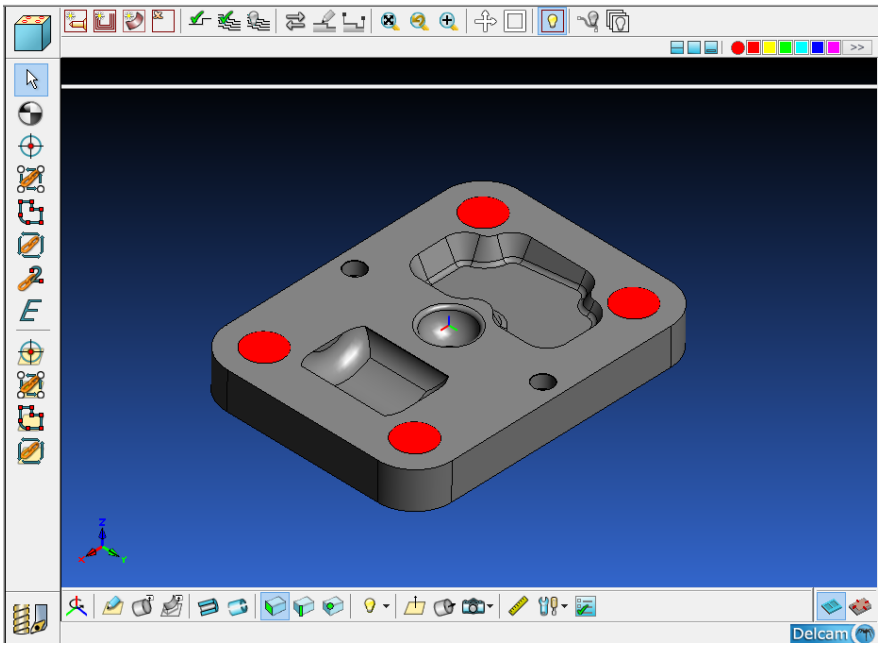
Icon for Group 1 holes

Important! *The part feature creation selections (i.e.* ***New Hole Group, New Profile Group*** *and* ***New Surface Group*** *functions) found under the* ***Part Features*** *menu can also be accessed by clicking your right mouse button in the CAM Face Window.*

Selecting the Bored Holes

Now that the Hole feature is created, you can assign it to the part by clicking directly on the solid model. You can do so by:

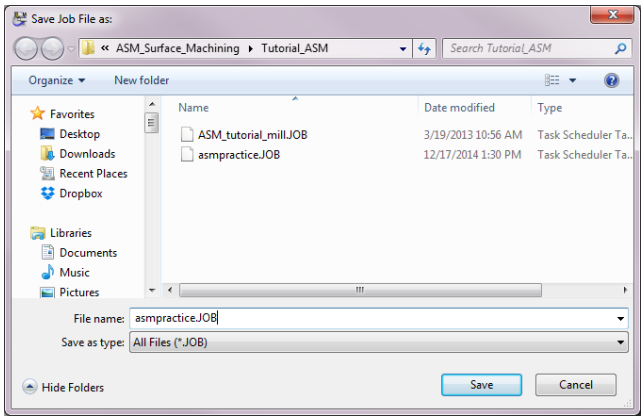
1. Select the Chain Holes on Solid Model icon
2. Click on any of the four bored holes on the solid model. Your **Face Window** will appear as shown below.



Save Your Work

It is always a good idea to save your work at various points in the part creation process in PartMaker. To do so:

1. From the **File** menu choose **Save** or press **<CTRL + S>** on your keyboard and enter the name of your job. Here, save your job file as **asmpractice** as shown below:

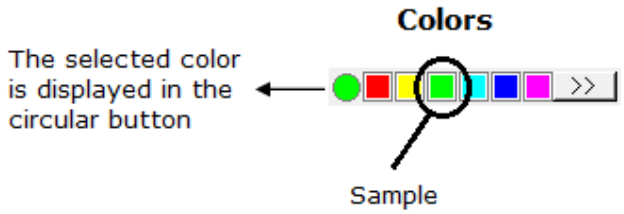


2. Click the **<Save>** button.

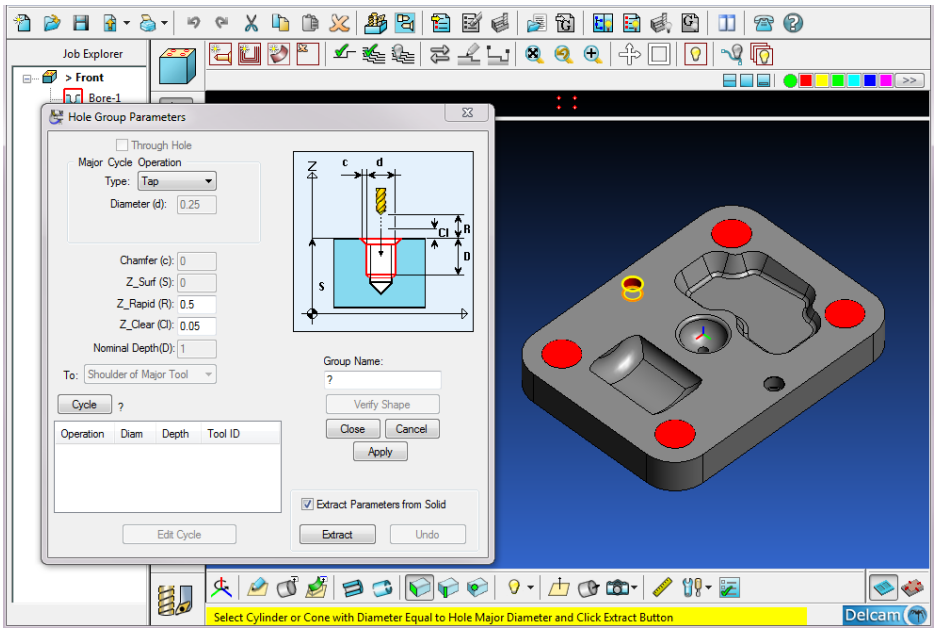
Define Holes for the Tapping Cycle

In this section, you will specify the holes for Group 2, the tapping cycle. To do so:

1. Choose a **New Color**. Click a color square in the color bar at the top of the **Face Window** different from the Sample Color.

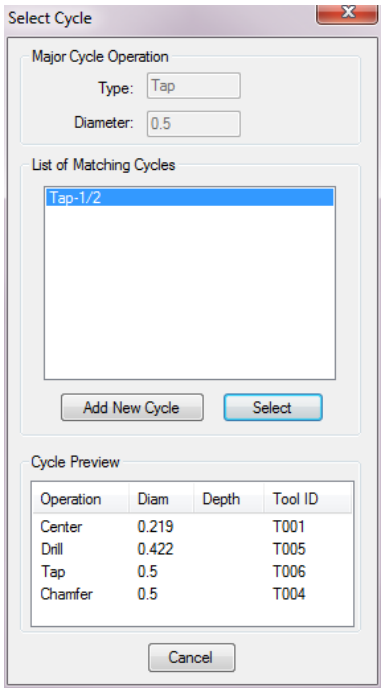


2. Choose **New Hole Group** from the **Part Features** menu to display the **Hole Group Parameters** dialog.
3. Uncheck the **Through Hole** check box.
4. Choose **Tap** from the **Type** drop down menu.
5. Click the Extract Parameters from Solid box.
6. Click on the tapped hole in the upper half of the part as shown below. It will be highlighted in red:

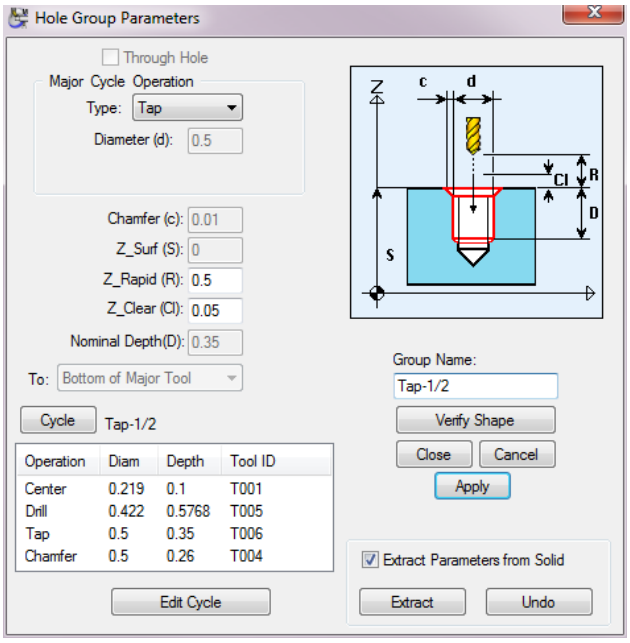


7. Click the **Extract** button to automatically extract all the geometric information about the hole into the **Hole Group Parameters** dialog.

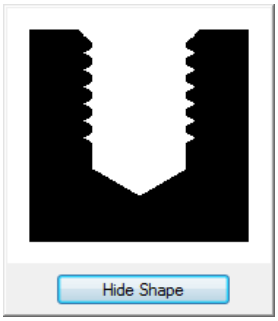
8 Click **<Select>** to select the Tap-1/2 cycle



9 **PartMaker** finds the appropriate cycle in the database, Tap-1/2. Calculated depths for each tool are displayed automatically at the bottom of the dialog when the appropriate cycle is found. Your completed Hole Group Parameters dialog should appear as shown below:



10 To verify the shape of the hole, click the **<Verify Shape>** button; click the **<Hide Shape>** button under the picture to return to the dialog.



11 Click the **<Close>** button to exit the dialog and return to the **Face Window**.

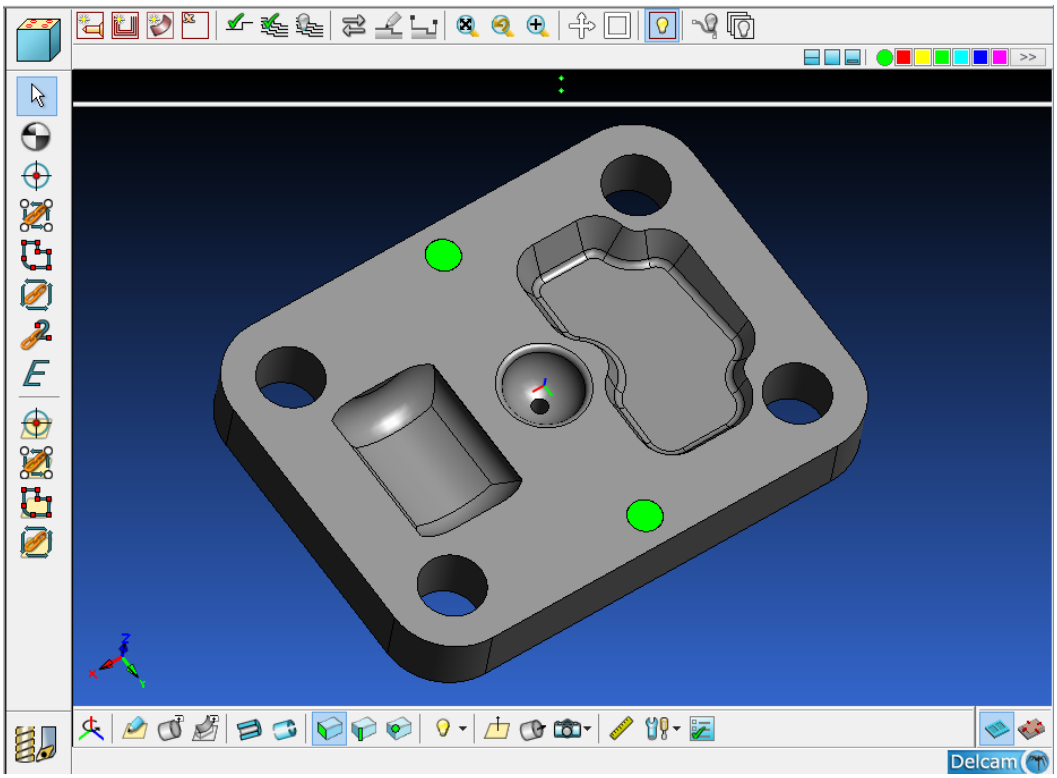
A group symbol for Group 2 displays in the upper-left corner of the **Face Window** under the Tap-1/2 group icon. The cycle name Tap-1/2 appears to the right of the group symbol.

Selecting the Tapped Holes

Now that the Hole feature is created, you can assign it to the part by clicking directly on the solid model. You can do so by:

1 Select the Chain Holes on Solid Model icon

2 Click on either of the two tapped holes on the solid model. Your **Face Window** will appear as shown below.



Define Machining Strategies for the Tapered Pocket

In this section of the tutorial you will create, bound and verify a tool path for roughing the tapered pocket on the imported solid model.

Roughing the Tapered Pocket Using the Model Area Clearance Strategy

1 Choose a **New Color**: 

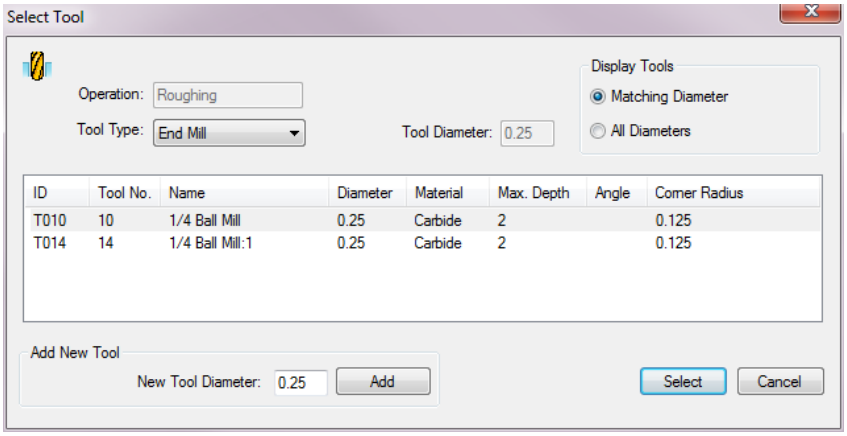
2 Choose **New Surface Group** from the **Face Window** tool bar to display the **Surface Group Parameters** dialog.

3 Make sure that there is a checkmark next to **Roughing** in the **Operations** column.

4 Enter a tool diameter of 0.25 (6.35)

5 Click the **<Select Tools>** to view the **Select Tool** dialog.

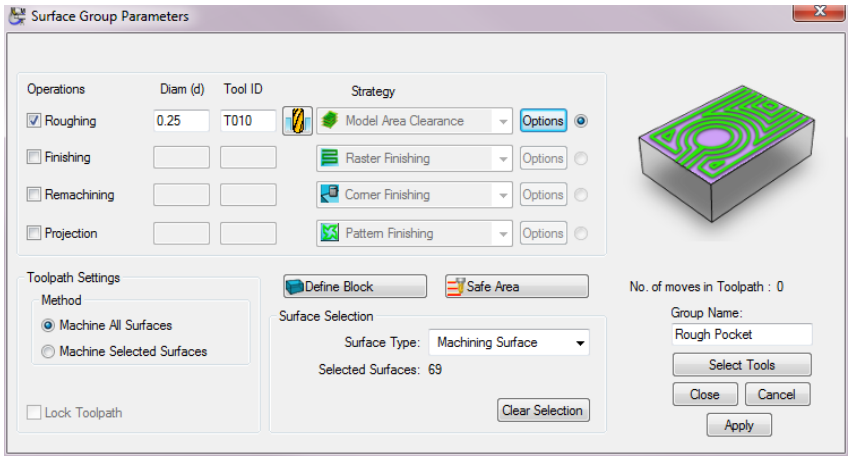
6 Choose the tool T010 1/4Ball Mill and click on <**Select**> to return to **Surface Group Parameters** dialog.



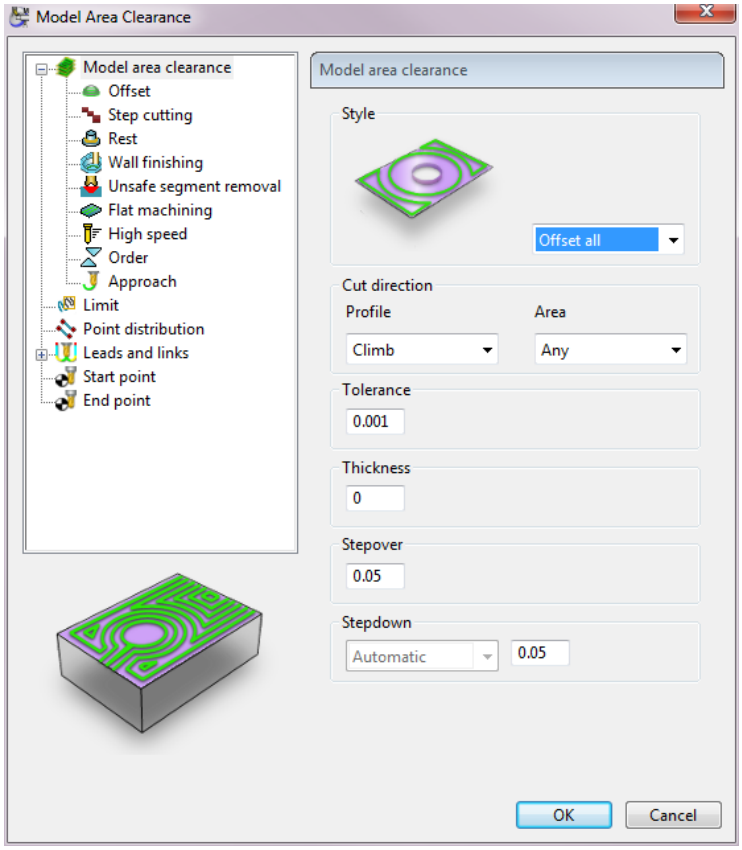
7 The **Strategy** drop down menu is disabled and **Model Area Clearance** is selected by default.

***Note**: The **Strategy** drop down menu is active only for Finishing and Roughing operations that provide multiple strategies.*

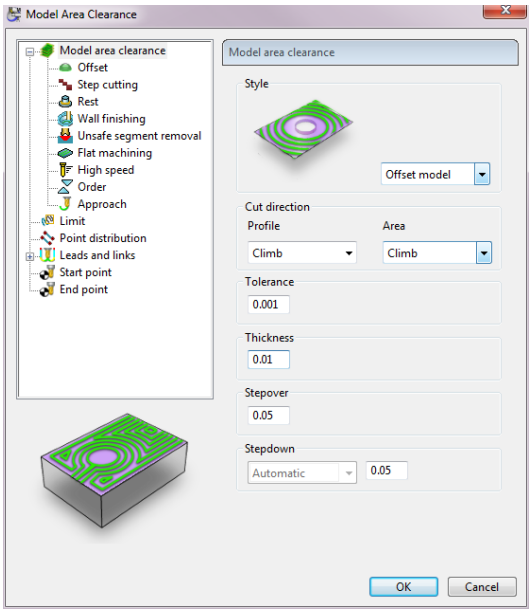
8 In the **Group Name** field enter "**Rough Pocket**" and press the **<Enter>** key on your keyboard or click the **<Apply>** button.



9 Click the **<Options>** button to display **Model Area Clearance** dialog as shown below.



10 From the drop down menu, set the toolpath **Style** to **Offset model** and set Cut direction to **Climb** for both **Profile** and **Area** as shown below. Change the value of **Thickness** to 0.01 (0.254) for finishing.



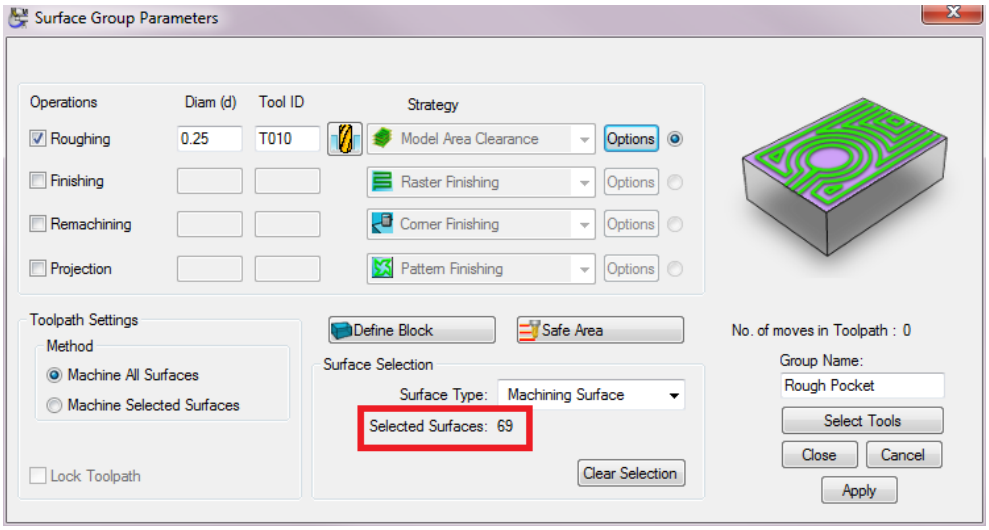
11 Click <**OK**> to return to the **Surface Group Parameters** dialog.

Select Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**.

Click the **Machine All Surfaces** radio button. As a result you will have selected every surface of the part to be machined. Here we will leave all the surfaces selected and just bound machining using a “Boundary Curve”

Your **Surface Group Parameters** dialog should appear as shown below:



Notice the number of selected surfaces is 69.

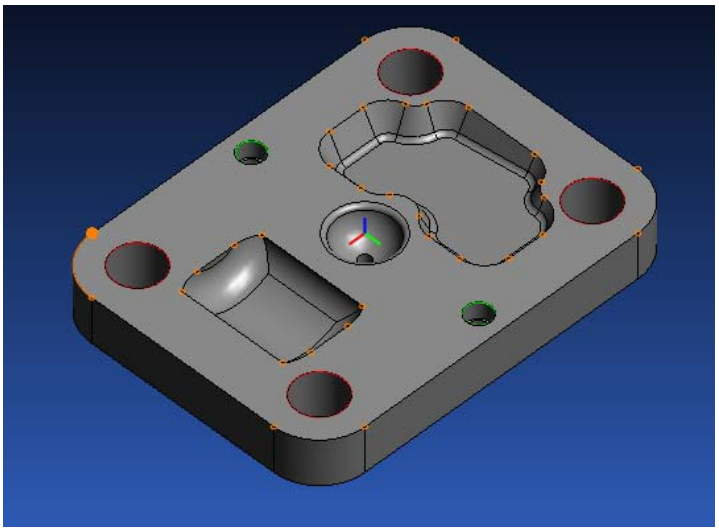
Click on the <**Close**> button in **Surface Group Parameters** dialog to return to the Face Window.

Create a Boundary Curve

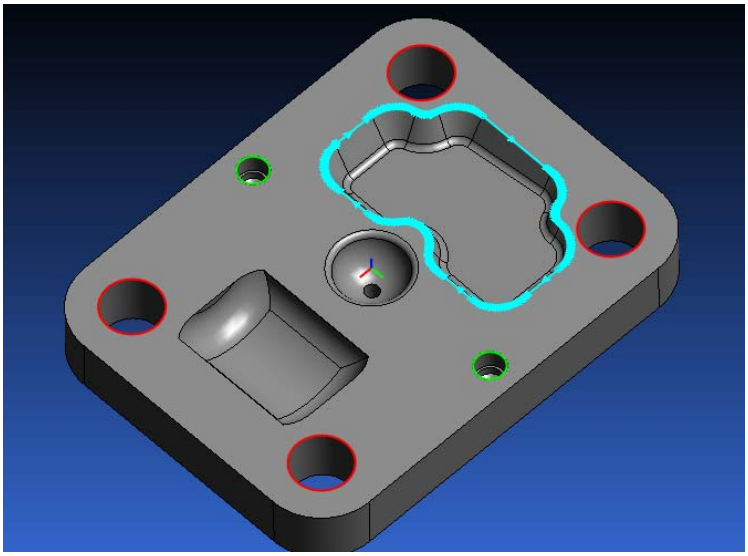
Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict the machining area. To create a boundary curve to bound rough machining of the Tapered Pocket:

1 Choose the Chain Geometry on Solid Model icon.

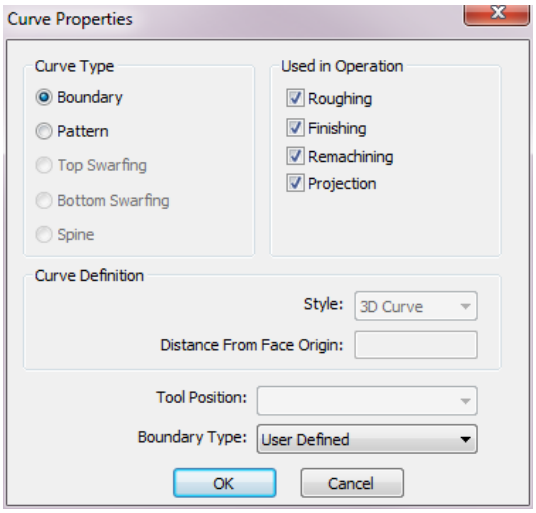
2 Notice the edge end points are highlighted by orange circles as shown below. You can select any edge end point on the tapered pocket.



3 Double click on any of the orange points of the tapered pocket. Your model should now appear as shown below:



4 Double click on the **Boundary Curve** you have just created with the **Selection** icon to display the **Curve Properties** dialog as shown below.



5 In the **Curve Properties** dialog, notice the Curve Type is set as **Boundary** and the Boundary Type is set to **User Defined.**

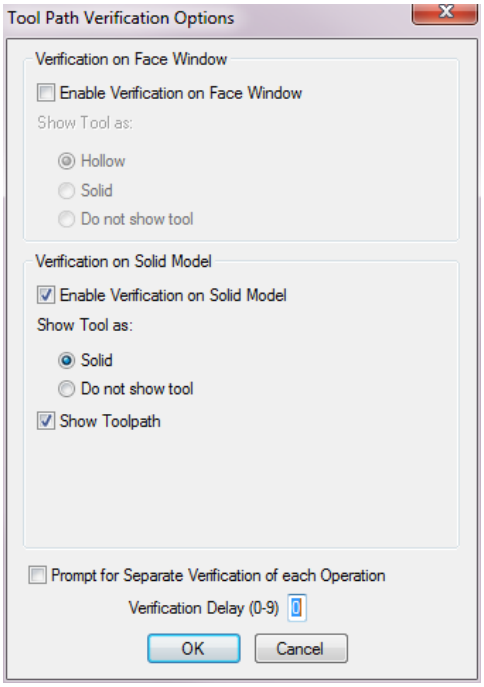
6 Click **<OK>** to close the Curve Properties dialog.

Verify the Tool Path

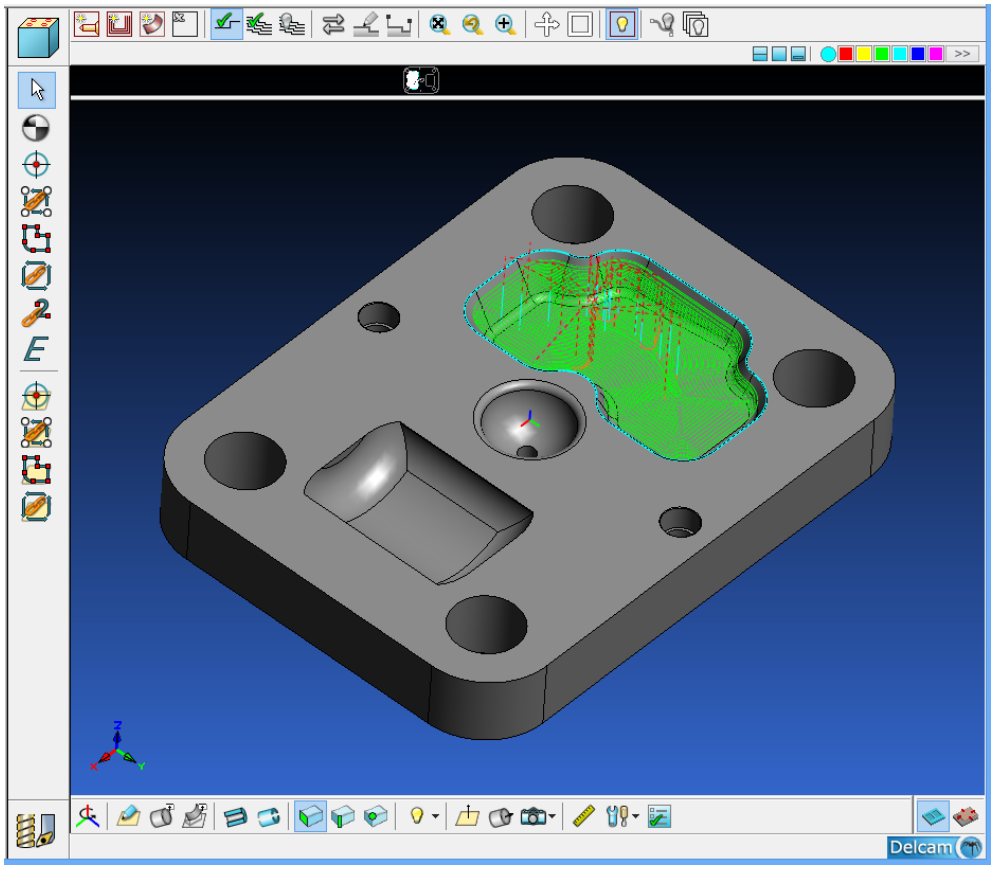
Once you have created the **Part Feature** and established a **Boundary Curve** you can visually verify the tool path PartMaker has calculated. To do so:

1. From the Face Window tool bar choose **Verify Work Group Tool path**.
2. Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.
3. Enter a Verification Delay of 0.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see a 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:

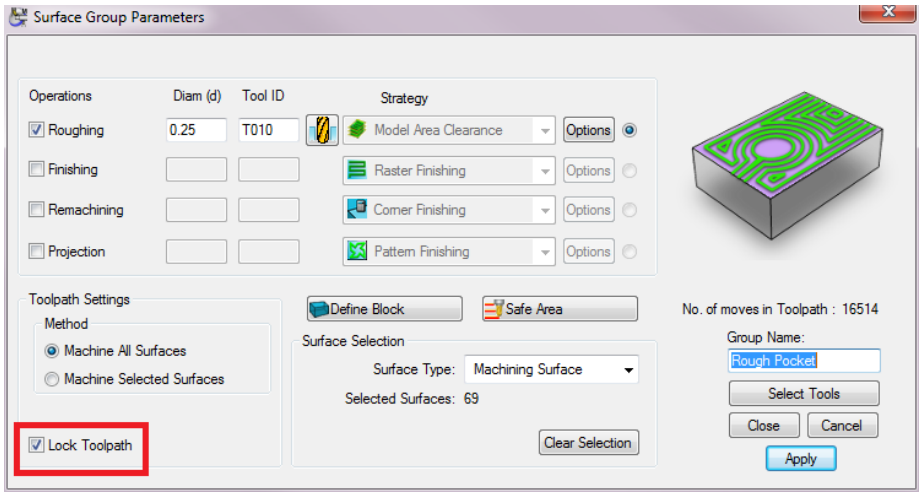


5 To hide toolpath, from the **Face Window** tool bar click on **Hide Every Tool path**.

Lock the Tool Path

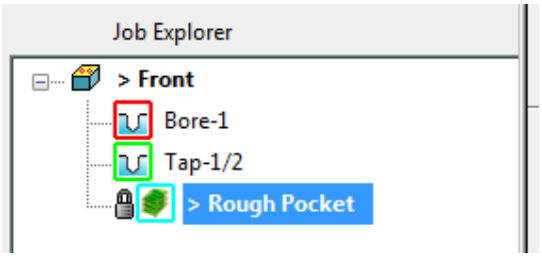
Whenever creating a surfacing tool path in ASM, it is always a good idea to "lock" the tool path once you are satisfied with result to avoid it being regenerated later. To "lock" a tool path

1. Double click on the feature in the **Job Explorer** tree
2. Click the **Lock Toolpath** button in the **Surface Group Parameters** dialog as shown here:



Once a tool path has been "locked" it will appear with a small lock next to it on the Feature Tree.

Your Job Explorer tree should now appear as shown below:



3. Click the **<Close>** button to close the **Surface Group Parameters** dialog.

Finishing the Tapered Pocket using Steep and Shallow Finishing Strategy

In this section of the tutorial you will create, bound and verify the tool path for finishing the steep and shallow areas of the tapered pocket on the imported solid model.

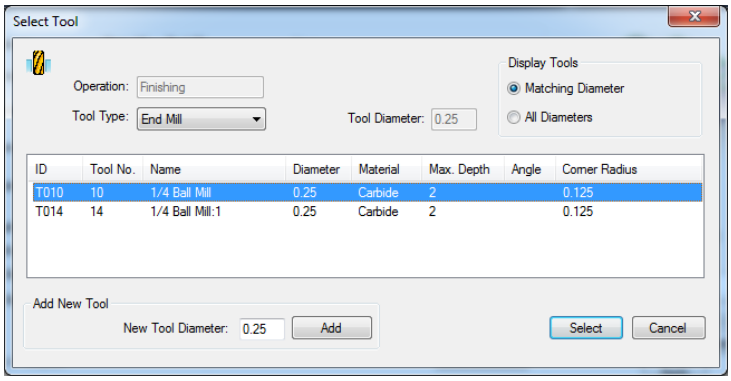
1 Choose a **New Color**.

2 Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.

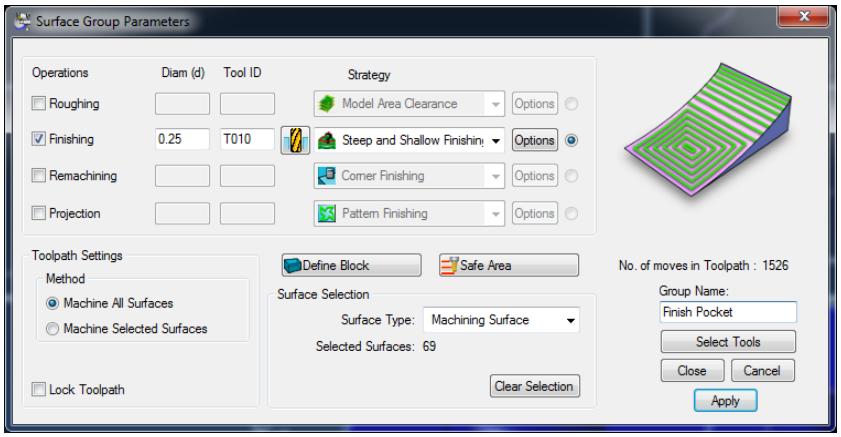
3 Uncheck the **Roughing** box and make sure **Finishing** is checked.

4 Tool diameter should be already set to 0.25 (6.35).

5 Click the <**Select Tools>** button and make sure Tool Type is set to **End Mill**. Click <**Select**> to choose the tool T010 1/4 Ball Mill and return to **Surface Group Parameters** dialog.



6 From the **Strategy** drop down menu choose **Steep and Shallow Finishing** as shown below.



7 Enter “Finish Pocket” in the **Group Name** field. Press the **<Enter>** key or click the **<Apply>** button.

8 Click the **<Options>** button next to **Strategy** drop down menu to display the **Steep and shallow finishing** dialog.

9 Set the Order using the drop down menu to Steep first with zero Additional stock. The Order determines the machining order of shallow and steep areas in the pocket.

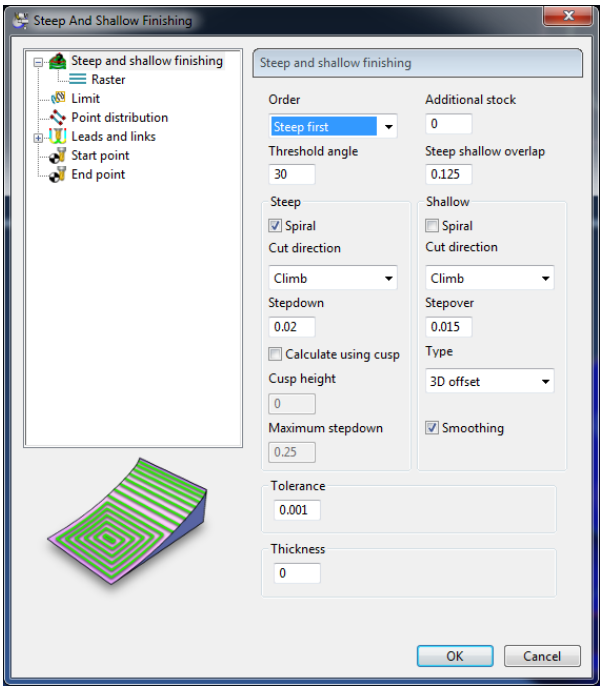
10 Set the **Threshold angle** to 30 degrees and the **Steep and shallow overlap** to 0.125 (3.175).

11 Set the value of **Stepdown** under **Steep** group to 0.02 (0.508).

12 Set the value of **Stepover** under **Shallow** group to 0.015 (0.381).

13 Check **Spiral** toolpath for the **Steep** group as shown below.

14 Under **Shallow** group, choose 3D offset for **Type** and check **Smoothing**.



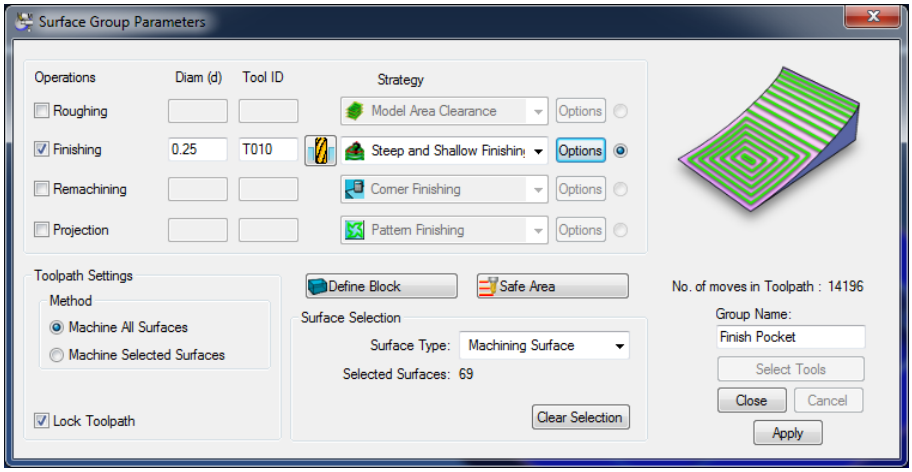
15 Click **<OK>** to return to the **Surface Group Parameters** dialog

Select Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**.

Make sure the **Machine All Surfaces** radio button is checked. As a result you will have selected every surface of the part to be machined.

Your dialog should appear as below, where the number of **Selected Surfaces** is 69.



Click on the <**Close**> button in **Surface Group Parameters** dialog to return to the Face Window.

Create a Boundary Curve

Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict machining area.

1 From the **Face Window** tool bar choose **Show Holes and Profiles for Workgroup Only** to enter the mode for showing the boundary curves in just the selected group.

2 Choose the Chain Geometry on Solid Model icon

3 Double click on any of the orange points of the tapered pocket as you did in the previous section.

4 From the **Face Window** tool bar click back on the **Show Holes and Profiles for Workgroup Only** button to return to mode where you can see the tool path profiles for all of your groups.

Verify the Tool Path

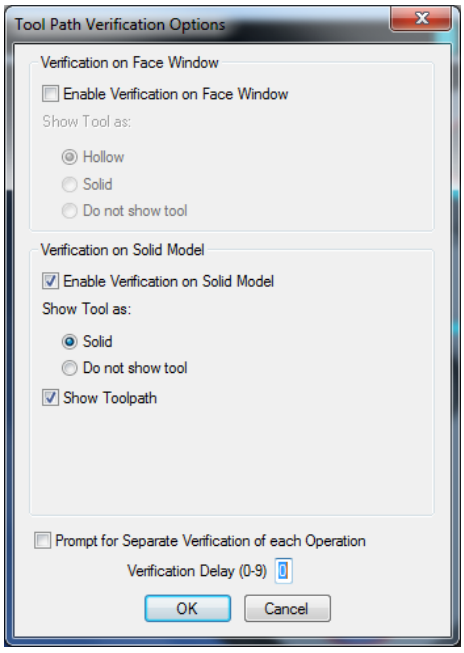
Once you have created the **Part Feature** and established a **Boundary Curve** you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Face Window tool bar choose Verify Work Group Tool path.

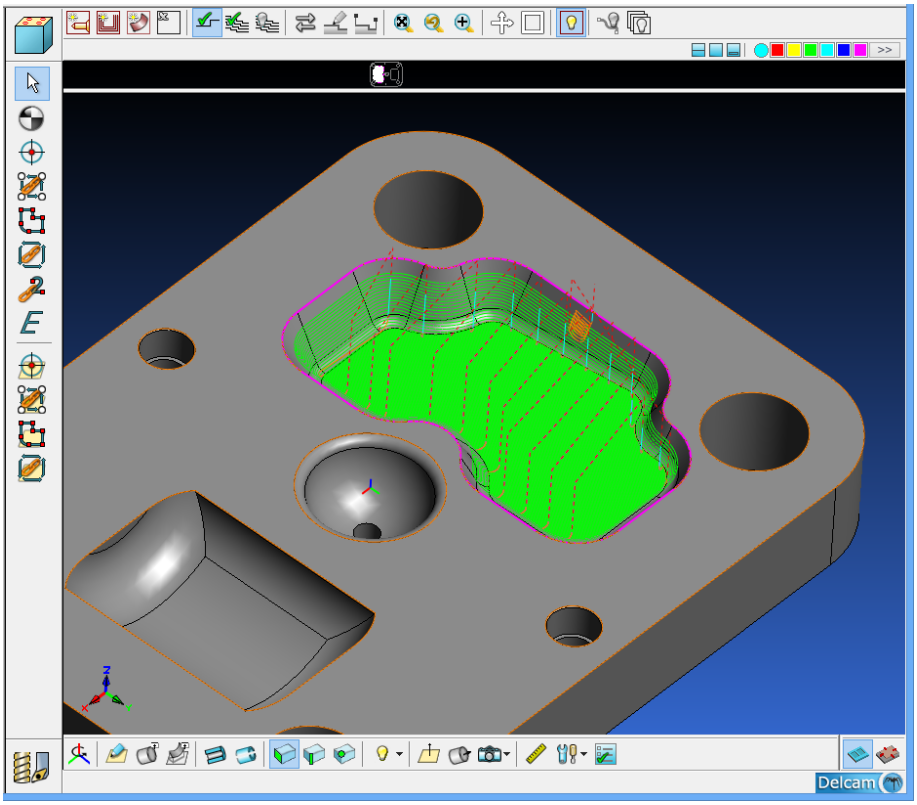
2 Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.

3 Enter a Verification Delay of 0.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see a 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



5 To hide toolpath, from the **Face Window** tool bar click on **Hide Every Tool path**.

Lock the Tool Path

As you did in the previous step, once you are satisfied with the verified tool path, you should now lock the tool path. To do so:

1 Double click on the feature in the **Job Explorer** tree

2 Click the Lock Toolpath check box in Surface Group Parameters dialog

3 Click the **<Close>** button to close the Surface Group Parameters dialog

Define Surface Machining Strategies for the Side Cavity

In this section of the tutorial you will create, bound and verify a toolpath for roughing, finishing and remachining of the side cavity with the rounded bottom. In this section, roughing, finishing and remachining will be done using only one **Part Feature**.

1 Choose a **New Color**.

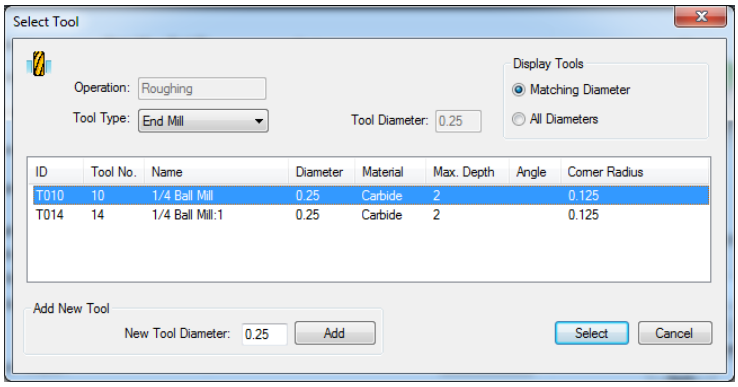
2 Choose **New Surface Group** from the **Face Window** tool bar to display the **Surface Group Parameters** dialog.

3 Check the **Finishing** and **Remachining** radio button. Make sure that **Roughing** is still checked.

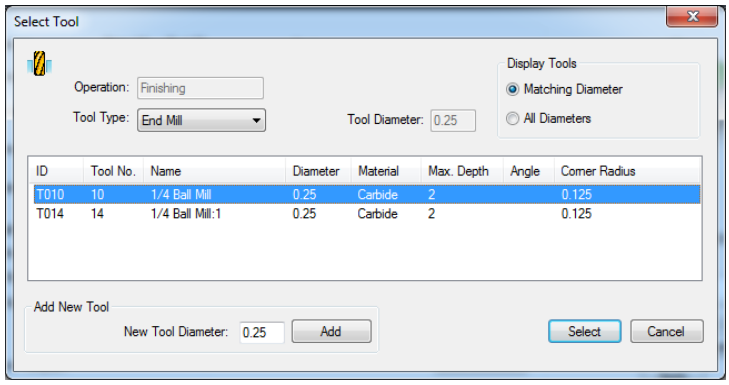
4 Enter a tool diameter of .25 (6.35) for **Roughing**, **Finishing,** and **Remachining** operations**.**

5 Click the **<Select Tools>** button.

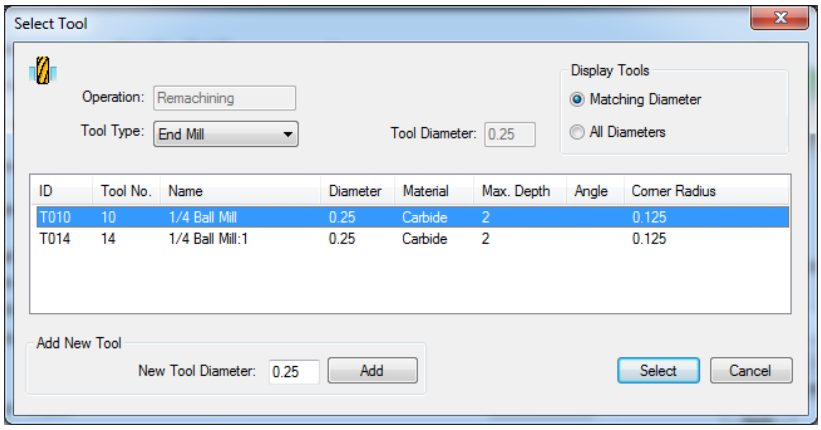
Select T010 1/4 Ball Mill for **Roughing** operation by clicking <**Select>** button as shown below.



Upon clicking <**Select**> button, the dialog box closes and reopens for **Finishing** operation as shown below. Select the tool T010 1/4 Ball Mill for **Finishing** and click <**Select>** button.



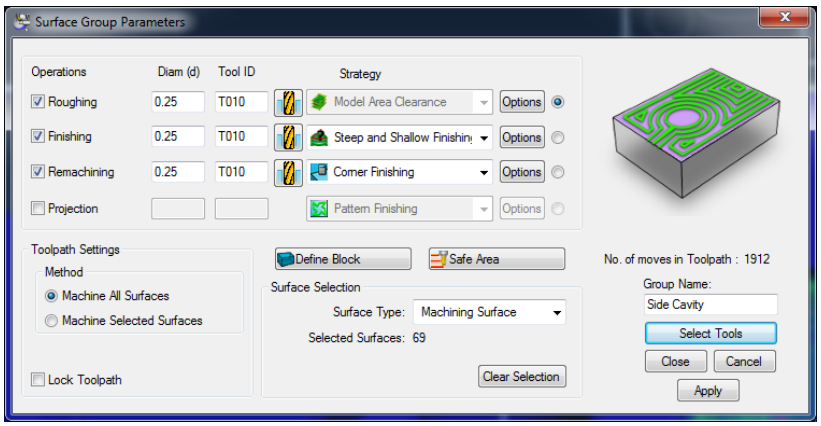
The **Select Tool** dialog closes and reopens for the **Remachining** operation, click on T010 1/4 Ball Mill tool and click <**Select**> button as shown below. You will return to Surface Group Parameters dialog.



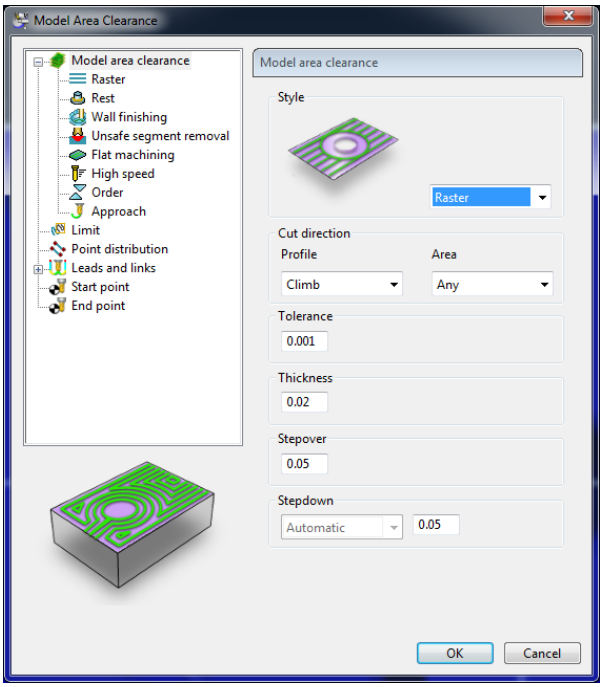
6 From the Strategy drop down menu for Finishing choose Steep and Shallow Finishing.

7 From the Strategy drop down menu for Remachining choose Corner Finishing.

8 In the Cycle Name field enter "**Side Cavity**". Press the **<Enter>** key or click the **<Apply>** button.



9 Click the **<Options>** button next to the **Model Area Clearance** strategy to display the **Surface Machining Options** dialog as shown below.



10 Change the **Style** to **Raster** and set the **Thickness** value to 0.02 (0.508).

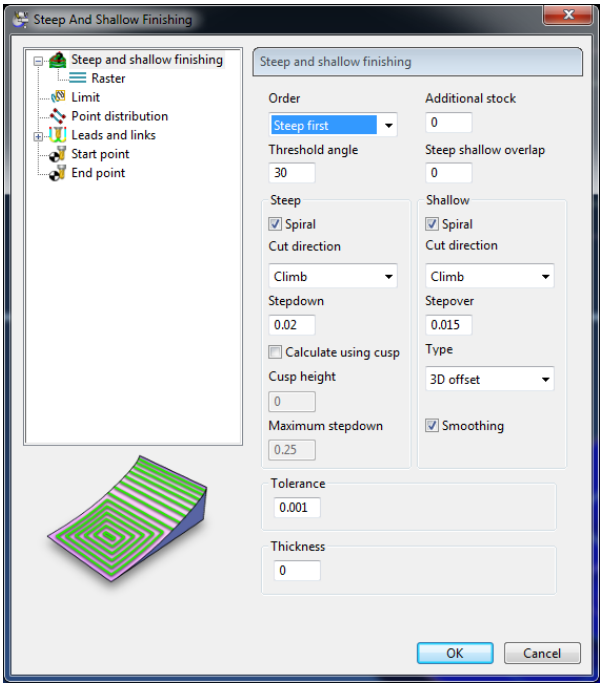
11 Click <OK> to return to the **Surface Group Parameters** dialog.

12 Click the <Options> button next to Steep and Shallow Finishing strategy to display the Surface Machining Options dialog as shown below.

13 Set the value of Stepdown under Steep group to 0.02 (0.508).

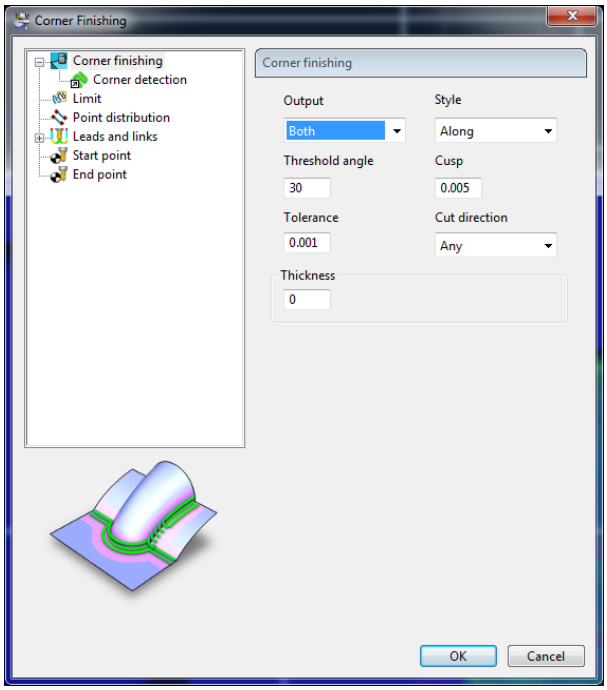
14 Set the value of Stepover under Shallow group to 0.015 (0.381).

15 Under Steep group, check Spiral option. For the Shallow group, check Spiral and Smoothing options.



16 Click **<OK>** to return to the **Surface Group Parameters** dialog.

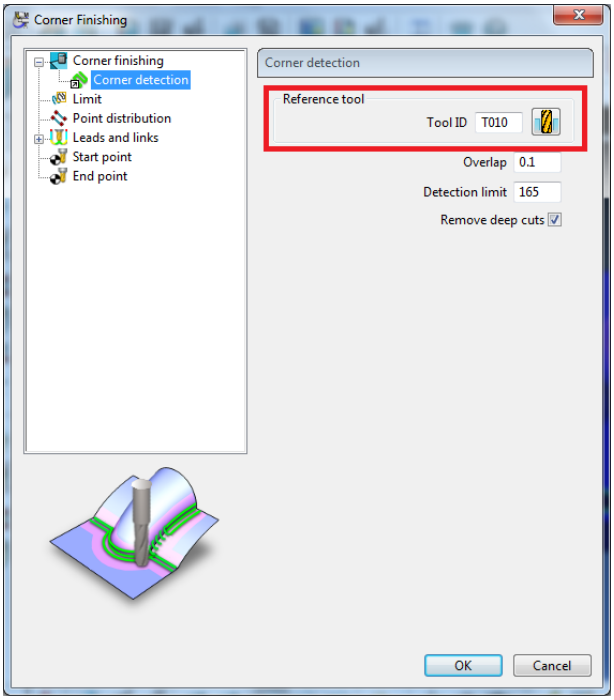
17 Click the **<Options>** button next to **Corner Finishing** strategy to display the **Surface Machining Options** dialog as shown to the right.



18 In the left panel of the dialog, click on Corner detection item under Corner finishing tree.

19 Note the reference tool icon. This shows the tool being referenced for the remachining.

20 Since you are using the same sized tool for remachining as you are for roughing and finishing, you will need to specify and overlap. Enter an overlap 0.1



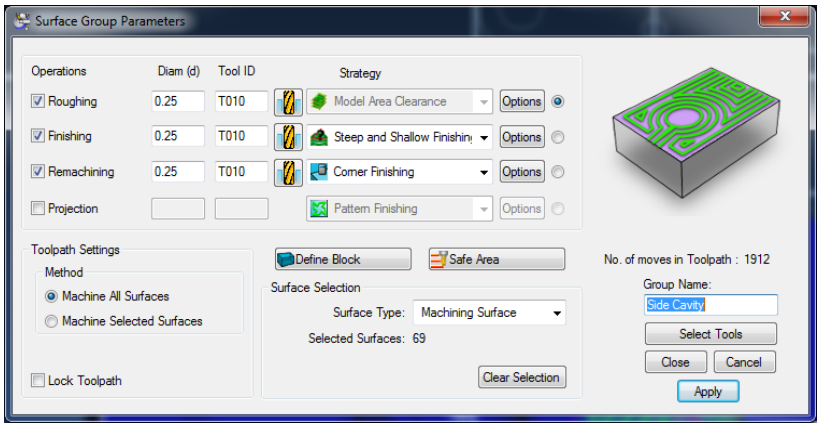
21 Click **<OK>** to return to the **Surface Group Parameters** dialog.

Select Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**.

Make sure the **Machine All Surfaces** radio button is checked. As a result you will have selected every surface of the part to be machined.

Your dialog should appear as below where the number of **Selected Surfaces** is 69.

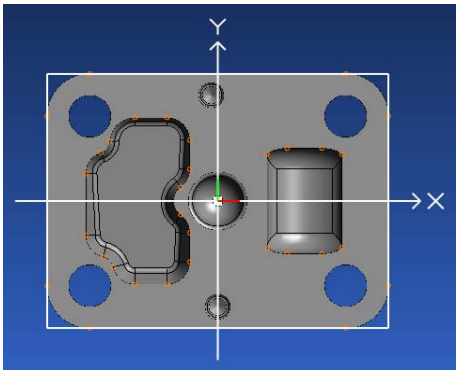


Click on the <**Close**> button in **Surface Group Parameters** dialog to return to the Face Window.

Create a Boundary Curve

Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict machining area.

1. Choose the **Chain Geometry on Solid Model** icon
2. Double click on any of the orange points of the side cavity as you did in the previous section.

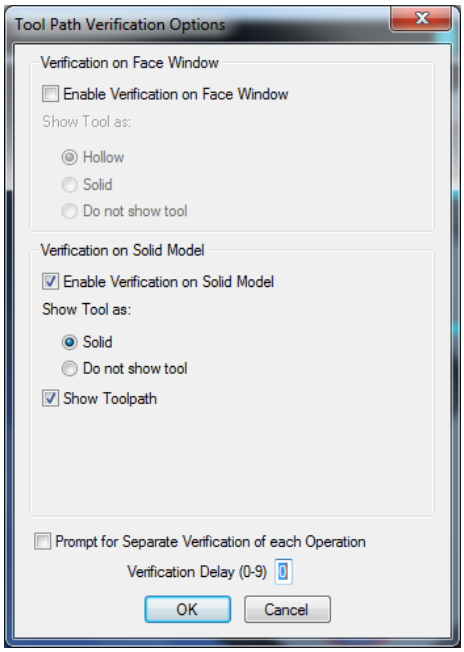


Verify the Tool Path

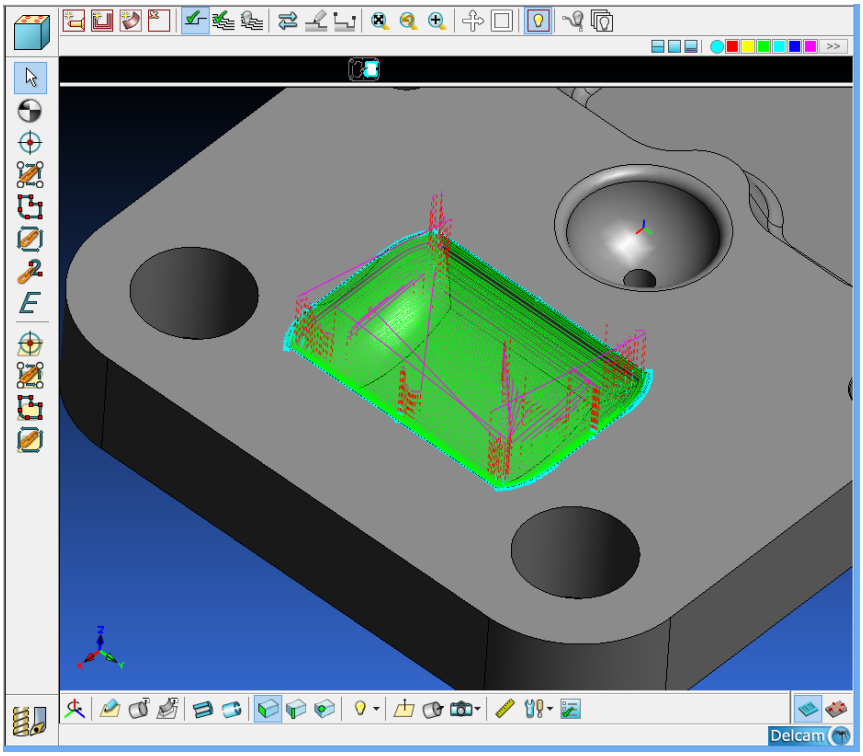
Once you have created the **Part Feature** and established a **Boundary Curve** you visually verify the tool path PartMaker has calculated. To do so:

1. From the Face Window tool bar choose Verify Work Group Tool path.
2. Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.
3. Enter a Verification Delay of 0.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see a 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



5 To hide toolpath, from the **Face Window** tool bar click on **Hide Every Tool path**.

Lock the Tool Path

As you did in the previous step, once you are satisfied with the verified tool path, you should now lock the tool path. To do so:

1 Double click on the feature in the **Job Explorer** tree

2 Click the Lock Toolpath check box in Surface Group Parameters dialog

3 Click the **<Close>** button to close the **Surface Group Parameters** dialog

Surface Machining Strategies for the Center Cavity

In this section of the tutorial you will create a roughing operation using Model Area Clearance strategy and a finishing operation using Spiral Finishing strategy for the circular cavity in the middle of the part. Both, roughing and finishing operations will be done using only one **Part Feature**

1 Choose a **New Color**.

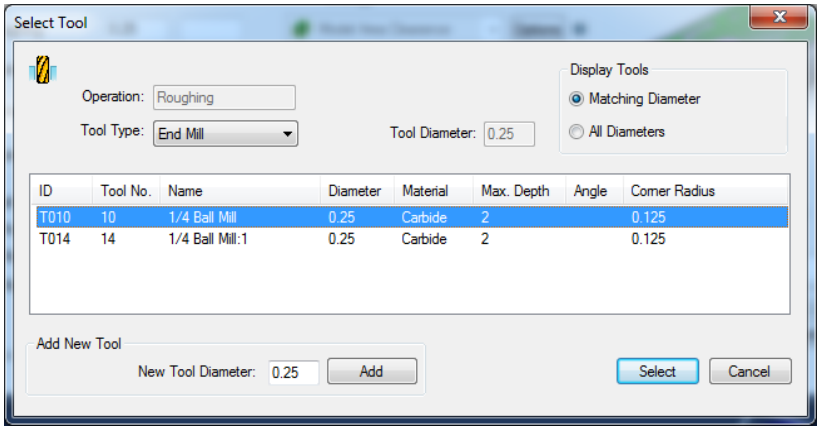
2 Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will open.

3 Check the **Finishing** radio button. Make sure that **Roughing** is still checked.

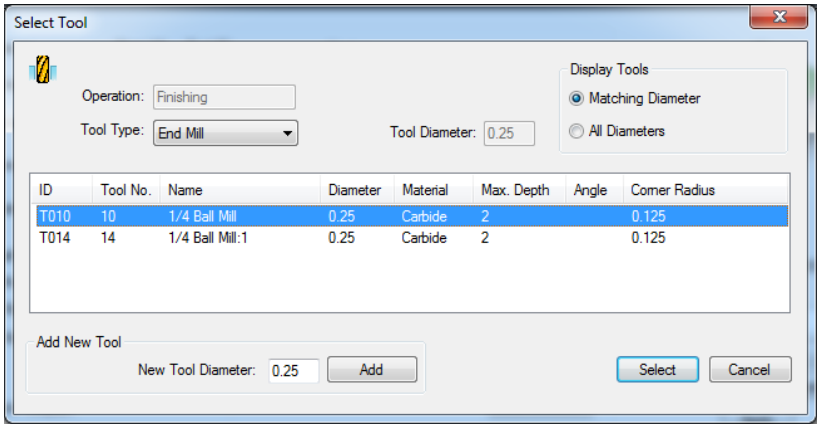
4 Enter tool diameter of 0.25 (6.35) for **Roughing** and **Finishing** operations.

5 Click the <**Select Tools>** button.

Select T010 1/4 Ball Mill for **Roughing** operation by clicking <**Select**> button as shown below.



Upon clicking <Select> button, the dialog box closes and reopens for **Finishing** operation as shown below. Select the tool T010 1/4 Ball Mill for **Finishing** and click <**Select**> button to return to **Surface Group Parameters** dialog.

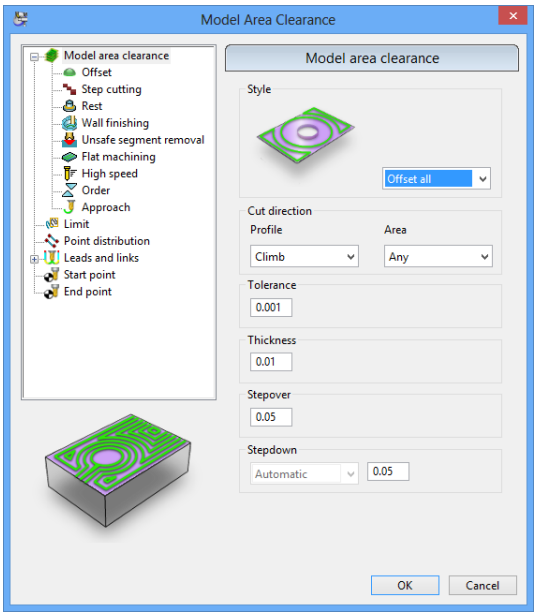


6 In the **Surface Group Parameters** dialog, choose **Spiral Finishing** from the **Strategy** drop down menu.

7 In the Group Name field enter "**Center Cavity**". Press the **<Enter>** key or click the **<Apply>** button.

8 Click the **<Options>** button next to the **Model Area Clearance** strategy to display the **Surface Machining Options** dialog as shown to the right.

9 Change the **Style** to **Offset All** and set the **Thickness** value to 0.01 (0.254).

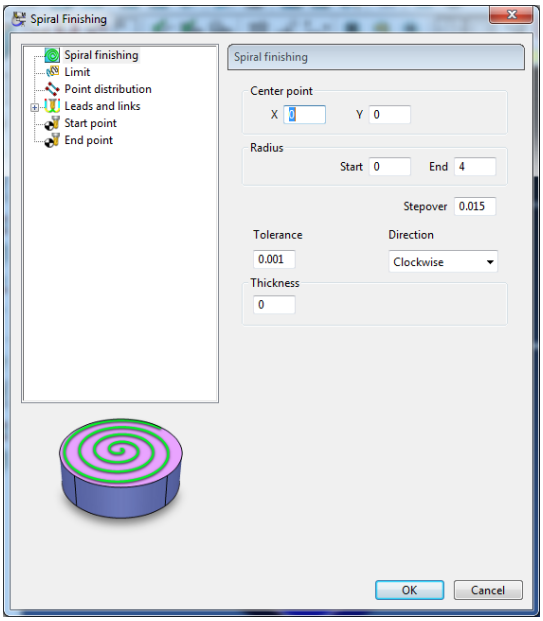


10 Click <OK> to return to the **Surface Group Parameters** dialog.

11 Click the **<Options>** button next to **Spiral Finishing** strategy to display the **Surface Machining Options** dialog as shown to the right.

12 Set the value of **Stepover** to 0.015 (0.381).

13 Make sure the **Thickness** value is set to zero.



14 Click <**OK**> to return to the **Surface Group Parameters** dialog.

Different Types of Surfaces in ASM

ASM allows you to select a few different types of surfaces, including:

- **Machining Surface** – A surface that will be machined with a surfacing tool path
- **Hole Surface** – A hole that will machined in another process, typically some sort of hole making (i.e. drilling) or 2D milling operation.
- **Collision** - A surface that will not be machined, but must be avoided (for example, a clamp).
- **Ignore** - A surface that will not be machined and was created only for construction purposes.

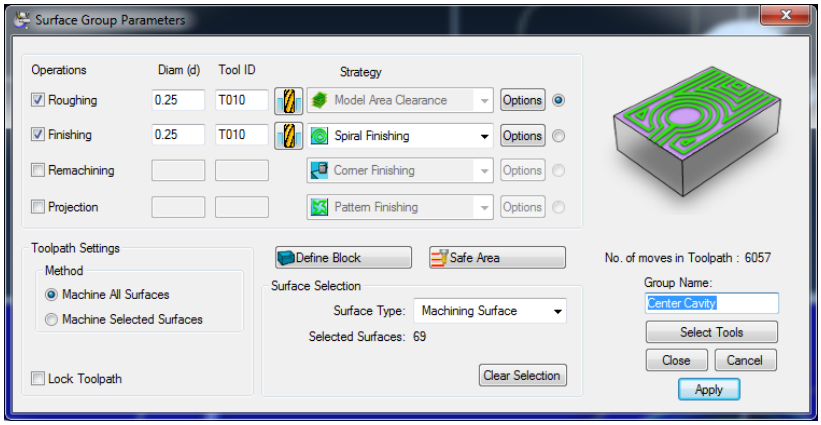
In this section, we will create both a Machining surface and a Hole surface

Select Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**.

Make sure the **Machine All Surfaces** radio button is checked. As a result you will have selected every surface of the part to be machined.

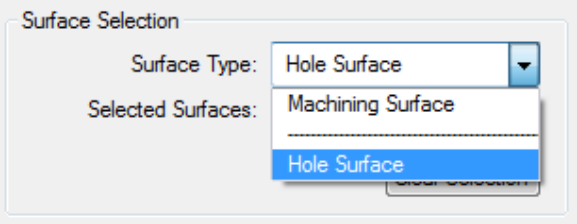
Your dialog should appear as below, where the number of **Selected Surfaces** is 69.



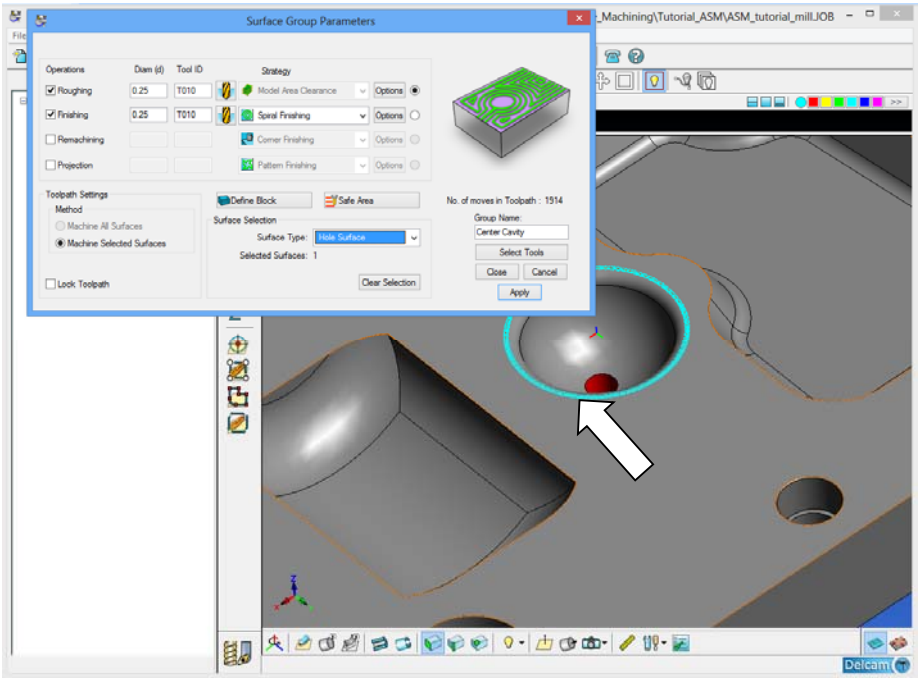
Select a Hole Surface

In the case of the center cavity, there is a small drilled hole at the bottom. We want to make sure our end mill does not go into this hole. In order to avoid having the tool defined in this feature go into the hole, we are going to define this as a **Hole Surface**. To do so:

1 In the **Surface Group Parameters** dialog, choose the **Hole Surface** from the **Surface Type** drop down as show here:



2 Select the hole at the bottom of the center cavity by clicking on it with your left mouse button as shown below:

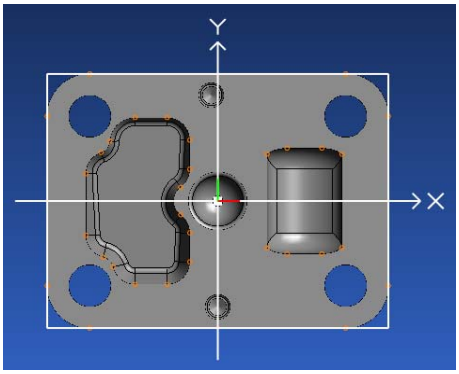


3 Click on the <**Close**> button in **Surface Group Parameters** dialog to return to the Face Window.

Create a Boundary Curve

Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict machining area.

1. Choose the **Chain Geometry on Solid Model** icon
2. Double click on any of the orange points of the side cavity as you did in the previous section.

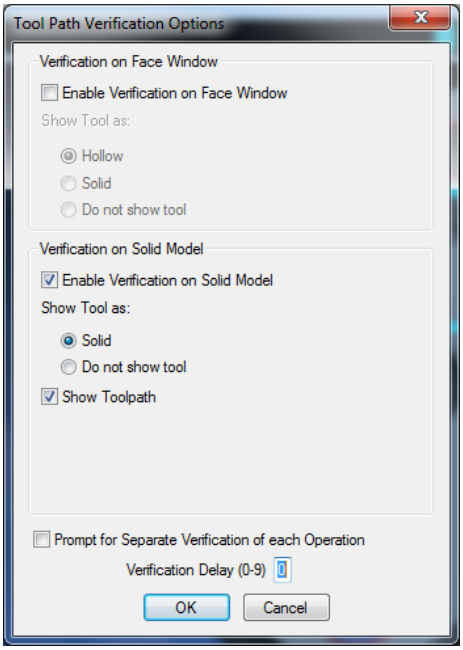


Verify the Tool Path

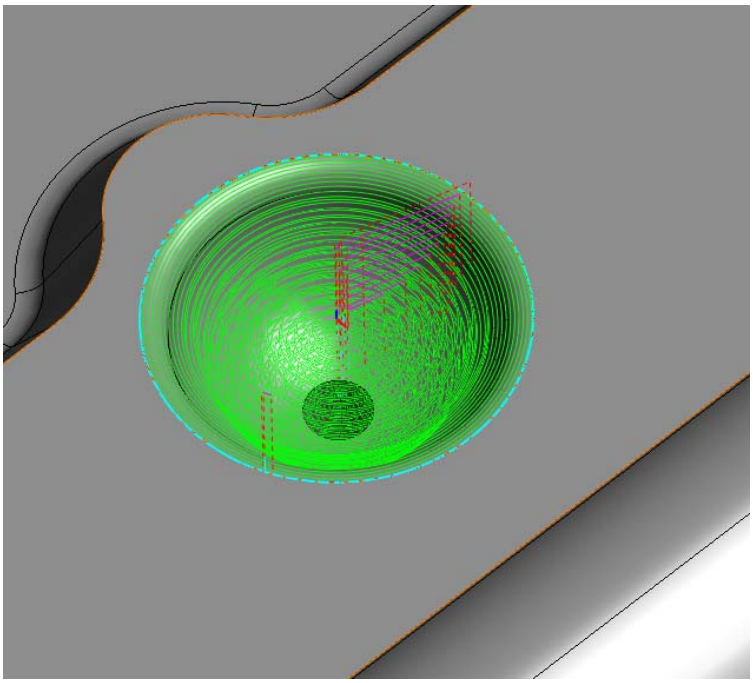
Once you have created the **Part Feature** and established a **Boundary Curve** you visually verify the tool path PartMaker has calculated. To do so:

1. From the Face Window tool bar choose Verify Work Group Tool path.
2. Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.
3. Enter a Verification Delay of 0.

***Note:** When entering a **Verification Delay** of greater than 0, you will see a 3D representation of the cutting tool moving along the Solid Model. When **Verification Delay** is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



5 From the Face Window tool bar choose Hide Every Tool Path.

Lock Toolpaths

As you did in the previous step, once you are satisfied with the verified tool path, you should now lock the tool path. To do so:

1 Double click on the feature in the **Job Explorer** tree

2 Click the Lock Toolpath check box in Surface Group Parameters dialog

3 Click the <Close> button to close the Surface Group Parameters dialog

Define Profiles for Contour Mill Cycle

In this section of the tutorial you will mill the outside of the part using a 2 ½ axis milling feature

1 Choose a **New Color**.

2 Choose New Profile Group from the Part Features menu to display the Profile Group Parameters dialog.

3 Choose **Contour Mill** from the **Strategy** drop down menu.

4 Choose **Left** from the **Tool Position** drop down menu.

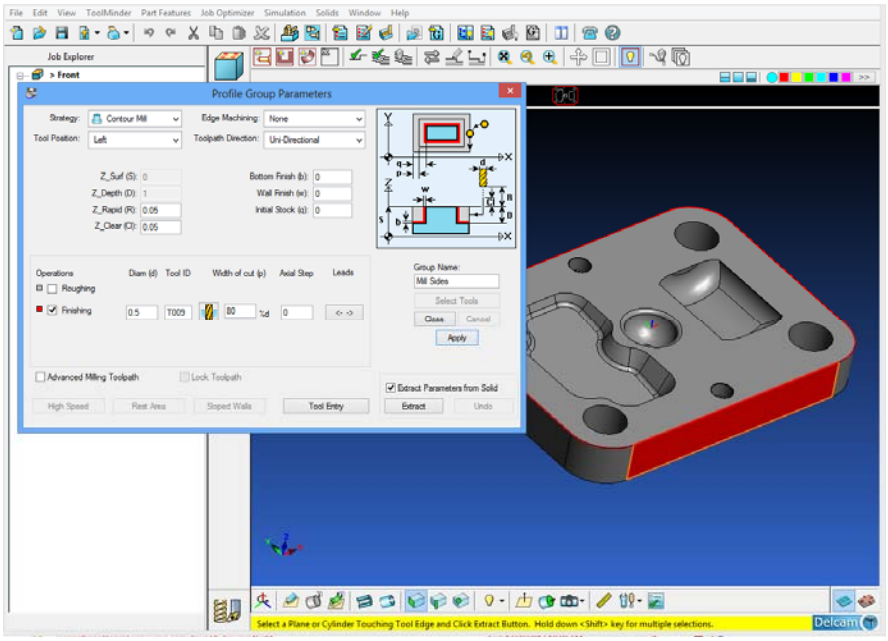
5 Enter 0.5 (12.7) in the **Diam (d)** field.

6 Click the **<Select Tools>** button.

7 From the **Select Tool** dialog, click on the End Mill_1/2 and click the **<Select>** button

8 Click the Extract Parameters from Solid box

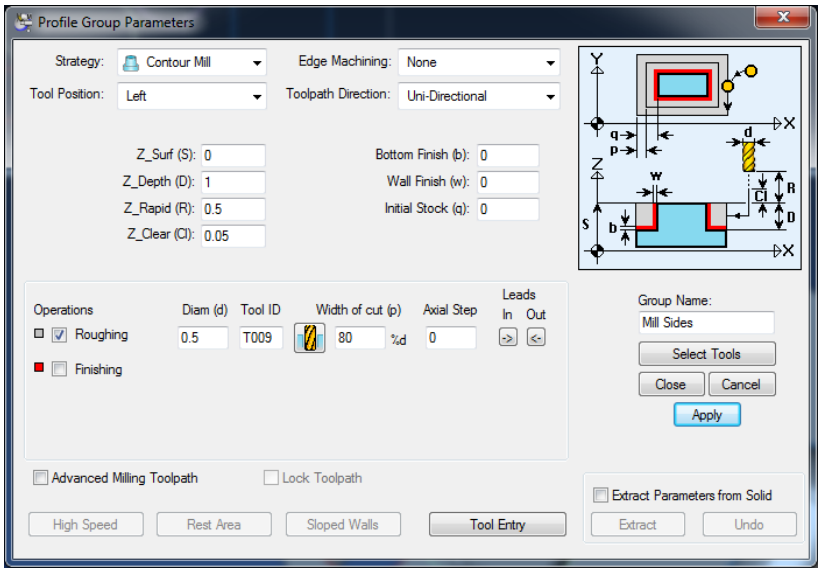
9 Select the side of the part as shown below. Once selected, it will be highlighted in red.



10 Click the <**Extract>** button and the machining **Z_Depth** will be populated automatically.

11 Enter "**Mill Sides**" in the **Group Name** field. Press **<Enter**> on your keyboard or click **<Apply>**.

When you are finished your dialog should appear as shown below:

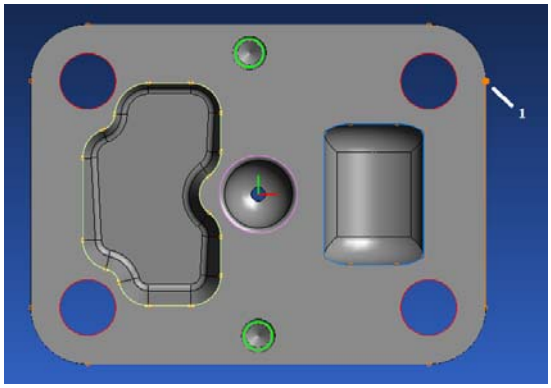


12 Click the **<Close>** button.

Create Profiles Using Chain Geometry Icon

1 Select the **Chain Geometry on Solid Model** icon on the left side of the **Face Window**.

2 Double click on the location marked “1” in the following illustration. A profile is automatically created.

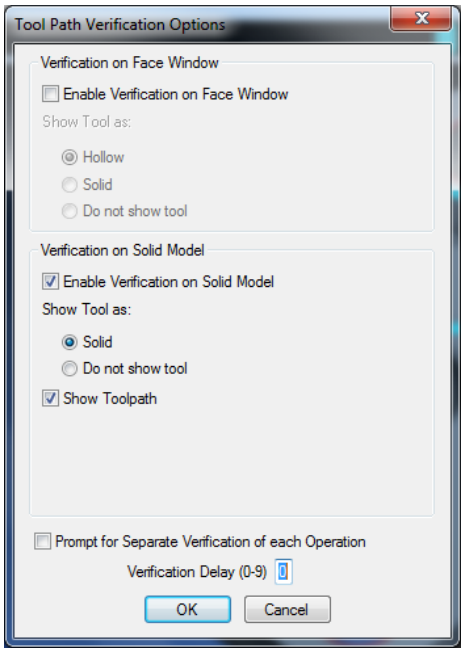


Note: *The profile shows direction arrows. This means it is selected and can be reversed, translated or rotated.*

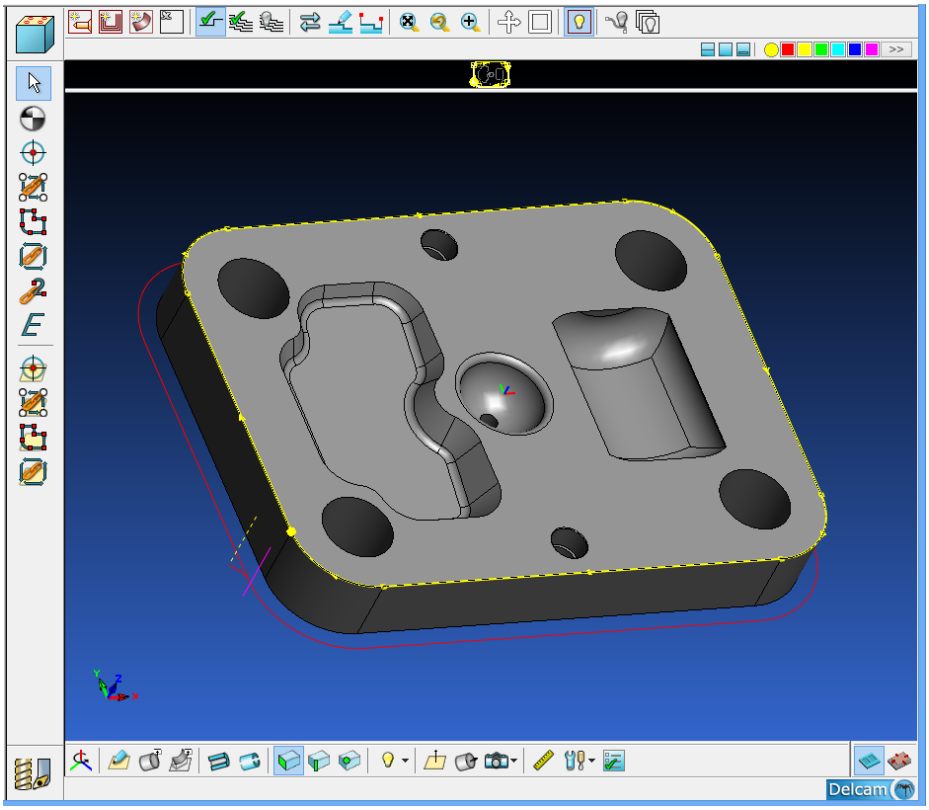
Verify the Tool Path

1. From the Face Window tool bar choose Verify Work Group Tool path.
2. Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.
3. Enter a Verification Delay of 0.

***Note:** When entering a **Verification Delay** of greater than 0, you will see a 3D representation of the cutting tool moving along the Solid Model. When **Verification Delay** is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



5 To hide toolpath, from the **Face Window** tool bar click on **Hide Every Tool path**.

Drill a Hole in the Center Cavity

In this section of the tutorial you will drill the hole in the center cavity

1 Choose **New Color**.

2 Choose **New Hole Group** from the **Part Features** menu to display the **Hole Group Parameters** dialog.

3 Leave the **Through Hole** box checked.

4 Leave the **Type** drop down menu set to **Drill.**

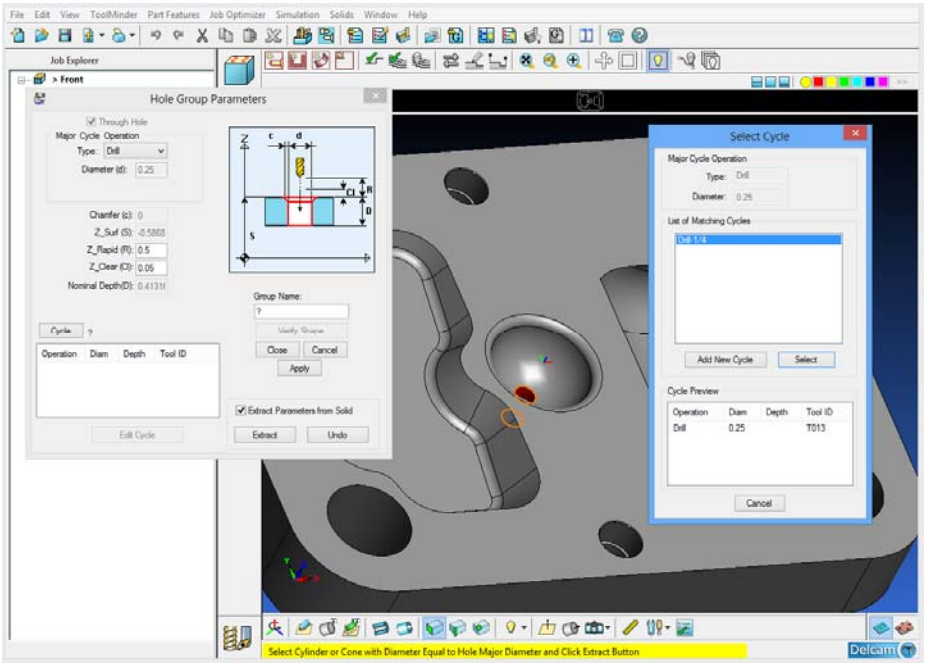
5 Leave the **Diameter (d)** field set to 0.25 (6.35).

6 Click on the “Extract Parameters from Solid” check box.

7 Click on the Solid Model inside the hole in Center Cavity.

8 Click on the “Extract” button and select the Drill-1/4 (Drill_6.35) cycle.

9 In the **Select Cycle** dialog click the **<Select>** button to choose selected cycle

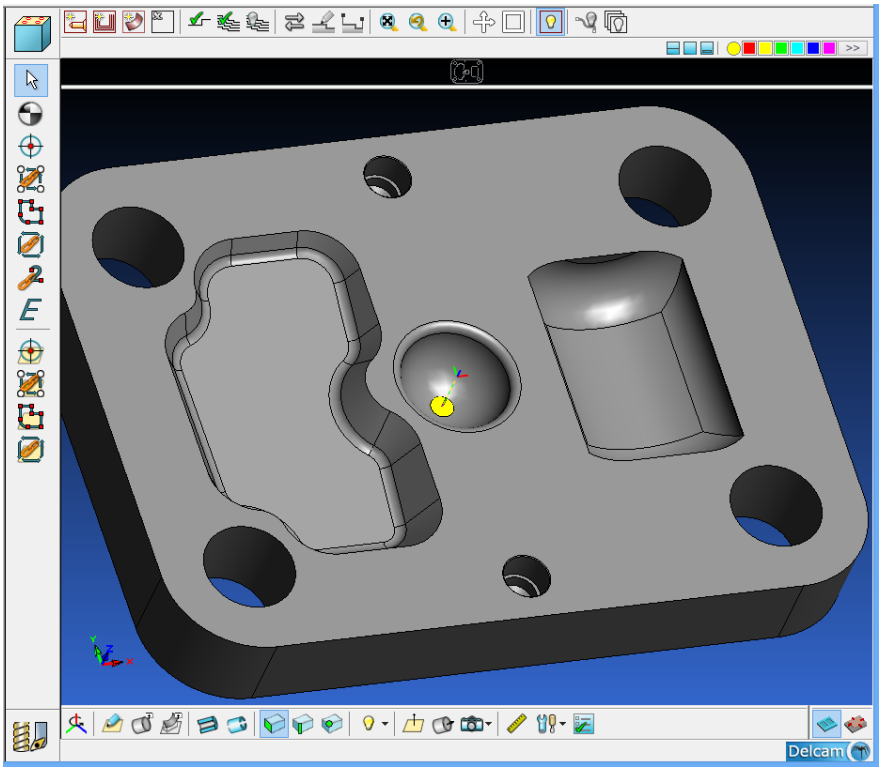


10 The **Hole Group Parameters** dialog has been filled in using information from the solid model.

11 Click the **<Close>** button to close the **Hole Group Parameters** dialog.

12 Choose the **Single Hole on Solid Model** icon and click on the .25 (6.35) inch drilled hole in the middle of the part.

Your screen should now appear as shown below:

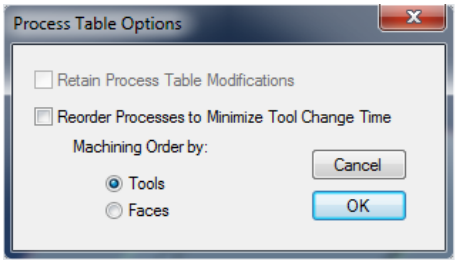


Generate a Process Table

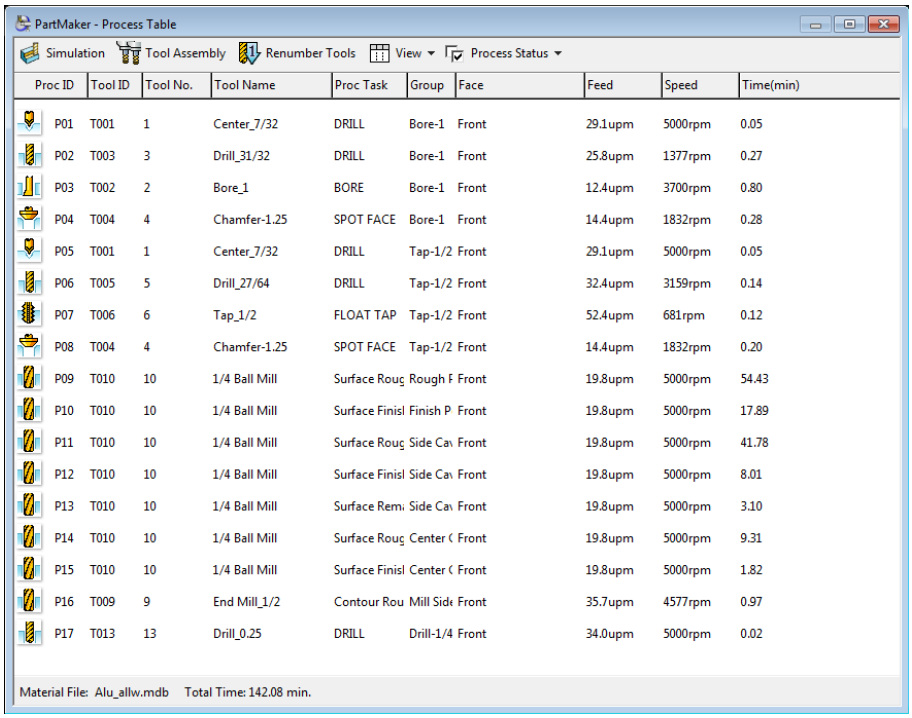
The *process table* shows you all of the processes for a part. Before generating a process table, you can choose to reorder processes automatically to minimize tool change time and optimize machining.

When **PartMaker** generates a process table, all cutting conditions such as feed rate and spindle speed RPM are calculated automatically based on the tools and material information.

1 Choose **Generate Process Table** from the **Main** tool bar to display the Process Table Options dialog as below:



2 Click <**OK**> to display the **Process Table** window as shown below.

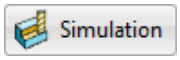


Proc ID	Tool ID	Tool No.	Tool Name	Proc Task	Group	Face	Feed	Speed	Time(min)
P01	T001	1	Center_7/32	DRILL	Bore-1	Front	29.1upm	5000rpm	0.05
P02	T003	3	Drill_31/32	DRILL	Bore-1	Front	25.8upm	1377rpm	0.27
P03	T002	2	Bore_1	BORE	Bore-1	Front	12.4upm	3700rpm	0.80
P04	T004	4	Chamfer-1.25	SPOT FACE	Bore-1	Front	14.4upm	1832rpm	0.28
P05	T001	1	Center_7/32	DRILL	Tap-1/2	Front	29.1upm	5000rpm	0.05
P06	T005	5	Drill_27/64	DRILL	Tap-1/2	Front	32.4upm	3159rpm	0.14
P07	T006	6	Tap_1/2	FLOAT TAP	Tap-1/2	Front	52.4upm	681rpm	0.12
P08	T004	4	Chamfer-1.25	SPOT FACE	Tap-1/2	Front	14.4upm	1832rpm	0.20
P09	T010	10	1/4 Ball Mill	Surface Roug	Rough F	Front	19.8upm	5000rpm	54.43
P10	T010	10	1/4 Ball Mill	Surface Finisl	Finish P	Front	19.8upm	5000rpm	17.89
P11	T010	10	1/4 Ball Mill	Surface Roug	Side Ca	Front	19.8upm	5000rpm	41.78
P12	T010	10	1/4 Ball Mill	Surface Finisl	Side Ca	Front	19.8upm	5000rpm	8.01
P13	T010	10	1/4 Ball Mill	Surface Rem	Side Ca	Front	19.8upm	5000rpm	3.10
P14	T010	10	1/4 Ball Mill	Surface Roug	Center (	Front	19.8upm	5000rpm	9.31
P15	T010	10	1/4 Ball Mill	Surface Finisl	Center (	Front	19.8upm	5000rpm	1.82
P16	T009	9	End Mill_1/2	Contour Rou	Mill Side	Front	35.7upm	4577rpm	0.97
P17	T013	13	Drill_0.25	DRILL	Drill-1/4	Front	34.0upm	5000rpm	0.02

Note: *If tool numbers are not in the proper order, you can choose* ***Renumber Tools*** *from the* ***Job Optimizer*** *menu to automatically renumber the tools before generating an NC part program.*

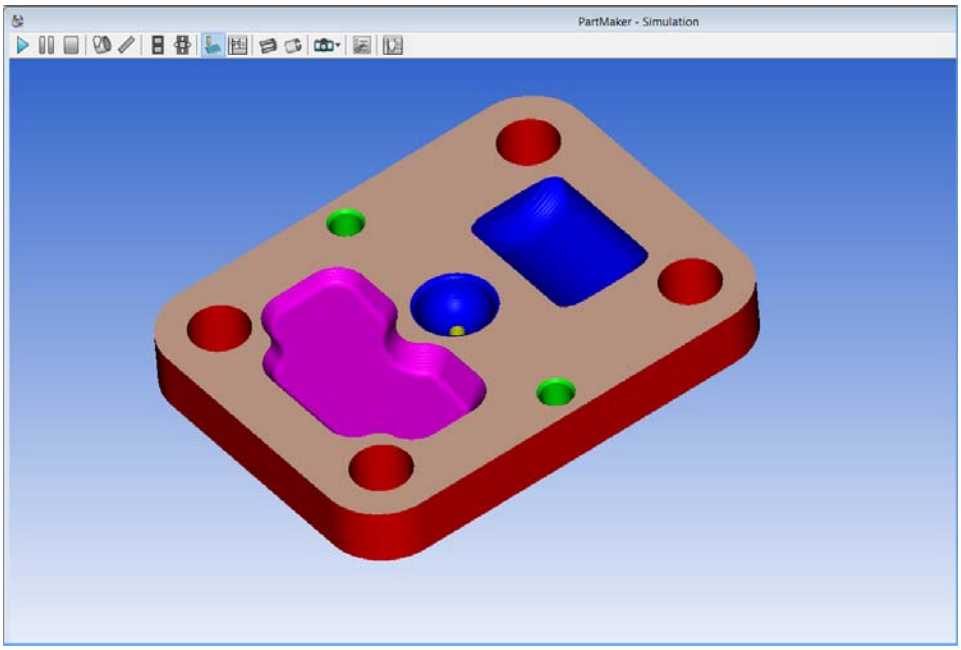
Simulate the Cutting Process

Once you are satisfied with the appearance of your Process Table you can simulate cutting. To do so:



1 From the Process Table, click the <**Simulation**> button in the lower left hand corner of the Process Table or just press the **<Space Bar>**.

2 When the **Simulation** window appears, press the **<Space Bar>** or choose **Start Simulation** from the **Simulate** menu to initiate 3D simulation. Your completed part should appear as shown below:



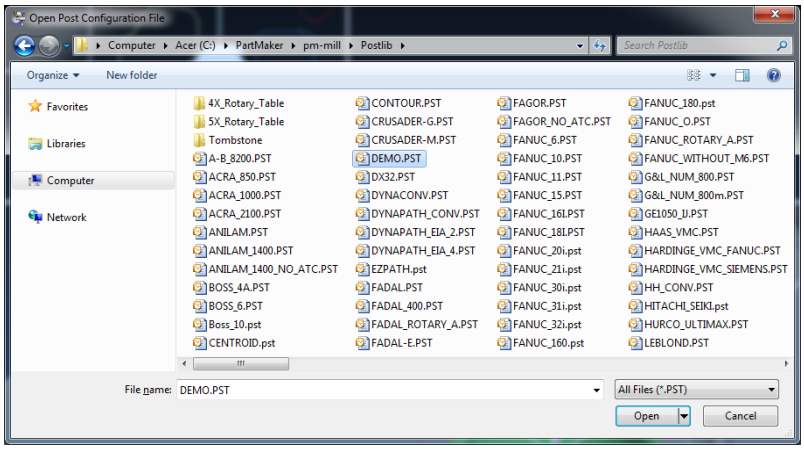
Generate an NC Program

1 Choose Post Config File= ? From the Job Optimizer menu to display the Open Post Configuration File dialog.

2 Locate and double-click the **Postlib** folder to open the Post Configuration file.

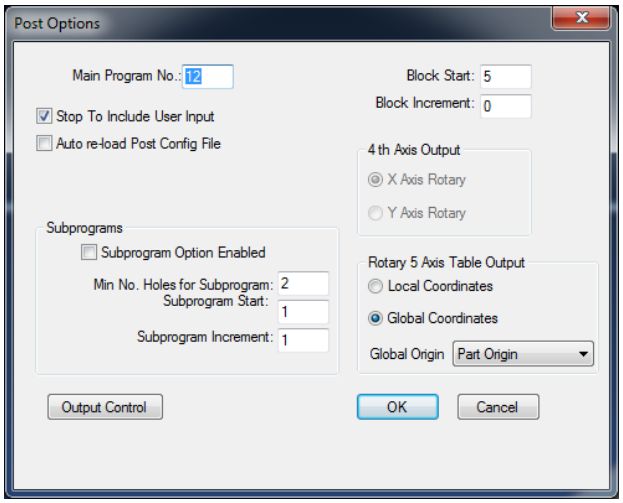
3 Select the DEMO.PST post processor.

4 Click the **<Open>** button to load the DEMO.PST.



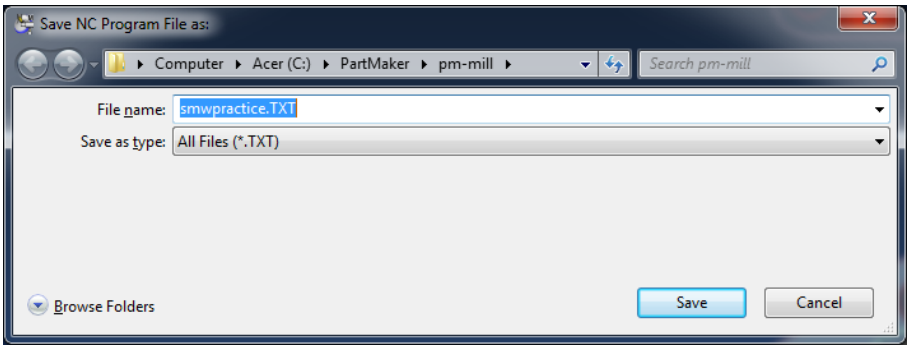
Note: *If you use the same .PST file all the time, move it from POSTLIB (the postprocessor library) to your working directory/folder where it will be loaded automatically when you generate an NC program.*

5 Click on **Generate NC-Program** icon from the **Main** tool bar to display the **Save NC Program** dialog. When you generate an NC Program for the first time during a programming session the **Post Options** dialog is displayed before the **Save NC Program** dialog.



6 Choose the Post Options you prefer. If you leave Block Increment at zero there will be no sequence numbers in your part program.

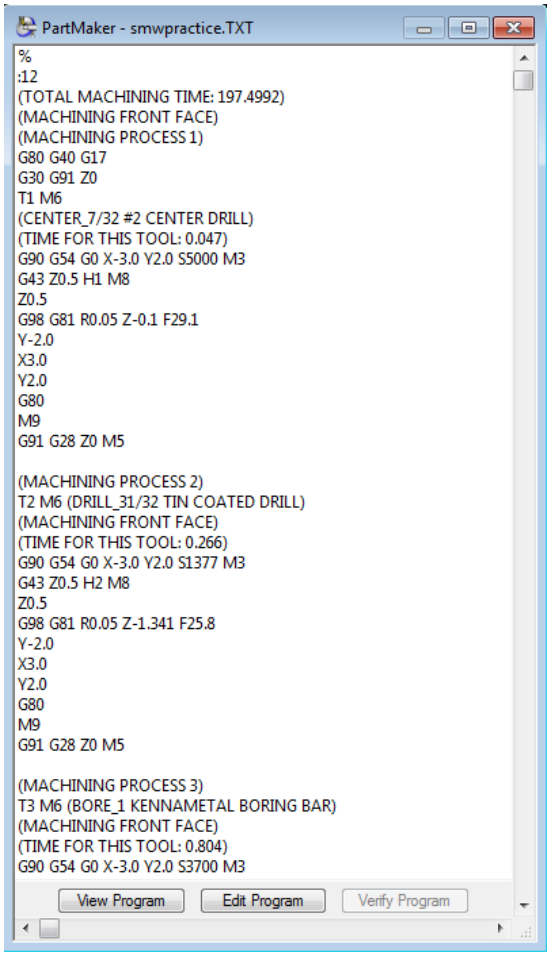
7 Click **<OK>** to proceed to the Save NC Program dialog:



8 Enter the name smwpractice.TXT for the NC program.

9 Click **<Save>** to generate the NC Program.

NC Program generated is shown below.



Chapter 2: Creating a Sample Part using PartMaker Advanced Surface Machining (ASM) for SwissCAM and Turn-Mill

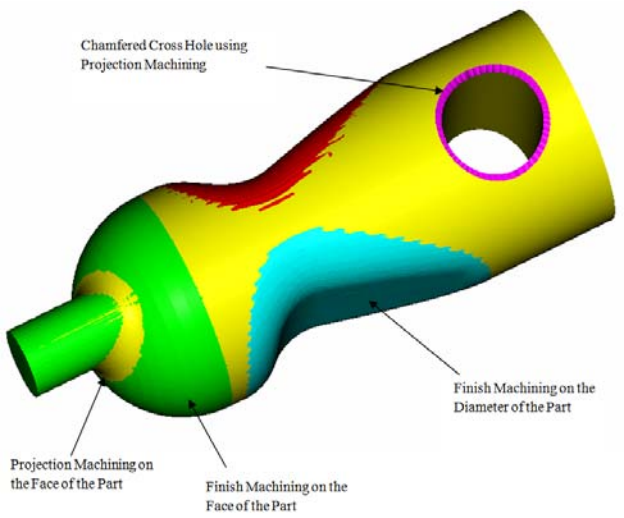
Introduction

This tutorial is designed to help you learn the steps to go through in using PartMaker SwissCAM or PartMaker Turn-Mill to program parts on a lathe utilizing PartMaker's Advanced Surface Machining. This tutorial assumes that you have a good working knowledge of either PartMaker SwissCAM or Turn-Mill. Though the PartMaker SwissCAM module is used for this tutorial, the steps can be identically replicated if working in PartMaker Turn-Mill.

This tutorial has been developed for use with PartMaker Versions 2013 and higher.

How You Will Create the Sample Part

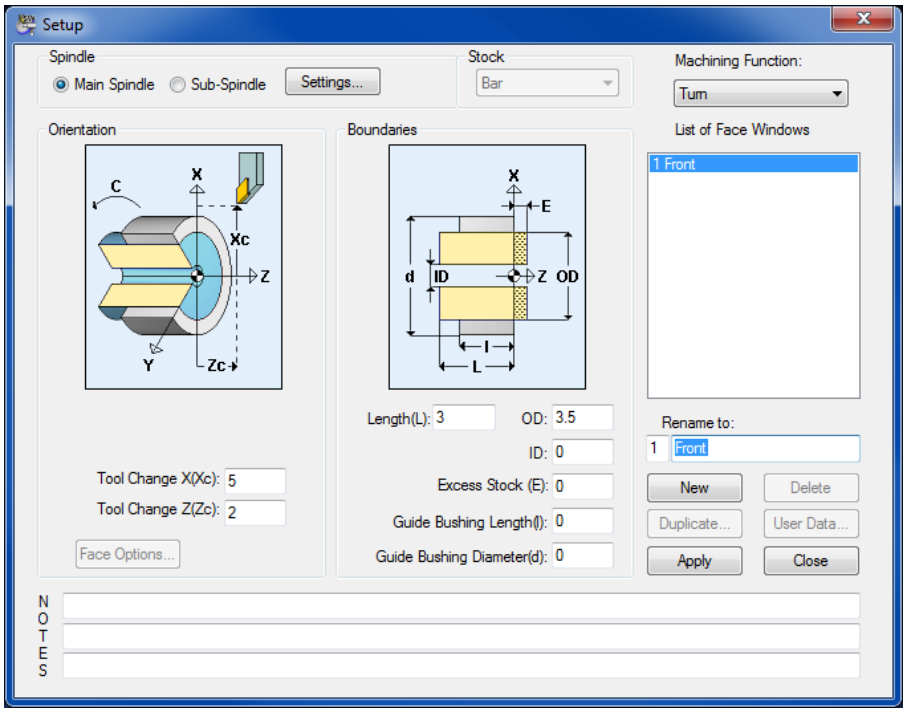
Below are the major concepts that will be covered in this tutorial.



- Solid Model Importing
- Establishing part boundaries from a Solid Model
- Transferring Geometry from a Solid Model
- Opening Tools, Cycles and Material files
- Performing Turning operations in the context of surface machining
- Creating Surface Finishing tool paths on the face of the part
- Creating Surface Projection tool paths on the face of the part
- Creating Surface Finishing tool paths on the outside diameter of the part
- Duplicating identical face windows
- Creating a cross hole
- Chamfering a cross hole using a Surface Projection Machining Strategy

Start PartMaker

1. Double click the PartMaker SwissCAM icon (or PartMaker Turn-Mill icon if using PartMaker Turn-Mill) on your Windows desktop.
2. When PartMaker launches, the **Setup** dialog will open as shown below:



3. Click **<Close>** to close the **Setup** dialog.

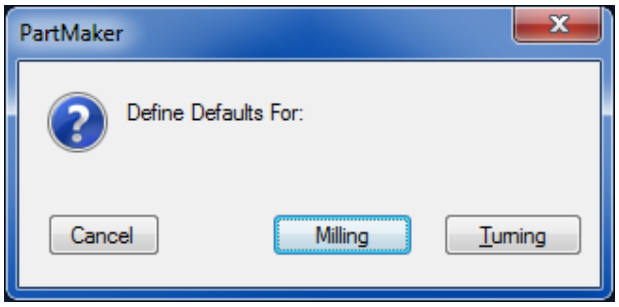
Set Defaults

Before getting started with this tutorial, it is important to set the correct defaults. Doing so is explained below:

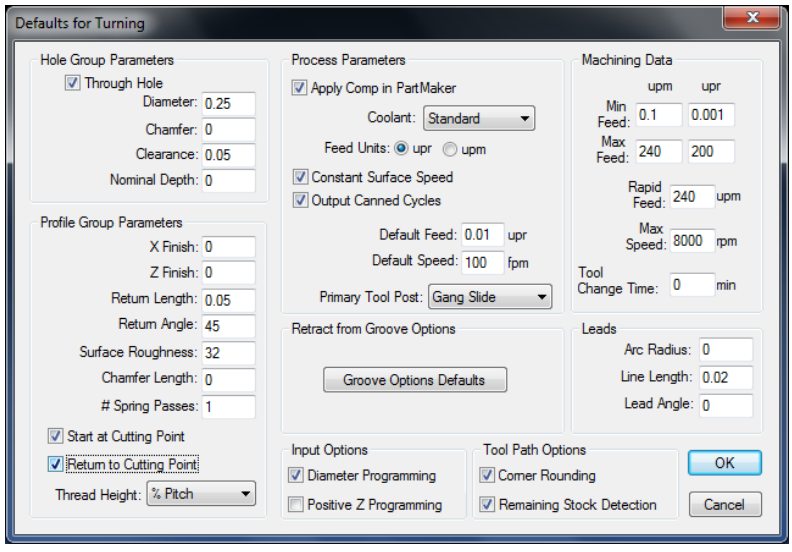
Defaults for Turning (SwissCAM users)

If you are using PartMaker SwissCAM to complete the steps of this tutorial, please read below. If you are using PartMaker Turn-Mill please skip to the next section called **Defaults for Turning (SwissCAM users)**:

1. From **Job Optimizer** menu choose the **Defaults** command. You will be prompted with the dialog below:



2. Click the **<Turning>** button to enter the **Defaults of Turning** dialog.
3. In the **Input Options** section of the **Defaults for Turning** dialog, check the **Diameter Programming** box (see below)
4. In the **Input Options** section of the **Defaults for Turning** dialog, make sure to uncheck the **Positive Z Programming** box as shown below:

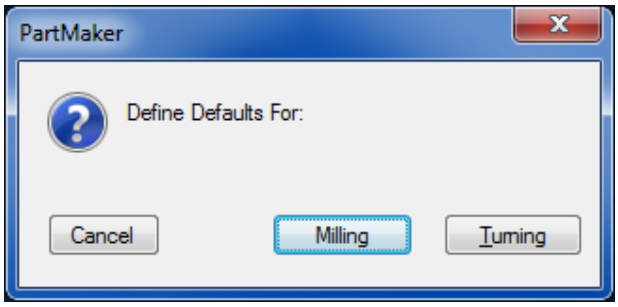


5. Click the **<OK>** button to close the **Defaults for Turning** dialog.

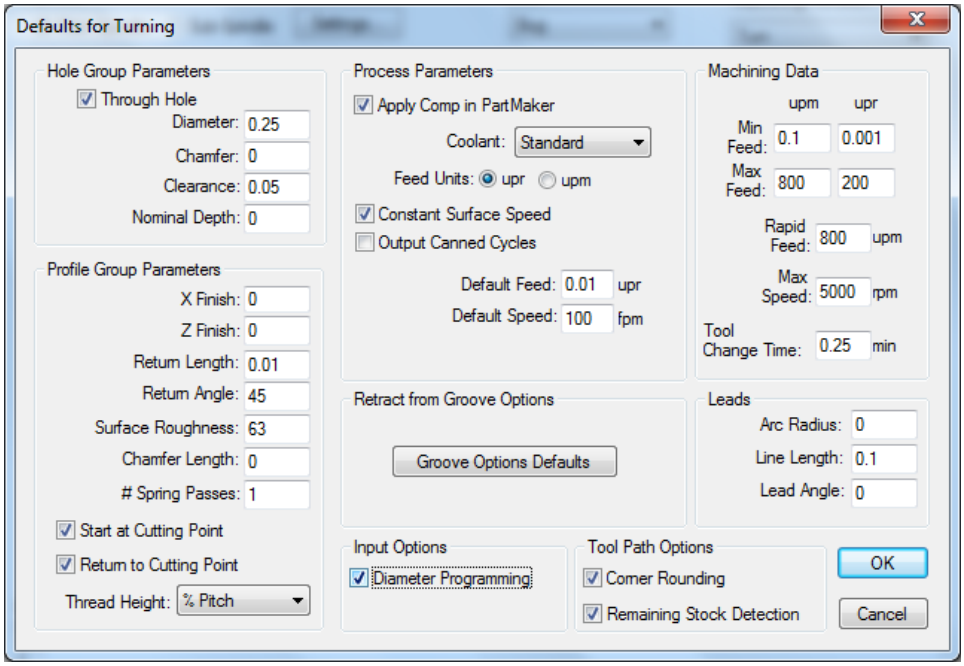
Defaults for Turning (Turn-Mill users)

If you are using PartMaker Turn-Mill to complete the steps of this tutorial, please read below. If you are using PartMaker SwissCAM please refer to the previous section called **Defaults for Turning (SwissCAM users)**:

1. From **Job Optimizer** menu choose the **Defaults** command. You will be prompted with the dialog below:



2. Click the **<Turning>** button to enter the **Defaults of Turning** dialog.
3. In the **Input Options** section of the **Defaults for Turning** dialog, check the **Diameter Programming** box (see below)

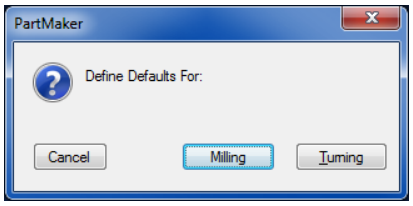


4. Click the **<OK>** button to close the **Defaults for Turning** dialog.

Defaults for Milling (SwissCAM and Turn-Mill users)

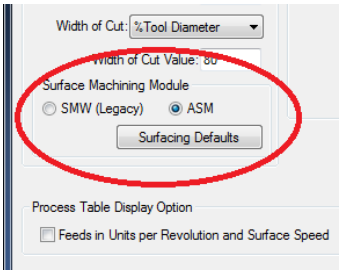
Both PartMaker Turn-Mill and SwissCAM users will be prompted with identical dialogs for setting default parameters for milling. To view these parameters:

1. From **Job Optimizer** menu choose the **Defaults** command. You will be prompted with the dialog below:

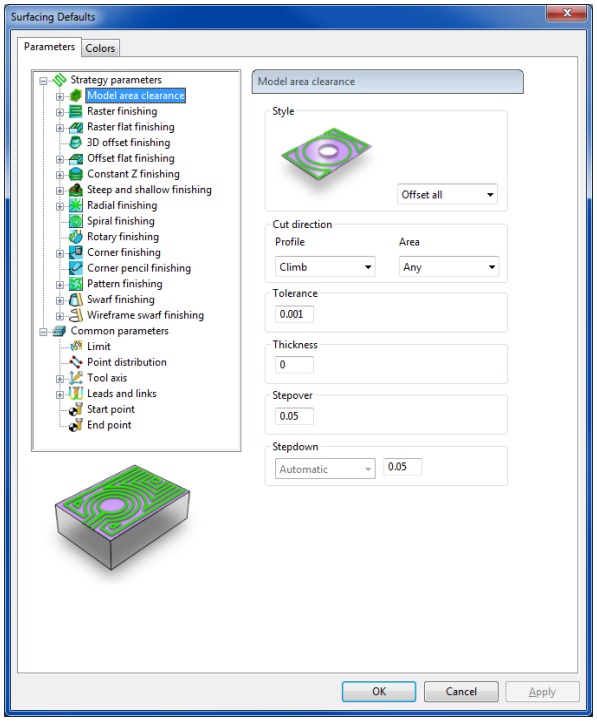


2. Click the **<Milling>** button to enter the **Defaults of Milling** dialog.

3. Make sure ASM Radio button is selected under Surface Machining Module to use ASM.



4. Click the **<Surfacing Defaults>** button in the **Defaults for Milling** dialog. The Surfacing Defaults dialog will display as shown below.



Importing the Solid Model

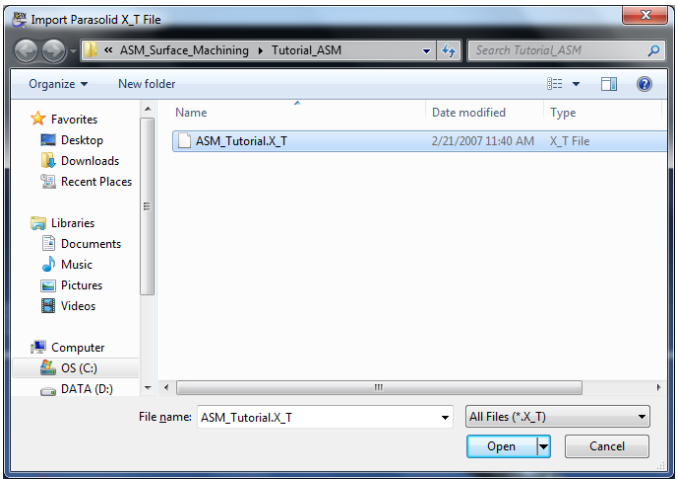
1 From the **File** menu choose the **Import** submenu as shown below:

2 From the **Import** submenu choose the **X_T Parasolid Transmit Text File...** command.

3 Open the Parasolid file at the following directory path:

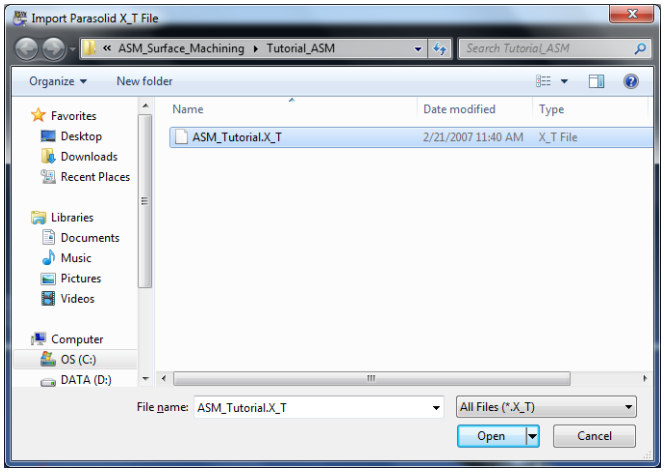
a If you are using PartMaker SwissCAM:

C:\PartMaker_2014\pm-swiss\ASM_Surface_Machining\Tutorial_ASM \ASM_Tutorial.X_T as shown below:



b If you are using PartMaker Turn-Mill:

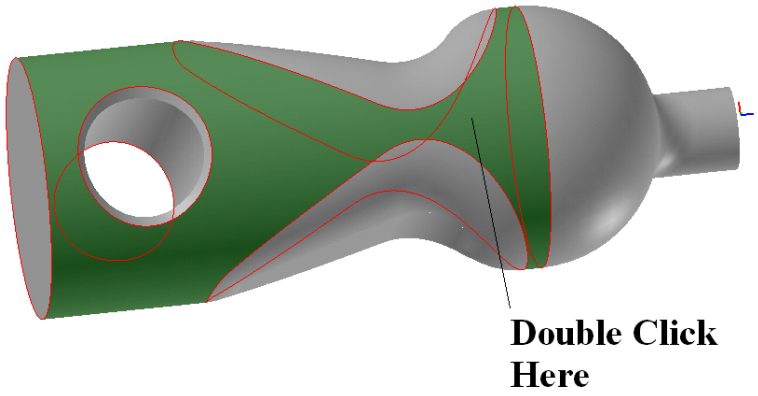
C:\PartMaker\pm-tm\ASM_Surface_Machining\Tutorial_ASM \ASM_Tutorial.X_T as shown below:



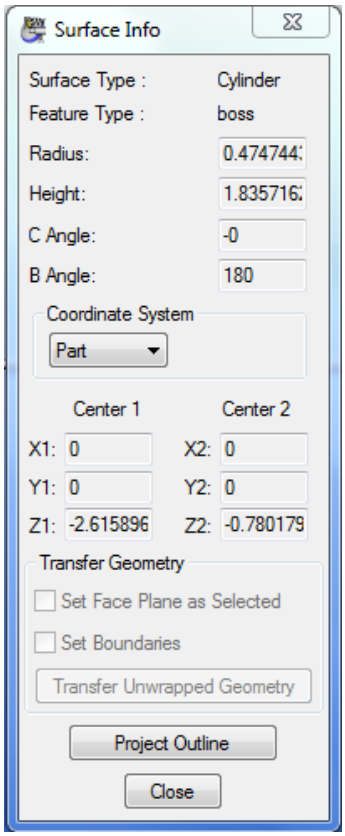
4 Click the **<Open>** button to load the Solid Model into PartMaker.

Setting Part Boundaries from the Solid Model

1 Double click on the OD of the Solid Model in the **Solid Model** window as shown below:



2 Double clicking on this surface will display the **Surface Info** dialog as shown below:



3 Take note of value of -2.6159 (-66.444) in the **Z1** field in the **Surface Info** dialog. This corresponds to the finished length of the part.

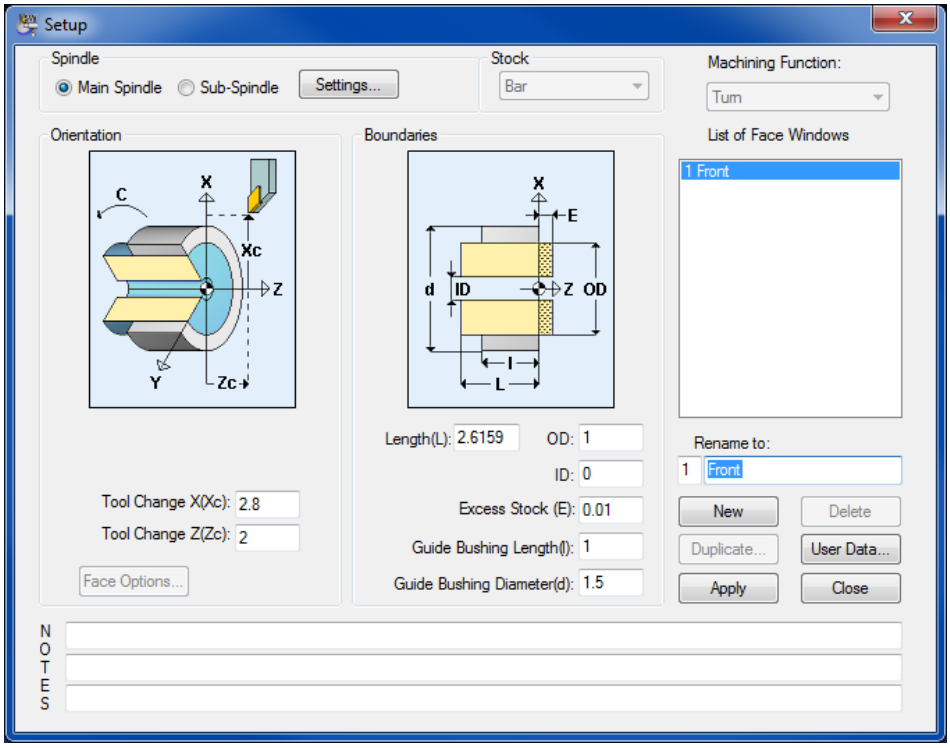
4 Take note of value of .4747 (12.059) in the **Radius** field in the **Surface Info** dialog. This corresponds to the largest radius of the part. As a result, for purposes of this tutorial it will be assumed that you will start from 1" (25.00) diameter bar stock (as the largest diameter is twice the value of the parts largest radius, i.e. 2*.4747 or .9494 (12.059*2=24.118)

5 Click the **<Close>** button to close the **Surface Info** dialog.

The Setup Dialog: SwissCAM Users

If you are using PartMaker SwissCAM to complete the steps of this tutorial, please read below. If you are using PartMaker Turn-Mill please skip to the next section called **The Setup Dialog: Turn-Mill Users**:

1. Choose **Setup** from the **View** menu.
2. Enter 2.6159 (66.444) in the **Length (L)** field in the **Setup** dialog.
3. The part will be made out of 1inch (25mm) bar stock. Enter 1(25) in the **OD** field of the **Setup** dialog.
4. Enter .01 (2.0) in the **Excess Stock (E)** field of the **Setup** dialog.
5. Click the **<Apply>** button. Your completed **Setup** dialog should appear as shown below:

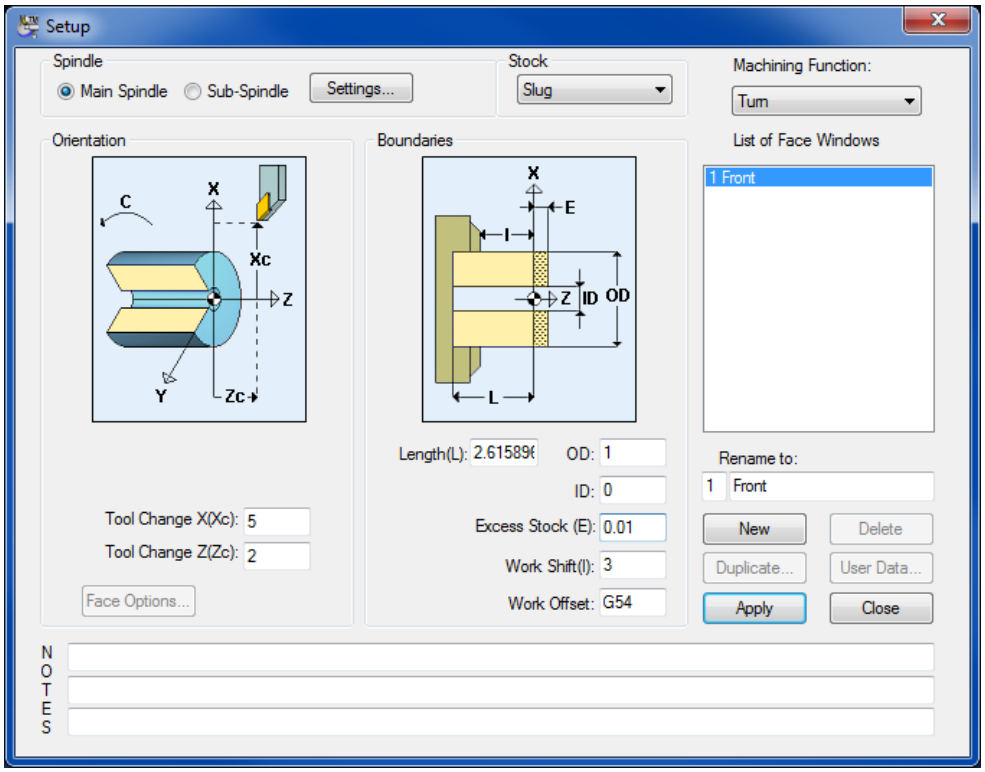


6. Click the **<Close>** button to close the **Setup** dialog.

The Setup Dialog: Turn-Mill Users

If you are using PartMaker Turn-Mill to complete the steps of this tutorial, please read below. If you are using PartMaker SwissCAM please see the previous section called **The Setup Dialog: SwissCAM Users**:

1. Choose **Setup** from the **View** menu
2. Check the **Use Bar Stock** box
3. Enter 2.6159 in the **Length (L)** field in the **Setup** dialog
4. The part will be made out of 1inch bar stock. Enter 1 in the **OD** field of the **Setup** dialog.
5. Enter .01 in the **Excess Stock (E)** field of the **Setup** dialog.
6. Enter 3 in the **Work Shift (I)** field of the **Setup** dialog.
7. Click the **<Apply>** button. Your completed **Setup** dialog should appear as shown below:

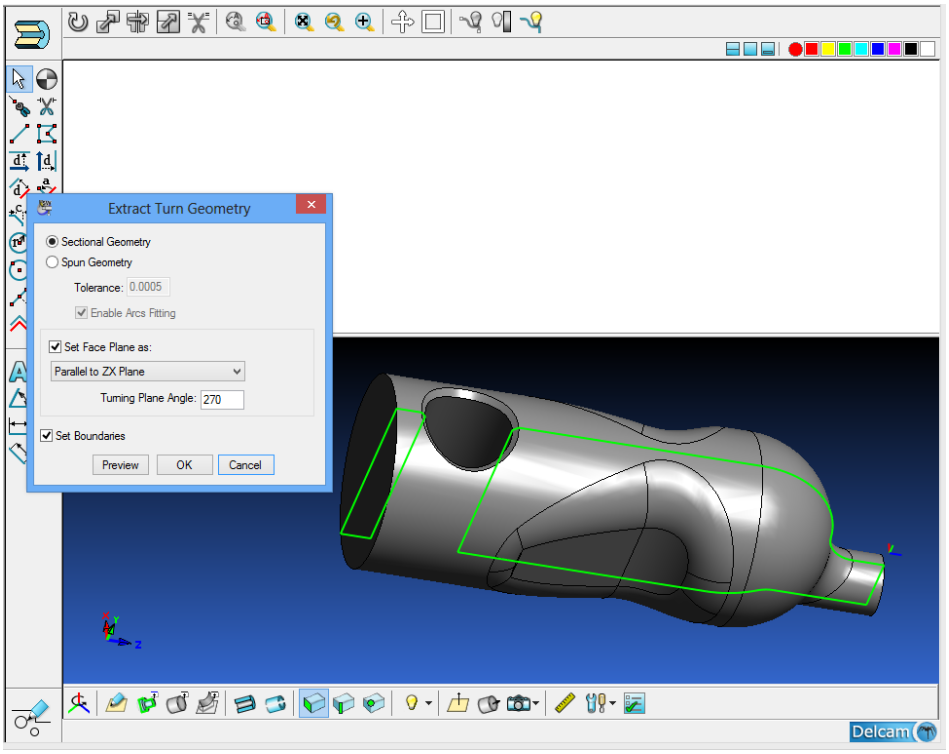


8. Click the **<Close>** button to close the **Setup** dialog.

Transferring Geometry from the Solid Model

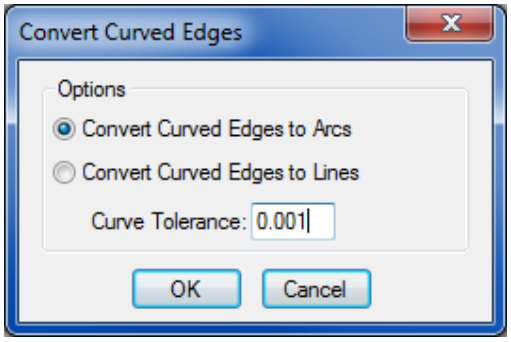
1 Use the **CAD / CAM Switch** icon to switch to CAD Mode.

2 On the Solids Window toolbar click the Extract Turn Geometry icon.

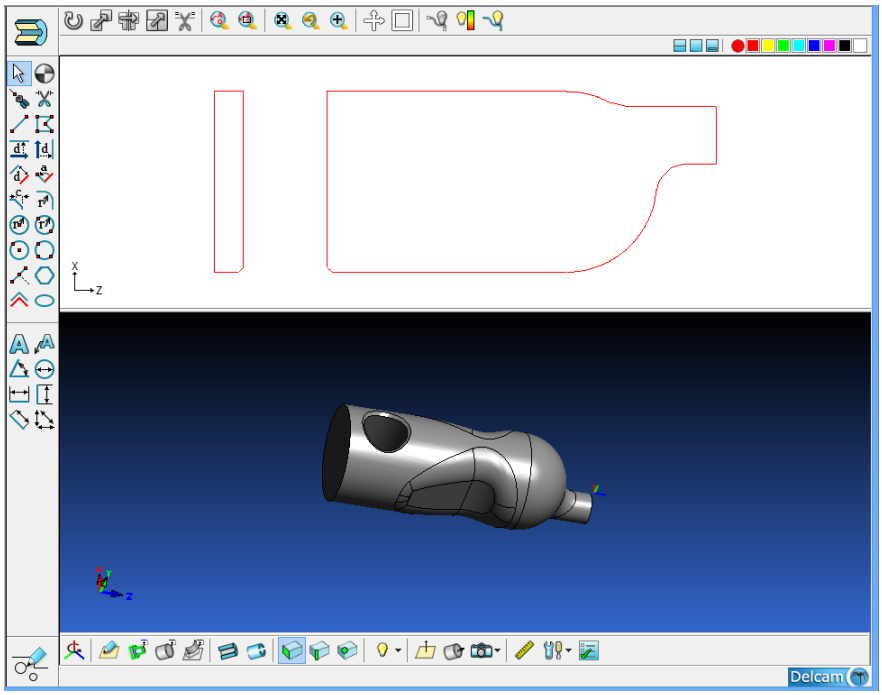


3 Set the "**Turning Plan Angle"** to 270 and click **<OK>**

4 Click the <**Convert Curves to Arcs**> button and click **<OK>** as shown below:



5 Click anywhere in the CAD Face Window to deselect the geometry you have transferred. Your CAD Face Window should now appear as shown below:



6 Use the **CAD / CAM Switch** icon to switch back to CAM Mode.

Open Tools, Cycles and Material Files

If you are using **PartMaker SwissCAM**:

From the **File** menu choose **Open Tools File** and open

C:\PartMaker\PM-Swiss\ASM_Surface_Machining\

Tutorial_ASM\ASM_Tutorial.tdb

From the **File** menu choose **Open Cycles File** and open

C:\PartMaker\PM-Swiss\ASM_Surface_Machining\

Tutorial_ASM\ASM_Tutorial.cdb

From the **File** menu choose **Open Material File** and open

C:\PartMaker\PM-Swiss\Material\Alu_allw.mdb

If you are using **PartMaker Turn-Mill**:

From the **File** menu choose **Open Tools File** and open

C:\PartMaker\pm-tm\ASM_Surface_Machining\

Tutorial_ASM\ASM_Tutorial.tdb

From the **File** menu choose **Open Cycles File** and open

C:\PartMaker\PM-TM\ASM_Surface_Machining\

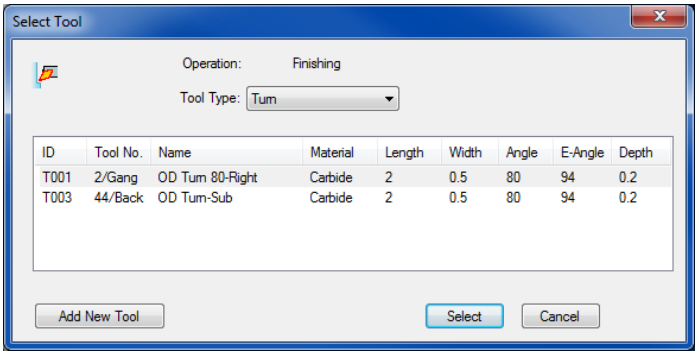
Tutorial_ASM\ASM_Tutorial.cdb

From the **File** menu choose **Open Material File** and open

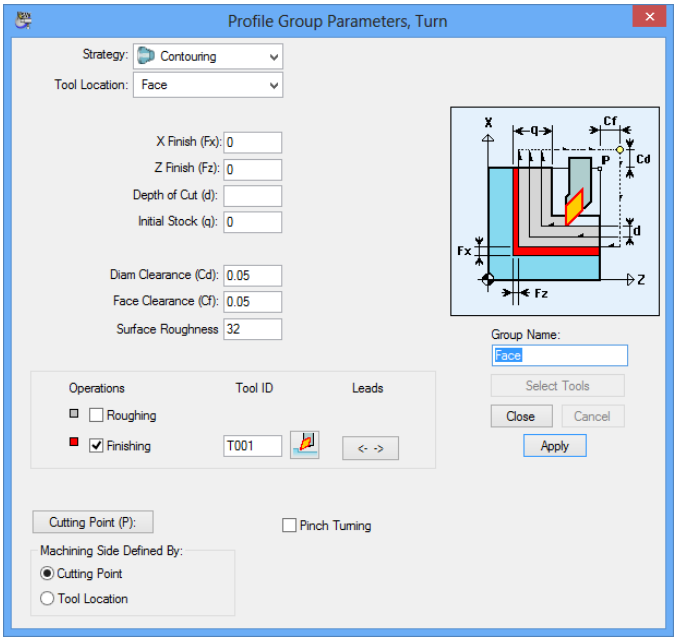
C:\PartMaker\PM-TM\Material\Alu_allw.mdb

Creating a Facing Operation

1. Choose New Profile Group from the Part Features menu.
2. Leave the **Strategy** selection as **Contouring**
3. From the **Tool Location** drop down menu, choose **Face**
4. Uncheck the **Roughing** box so **Finishing** is checked.
5. Click the <**Select Tools**> button.
6. Choose **T001** from the **Select Tool** dialog and click **<OK>**



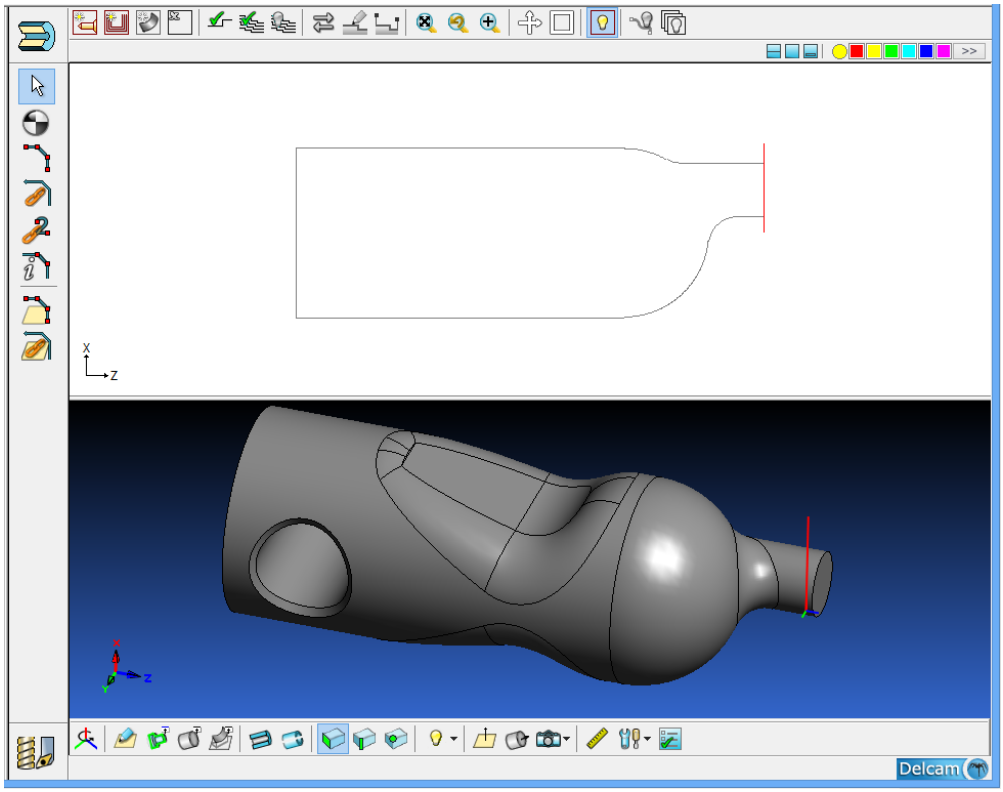
7. In the **Group Name** field, type "Face" and click the **<Apply>** button. Your **Profile Group Parameters** dialog should appear as shown below:



8. Click the **<Close>** button to return to the CAM Face Window.

Create the Profile for Facing

Because you are facing off the part, PartMaker will create the profile automatically. Your face window should appear as shown below:

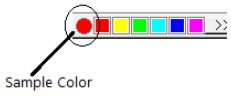


Creating the First OD Turning Operation

1 Choose a **New Color**.

To do so click a color square in the color bar at the top of the **Face Window** different from the Sample Color.

Important! *Do not select a color that is the same as the background color of your Face Window. Doing so will make your profile difficult to see.*

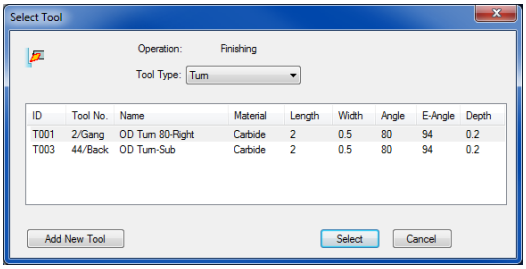


2 Choose New Profile Group from the Part Features menu.

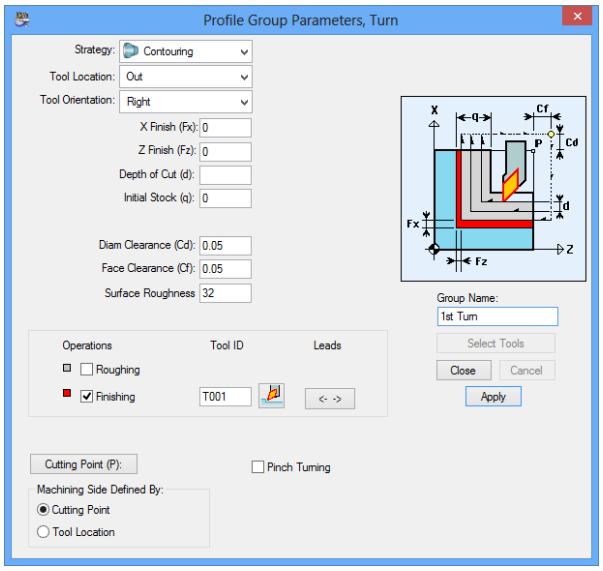
3 Uncheck the **Roughing** box and **Finishing** will automatically be selected.

4 Click the <**Select Tools**> button.

5 Choose **T001** from the **Select Tool** dialog and click **<OK>**



6 In the **Group Name** field, type "1st Turn" and click the **<Apply>** button. Your **Profile Group Parameters** dialog should appear as shown below:



7 Click the **<Close>** button to return to the "Front" Face Window.

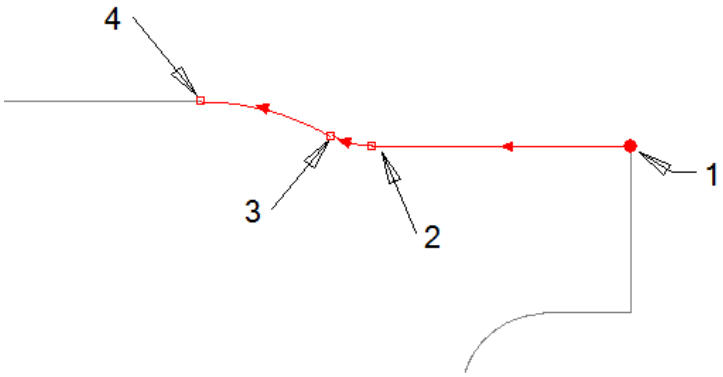
Create the Profile for OD Turning

Now that you have created the part feature, you are ready to assign it to your part geometry to create a tool path. To do so:

1 Choose the **Profile** icon from the graphics icons on the left side of the Face Window

2 Click the **Closest Intersection** Snap Mode from the Snap Mode icons at the top of the screen

Your cursor now appears as a small X. Click on positions 1, 2, 3 and 4 respectively as shown in the picture below to assign the tool path to the geometry.

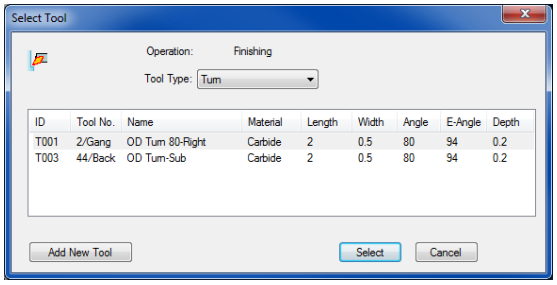


In this operation, you will only turn just past the first radius so there is adequate support for the milling operations. In the next operation, you will turn the rest of the part

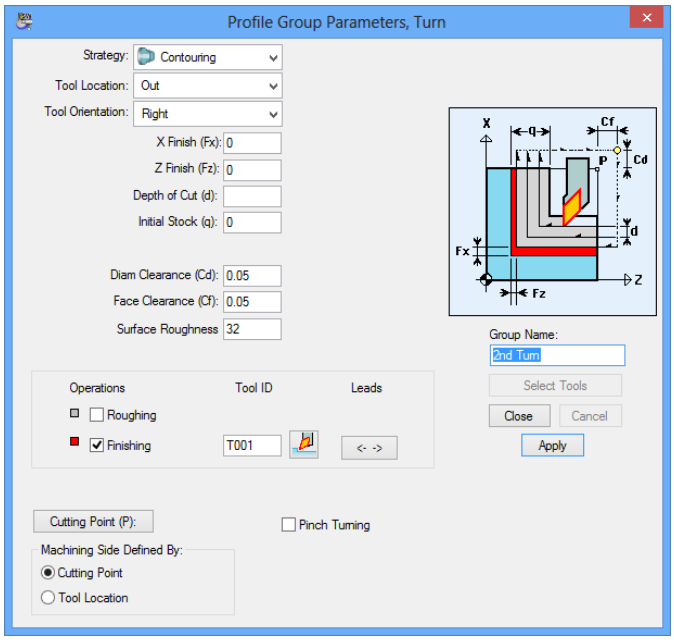
3 Click the **Selection** icon.

Creating the Second OD Turning Operation

1. Choose a **New Color**.
2. Choose New Profile Group from the Part Features menu.
3. Uncheck the **Roughing** box so **Finishing** is checked.
4. Click the <**Select Tools**> button.
5. Choose **T001** from the **Select Tool** dialog and click **<OK>**



6. In the **Group Name** field, type "2nd Turn" and click the **<Apply>** button. Your **Profile Group Parameters** dialog should appear as shown below:



7. Click the **<Close>** button

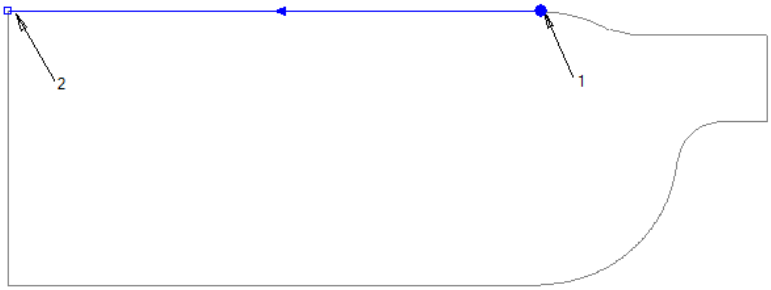
Create the Profile for OD Turning

Now that you have created the part feature, you are ready to assign it to your part geometry to create a tool path. To do so:

1 Choose the **Profile** icon from the graphics icons on the left side of the Face Window

2 Click the **Closest Intersection** Snap Mode from the Snap Mode icons at the top of the screen

3 You cursor now appears as a small X. Click on positions 1 and 2 respectively as shown in the picture below to assign the tool path to the geometry.

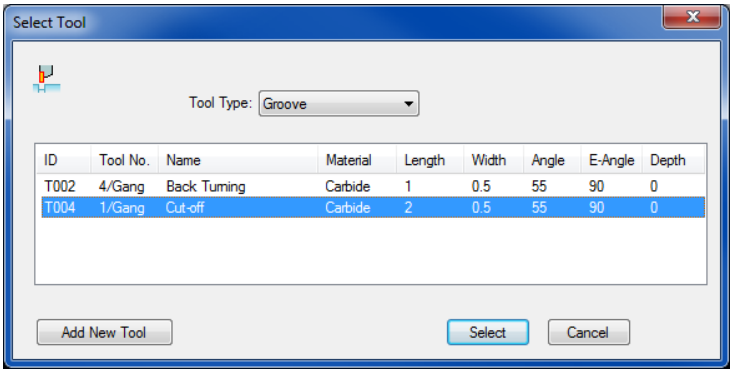


4 Click the **Selection** icon to deselect the profile.

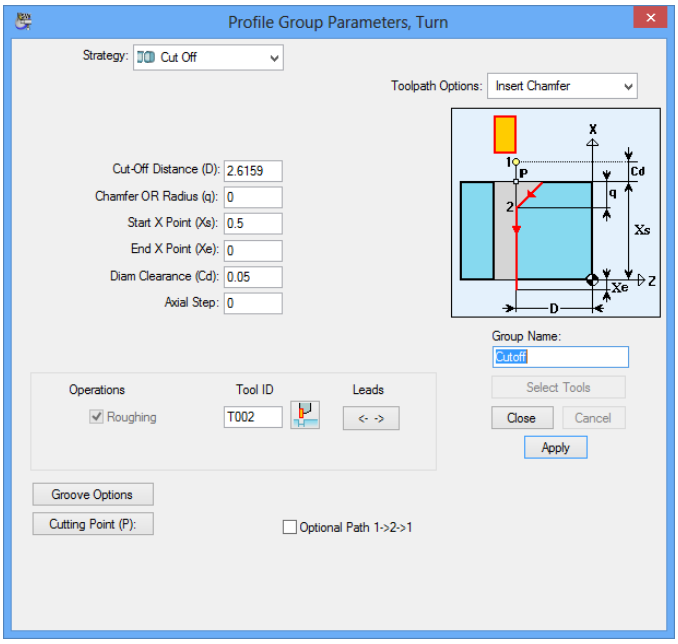
Creating a Cutoff Work Group

1. Choose a **New Color**.

2. Choose New Profile Group from the Part Features menu.
3. From the **Strategy** drop menu, choose **Cut-Off**.
4. Click the <**Select Tools**> button.
5. Choose **T004** from the **Select Tools** dialog and click **<OK>**



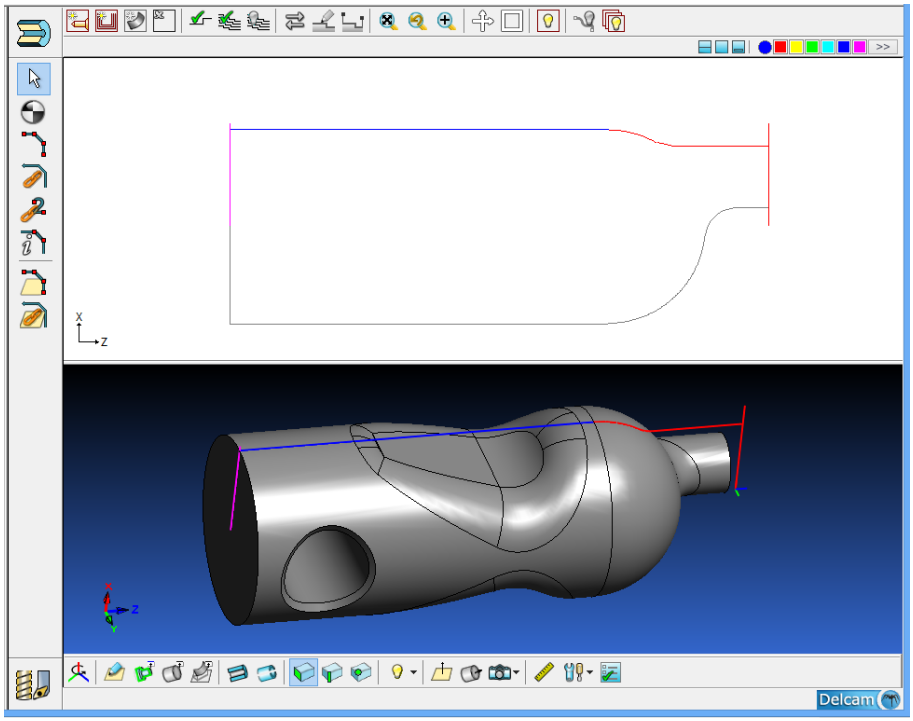
6. In the **Cycle** Name field, type “Cut-Off” and click the **<Apply>** button. Your **Profile Group Parameters dialog** should appear as shown below:



7. Click the **<Close>** button to close the **Profile Group Parameters** dialog.

Create the Profile for the Cut-Off Operation

Because you are cutting off the part, PartMaker will create the profile automatically. Your face window should appear as shown below:



Save Your Work

It is always a good idea to save your at various points in the part creation process in PartMaker. To do so:

1. From the **File** menu choose **Save** or press **<CTRL + S>** on your keyboard
2. Enter the name of your job. Here, save your job file as **asmpractice**
3. Click the **<Save>** button.

Creating Surface Projection Tool Paths on the Face of the Part

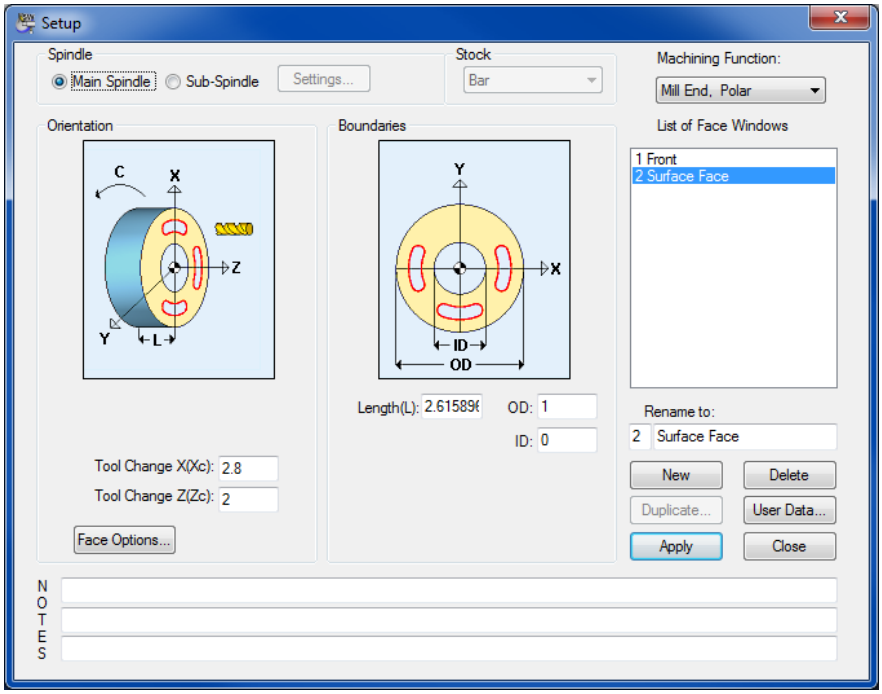
In this section of the tutorial you will create, bound and verify surfacing tool paths for projection and finish machining on the face of the imported solid model using ASM. You will use this projection to rough away material for the finishing operation that will be made in the next section.

***Important!** Before moving on, make sure your Solid Model is set to Full View. If your Solid Model currently appears in a cross sectional view, click on it with your right mouse button and choose **Full View** from the drop down menu or select the **Full View** button on the Solids tool bar.*

Create a New Face Window for Surfacing on the Face

The first step in creating a surfaced feature in PartMaker Turn-Mill or SwissCAM is to create a new Face Window. In this case, you will use the Mill End Polar Face Window so you can use
X, Z and C tool motion on the face of the part. To create the Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the **<New>** button to create a new Face Window which appear with the name "**Untitled 2**"
3. From the **Machining Function** drop down menu, choose **Mill End Polar**.
4. In the **Rename To:** field, type "**Surface Face**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:

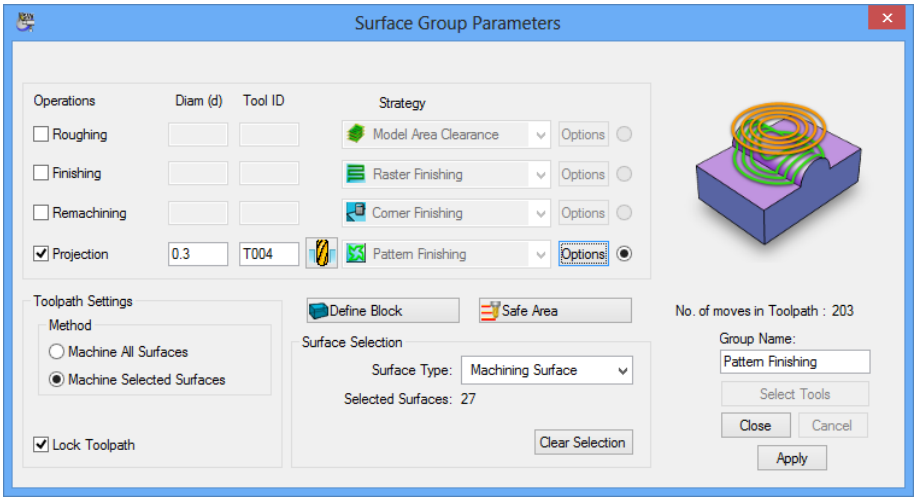


5. Click the **<Close>** button to close the **Setup** dialog.

Create a New Surface Group for Projection Machining

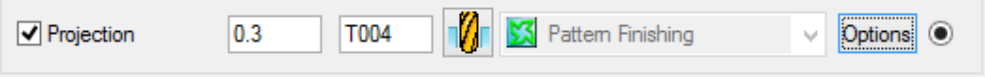
In this section of the tutorial you will create, bound and verify a tool path for projection machining on the face of the imported solid model. Projection machining allows you to "project" a pattern curve onto a 3D surface. In this example, the projection tool path will be used to "rough" away material for the finishing tool paths to be created in the next section.

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Check the **Projection** check box.
3. Uncheck the **Roughing** check box.
4. Enter a tool diameter of 0.3 in (7.6 mm) in the **Diam (d)** field and click the **<Select Tools>** button.
5. In the **Group Name** field enter "**Pattern Finishing**" and click **<Apply>**. The dialog should now appear as shown below:

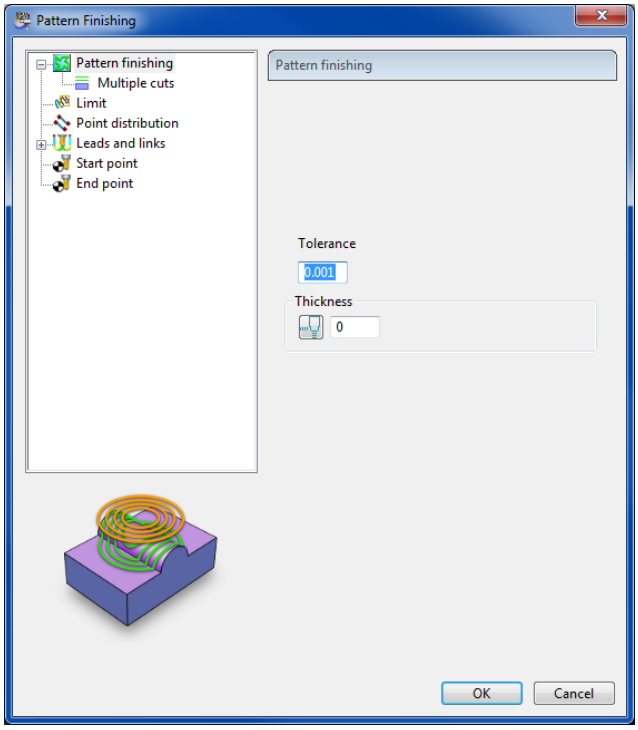


Establishing the Tool's Approach to the Stock

When machining this projection feature, it will be necessary to have the tool correctly approach the part to avoid a collision between the tool and the stock. To do so, click the <**Options>** button in the **Surface Group Parameters** dialog which will open the dialog for the Pattern Finishing strategy as shown below in the picture.

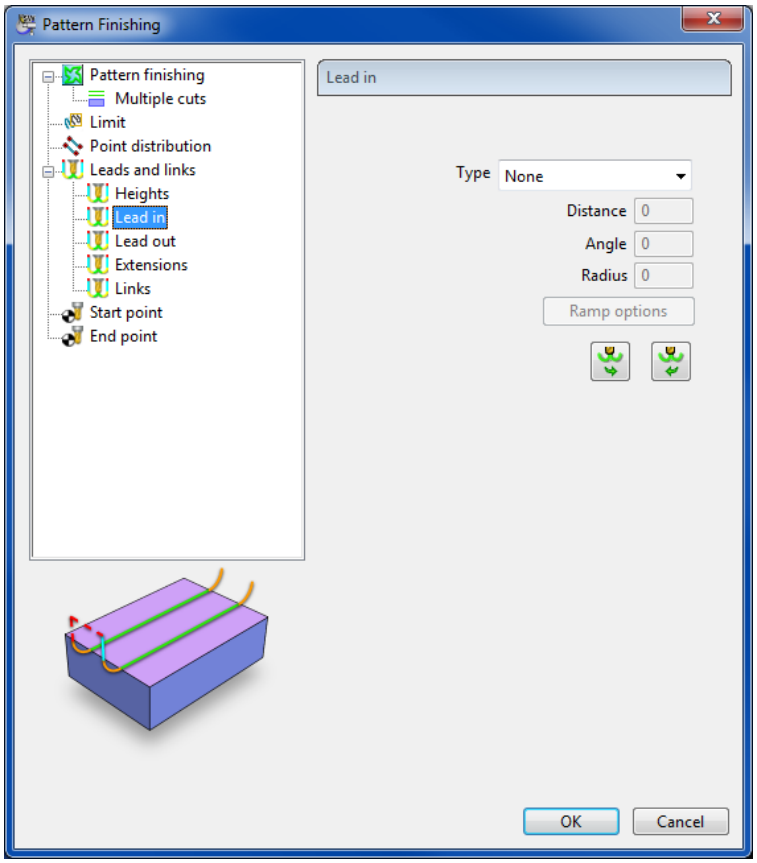


This will open the Pattern Finishing strategy dialog

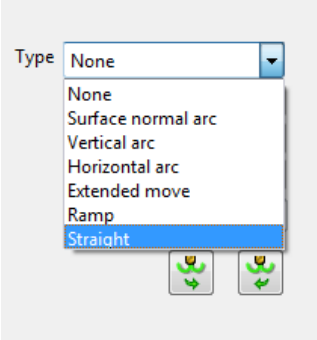


1 Click the **Leads and Links** tab in the tree on the left in the **Pattern finishing** dialog to expand the **Lead and Links** features.

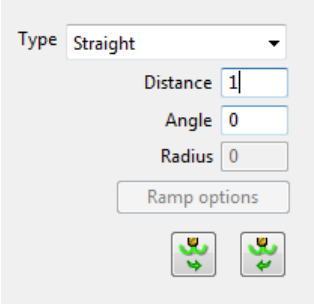
2 Click on **Lead in** the tree to see the lead-in options.



3 Set Type to Straight



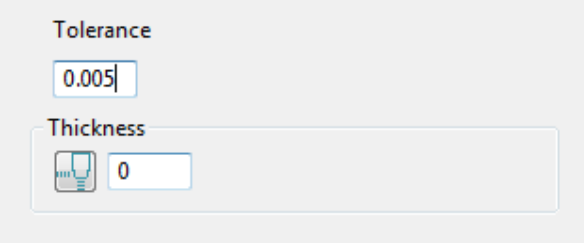
4 The **Distance** field would now be enabled. Enter a value of 1 (25 mm) in the distance field.



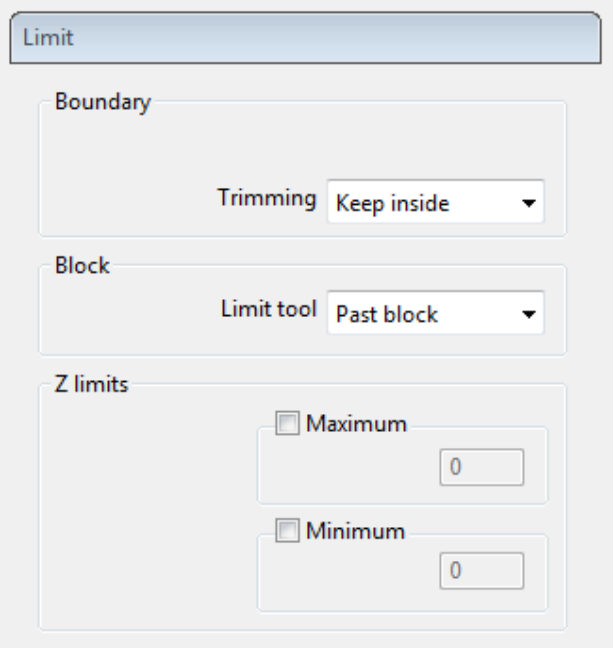
5 Click on  to copy the same parameters from **Lead In** to **Lead Out.**

General Pattern Finishing Settings

1 In the Pattern finishing dialog, on the main page enter the Tolerance value of 0.005 (0.125 for metric) as shown below



2 In the Pattern Finishing dialog click on Limit tab to see the toolpath limit parameters as shown below.



3 Make sure the **Limit Tool** option is set to Past block to allow the tool to move outside the block in order to follow the pattern.

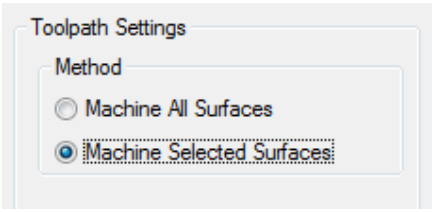
4 Click the **<OK>** button to close the **Pattern Finishing** strategy dialog and return to the Surface Group Parameters dialog

Select the Surfaces for Machining

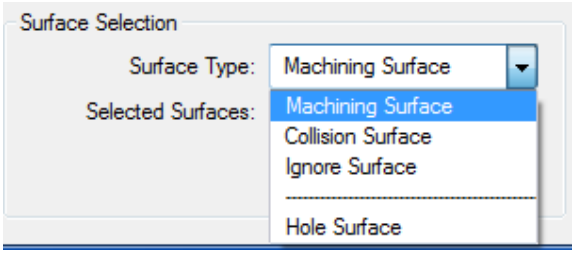
At this point you should still have the **Surface Group Parameters** dialog open. In the lower left hand corner of this dialog there is a framed section called **Toolpath Settings**. In this section you will control how you select surfaces to be machined by the current operation. There are two choices: **Machine All Surfaces** and **Machine Selected Surfaces**. By default **Machine All Surfaces** will be selected which will select all surfaces of the imported solid model as Machining Surfaces.

In this case, you will select only the surfaces you wish to machine, i.e. you will select the surfaces manually. To do so:

1. Click the **Machine Selected Surfaces** radio button in the **Toolpath Settings** section of the **Surface Group Parameters** dialog as shown below:

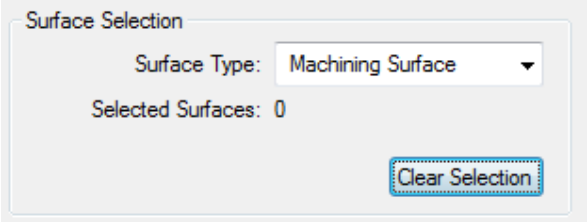


2. The type of the surface you would be selecting can be changed from the **Surface Type** combo box in the **Surface Selection** section as shown in figure below:

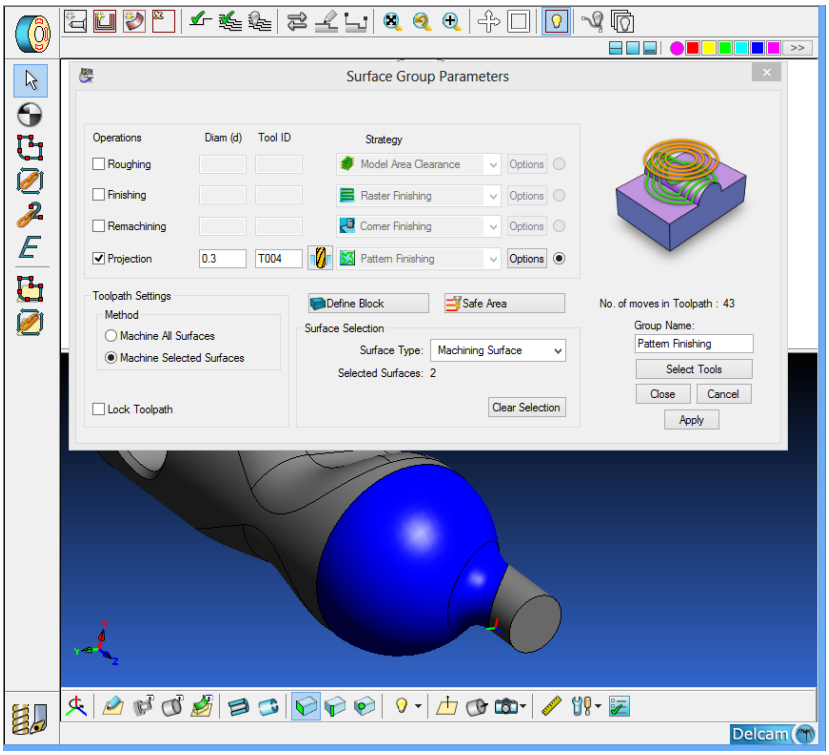


3. For current operation you need to select **Machining Surface** which would be selected by default.

4. Click on the **Clear Selection** button in the **Surface Selection** section to clear the selected surfaces from the solid model. As you click on the **Clear Selection** button, the **Selected Surfaces** will show the current number of surfaces selected for the current surface type selected in the **Surface Type** combo box as shown below:



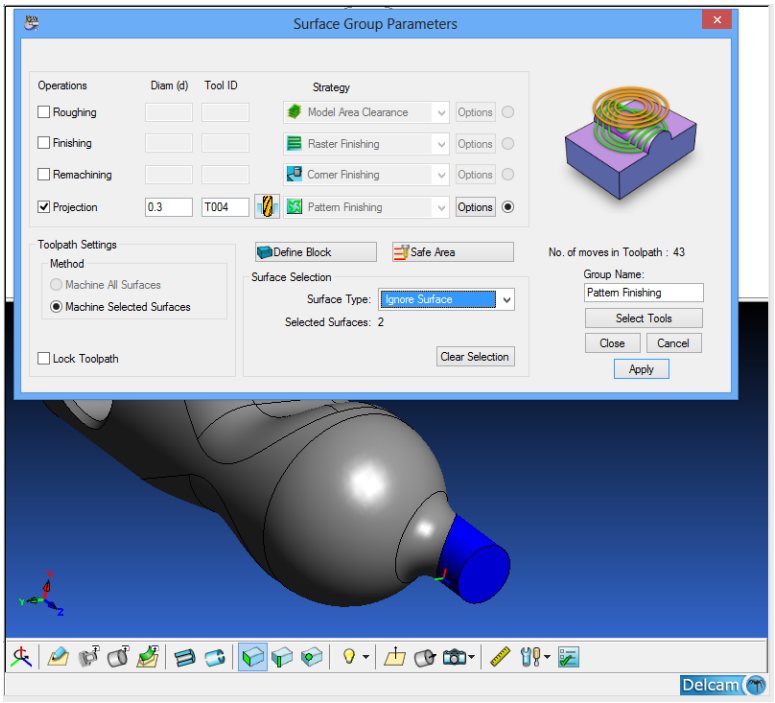
5 In the **Solid Model File** window, click on the base of the boss and the surface adjacent to it as indicated in the picture below:



The **Selected Surfaces** text in the **Surface Group Parameters** dialog should display the value **2**.

6 Now select ignore surfaces of the model so that the toolpath can follow the pattern without detecting gouging with the solid model. Select **Ignore Surfaces** in **Surface Type** combo box.

Select the surfaces as shown below as ignore surfaces



7 Click the **<Close>** button to return to the CAM Face Window.

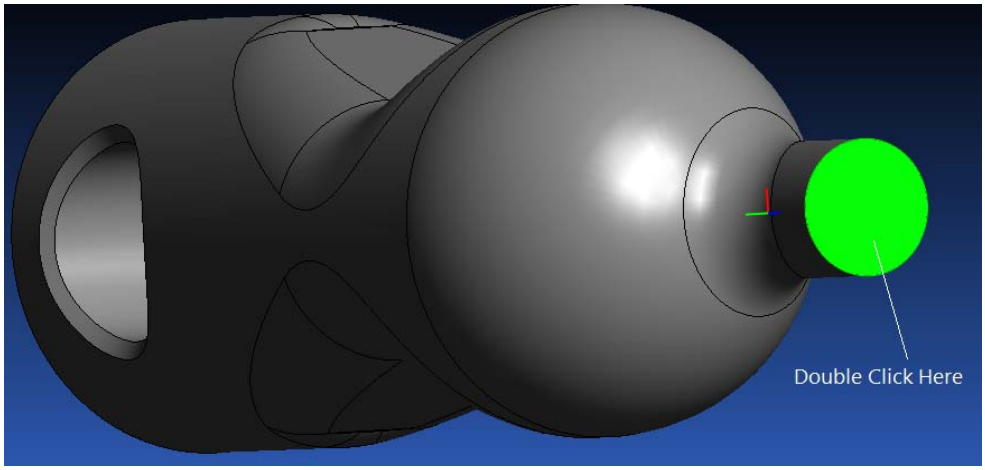
Create Surface Machining Pattern Curve

Pattern curves are created the same way as 2D profile curves. Pattern curves are 2D/3D profiles that are projected onto a 3D surface in order to calculate a tool path.

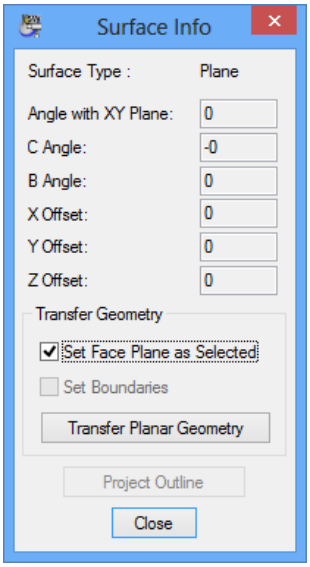
To create a pattern curve for the feature defined in this section:

1 Click the CAD/CAM switch to enter the CAD mode.

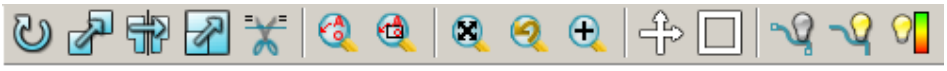
2 Double click on the face of the solid model as indicated below to open the **Surface Info** dialog.



3 Click the <**Transfer Planar Geometry** > button in the **Surface Info** dialog shown below to transfer the boss's profile into the CAD Face Window.



The circle you have just transferred should now be highlighted indicating that it is selected. If it is not highlighted, select it by clicking on it so that it is highlighted.



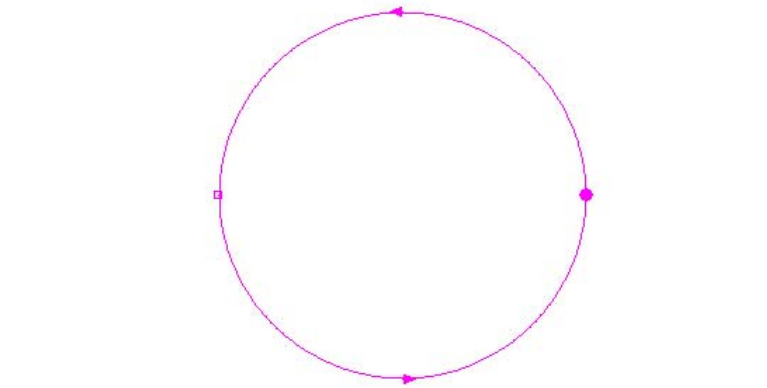
4 Click anywhere in the Face Window to deselect the geometry.

5 Switch to the **CAM** mode.

6 Choose the **Chain Geometry** icon

7 Click your cursor at the 9 o'clock position on the circle as indicated below to apply the pattern curve to the geometry.

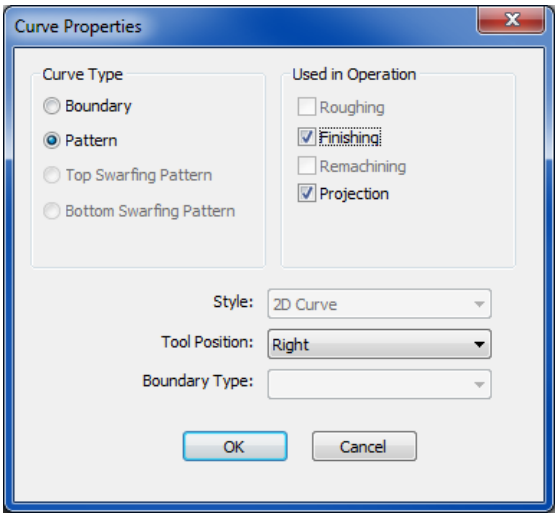
Your Face Window should now appear as shown below:



8 Double click on the pattern curve using the **Selection** icon to display the **Curve Properties** dialog below.

9 The direction of the arrows on the projection curve will dictate the **Tool Position** you choose. If the arrows are pointing in a counter clockwise manner, choose **Right** from the **Tool Position** drop down menu. If the arrows are pointing in a clockwise manner, choose **Left** from the **Tool Position** drop down menu.

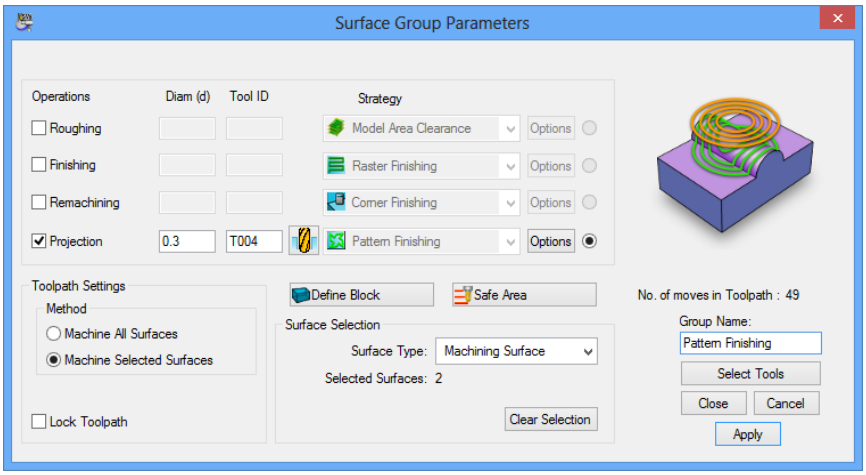
In the picture above, the directional arrows are pointing counter clockwise, hence **Tool Position Right** has been chosen as shown below:



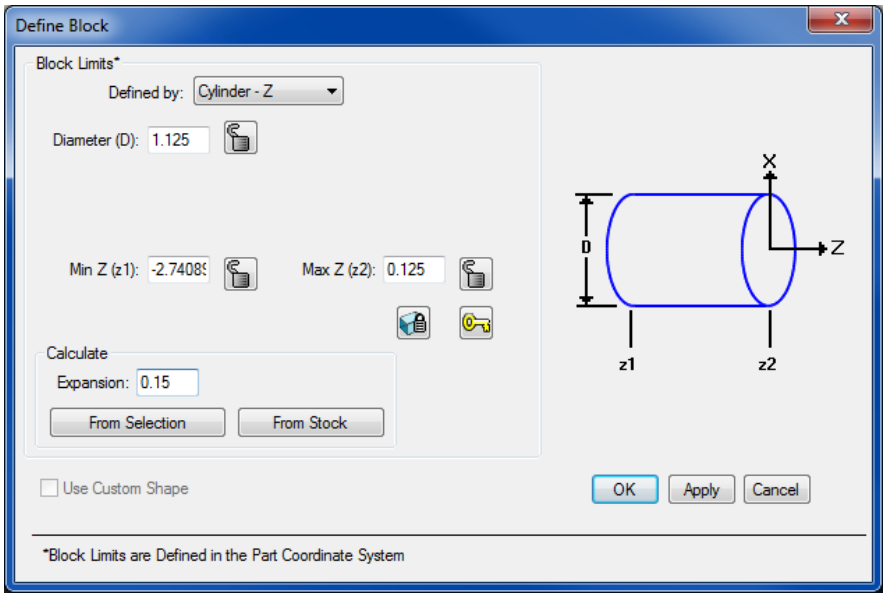
10 Click the **<OK>** button to return to the CAM Face Window.

Change Block Size to Contain the Pattern

1. As we will be machining to the Right of the defined Pattern, we need to make sure that the Block size for the ASM group would contain the tool if it moves to the right of the Pattern.
2. To increase the block size, open the **Surface Group Parameters** dialog by double clicking on the group in the **Job Explorer.**
3. The dialog as shown below will open. Click on **Define Block** to open the Block dialog

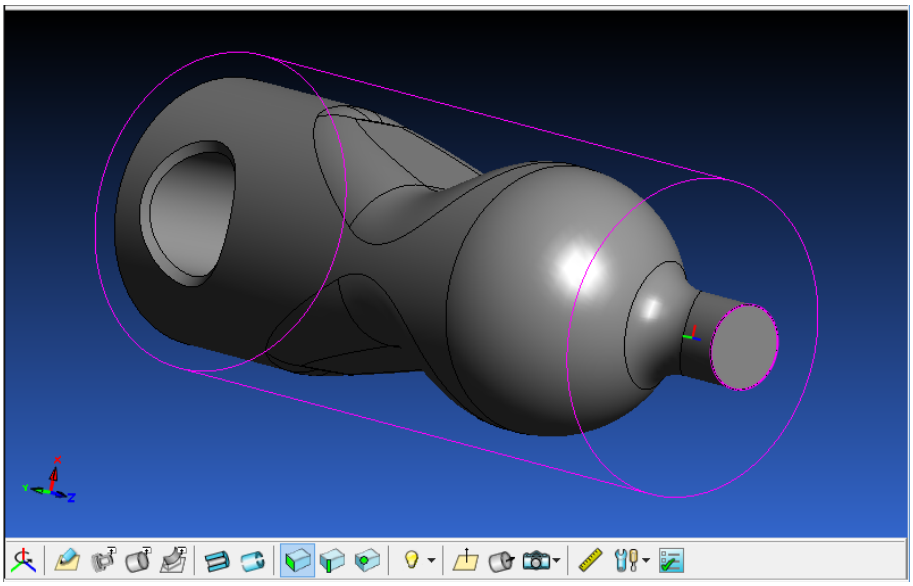


4. The block dialog is as shown below. Enter the value in **Expansion** field to increase the block in all directions by the value. The expansion value in this case would be half the diameter of the tool being used i.e. 0.3/2 = 0.15. Your **Define Block** dialog should look as below:



5. Now click on <**From Stock**> button and then click the **<Apply>** button to apply the value defined in the **Expansion** field to the block and click the.

6 You would be able to see the increase in the block size as shown below in the Solids Window. You can see the block limits by choosing **Show Block Limits** from the **Part Features** menu.



You can hide the block limits by choosing **Hide Block Limits** from the **Job Optimizer** menu

7 Now click <**OK**> in the **Define Block** dialog to return to **Surface Group Dialog** and click on <**Close**> to close that dialog.

Verify the Tool Path

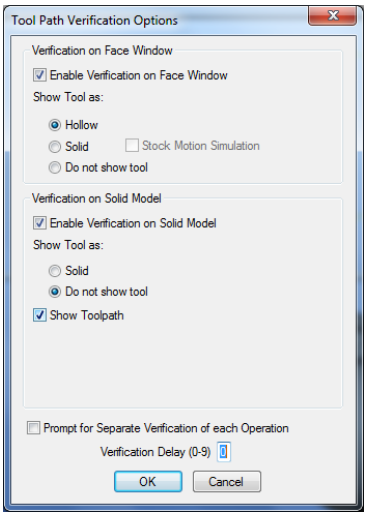
Once you have created the **Part Feature** and established a **Boundary Curve** you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Part Features menu choose Verify Work Group Tool Path.

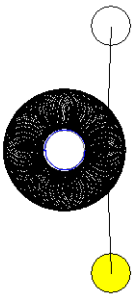
2 In the **Tool Path Verification Options** click the **Do not show tool** radio button as shown below.

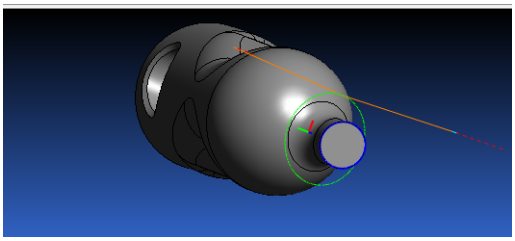
3 Enter a Verification Delay of 0.

Note: *When entering a* ***Verification Delay*** *of greater than 0, you will see the 3D representation of the cutting tool moving along the Solid Model. When* ***Verification Delay*** *is set to 0, you will only see the path of the tool on the Solid Model.*



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:





Note: *Notice above, the line that extends into the programmed tool path. This line represents the 1-inch* ***Lead In & Lead Out*** *set in the* ***Leads and Links*** *property page. Creating this line assures that the tool will safely approach the stock without risking a tool collision.*

5 From the Part Features menu choose Hide Work Group Tool Path.

Creating Surface Finishing Tool Paths on the Face of the Part

In this section of the tutorial you will create, bound and verify surfacing tool paths for finish machining on the face of the imported solid model. The finishing operations will be created in the same Face Window as the projection features created in the previous. As a result, it is not necessary to create a new Face Window.

Create a New Surface Group for Finishing

In this section of the tutorial you will create, bound and verify a tool path for finish machining on the face of the imported solid model.

1 Choose a **New Color**.

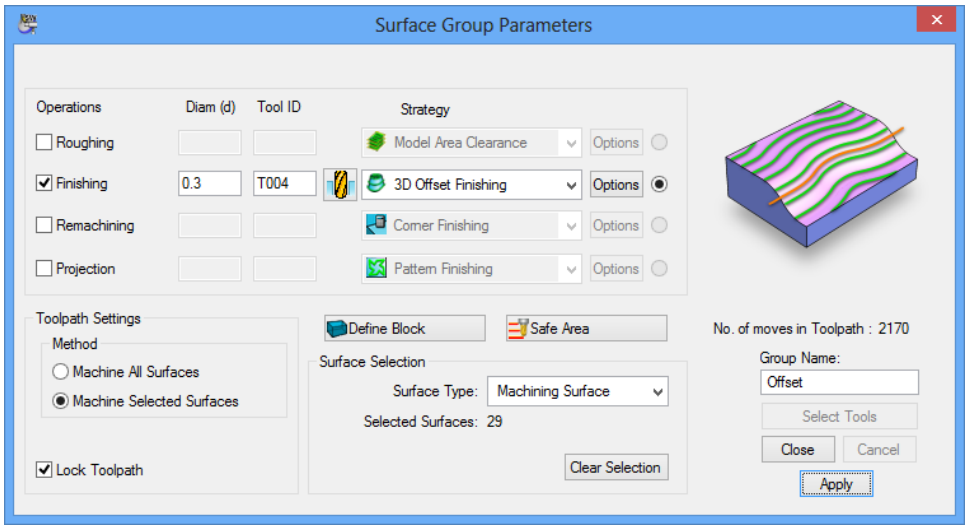
2 Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.

3 Uncheck the **Roughing** radio button and make sure **Finishing** is checked.

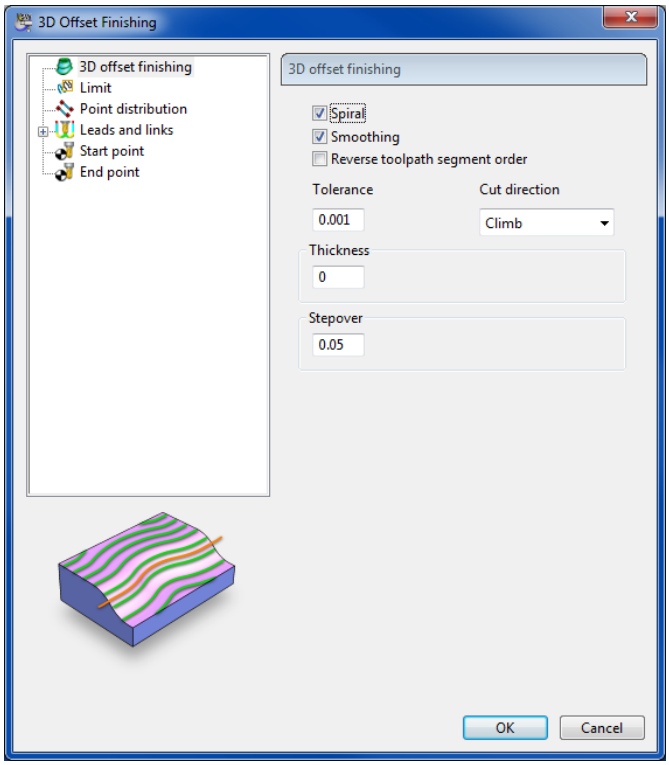
4 Enter a tool diameter of 0.3 (7.6) in the **Diam (d)** field and click the **<Select Tools>** button.

5 From the **Strategy** drop down menu choose **3D Offset Finishing**.

6 In the **Group Name** field enter "**Offset**" and click **<Apply>**. Your completed dialog should appear as shown below:

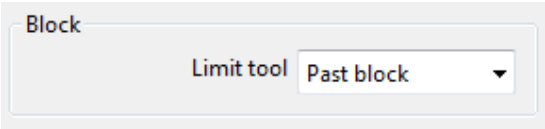


7 Click the **<Options>** button to display **3D Offset Finishing** dialog as shown below:



8 Enter 0.05 (1.25 MM) in the **Stepover** field.

9 Go to **Limit** tab and change **Limit tool** to **Past Block** in **Block** section to allow the tool outside the block



10 Click the <**OK**> button to return to the **Surface Group Parameters** dialog.

Select Surfaces for Machining

At this point you should still have the **Surface Group Parameters** dialog open. Make sure **Machine All Surfaces** is selected in the **Toolpath Settings** section which sets all surfaces of the solid to machine surfaces.

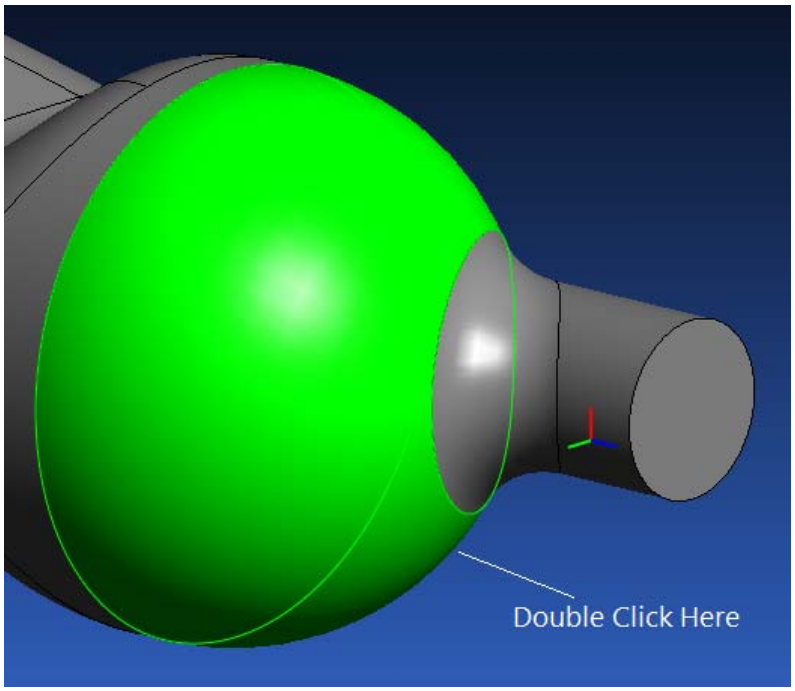
Click the **<Close>** button to close the **Surface Group Parameters** dialog.

Create Boundary Curves

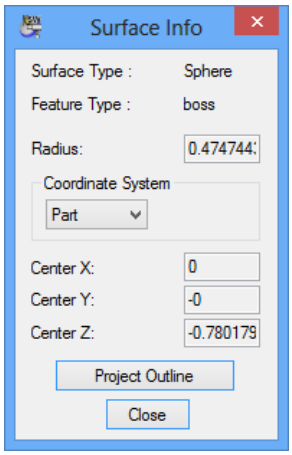
Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict the machining area. In this example, you will create two boundary curves.

To create the boundary curves for this feature:

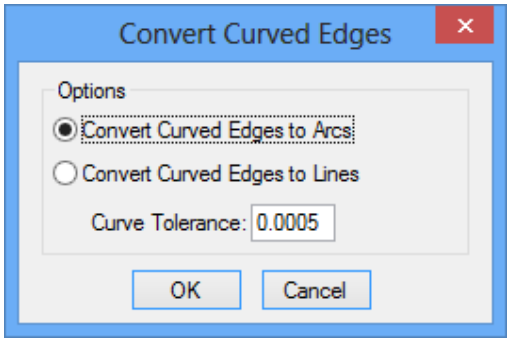
1. Click the CAD/CAM switch to enter the CAD mode.
2. Double click on the face of the solid model as indicated below to open the **Surface Info** dialog.



3 Click the <**Project Outline** > button in the **Surface Info** dialog shown below to transfer the boss's profile into the CAD Face Window.



4 Click the Convert Curved Edges to Arcs radio button

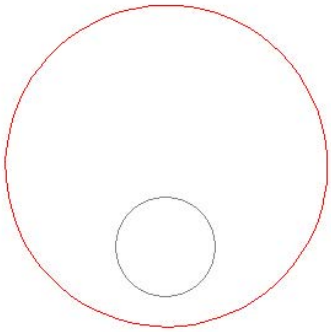


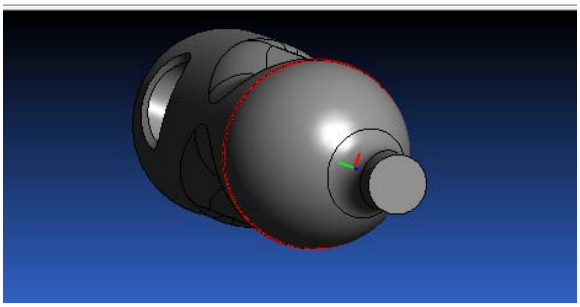
5 Click anywhere in the Face Window to deselect the geometry.

6 Switch to the **CAM** mode.

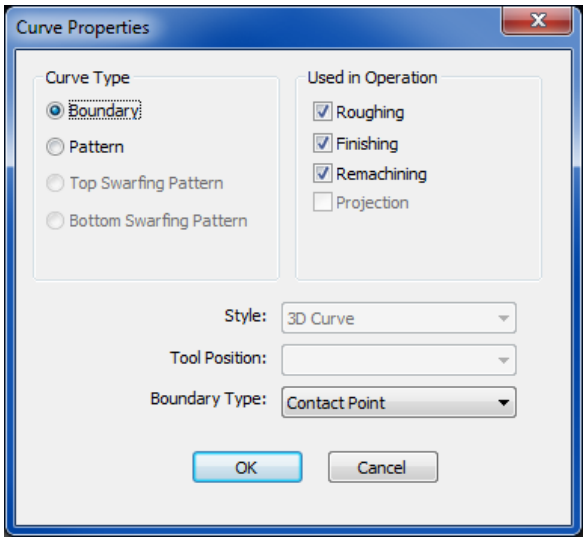
7 Choose the **Chain Geometry** icon

8 Click your cursor on the outer profile just created as shown in the picture below to create a boundary curve for the toolpath.





9 Double click on the selected curve to open the **Curve Properties** dialog. Select the **Boundary Type** as **Contact Point** as shown in figure below. For more information on boundary types, please refer help.



10 Click the **<OK>** button to close the **Curve Properties** dialog.

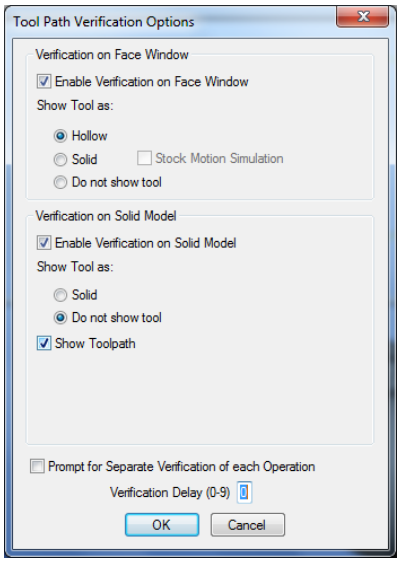
Verify the Tool Path

Once you have created the **Part Feature** and established a **Boundary Curve** you can visually verify the tool path PartMaker has calculated. To do so:

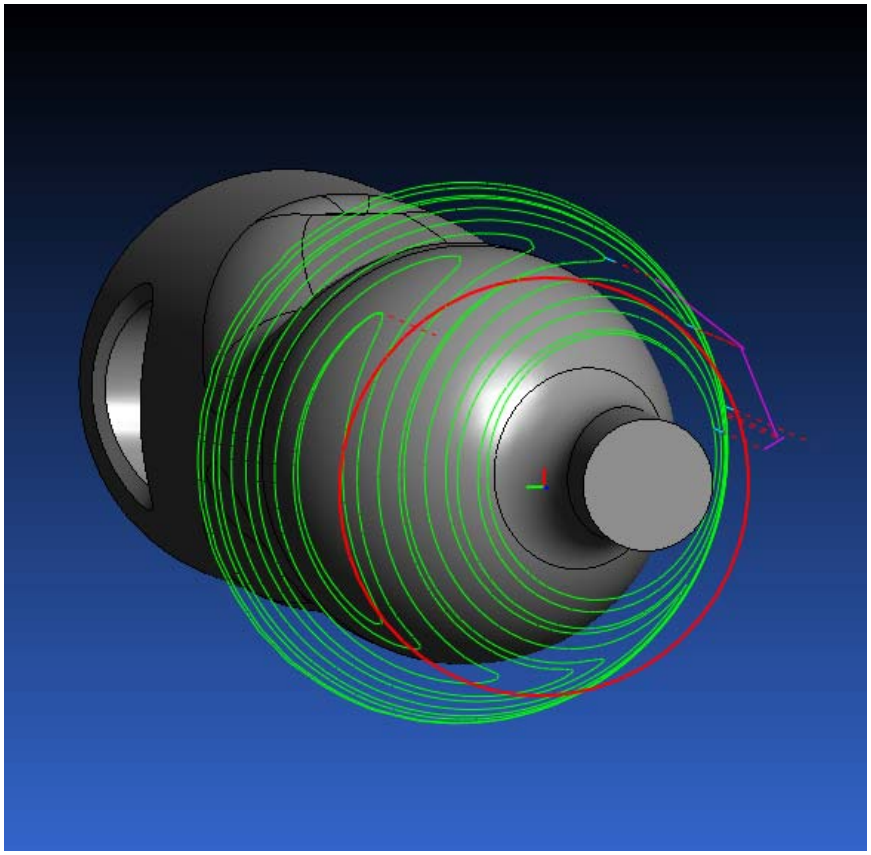
1 From the Part Features menu choose Verify Work Group Tool Path.

2 In the **Tool Path Verification Options** click the **Do not show tool** radio button as shown below.

3 Enter a Verification Delay of 0.



4 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



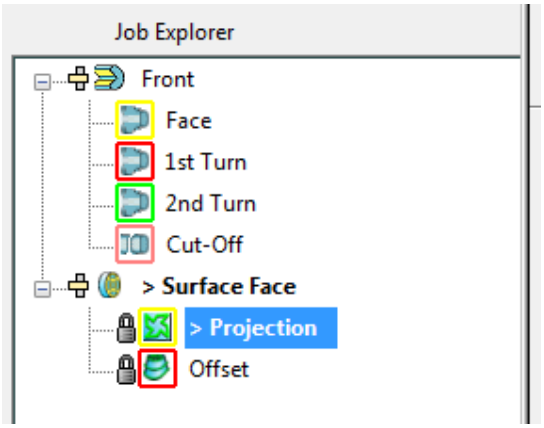
5 From the Part Features menu choose Hide Work Group Tool Path.

Note: *You should always hide your tool path after verifying it.*

Note*: Whenever creating a surfacing tool path in ASM, it is always a good idea to "lock" the tool path once you are satisfied with result to avoid it being regenerated later. To "lock" a tool path, simply double click on the feature in the Job Explorer tree and click the Lock Toolpath button in Surface Group Parameters dialog.*

Once a tool path has been "locked" it will appear with a small lock next to it on the Feature Tree.

As an exercise, go back and "lock" the first projection tool path you made. Your Job Explorer tree should then appear as shown below:



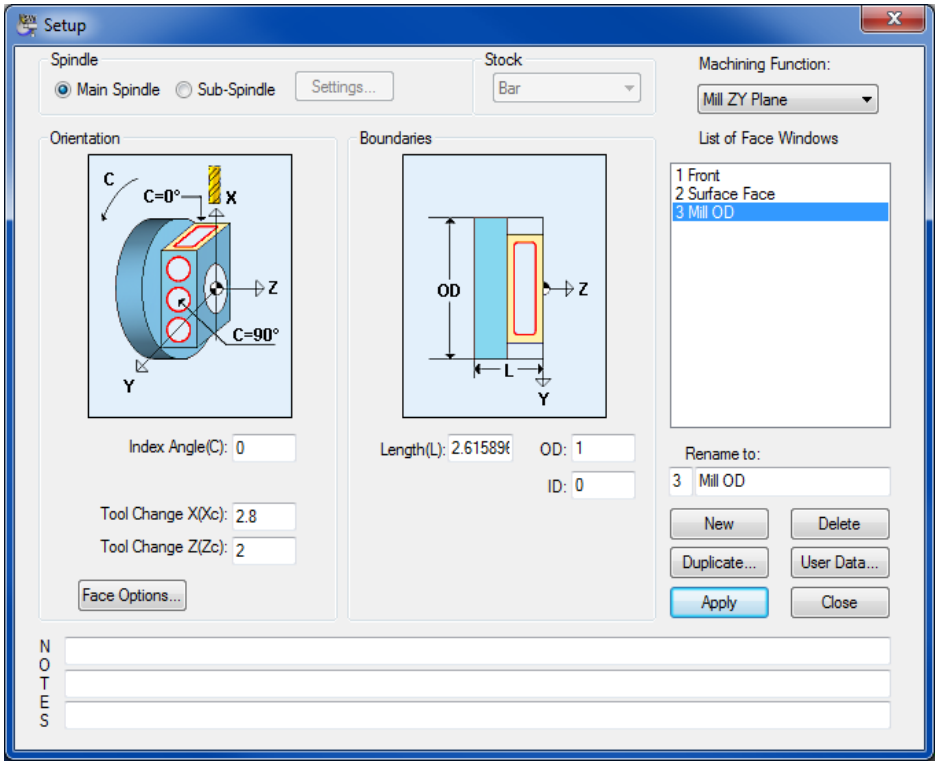
Creating Surface Finishing Tool Paths on the Diameter of the Part

In this section of the tutorial you will create, bound and verify a tool path for finishing operations on the outside diameter of the imported solid model.

Create a New Face Window for Finish Machining on the Diameter

The first step in creating the finish machining feature on the diameter is to create an appropriate Face Window. In this case, you will use the Mill ZY Face Window to use a vertically oriented tool on the diameter of the part. To create the Face Window:

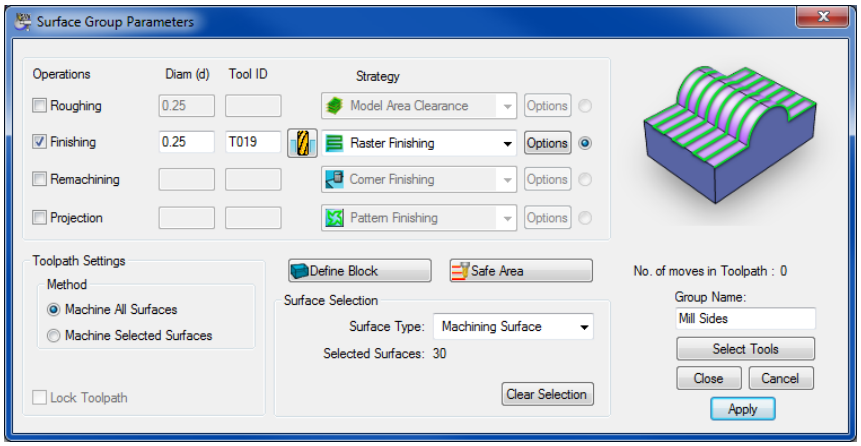
1. Choose Setup from the View menu
2. In the **Setup** dialog, click the **New** button to create a new Face Window which appear with the name "Untitled 3"
3. From the **Machining Function** drop down menu, choose Mill ZY plane.
4. In the Rename To: field, type "**Mill OD**" and click **<Apply>**. Your completed **Setup** dialog should appear as shown below:



5. Click **<Close>** to close the **Setup** dialog.

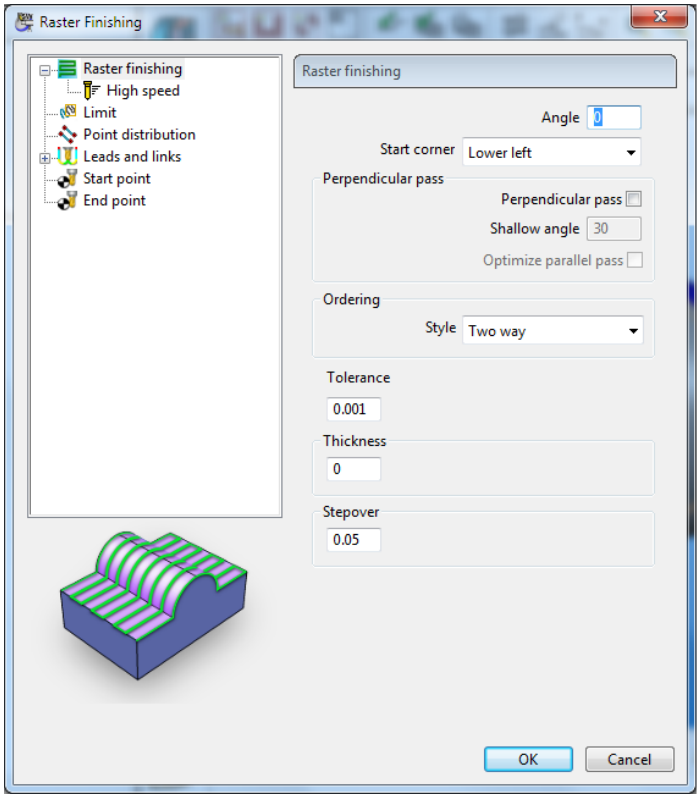
Create a New Surface Group for Finishing

1. Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.
2. Uncheck the **Roughing** radio button and make sure **Finishing** is checked.
3. Enter a tool diameter of 0.25 (6.0) in the **Diam (d)** field and click the **<Select Tools>** button.
4. From the **Strategy** drop down menu choose **Raster Finishing**.
5. In the **Group Name** field enter "**Mill Sides**" and click **<Apply>**. Your completed dialog should appear as shown below:



6. Click the active **<Options>** button to display **Raster Finishing** dialog.
7. Enter 0.05 (.50) in the **Stepover** field.
8. Select the Ordering Style as Two Way.

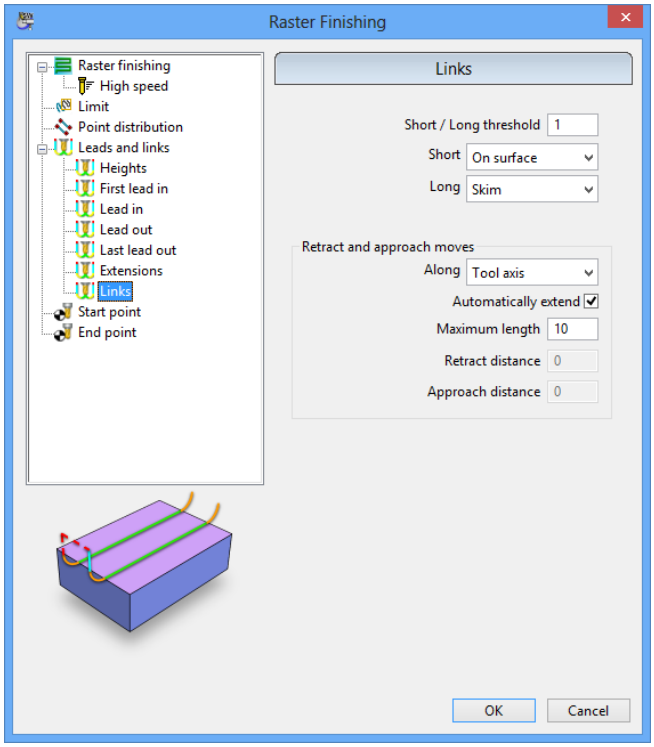
Your completed **Raster Finishing** dialog should appear as shown below:



9 Click on the **Limit** tab and select **Limit Tool** under **Block** section to **Past Block**

10 Click on the **Leads and Links** tab and select **Links** from the tree.

11 Choose the **Short** link to be **On Surface** as shown in figure below:



12 Click <**OK**> to close the **Raster Finishing** dialog.

Select Surfaces for Machining

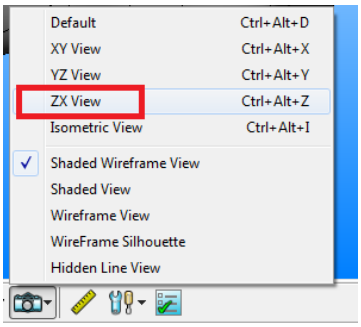
PartMaker's ASM makes it very easy to select just the surfaces you wish to machine. To do so:

1 Click the Machine Selected Surfaces radio button in the Surface Group Parameters dialog

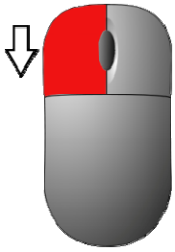
2 Click the **<Clear Selection>** button to clear the currently selected surfaces

3 Move the **Surface Group Parameters** dialog out of the way and position the solid so you can see the surfaces you wish to select

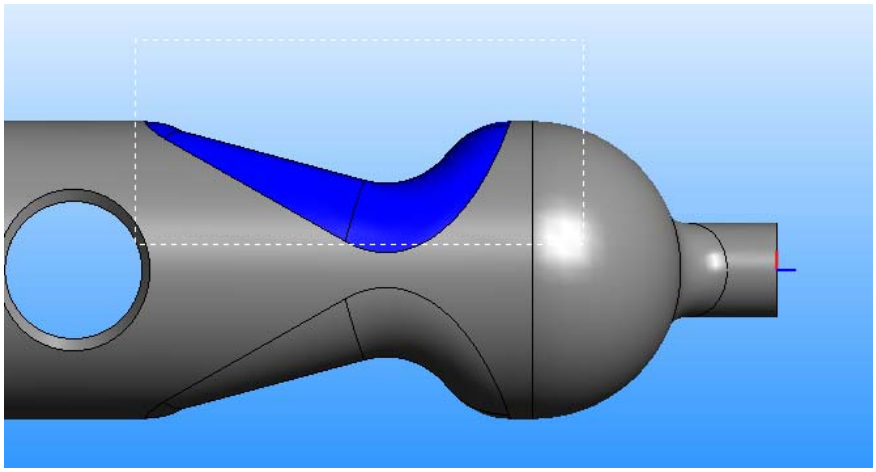
4 To assure you select the correct surfaces, click the **Set View** icon from the solids icon toolbar at the bottom of the screen and choose the **ZX View** as shown below:



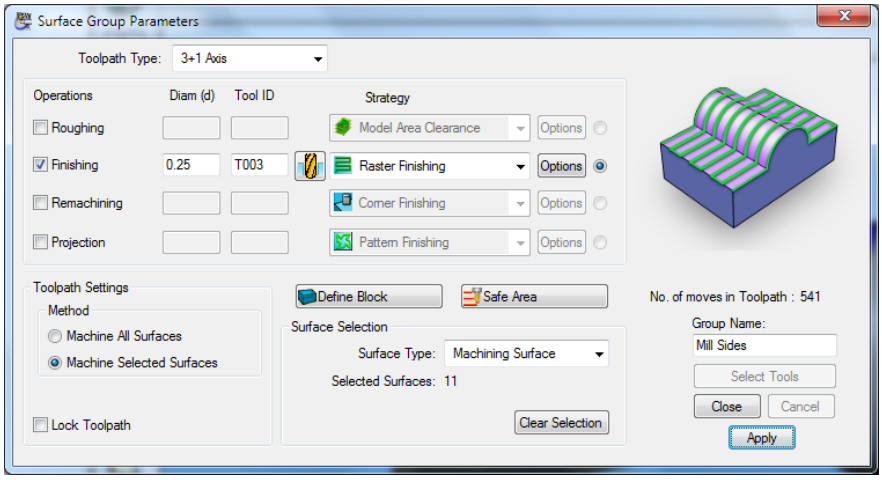
5 Hold the **<Shift>** key down on your keyboard while depressing the left mouse button



and drag a box from the upper right side of the surfaces to the lower left side of the surfaces and they will be highlighted blue as shown below:



6 Click the **<Apply>** button and you will see the exact number of surface you have selected. You should have 11 surfaces selected. Your **Surface Group Parameters** dialog should appear as shown below:



Note: *If you have more than 11 surfaces selected, you have probably selected more surfaces than you wanted. You can tell a surface is selected because it is highlighted in blue. You can easily deselect a surface by just clicking the extra surface.*

7 Click the **<Close>** to close the **Surface Group Parameters** dialog.

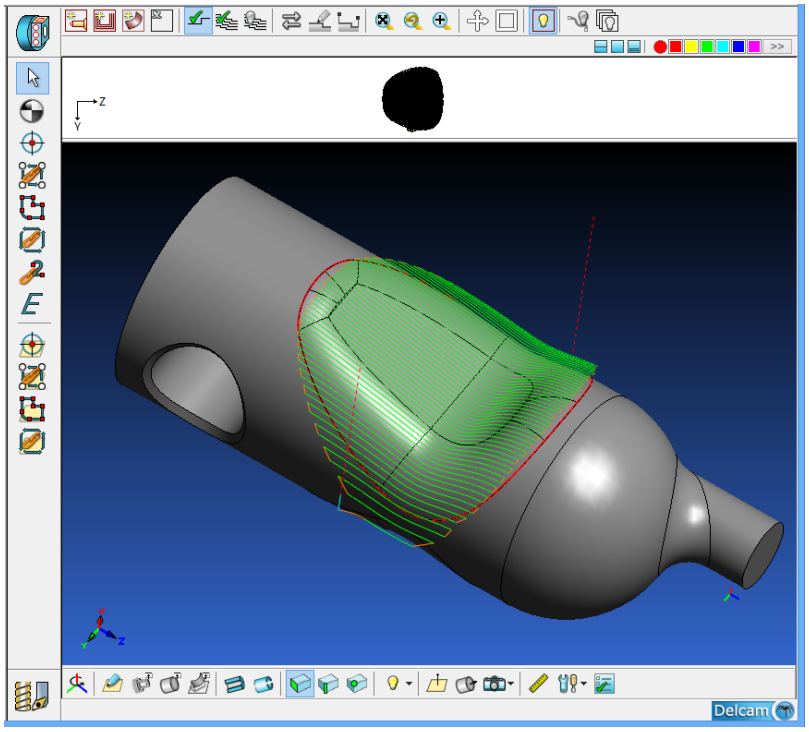
Verify the Tool Path

Once you have created the **Part Feature** selected the surfaces to machine,m you can visually verify the tool path PartMaker has calculated. To do so:

1 From the Part Features menu choose Verify Work Group Tool Path.

2 In the Tool Path Verification Options click the Do not show tool radio button Enter a Verification Delay of 0.

3 Click **<OK>** to show the calculated tool path. Your screen should appear as shown below after the tool path has been calculated:



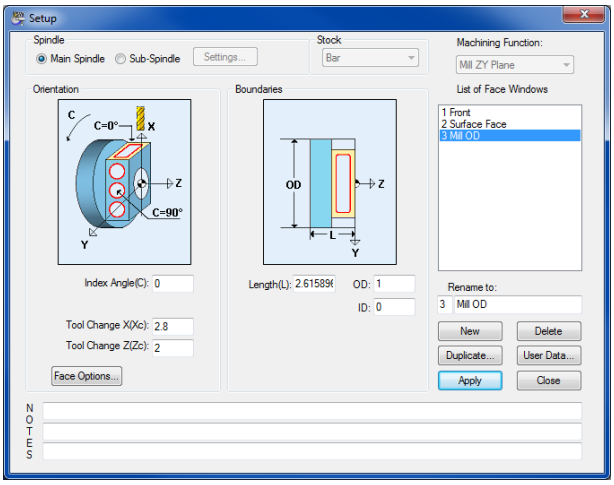
4 From the Part Features menu choose Hide Work Group Tool Path.

5 As before, you should now “lock” your tool path by opening the **Surface Group Parameters** dialog and clicking the “Lock Toolpath” button

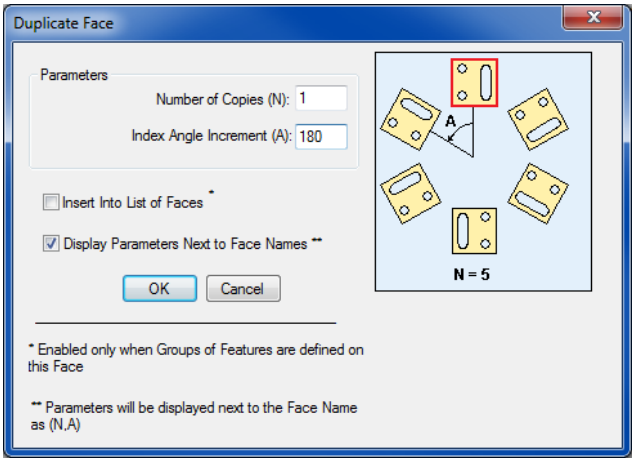
Duplicating Identical Face Windows

Having programmed the finishing operation on one side of the outside diameter of the part, you now need to create the same feature on the other side of the part, i.e. 180 degrees apart. Instead of having to go through the process of recreating the feature in a new window, you can automatically duplicate it. To do so:

1. Choose **Setup** from the **View** menu.
2. In the **Setup** dialog, select the window called “Mill OD” as shown below (it should already be selected).

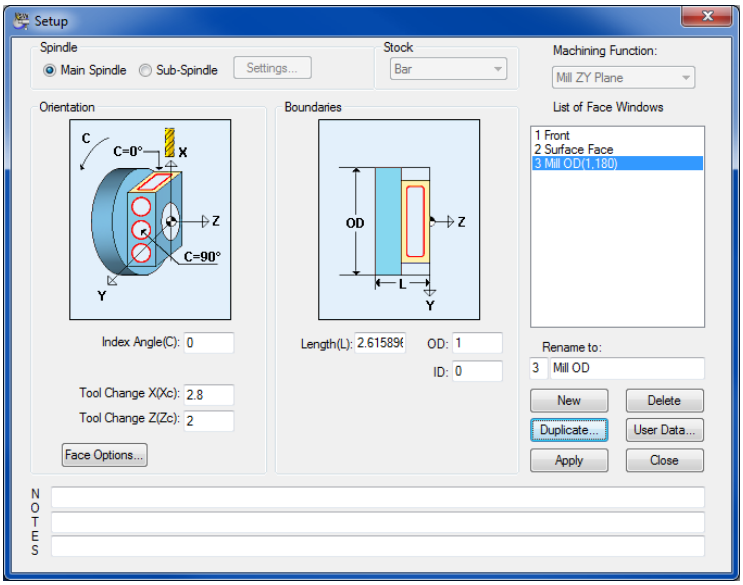


3. Click the **<Duplicate>** button.
4. In the **Duplicate Face** dialog, enter 1 in the **Number of Copies** field and 180 in the **C Angle Increment** field. The completed dialog should appear as shown below:



If the “**Insert Into List of Faces”** box is checked, then PartMaker will permanently insert new faces into list of faces. If this button is not checked then PartMaker will apply that number of copies and index increment angle on the Process Table. The process with duplicate copies will have a special icon next to the face name. Leave the “**Insert Into List of Faces”** box unchecked.

5 Click the **<OK>** button in the **Duplicate Face** dialog. Your completed **Setup** dialog should appear as shown below:



Notice the name of the **Mill OD** Face Window has (1,180) appended to it, this corresponds to the number of copies and index angle increment.

6 Click the **<Close>** button to close the **Setup** dialog.

Creating a Chamfered Cross Hole

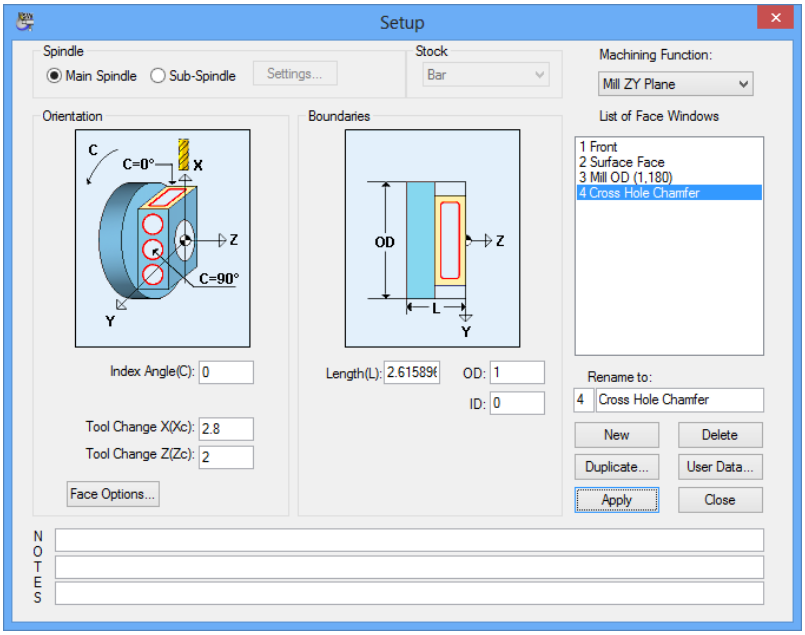
Method 1: When the Solid Model already has a Chamfer

In this section of the tutorial you will create a cross hole and create a uniform chamfer on that cross hole using a Projection surface machining strategy. The steps explained below are to be used when the solid model being machine already has a chamfer defined on it.

Create a New Face Window for Projection Machining on the Diameter

The first step in creating the chamfered cross-hole on the diameter is to create an appropriate Face Window. In this case, you will use the Mill ZY Face Window to use a vertically (i.e. X) oriented tool on the diameter of the part. To create the Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the **New** button to create a new Face Window which appears with the name "Untitled 5"
3. From the **Machining Function** drop down menu, choose Mill ZY plane (it should already be selected)
4. Enter 90 in the **Angle (C)** field. Doing so will index the spindle to 90 degrees when creating the cross hole and the chamfer.
5. In the **Rename To:** field, type "**Cross Hole Chamfer**" and click **<Apply>**. Your completed Setup dialog should appear as shown below:



6. Click the **<Close>** button to go to the new Face Window you have just created.

Create a Drilled Cross Hole

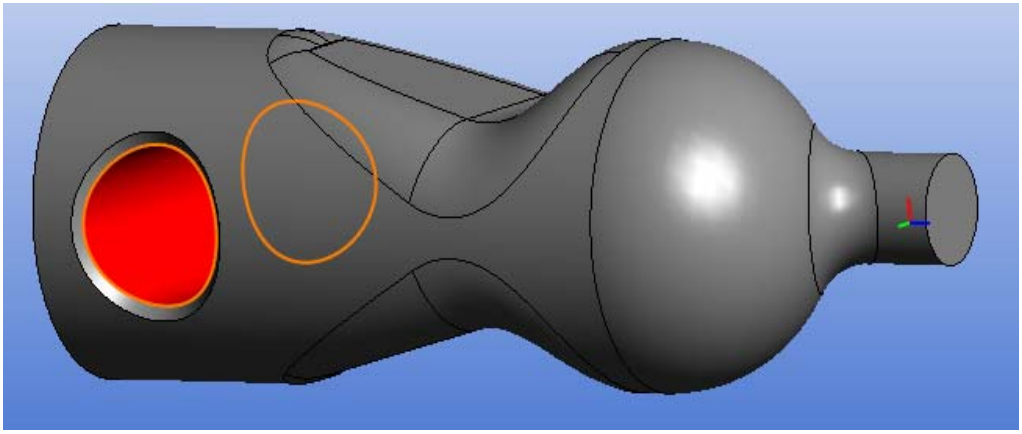
1 Choose New Hole Group from the Part Features menu to display the Hole Group Parameters dialog.

Here you will be making a through hole, so make sure to leave the **Through Hole** box checked

2 Choose **Drill** from the **Type** drop down menu.

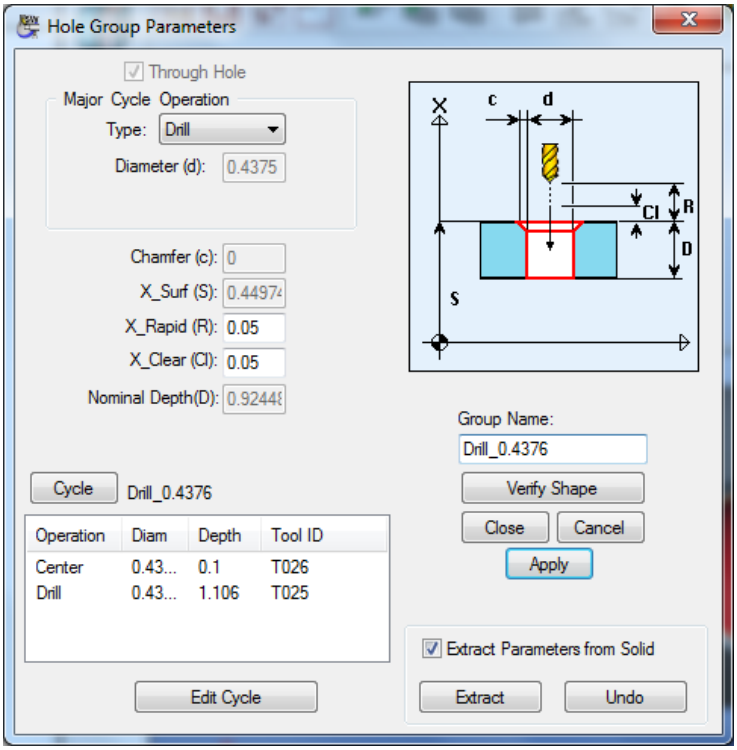
3 Check the Extract Parameters from Solid box

4 Select the hole surface you wish to drill. The hole surface will turn red once you do so.



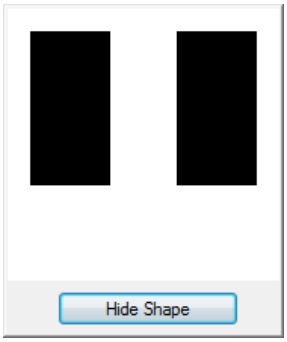
5 Once the hole is selected, click the <**Extract**> button

6 From the **Select Cycle** dialog, choose the Drill_0.4376 Cycle and click the **<Select>** button. Your completed **Hole Group Parameters** dialog should appear as shown below.



7 To verify the hole shape, click the **<Verify Shape>** button. The dialog below will display.

8 click the **<Hide Shape>** button under the picture to hide the hole shape and return to the **Hole Group Parameters** dialog.



9 Click the **<Close>** button in the **Hole Group Parameters** dialog to return to the Face Window.

Assign the Hole Feature

To assign the hole feature:

1 Click the Single Hole on Solid Model icon

2 Click the arrow head of your cursor approximately in the middle of the hole to be drilled on the solid model

3 Click the **Selection** icon and click anywhere the Face Window away from hole profile to deselect the hole

Create a New Surface Group for Projection Machining

Now that you have drilled the cross hole, you are ready to use a Projection machining strategy to chamfer this cross hole. To do so:

1 Choose a **New Color**.

2 Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.

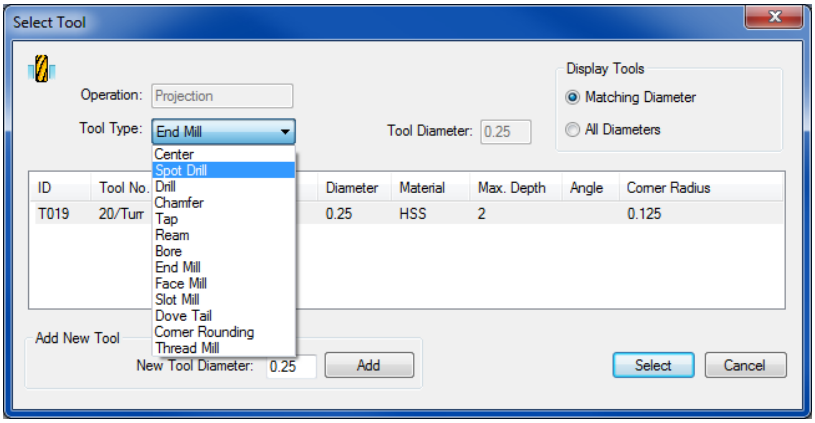
3 Check the **Projection** check box.

4 Uncheck the **Roughing** check box.

5 Enter a tool diameter of 0.25 (6.0) in the **Diam (d)** field.

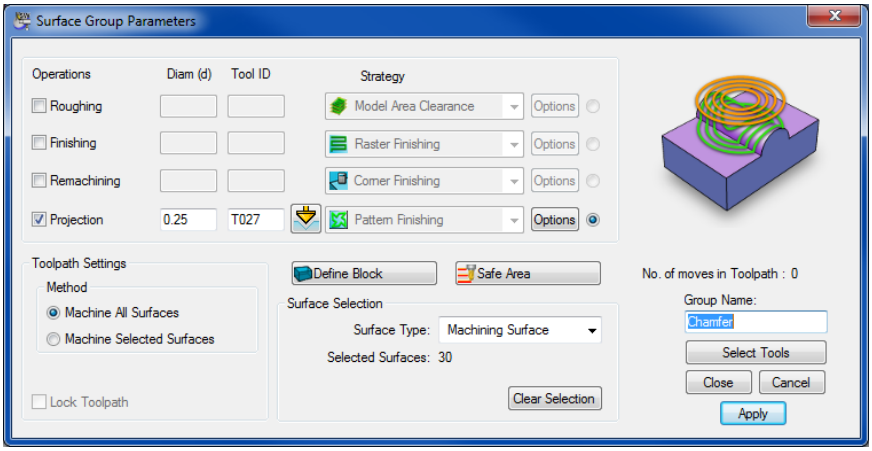
6 Here you will use a Spot Drill to create the feature.

7 Click the **<Select Tools>** button. From the <**Tool Type**> drop down menu select Spot Drill.



8 Click the **<Select>** button to select this tool.

9 In the **Group Name** field enter "**Chamfer**" and click **<Apply>**. The dialog should now appear as shown below:



Select Surfaces for Machining

At this point you should still have the Surface Group Parameters dialog open. Make sure **Machine All Surfaces** is selected in the Toolpath Settings section which sets all surfaces of the solid to machine surfaces. Click the <**Close**> button to return to the Face Window. Click the <**Close**> button to return to the Face Window.

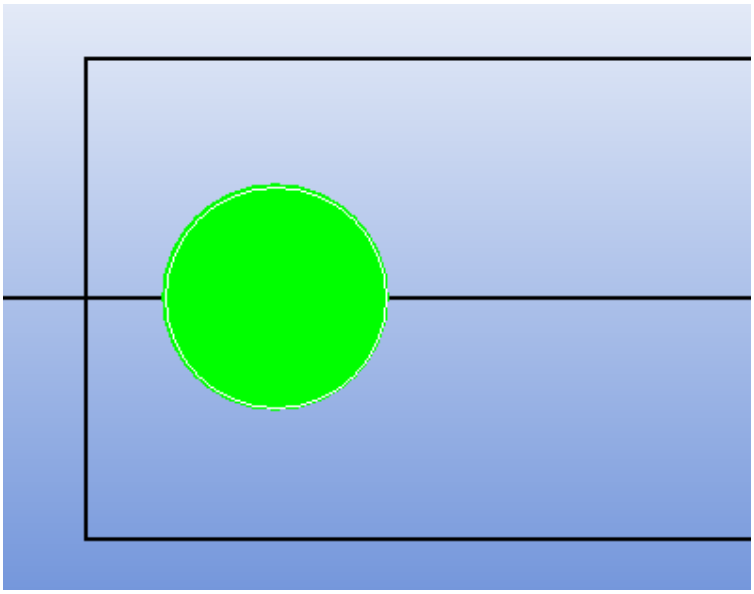
Click the <**Close**> button to return to the Face Window.

Create Surface Machining Pattern Curves

Pattern curves are created the same way as 2D profile curves. Pattern curves are 2D/3D profiles that are projected onto a 3D surface in order to calculate a tool path.

To create a pattern curve for the feature defined in this section:

1 In the 2D CAM Window, click on the Hole Feature created in the previous step so it is highlighted as shown here:



2 With hole selected, from the **Edit** menu, choose **Extract Geometry**

3 Make sure the **Chamfer** work group on the Job Explorer Tree is highlighted

4 From the **Face Window** Toolbar, choose the **Show Work Group Only** icon. This will show just 2D curve of the chamfer

5 Using the **Chain Profile** icon, click on the circular geometry

With your cursor, double click on the profile curve you have just created to enter the **Curve Properties** dialog. Notice that **Tool Position** has defaulted to **On**. When performing chamfering of a cross hole using a **Projection Machining** strategy, you should always leave **Tool Position** set to **On** so the tool machines directly on centerline along the **Pattern Curve**.

6 Click **<OK>** to close the **Curve Properties** dialog.

Verify the Tool Path

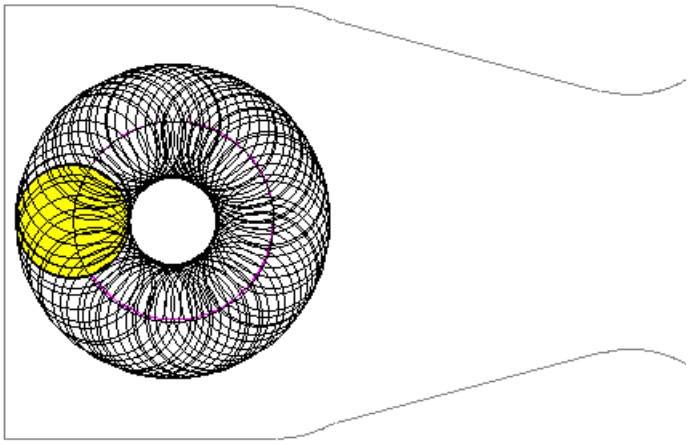
Once you have created the **Part Feature** and established a **Projection Curve** you can visually verify the tool path PartMaker has calculated. To do so:

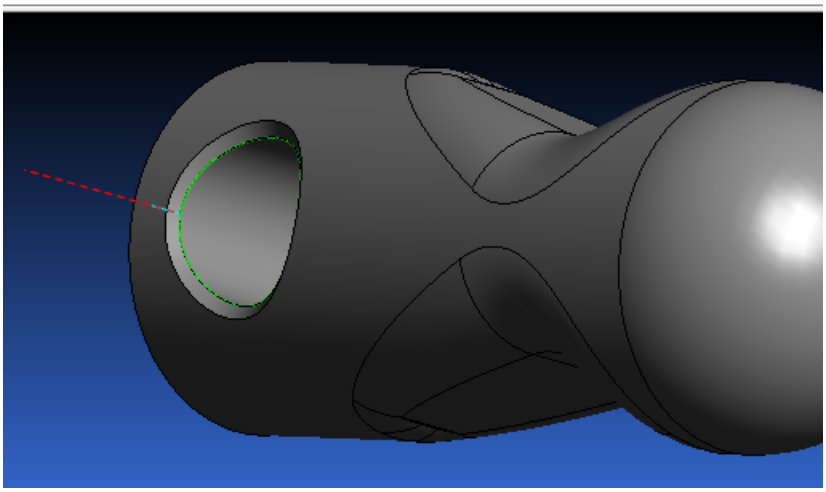
1 From the Part Features menu choose Verify Work Group Tool Path.

2 In the Tool Path Verification Options click the Do not show tool radio

3 Enter a Verification Delay of 0.

Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:





*Notice above that because the Solid Model is chamfered, the tool path was projected directly to diameter of the hole, thus automatically plunging the tool to the correct depth to create the chamfer. As a result, it was not necessary to specify an **Axial Thickness** in the **Pattern Finishing** dialog of the **Surface Group Parameters** dialog.*

4 From the Part Features menu choose Hide Work Group Tool Path.

As before, you should now “lock” your tool path by opening the **Surface Group Parameters** dialog and clicking the “Lock Toolpath” button

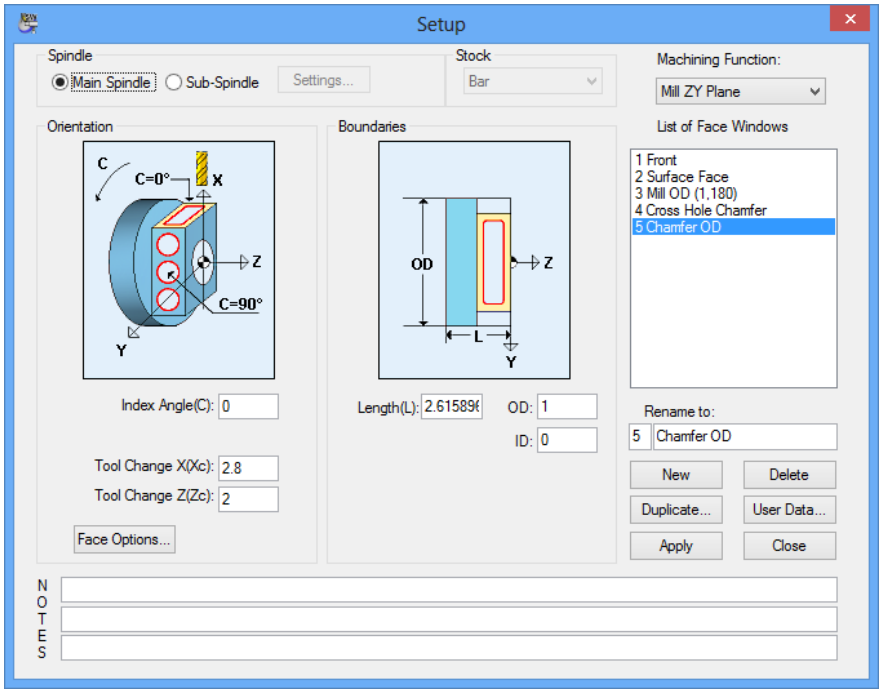
Method 2: When the Solid Model Does Not Have a Chamfer

In this section of the tutorial you will create a uniform chamfer on the cross hole created in the previous section using a Projection surface machining strategy. The steps explained below are to be used when the solid model being machined does not have a chamfer defined on it.

Create a New Face Window for Projection Machining on the Diameter

The first step in creating the chamfer on the diameter is to create an appropriate Face Window. In this case, you will use the Mill ZY Face Window to use a vertically (i.e. X) oriented tool on the diameter of the part, 180 degrees apart from the previously created Face Window. To create the Face Window:

1. Choose **Setup** from the **View** menu
2. In the **Setup** dialog, click the **New** button to create a new Face Window which appears with the name “Untitled 6”
3. From the **Machining Function** drop down menu, choose Mill ZY plane (it should already be selected)
4. Enter 270 in the **Angle (C)** field. Doing so will place the chamfer you are about to create 180 degrees apart from the chamfer created in the previous section.
5. In the **Rename To:** field, type “**Chamfer OD**” and click **<Apply>**. Your completed Setup dialog should appear as shown below:



6. Click the **<Close>** button to go to the new Face Window you have just created.

Create a New Surface Group for Projection Machining

Since you have already drilled a through cross hole, you are ready to use a Projection machining strategy to chamfer the hole. To do so:

1 Choose a **New Color**.

2 Choose New Surface Group from the Part Features menu. The Surface Group Parameters dialog will display.

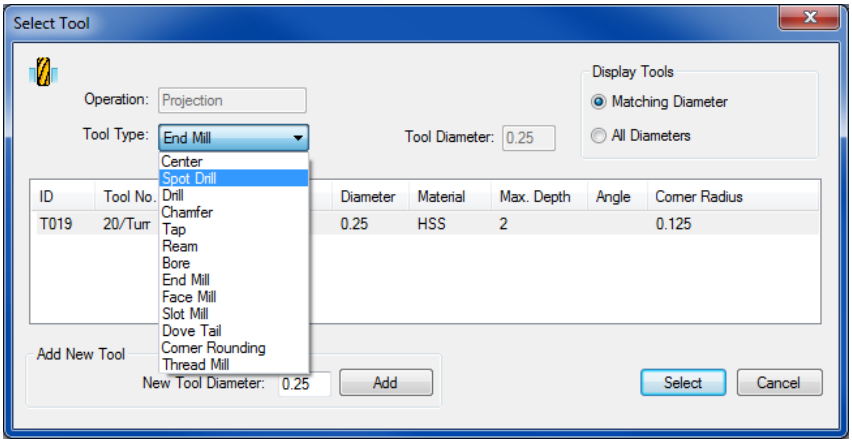
3 Check the **Projection** check box.

4 Uncheck the **Roughing** check box.

5 Enter a tool diameter of 0.25 in the **Diam (d)** field.

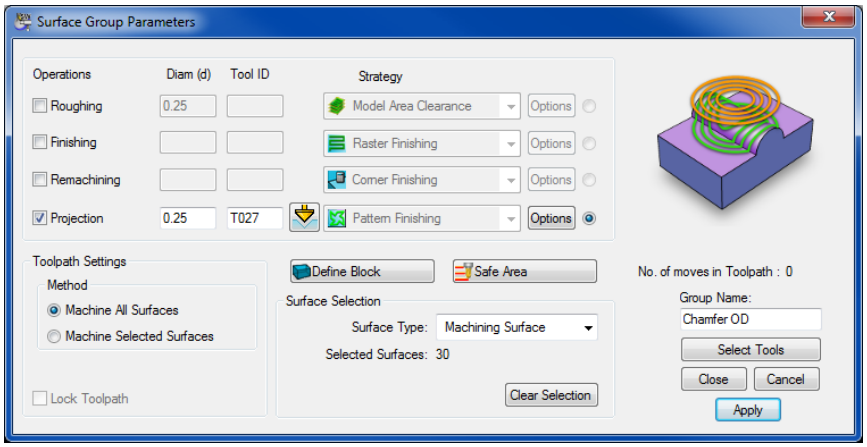
6 Here you will use a Spot Drill to create the feature.

7 Click the **<Select Tools>** button. From the <**Tool Type**> drop down menu select Spot Drill.



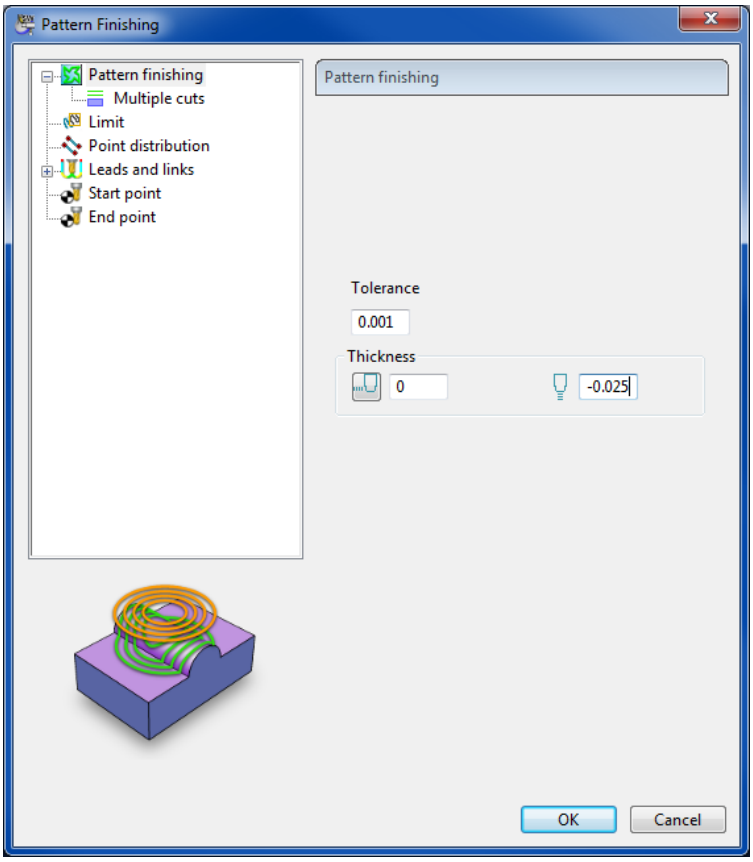
8 Click the **<Select>** button to select this tool.

9 In the **Group Name** field enter "**Chamfer OD**" and click **<Apply>**. The dialog should now appear as shown below:



10 Click on the active **<Options>** button to open the **Pattern Finishing** dialog.

11 Enter -.025 (-.63) in the **Axial Depth:** field. You activate this by clicking on the thickness button. Your dialog should appear as shown below:



12 Click the **<OK>** button to return the **Surface Group Parameters** dialog.

Select Surfaces for Machining

At this point you should still have the Surface Group Parameters dialog open. Make sure **Machine All Surfaces** is selected in the **Toolpath Settings** section which sets all surfaces of the solid to machine surfaces.

Click the <**Close**> button to return to the **Face Window**.

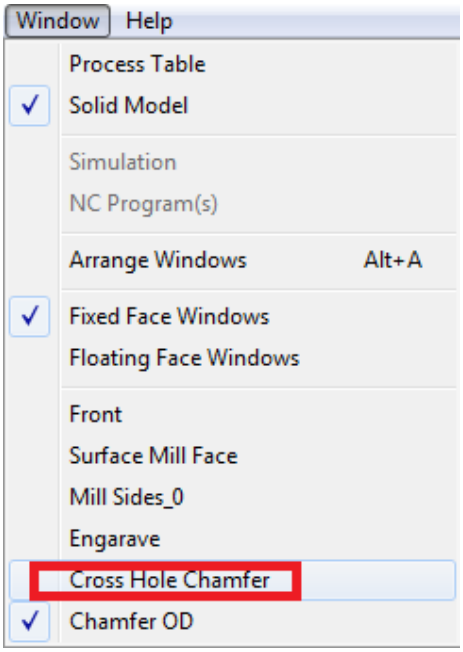
Create Surface Machining Pattern Curves

Pattern curves are created the same way as 2D profile curves. Pattern curves are 2D/3D profiles that are projected onto a 3D surface in order to calculate a tool path.

Create the Geometry for the Surface Machining Pattern Curves

The first step in creating this chamfer is to create the geometry for the projection curve for the chamfer. To do so, you will copy the circle created in the previous window. To do so:

1 From the **Window** menu, choose the **Cross Hole Chamfer** window as indicated below:

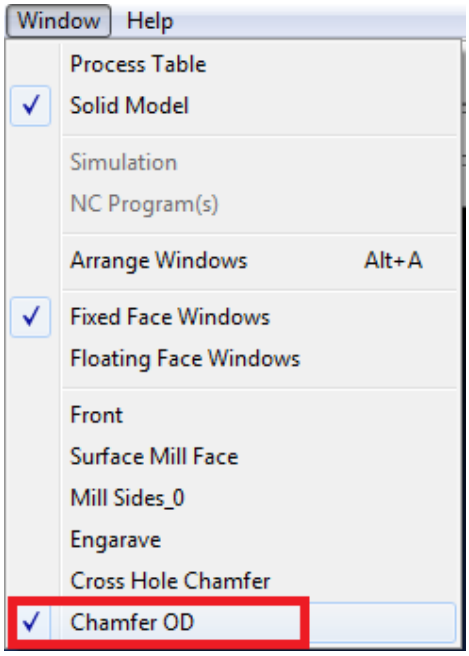


2 Once in the **Cross Hole Chamfer** window, click the CAD/CAM switch to enter the CAD mode

3 Using the **Selection** icon, click on the circle created previously. Doing so will select the geometry.

4 With the circular geometry selected, from the **Edit** menu choose **Copy**.

5 From the **Window** menu, choose the **Chamfer OD** window as indicated below:



6 In the **Chamfer OD** window, click the CAD/CAM switch to enter the CAD mode.

7 From the **Edit** menu, choose **Paste** to paste the circular geometry into the new window.

8 Click anywhere in the Face Window away from the shape to deselect the geometry.

9 Click the CAD/CAM switch to return to the CAM mode.

Create the Surface Machining Projection Curve Profile

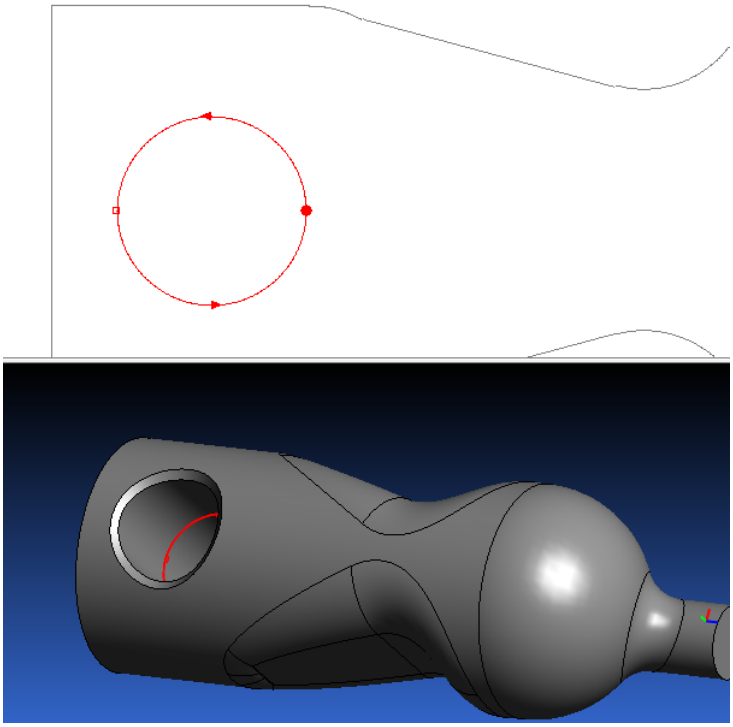
1 Choose the **Chain Geometry** icon

2 Click your cursor on the circle you just pasted into the face window. Doing so will create the projection curve. Your window should now appear as shown below:

Verify the Tool Path

Once you have created the **Part Feature** and established a **Projection Curve** you can visually verify the tool path PartMaker has calculated. To do so:

1. From the Part Features menu choose Verify Work Group Tool Path.
2. In the Tool Path Verification Options click the Do not show tool radio button
3. Enter a Verification Delay of 0.
4. Click **<OK>** to show the calculated tool path. Your screen should appear as shown below:



Notice above that because the Solid Model is not chamfered, you need to specify an **Axial Thickness** to create the chamfer.

5. From the Part Features menu choose Hide Work Group Tool Path.
6. As before, you should now “lock” your tool path by opening the **Surface Group Parameters** dialog and clicking the “Lock Toolpath” button

Save Your Work

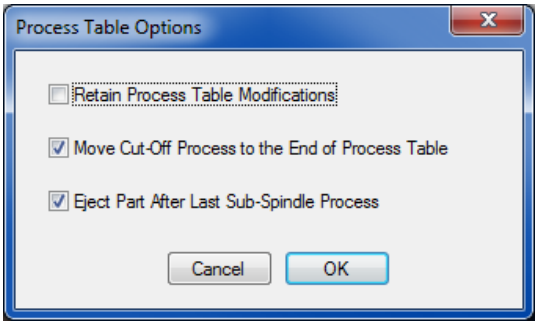
It is always a good idea to save your at various points in the part creation process in PartMaker. To do so:

From the **File** menu choose **Save** or press **<CTRL + S>** on your keyboard

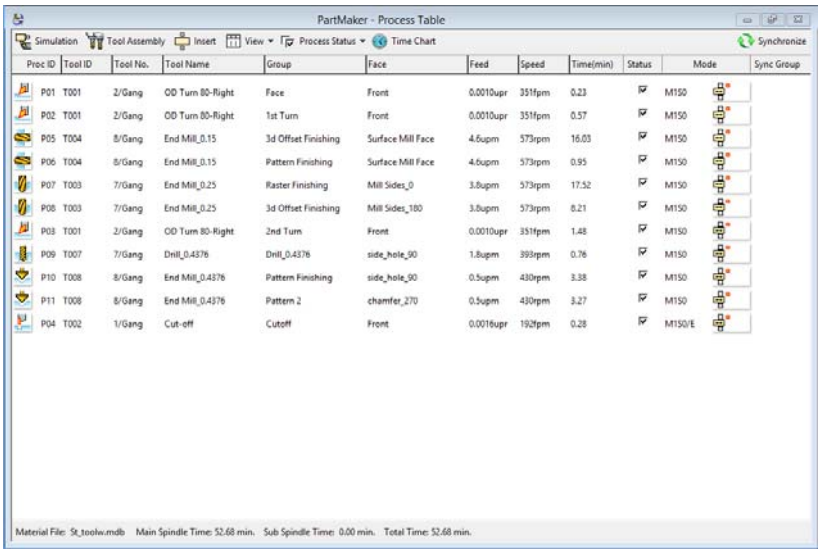
Generate a Process Table

A *process table* shows you all of the processes for a part. When **PartMaker** generates a process table, all cutting conditions such as feedrate and spindle speed RPM are calculated automatically based on the tools and material information.

1 Choose Generate Process Table from the Job Optimizer menu to display the Process Table Options dialog as below.



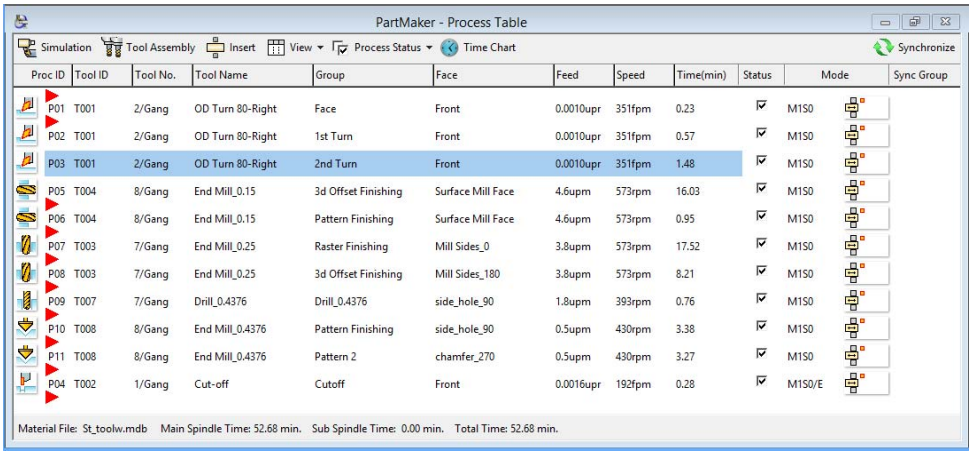
2 Click **<OK>** to display the process table window as shown below.



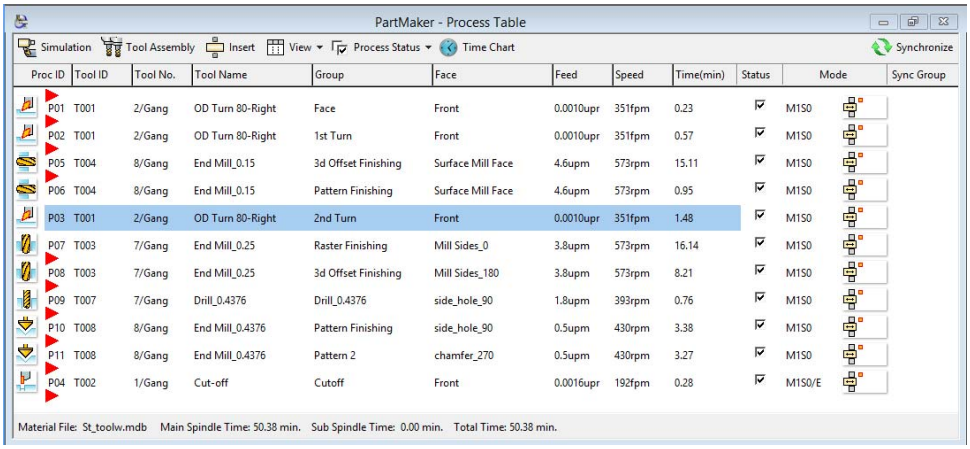
Reordering Operations

Next, you will reorder your operations so they are in the correct sequence, i.e. the second OD turning operation will be placed after the milling operations on front face of the part. To do so:

1 Click on process #3 (P03) so that is selected as shown below:



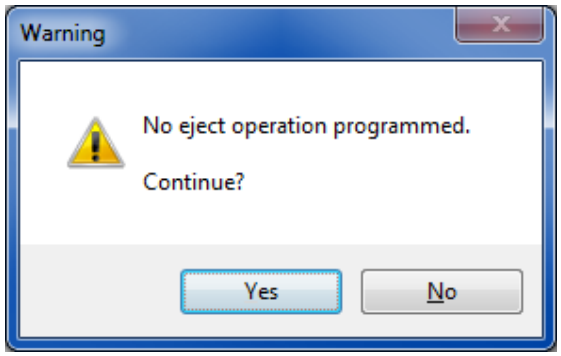
2 Click the arrow BENEATH process #6 (P06) to place process #3 (P03), the second turning process, after the face milling operations. Your completed **Process Table** window will appear as shown below:



Simulate the Cutting Process

Once you are satisfied with the appearance of your Process Table you can simulate cutting. To do so:

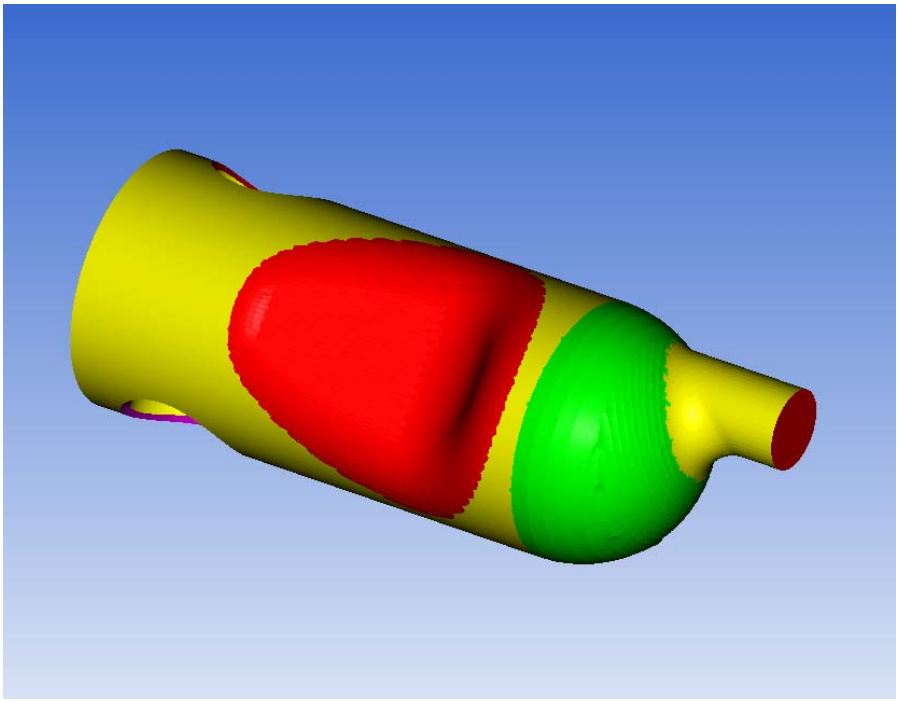
1. From the Process Table, click the **<Simulation>** button in the upper left hand corner of the Process Table or just press the **<Space Bar>**.
2. Because this part does not have any sub spindle operations and thus you are not ejecting it from the sub spindle upon completion of machining, you will be prompted with the warning below:



3. Click the **<Yes>** button to continue into **Simulation.**

4. When the Simulation Window appears, press the **<Space Bar>** or choose **Start Simulation** from the **Simulate** menu to initiate 3D simulation.

5. Once simulation is complete, choose **Show Finished Part** from the **Solids** menu. Your completed part should appear as shown below:

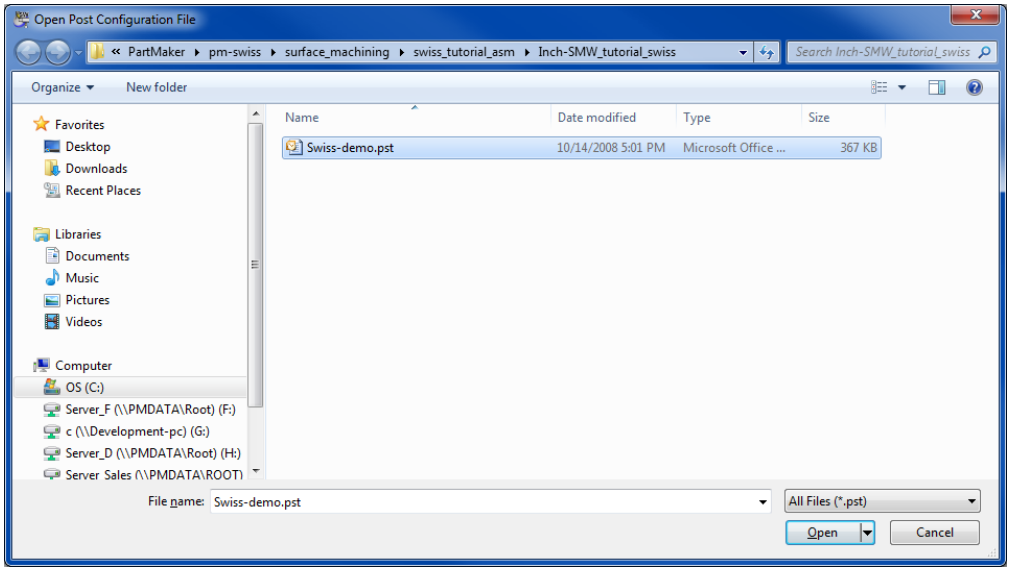


Generate an NC Program

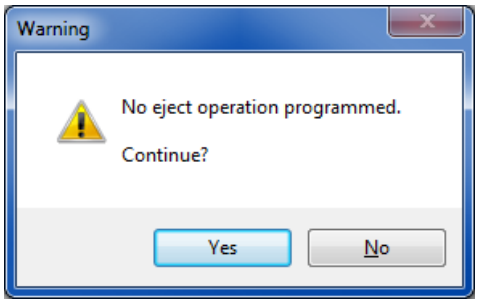
1 Choose Post Config File= ? from the Job Optimizer menu to display the Open Post Configuration File dialog.

2 Select the Swiss_Demo.PST post processor

3 Click the **<Open>** button as shown below to load the post processor.

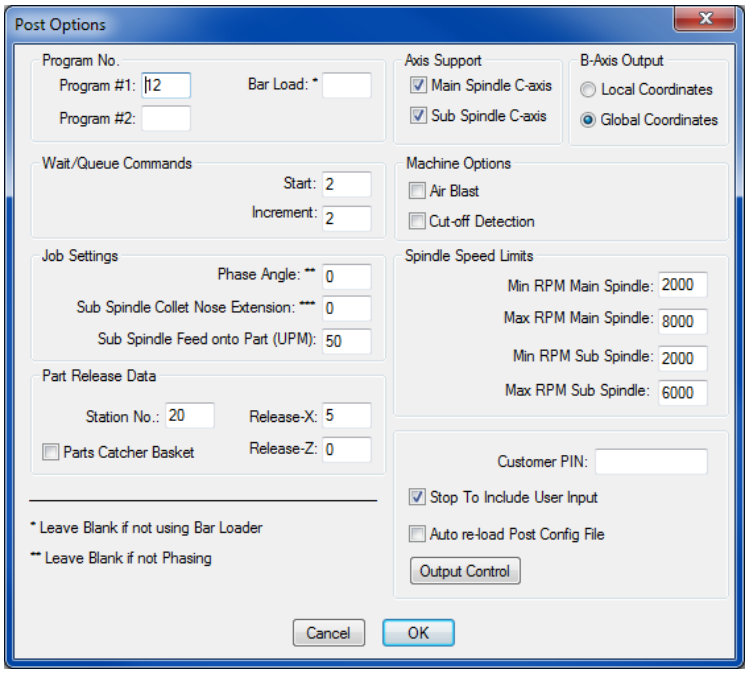


4 Choose **Generate NC Program** from the **Job Optimizer** menu. You may again be prompted with the **Warning** dialog below.



5 Click the **<Yes>** button to continue.

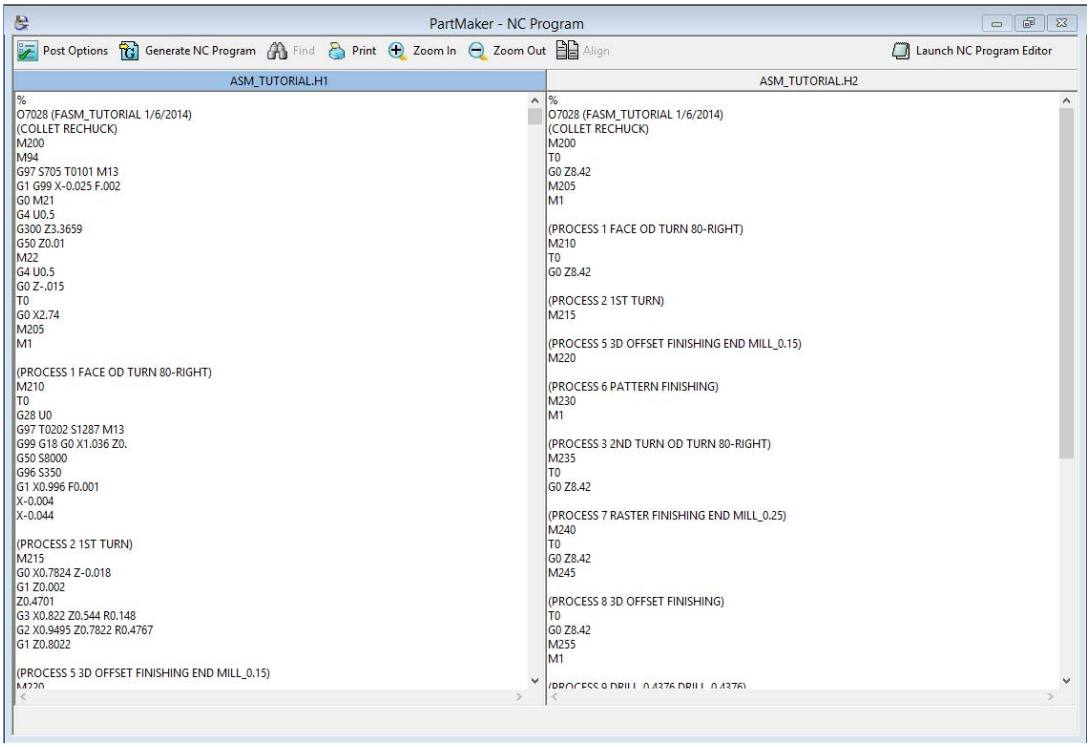
6 When you generate an NC Program for the first time during a programming session the **Post Options** dialog is displayed as shown below:

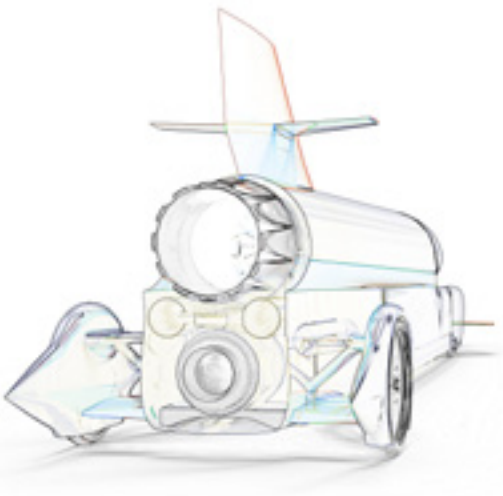


7 Click the **<OK>** button to open the **Save NC Program File As** dialog.

8 Type a name for your NC Program, such as "**ASM_Tutorial**" and click the **<Save>** button

9 Your NC Program files will be displayed as shown below:

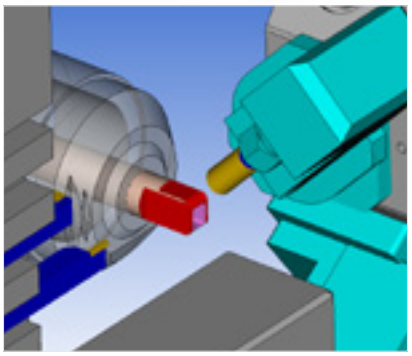




PartMaker 2015

World-leading CAM software for production machining

www.partmaker.com

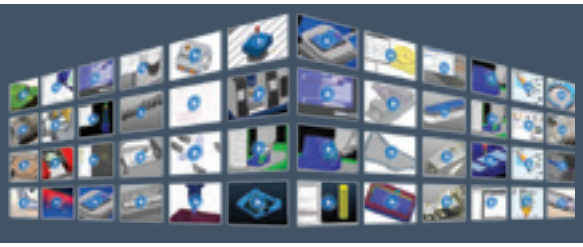


PartMaker 2015

www.partmaker.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

Visit **www.delcam.tv/lz** and see how Delcam software could benefit your business!

Delcam Advanced Manufacturing Solutions

Powering your productivity

Delcam
Small Heath Business Park, Birmingham, B10 0HJ, United Kingdom
T: +44 (0) 121 766 5544 | E: marketing@delcam.com
W: www.delcam.com

PartMaker Inc.
550 Pinetown Road, Suite 470, Ft. Washington PA 19034 USA
T: +215 643 5077 | E: info@partmaker.com
W: www.partmaker.com

To contact your local reseller, visit **www.delcam.com/resellers**

AUTODESK.