

## Advanced Surface Machining Version 2015 or Higher



# User Guide

## PartMaker 2015

# **User Manual**

User Guide/PartMaker Advanced Surface Machining



## **Important User Notices**

Copyright 1991-2014 Delcam Ltd. All rights reserved.

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.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence. Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

### Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

### 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 1 Introduction 1 Major Features 2 Input Options 2 Surface Machining Options 2 Cutting Limits and Constraints Options 2

Cutting Limits and Constraints Options	Z
Milling Tool Types Options	2
Surface and Tool Orientation Options	
What Makes ASM Unique?	4

## 

Introduction1
How you will Create the Sample Part 1
Start PartMaker® Mill 2
Import a Solid Model 2
Specifying the Part Boundaries
Set Defaults5
Define Holes for the Boring Cycle
Define Holes for the Tapping Cycle 12
Define Machining Strategies for the Tapered Pocket 15
Finishing the Tapered Pocket using Steep and Shallow Finishing Strategy
Define Surface Machining Strategies for the Side Cavity
Surface Machining Strategies for the Center Cavity
Define Profiles for Contour Mill Cycle
Drill a Hole in the Center Cavity
Generate a Process Table 47
Simulate the Cutting Process
Generate an NC Program

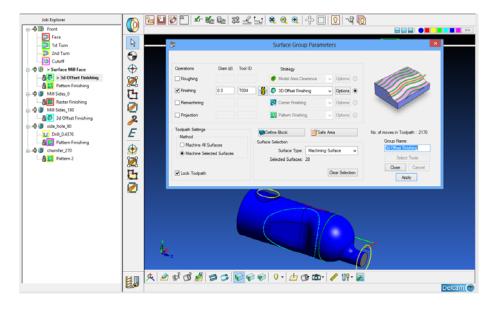
# 

Introduction	. 1
How You Will Create the Sample Part	. 1
Start PartMaker	. 2
Set Defaults	. 3
Defaults for Turning (SwissCAM users) Defaults for Turning (Turn-Mill users) Defaults for Milling (SwissCAM and Turn-Mill users)	4
Importing the Solid Model	. 6
Setting Part Boundaries from the Solid Model	. 7
The Setup Dialog: SwissCAM Users The Setup Dialog: Turn-Mill Users	
Transferring Geometry from the Solid Model	10
Creating a Facing Operation	12
Create the Profile for Facing	13
Creating the First OD Turning Operation	14
Create the Profile for OD Turning	15
Creating the Second OD Turning Operation	16
Create the Profile for OD Turning	
Creating a Cutoff Work Group	18
Create the Profile for the Cut-Off Operation	19
Creating Surface Projection Tool Paths on the Face of the Part	20
Create a New Face Window for Surfacing on the Face Create a New Surface Group for Projection Machining Create Surface Machining Pattern Curve	21
Creating Surface Finishing Tool Paths on the Face of the Part	32
Create a New Surface Group for Finishing Create Boundary Curves	32 34
Creating Surface Finishing Tool Paths on the Diameter of the Part	39
Create a New Face Window for Finish Machining on the Diameter	
Duplicating Identical Face Windows	44
Creating a Chamfered Cross Hole	46
Method 1: When the Solid Model already has a Chamfer Method 2: When the Solid Model Does Not Have a Chamfer	
Generate a Process Table	58
Reordering Operations	59
Simulate the Cutting Process	60
Generate an NC Program	61

## 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<sup>®</sup> Mill <u>Advanced Surface</u>** <u>Machining (ASM)</u> 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**.

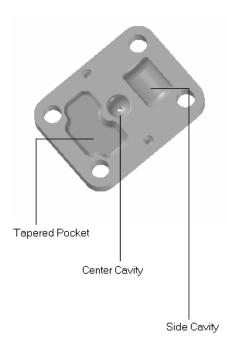


**Note:** You must install software from PartMaker<sup>®</sup> versions 2013 R1 or higher to have all support files and control functions referenced here.

## 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

M
MILL

Ŷ

T

- 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.

Computer Solution OS (C:) DATA (D:)

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:

👺 Import Parasolid X\_T File ≪ ASM\_Surface\_Machining ► Tutorial\_ASM - 4+ Q New folder = • 0 Organize 🔻 Date modified Туре Name 쑦 Favorites 📃 Desktop ASM\_tutorial\_mill.X\_T 11/28/2012 8:28 AM X\_T File 鷆 Downloads 📃 Recent Places ز Libraries Documents J Music E Pictures 📕 Videos

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.

All Files (\*.X\_T)

<u>O</u>pen 🔽

Cancel

•

File name: ASM\_tutorial\_mill.X\_T

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:

👺 Define Face Plane
Set Face Plane:
Parallel to XY Plane 👻
Reverse Machining Side
Z Face: 0
Face Orientation: FRONT    Stock Positioning Angle: 0
Set Face Origin:         New Origin:       Enter Coordinates         X:       0         Y:       0         Z:       0
Set Boundaries  Preview OK Cancel

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:

able	Stock	Part ID	Machining F	unction:
Settings Standard v	Box 💌	1	Mill XY Plan	e v
rientation	Boundaries	_	List of Face	Windows
	Y (x1,y1) Y (x2,y2) O O O V X		1 Front	
Height(H): 1	x1: -4 x2: 4		Rename to:	
Z Face(F): 0	y1: -3 y2: 3		1 Front	
			New	Delete
			Duplicate	User Data
Face Options	Work Offset: G54		Apply	Close

The Define Block dialog is also shown here:

Define Block	×
Block Limits* Defined by: Box ~	
Defined by: box *	
Min X (x1): 4 🖌 Max X (x2): 4	$\wedge$
Min Y (y1): -3 Max Y (y2): 3	-z2
Min Z (z1): -1 Max Z (z2): 0	-21
	y2 x2
Calculate	×
Expansion: 0	yî x1
From Selection From Stock	
Use Custom Shape	OK Apply Cancel
*Block Limits are Defined in the Part Coordinate System	

## **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

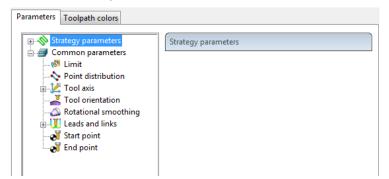
Using such a large number for your Rapid Plane may not be required for machining this part but it will help you see how the tool path is generated in the reference to the Rapid Plane.

Group Parameters	Process Parameters:	Machining Data:
Through Hole	Apply Comp in PartMaker	Maximum Speed: 5000
Diameter: 0.25	Coolant: Standard	Maximum Feed: 200
Chamfer: 0		Test Change Tree (ris): 0.25
Z_Surf: 0	Feed: 10	Tool Change Time (min): 0.25
Z Depth: 1	Speed: 1000	Rapid Feed: 1000
	Tool Ch X: 0	
Z_Rapid: 0.5	Tool Ch Y: 0	leads
Z_Clear: 0.05	Tool Ch Z: 5	20000
Width of Cut: %Tool Diameter	•	Arc Radius: 75 %dia
Width of Cut Value: 80		Line Length: 75 %dia
Surface Machining Module		Lead Angle: 45 deg
💿 SMW (Legacy) 🛛 💿 ASM 🗨		
Surfacing Default		
Process Table Display Option		Arc Tolerance: 0.005
Feeds in Units per Revolution an	d Surface Speed	
		Comer Rounding
	(upm)	

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.

Surfacing Defaults	×
Parameters Toolpath colors	
Parameters Toolpath colors	Strategy parameters
	OK Cancel Apply

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.

Parameters ·	Toolpath colors				
● ◆ Stra	tegy parameters nmon parameters	Lead in	Туре	None Distance Angle Radius Ramp options	

- 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:

	.ead in			
🚊 🗐 Common parameters 👘				
🕅 Limit				
🗞 Point distribution				
🗄 🖉 Tool axis		Туре	Vertical arc	
				_
			Distance	0.2
🖃 🔰 Leads and links			Angle	60
				0.5
- III First lead in			Radius	0.5
			Ramp options	
📲 🚺 Lead out				_
			<b></b>	

**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.

Parameters Toolpath colors	
Strategy parameters	Links
	Short / Long threshold 1 Short Circular arc Long Skim +
Heights     Heights     Ifirst lead in     Lead in     Lead out     Last lead out     Last lead out     Last lead out     Links	Retract and approach moves Along Tool axis Automatically extend Maximum length 10 Retract distance 0

- 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

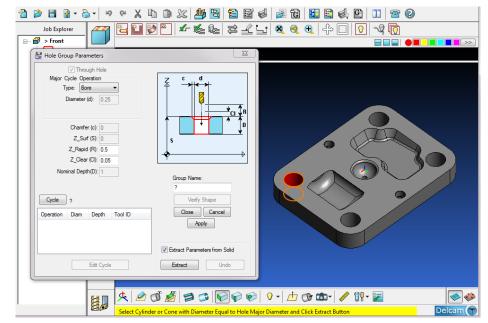
## **Define Holes for the Boring Cycle**

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

Major Cycle O	peration					
Type: Bore						
Diameter: 1						
List of Matchir	ng Cycles					
Bore-1						
Add N	lew Cycle		ielect			
Add N	lew Cycle		elect			
Add N Cycle Preview			elect			
		Depth	ielect Tool ID			
Cycle Preview	,					
Cycle Preview Operation	Diam		Tool ID			
Cycle Preview Operation Center	Diam 0.2188		Tool ID T001			
Cycle Preview Operation Center Drill	Diam 0.2188 0.9688		Tool ID T001 T003 T002			
Cycle Preview Operation Center Drill Bore	Diam 0.2188 0.9688 1		Tool ID T001 T003			
Cycle Preview Operation Center Drill Bore	Diam 0.2188 0.9688 1	Depth	Tool ID T001 T003 T002			

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:

– Major C Ty	✓ Throu ycle Ope pe: Bor Diameter (	eration re	•	
	Z_Rapid Z_Clear	(S): 0 (R): 0.5 (CI): 0.05		
Cycle	inal Deptł Bore-1	n(U):		Group Name: Bore-1 Verify Shape
Operation	Diam	Depth	Tool ID	Close Cancel
Center Drill Bore	0.21 0.96 1	0.1 1.3411 1.05	T001 T003 T002	Apply
	1	0	T004	Extract Parameters from Solid

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.

Hide Shape	

**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

7		٦.
λ.		,
-1		r -
	4	1

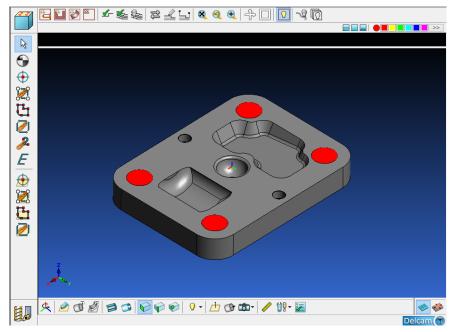
**Important!** The part feature creation selections (i.e. **New Hole Group**, **New Profile Group** and **New Surface Group** functions) found under the **Part Features** menu can also be accessed by clicking your right mouse button in the CAM Face Window.

### **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:

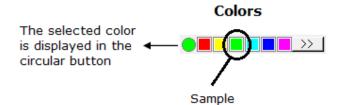
👺 Save Job File as:					×
🔾 🗸 🖉 🖉 🖉	M_Sur	face_Machining	• <del>•</del>	Search Tutorial	ASM 🔎
Organize 🔻 Ne	w folde				) <b>•</b> •
🔆 Favorites		Name	D	ate modified	Туре
🧮 Desktop	E	ASM_tutorial_mill.JOB	3	/19/2013 10:56 AM	Task Scheduler Ta.
鷆 Downloads		asmpractice.JOB	1	2/17/2014 1:30 PM	Task Scheduler Ta.
📃 Recent Places					
💱 Dropbox					
🔚 Libraries					
Documents					
👌 Music					
Pictures	~	•	ш		4
File name:	asmp	ractice.JOB			•
Save as type:	All Fil	es (*.JOB)			•
Hide Folders				Save	Cancel

2 Click the <Save> button.

## **Define Holes for the Tapping Cycle**

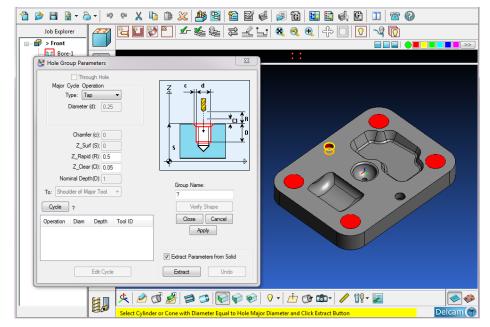
In this section, you will specify the holes for Group 2, the tapping cycle. To do so:

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.

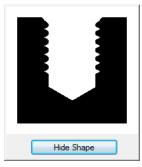
8 Click **<Select>** to select the Tap-1/2 cycle

		•	5					
Select Cycle				x				
Major Cycle Op	eration			_				
Туре	Туре: Тар							
Diameter	r: 0.5							
	List of Matching Cycles							
Tap-1/2								
Add Ne	ew Cycle	S	elect					
Cycle Preview								
Operation	Diam	Depth	Tool ID					
Center	0.219		T001					
Drill	0.422		T005					
Тар	0.5		T006					
Chamfer	0.5		T004					
	Car	icel						

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:

Ţ	Cycle Ope ype: Tap Diameter ( Chamfer Z_Surf	d): 0.5		
Norr To: Botto	ninal Deptł	(Cl): 0.05 n(D): 0.35	_	Group Name: Tap-1/2 Verify Shape
Operation Center Drill Tap	Diam 0.219 0.422 0.5	Depth 0.1 0.5768 0.35	Tool ID T001 T005 T006	Close Cancel Apply
Chamfer	0.5	0.26 Edit Cycle	T004	Extract Parameters from Solid Extract Undo

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**.

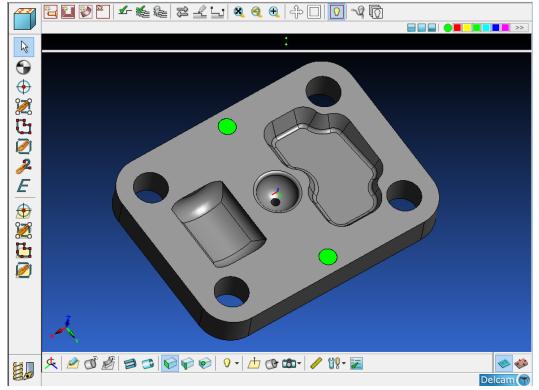
A group symbol for Group 2 displays in the upper-left corner of the **Face Window** under the Tap-1/2 group icon. The cycle name Tap-1/2 appears to the right of the group symbol.



### **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.

0	Operation:	Roughing				Display	Tools hing Diameter
T	Tool Type: (	End Mill 🔻	]	Tool Diamete	er: 0.25	) Al Di	ameters
ID	Tool No.	Name	Diameter	Material	Max. Depth	Angle	Comer Radius
T010	10	1/4 Ball Mill	0.25	Carbide	2		0.125
T014	14	1/4 Ball Mill:1	0.25	Carbide	2		0.125
Add Nev		w Tool Diameter: 0.25	Add				Select Cancel

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.

👺 Surface Group Parameters × Operations Diam (d) Tool ID Strategy Roughing 0.25 T010 Model Area Clearance - Options (0) Finishing 📕 Raster Finishing - Options Remachining Comer Finishing Options Projection Pattern Finishing - Options Toolpath Settings Define Block No. of moves in Toolpath : 0 Method Group Name: Surface Selection Machine All Surfaces Rough Pocket Surface Type: Machining Surface -Machine Selected Surfaces Select Tools Selected Surfaces: 69 Close Cancel Clear Selection Lock Toolpath Apply

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

👺 Model Area Clearance	×
Model area clearance	Model area clearance Style Offset all
Approach With Point distribution Carlot and links Start point End point	Cut direction Profile Area Climb Any Tolerance 0.001 Thickness 0
	Stepover 0.05 Stepdown Automatic 0.05 OK Cancel

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.

Model area clearance	Model area clearance
Oriset     Oriset     Step cutting     Rest     Wall finishing     Wall finishing     Flat machining     Flat machining     Flat machining     Flat speed     Approach     Wint     Approach     Oriet     Point distribution	Style Offset model Cut direction Profile Area
JU Leads and links – J Start point – J End point	Climb Climb Tolerance 0.001 Thickness 0.01 Stepover
	0.05 Stepdown Automatic = 0.05

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"

perations	Diam (d)	Tool ID	Strategy		
Roughing	0.25	T010	🌒 🧳 Model Area Clearance	- Options	
Finishing			Raster Finishing	▼ Options ○	
Remachining			Comer Finishing	▼ Options ◯	
Projection			Pattern Finishing	▼ Options ◯	$\checkmark$
oolpath Settings			Define Block	fe Area	No. of moves in Toolpath : 0
Method			Surface Selection		Group Name:
Machine All S Machine Sele			Surface Type: Machinir	ng Surface 👻	Rough Pocket
I I Machine Sele	cted Surfaces		Selected Surfaces: 69		Select Tools

Your Surface Group Parameters dialog should appear as shown below:

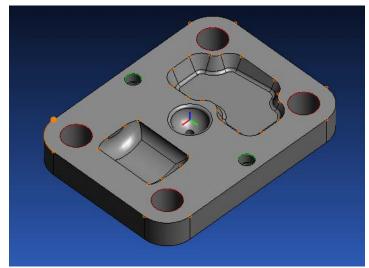
Notice the number of selected surfaces is 69.

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

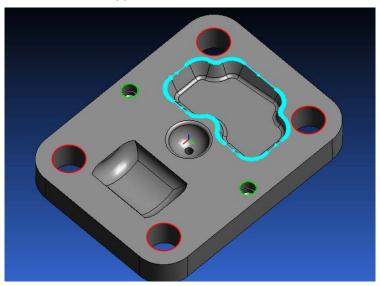
## **Create a Boundary Curve**

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:

- 0
- 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.

Curve Properties	×						
Curve Type	Used in Operation						
O Boundary	Roughing						
Pattern	V Finishing						
Top Swarfing	Remachining						
Bottom Swarfing	Projection						
Spine							
Curve Definition							
	Style: 3D Curve 👻						
Distance From Face Origin:							
Tool Position:							
Boundary Type:	User Defined 🔻						
ОК	Cancel						

- 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:

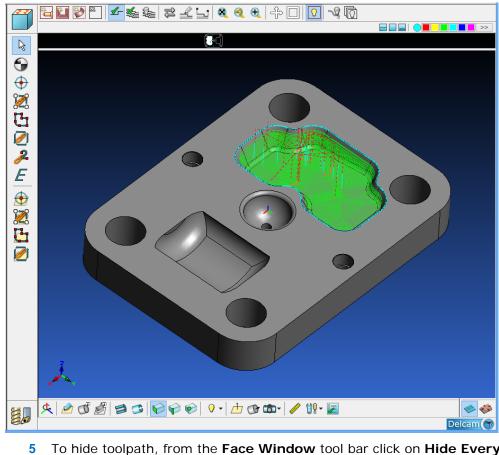


- 1 From the Face Window tool bar choose Verify Work Group Tool path.
- 2 Uncheck the **Enable Verification on Face Window**. As we are only working on the solid, we do not need to see the verification in 2D.
- 3 Enter a Verification Delay of 0.



**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.

Tool Path Verification Options
Verification on Face Window
Enable Verification on Face Window
Show Tool as:
Hollow
<ul> <li>Solid</li> </ul>
Do not show tool
Verification on Solid Model
Enable Verification on Solid Model
Show Tool as:
<ul> <li>Solid</li> </ul>
Do not show tool
✓ Show Toolpath
Prompt for Separate Verification of each Operation
Verification Delay (0-9)
OK Cancel



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

<u>\_</u>

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:

Surface Group Pa	rameters			
Operations	Diam (d)	Tool ID	Strategy	
Roughing	0.25	T010	💋 🥏 Model Area Clearance 🚽 Options	s 0
Finishing			Raster Finishing	
Remachining			Comer Finishing	s o
Projection			Pattern Finishing	s O
oolpath Settings			Define Block	No. of moves in Toolpath : 16514
Method Machine All Su	faces	S	urface Selection	Group Name:
Machine Selection			Surface Type: Machining Surface	✓ Rough Pocket
0 1400 110 00000			Selected Surfaces: 69	Select Tools
Lock Toolpath			Clear Selec	ction Close Cancel

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:

Job Explorer
🖃 🗊 > Front
Bore-1
<b>1</b> Tap-1/2
📖 🔮 🦻 > Rough Pocket

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.

		Finishing End Mill		Tool Diamete	er: 0.25	Oisplay	hing Diameter
ID	Tool No.	Name	Diameter	Material	Max. Depth	Angle	Comer Radius
T010	10	1/4 Ball Mill	0.25	Carbide	2		0.125
T014	14	1/4 Ball Mill:1	0.25	Carbide	2		0.125
Add Ne		w Tool Diameter: 0.2	5 Add				Select Cancel

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

🐇 Surface Group Pa	arameters		
Operations CupyIng Operations Ope	Diam (d) Tool I	D Strategy Model Area Clearance  Options  Model Area Clearance Options  Options  Model Area Clearance Options  Model Area Clearance Options  Options O	
Toolpath Settings Method Machine All Si Machine Select Lock Toolpath		Surface Block       Surface Selection       Surface Type:       Machining Surface       Selected Surfaces:       69	No. of moves in Toolpath : 1526 Group Name: Finish Pocket Select Tools Close Cancel Apply

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.

Steep And Shallow Finishing		<b>X</b>
Steep and shallow finishing	Steep and shallow finishing	,
Limit	Order	Additional stock
⊕-UU Leads and links Start point	Steep first   Threshold angle	0 Steep shallow overlap
End point	30 Steep	0.125 Shallow
	Spiral Cut direction	Spiral Cut direction
	Climb -	Climb -
	Stepdown 0.02	Stepover 0.015
	Calculate using cusp Cusp height	Type 3D offset ▼
	0 Maximum stepdown	Smoothing
	Tolerance 0.001	
	Thickness 0	
		OK Cancel
		Cancel

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.

Operations	Diam (d) Tool ID	Strategy	
Roughing		🔹 Model Area Clearance 🚽 Options 🔘	
Finishing	0.25 T010	👔 📤 Steep and Shallow Finishin 👻 Options 💿	
Remachining		Corner Finishing	
Projection		Pattern Finishing	~
Toolpath Settings		Define Block	No. of moves in Toolpath:14196
Method Machine All Su	-	Surface Selection	Group Name:
Machine Selected Surfaces		Surface Type: Machining Surface 🗸	Finish Pocket
	ated Sundces	Selected Surfaces: 69	Select Tools

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

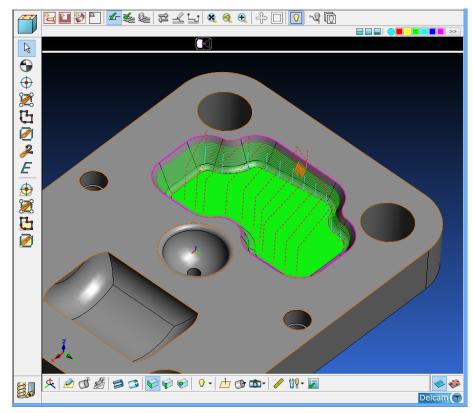
Once you have created the **Part Feature** and established a **Boundary Curve** you can visually verify the tool path PartMaker has calculated. To do so:

- 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.

Tool Path Verification Options
Verification on Face Window
Enable Verification on Face Window
Show Tool as:
Hollow
Solid
Do not show tool
Verification on Solid Model
Enable Verification on Solid Model
Show Tool as:
<ul> <li>Solid</li> </ul>
O not show tool
Show Toolpath
Prompt for Separate Verification of each Operation
Verification Delay (0-9)
OK Cancel





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



- 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.

Select To	Operation: Tool Type:	Roughing End Mill	<b>•</b>	1	Fool Diameter	. 0.25	Display T Match All Dia	hing Diameter	×
ID	Tool No.	Name		Diameter	Material	Max. Depth	Angle	Corner Radius	
T010	10	1/4 Ball Mill		0.25	Carbide	2		0.125	
T014	14	1/4 Ball Mill:1		0.25	Carbide	2		0.125	
Add N	ew Tool Ne	ew Tool Diameter:	0.25	Add			(	Select	Cancel

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.

Select T	ool							×		
-0	Operation: Tool Type:	Finishing End Mill	<b>•</b>	Т	ool Diameter:	0.25	Display T Match All Dia	ning Diameter		
ID	Tool No.	Name	1	Diameter	Material	Max. Depth	Angle	Comer Radius		
T010	) 10	1/4 Ball Mill		0.25	Carbide	2		0.125		
T014	14	1/4 Ball Mill:1	(	0.25	Carbide	2		0.125		
Add I	Add New Tool New Tool Diameter: 0.25 Add Select Cancel									

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.

		Remachining End Mill	<b>_</b>	-	Tool Diameter	: 0.25		Tools hing Diameter ameters	x
ID	Tool No.	Name		Diameter	Material	Max. Depth	Angle	Comer Radius	
T010	10	1/4 Ball Mill		0.25	Carbide	2		0.125	
T014	14	1/4 Ball Mill:1		0.25	Carbide	2		0.125	
Add Ne		w Tool Diameter:	0.25	Add			(	Select	ncel

- 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.

👺 Surface Group Pa	rameters			<b>—</b> ———————————————————————————————————
Operations	Diam (d) 0.25 0.25 0.25	Tool ID T010 T010 T010	Strategy	
Toolpath Settings     Method     Machine All Si     Machine Select     Lock Toolpath			Suface Selection Selected Surfaces: 69 Clear Selection	No. of moves in Toolpath : 1912 Group Name: Side Cavity Select Tools Close Cancel Apply

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

See Model Area Clearance	
Model area clearance  Rest  Nall finishing  Unsafe segment removal  Flat machining  Flat pseed  Order  J Approach	Model area clearance Style Raster
- 😵 Limit - Voint distribution UL Leads and links - UL Start point - End point	Cut direction Profile Area Climb  Any Tolerance 0.001 Thickness 0.02 Stepover
	0.05 Stepdown Automatic v 0.05 OK Cancel

- **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.

Steep And Shallow Finishing		
Steep and shallow finishing	Steep and shallow finishing	
Point distribution	Order Steep first	Additional stock
i U Leads and links	Threshold angle	Steep shallow overlap
	Steep	Shallow Spiral
	Climb	Cut direction
	Stepdown	Stepover
	0.02 Calculate using cusp	0.015 Туре
	Cusp height	3D offset 👻
	Maximum stepdown 0.25	Smoothing
	Tolerance 0.001	
	Thickness 0	
		OK Cancel

- 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.

Second Corner Finishing		
Corner finishing	Corner finishing	
	Output	Style
	Both 👻	Along 👻
	Threshold angle	Cusp 0.005
	Tolerance	Cut direction
	0.001	Any 👻
	Thickness 0	
		OK Cancel

- **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

Second Corner Finishing	×
Corner finishing	Corner detection
<ul> <li></li></ul>	Reference tool Tool ID T010
	Overlap 0.1
	Detection limit 165
	Remove deep cuts 📝
	OK Cancel

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.

👺 Surface Group Pa	rameters			×
Operations   Roughing    Finishing   Remachining   Projection	Diam (d) 0.25 0.25 0.25	Tool ID T010 T010 T010	Strategy	
Toolpath Settings Method Machine All Si Machine Select Lock Toolpath			Suface Selection Suface Type: Machining Suface Selected Surfaces: 69 Clear Selection	No. of moves in Toolpath : 1912 Group Name: Side Cavity Select Tools Close Cancel Apply

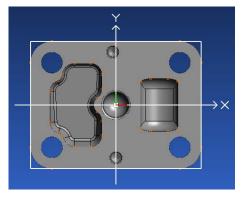
Click on the **<Close>** button in **Surface Group Parameters** dialog to return to the Face Window.

### **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

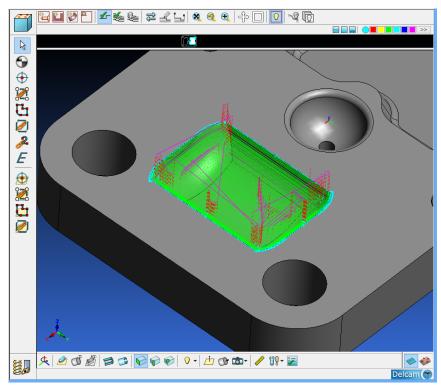
Once you have created the **Part Feature** and established a **Boundary Curve** you visually verify the tool path PartMaker has calculated. To do so:

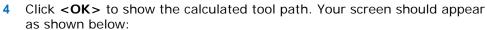


- 1 From the Face Window tool bar choose Verify Work Group Tool path.
- 2 Uncheck the **Enable Verification on Face Window**. As we are only working on the solid, we do not need to see the verification in 2D.
- **3** Enter a Verification Delay of 0.

**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.

Tool Path Verification Options	×
Verification on Face Window	_
Enable Verification on Face Window	
Show Tool as:	
Hollow	
Solid	
Do not show tool	
Verification on Solid Model	
Enable Verification on Solid Model	
Show Tool as:	
Solid	
O not show tool	
Show Toolpath	
Prompt for Separate Verification of each Operation	
Verification Delay (0-9)	
OK Cancel	
Calibei	







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.

Select To		Roughing End Mill	•	Tool Diamet	er: 0.25	-	Tools hing Diameter ameters	×
ID	Tool No.	Name	Diame	eter Material	Max. Depth	Angle	Comer Radius	
T010		1/4 Ball Mill	0.25	Carbide			0.125	
T014	14	1/4 Ball Mill:1	0.25	Carbide	2		0.125	
Add N	New Tool Ne	w Tool Diameter:	0.25	Add		(	Select	àncel

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.

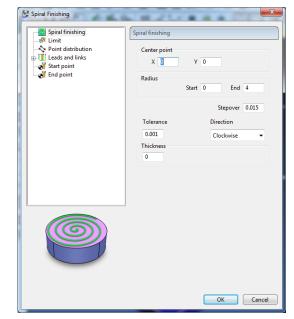
Select To	ol								X
0		Finishing End Mill	•	ī	Fool Diameter	0.25		Fools hing Diameter ameters	
ID	Tool No.	Name		Diameter	Material	Max. Depth	Angle	Comer Radius	
T010	10	1/4 Ball Mill		0.25	Carbide	2		0.125	
T014	14	1/4 Ball Mill:1		0.25	Carbide	2		0.125	
Add N	lew Tool Ne	w Tool Diameter:	0.25	Add			(	Select	ancel

- 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).

🖉 Mo	odel Area Clearance ×
Model area clearance Grief Step cuting Fest Wall finishing Fist machining Fist machining	Model area clearance         Style         Ciffset all         V         Cut direction         Profile       Area         Climb       Any         Tolerance         0.01         Thickness         0.01         Stepover         0.05         Stepdown
	Automatic v 0.05

- **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.

Operations	Diam (d)	Tool ID	Strategy	
Roughing	0.25	T010	Model Area Clearance 🗸 Options 💿	
Finishing	0.25	T010	🚺 💽 Spiral Finishing 🗸 Options 🔿	
Remachining			Comer Finishing 🗸 Options	
Projection			Pattem Finishing	
Toolpath Settings Method			Surface Selection	No. of moves in Toolpath : 6057 Group Name:
Machine All Su			Surface Selection Surface Type: Machining Surface -	Center Cavity
Machine Selected Surfaces			Selected Surfaces: 69	Select Tools

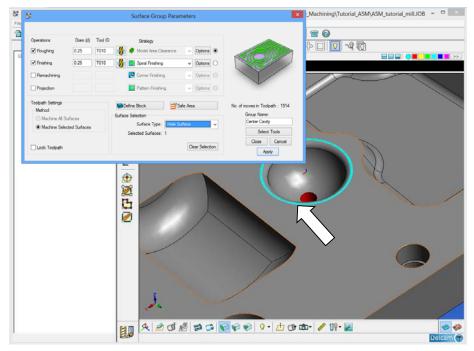
### **Select a Hole Surface**

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:

Hole Surface
Machining Surface
Hole Surface

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



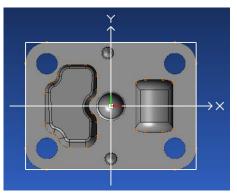
3 Click on the <**Close>** button in **Surface Group Parameters** dialog to return to the Face Window.

## **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:

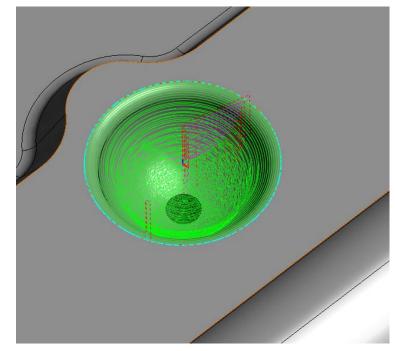
- 1 From the Face Window tool bar choose Verify Work Group Tool path.
- 2 Uncheck the **Enable Verification on Face Window.** As we are only working on the solid, we do not need to see the verification in 2D.
- **3** Enter a Verification Delay of 0.

**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.

Tool Path Verification Options	×
Verification on Face Window	
Enable Verification on Face Window	
Show Tool as:	
Hollow	
<ul> <li>Solid</li> </ul>	
O Do not show tool	
Verification on Solid Model	
Enable Verification on Solid Model	
Show Tool as:	
Solid	
Do not show tool	
Show Toolpath	
Prompt for Separate Verification of each Operation	
Verification Delay (0-9) 🚺	
OK Cancel	
UN Candel	



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

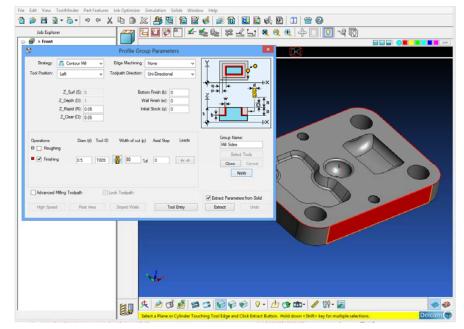
# **Define Profiles for Contour Mill Cycle**

In this section of the tutorial you will mill the outside of the part using a 2  $\frac{1}{2}$  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>.



-			-			
👺 Profile Grou	p Parameters					
Strategy:	🚨 Contour Mill	- Edg	ge Machining:	None	•	
Tool Position:	Left	- Toolp	ath Direction:	Uni-Directional	-	
	Z_Surf (S):         0           Z_Depth (D):         1           Z_Rapid (R):         0.5           Z_Clear (Cl):         0.05		Wal	n Finish (b): 0 I Finish (w): 0 I Stock (q): 0		$\begin{array}{c} \bullet & \bullet \\ P \rightarrow & \bullet \\ Z \rightarrow & \bullet \\ S \rightarrow & \bullet \\ b \neq \\ b \neq \\ \bullet \\ \bullet \\ \bullet \\ \bullet \\ \end{array}$
Operations V Roughi Finishin	ing 0.5	Tool ID T009	Width of cut (p)	Axial Step 0	Leads In Out -> <=	Group Name: Mill Sides Select Tools Close Cancel Apply
High Speed	Milling Toolpath	Lock	Toolpath oped Walls	Тоо	l Entry	Extract Parameters from Solid Extract Undo

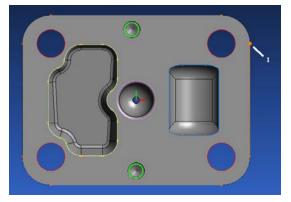
When you are finished your dialog should appear as shown below:

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

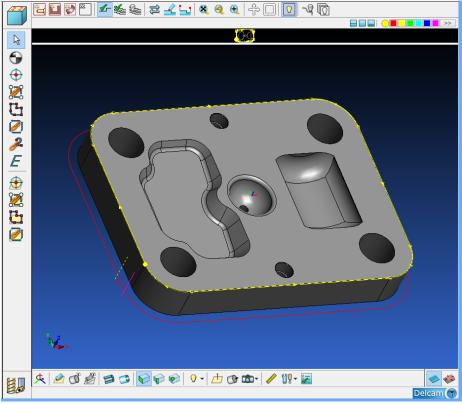


- 1 From the Face Window tool bar choose Verify Work Group Tool path.
- 2 Uncheck the **Enable Verification on Face Window**. As we are only working on the solid, we do not need to see the verification in 2D.
- **3** Enter a Verification Delay of 0.

**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.

Tool Path Verification Options	x
Verification on Face Window	
Enable Verification on Face Window	
Show Tool as:	
Hollow	
<ul> <li>Solid</li> </ul>	
O not show tool	
Verification on Solid Model	
Enable Verification on Solid Model	
Show Tool as:	
<ul> <li>Solid</li> </ul>	
Do not show tool	
☑ Show Toolpath	
Prompt for Separate Verification of each Operation	
Verification Delay (0-9)	
OK Cancel	

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**

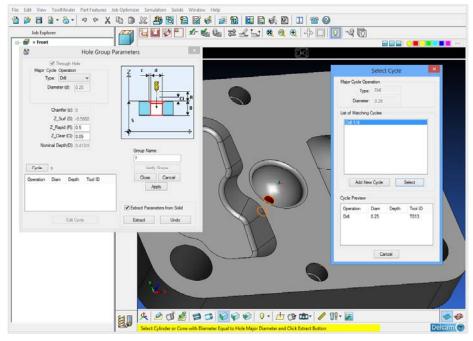
In this section of the tutorial you will drill the hole in the center cavity

1 Choose New Color.



ſ	000	_
н	35	
Ŀ	000	_1
L		
		-0

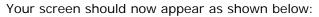
- 2 Choose New Hole Group from the Part Features menu to display the Hole Group Parameters dialog.
- 3 Leave the Through Hole box checked.
- 4 Leave the **Type** drop down menu set to **Drill**.
- 5 Leave the **Diameter (d)** field set to 0.25 (6.35).
- 6 Click on the "Extract Parameters from Solid" check box.
- 7 Click on the Solid Model <u>inside</u> the hole in Center Cavity.
- 8 Click on the "Extract" button and select the Drill-1/4 (Drill\_6.35) cycle.
- In the Select Cycle dialog click the <Select> button to choose selected cycle

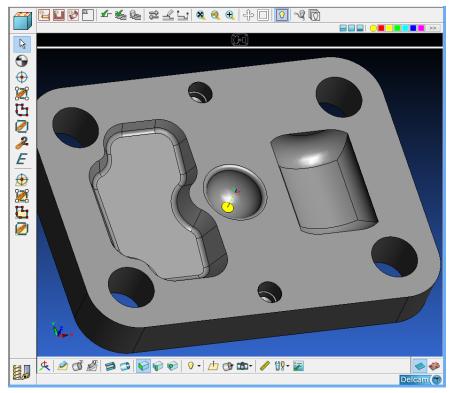


- **10** The **Hole Group Parameters** dialog has been filled in using information from the solid model.
- 11 Click the **<Close>** button to close the **Hole Group Parameters** dialog.



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.





## **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

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

Process Table Options	×
Retain Process Table Modifications     Reorder Processes to Minimize Tool     Machining Order by:	Change Time
<ul> <li>Tools</li> <li>Faces</li> </ul>	Cancel OK

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

🥩 Simulation 📅 Tool Assembly 🚺 Renumber Tools 🏢 View 🔻 🥞 Process Status 👻										
Pro	oc ID	Tool ID	Tool No.	Tool Name	Proc Task	Group	Face	Feed	Speed	Time(min)
<b>9</b>	P01	T001	1	Center_7/32	DRILL	Bore-1	Front	29.1upm	5000rpm	0.05
Ø	P02	T003	3	Drill_31/32	DRILL	Bore-1	Front	25.8upm	1377rpm	0.27
<u>)</u>	P03	T002	2	Bore_1	BORE	Bore-1	Front	12.4upm	3700rpm	0.80
9	P04	T004	4	Chamfer-1.25	SPOT FACE	Bore-1	Front	14.4upm	1832rpm	0.28
<b>8</b>	P05	T001	1	Center_7/32	DRILL	Tap-1/2	Front	29.1upm	5000rpm	0.05
Ø	P06	T005	5	Drill_27/64	DRILL	Tap-1/2	Front	32.4upm	3159rpm	0.14
8	P07	T006	6	Tap_1/2	FLOAT TAP	Tap-1/2	Front	52.4upm	681rpm	0.12
7	P08	T004	4	Chamfer-1.25	SPOT FACE	Tap-1/2	Front	14.4upm	1832rpm	0.20
0	P09	T010	10	1/4 Ball Mill	Surface Roug	Rough F	Front	19.8upm	5000rpm	54.43
0	P10	T010	10	1/4 Ball Mill	Surface Finisl	Finish P	Front	19.8upm	5000rpm	17.89
0	P11	T010	10	1/4 Ball Mill	Surface Roug	Side Cav	Front	19.8upm	5000rpm	41.78
0	P12	T010	10	1/4 Ball Mill	Surface Finisl	Side Cav	Front	19.8upm	5000rpm	8.01
0	P13	T010	10	1/4 Ball Mill	Surface Rema	Side Cav	Front	19.8upm	5000rpm	3.10
0	P14	T010	10	1/4 Ball Mill	Surface Roug	Center (	Front	19.8upm	5000rpm	9.31
0	P15	T010	10	1/4 Ball Mill	Surface Finisl	Center (	Front	19.8upm	5000rpm	1.82
0	P16	T009	9	End Mill_1/2	Contour Rou	Mill Side	Front	35.7upm	4577rpm	0.97
Į.	P17	T013	13	Drill_0.25	DRILL	Drill-1/4	Front	34.0upm	5000rpm	0.02
			undh Tota	l Time: 142.08 min.						

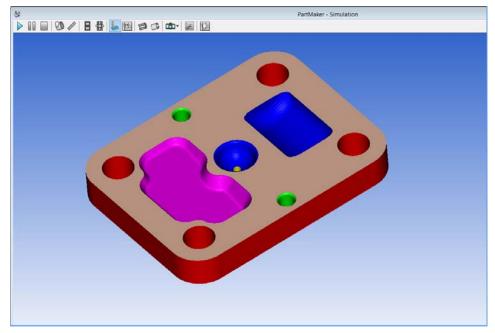
**Note:** If tool numbers are not in the proper order, you can choose **Renumber Tools** from the **Job Optimizer** menu to automatically renumber the tools before generating an NC part program.

# **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.

Organize 🔻 New folde	r			** 🗖
☆ Favorites	34X_Rotary_Table	CONTOUR.PST	FAGOR.PST	G FANUC_180.pst
	5X_Rotary_Table	CRUSADER-G.PST	FAGOR_NO_ATC.PST	FANUC_0.PST
🥽 Libraries	퉲 Tombstone	CRUSADER-M.PST	FANUC_6.PST	FANUC_ROTARY_A.PST
	A-B_8200.PST	DEMO.PST	FANUC_10.PST	FANUC_WITHOUT_M6.PS
Computer	ACRA_850.PST	DX32.PST	FANUC_11.PST	🗐 G&L_NUM_800.PST
	ACRA_1000.PST	DYNACONV.PST	FANUC_15.PST	🗐 G&L_NUM_800m.PST
辑 Network	ACRA_2100.PST	DYNAPATH_CONV.PST	FANUC_16LPST	💿 GE1050_U.PST
1	ANILAM.PST	DYNAPATH_EIA_2.PST	FANUC_18LPST	HAAS_VMC.PST
	MILAM_1400.PST	DYNAPATH_EIA_4.PST	FANUC_20i.pst	HARDINGE_VMC_FANUC.
	ANILAM_1400_NO_ATC.PST	EZPATH.pst	FANUC_21i.pst	HARDINGE_VMC_SIEMENS
	BOSS_4A.PST	FADAL.PST	FANUC_30i.pst	HH_CONV.PST
	BOSS_6.PST	FADAL_400.PST	FANUC_31i.pst	HITACHI_SEIKI.pst
	Boss_10.pst	FADAL_ROTARY_A.PST	FANUC_32i.pst	HURCO_ULTIMAX.PST
	CENTROID.pst	FADAL-E.PST	FANUC_160.pst	LEBLOND.PST
	•			
<b>5</b> 1	me: DEMO.PST			All Files (*.PST)

G

**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.

Post Options	<b>— X</b> —
Main Program No.:	Block Statt: 5 Block Increment: 0 4 th Axis Output (a) X Axis Rotary (b) Y Axis Rotary (c) Y Axis Rotary Rotary 5 Axis Table Output (c) Local Coordinates (c) Global Coordinates (c) Global Origin (c) Part Origin (c) V Axis (c) V Axis
Output Control	OK Cancel

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:

⊱ Save NC Program F	ile as:		×
🕘 🕘 - 🚺 🕨 Ca	omputer 🕨 Acer (C:) 🕨 PartMaker 🕨 pm-mill 🕨 🔍 👻 🤹	Search pm-mill	<mark>ب</mark>
File <u>n</u> ame:	smwpractice.TXT		•
Save as <u>t</u> ype:	All Files (*.TXT)		•
💌 <u>B</u> rowse Folders		Save	incel

- 8 Enter the name smwpractice.TXT for the NC program.
- 9 Click **<Save>** to generate the NC Program.

NC Program generated is shown below.

👺 PartMaker - smwpractice.TXT	
% :12 (TOTAL MACHINING TIME: 197.4992) (MACHINING FRONT FACE) (MACHINING PROCESS 1) G80 G40 G17 G30 G91 Z0 T1 M6 (CENTER_7/32 #2 CENTER DRILL) (TIME FOR THIS TOOL: 0.047) G90 G54 G0 X-3.0 Y2.0 S5000 M3 G43 Z0.5 H1 M8 Z0.5 G98 G81 R0.05 Z-0.1 F29.1 Y-2.0 X3.0 Y2.0 G80 M9 C01 G28 Z0 M5	
G91 G28 Z0 M5 (MACHINING PROCESS 2) T2 M6 (DRILL_31/32 TIN COATED DRILL) (MACHINING FRONT FACE) (TIME FOR THIS TOOL: 0.266) G90 G54 G0 X-3.0 Y2.0 S1377 M3 G43 Z0.5 H2 M8 Z0.5 G98 G81 R0.05 Z-1.341 F25.8 Y-2.0 X3.0 Y2.0 G80 M9 G91 G28 Z0 M5	
(MACHINING PROCESS 3) T3 M6 (BORE 1 KENNAMETAL BORING BAR) (MACHINING FRONT FACE) (TIME FOR THIS TOOL: 0.804) G90 G54 G0 X-3.0 Y2.0 S3700 M3 View Program Edit Program Verifi	y Program

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

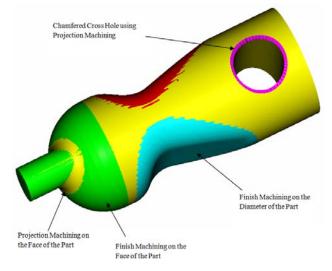
## Introduction

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

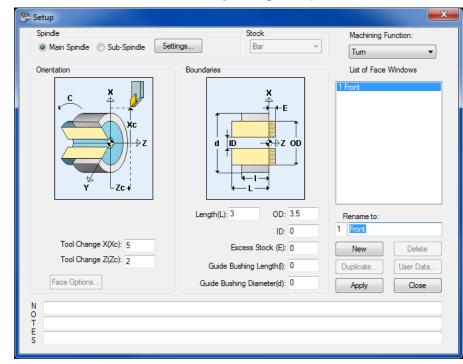
Below are the major concepts that will be covered in this tutorial.



- 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.

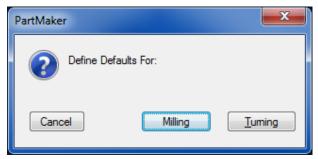
## **Set Defaults**

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:

Hole Group Parameters		Process Parameters	Machining Data
Through Hole Diameter:	0.25	Apply Comp in PartMaker	Upm Upr
Chamfer:	0.20	Coolant: Standard 🗸	Feed: 0.1 0.001
Clearance:	0.05	Feed Units: 💿 upr 💿 upm	Max Feed: 240 200
Nominal Depth:	0	Constant Surface Speed	Rapid Feed: 240 upn
Profile Group Parameters		Default Feed: 0.01 upr	Max
X Finish:	0		Speed: 8000 rpm
Z Finish:	0	Default Speed: 100 fpm	Tool Change Time: 0 min
Return Length:	0.05	Primary Tool Post: Gang Slide 🔹	Change Time: 0 min
Return Angle:	45	Retract from Groove Options	Leads
Surface Roughness:	32		Arc Radius: 0
Chamfer Length:	0	Groove Options Defaults	Line Length: 0.02
# Spring Passes:	1		Lead Angle: ()
<ul> <li>Start at Cutting Point</li> <li>Return to Cutting Point</li> <li>Thread Height: % Pitch</li> </ul>	•	Input Options Tool Path Opt Diameter Programming Corner Rou	OK

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:

PartMaker		×
Define De	faults For:	
Cancel	Milling	<u>T</u> uming

- 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)

Hole Group Parameters	Process Parameters	Machining Data
Through Hole	Apply Comp in PartMaker	upm upr
Diameter: (	.25 Coolant: Standard 🔻	Min Feed: 0.1 0.001
Chamfer: 0		
Clearance: (	.05 Feed Units:  upr  upm	Feed: 800 200
Nominal Depth: (		Rapid Feed: 800 upm
Profile Group Parameters	Output Canned Cycles	
X Finish: 0	Default Feed: 0.01 upr	Max Speed: 5000 rpm
Z Finish: 0	Default Speed: 100 from	Tool
		Change Time: 0.25 min
Return Length: (		
Return Angle: 4	5 Retract from Groove Options	Leads
Surface Roughness: 6	3	Arc Radius: 0
Chamfer Length: ()	Groove Options Defaults	Line Length: 0.1
# Spring Passes: 1		Lead Angle: 0
Start at Cutting Point		
Return to Cutting Point	Input Options Tool Path C	. OK
Thread Height: % Pitch	Diameter Programming     Comer R	lounding
mead height: Fitch	Remainir	ng Stock Detection Cancel

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:

PartMaker		<b></b>
Define De	faults For:	
Cancel	Milling	<u>T</u> uming

- 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.



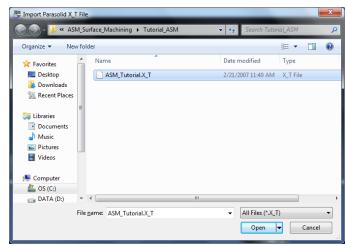
4 Click the **<Surfacing Defaults>** button in the **Defaults for Milling** dialog. The Surfacing Defaults dialog will display as shown below.

Surfacing Defaults		x
Parameters Colors		
Stratey parameters Stratey parameters Rest efficienting Rest efficienting Constant Z finishing Step and shallow finishing Step and shallow finishing Step and shallow finishing Step and shallow finishing Constant Z finishing Corner pencil finishing Corner pencil finishing Corner pencil finishing Corner pencil finishing Dimit Swarf finishing Dimit Load and Corner pencil finishing Dimit Load and links Start point Start point Start point	Model area clearance         Skyle         Offset all         Offset all         Cut direction         Profile       Area         Climb       Any         Climb       Any         Toickness       0         005       Stepover         005       Stepdown         Automatic       0.05	
	OK Cancel App	ly

# **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:

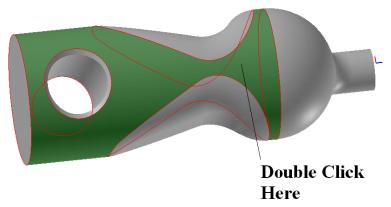
C:\PartMaker\pm-tm\ASM\_Surface\_Machining\Tutorial\_ASM \ASM\_Tutorial.X\_T as shown below:

	SM_Surface_	Machining   Tutorial_AS	M	<b>▼</b> <del>\$</del> 9	Search Tutor	ial_ASM	_	
Organize 🔻 🛛 N	ew folder					•		2
🚖 Favorites	^ Nar	ne		Date n	nodified	Туре		
🧮 Desktop		ASM_Tutorial.X_T		2/21/2	007 11:40 AM	X_T File		
鷆 Downloads								
📃 Recent Place	s							
	=							
🧊 Libraries								
Documents								
🁌 Music								
Pictures								
😸 Videos								
👰 Computer								
🏝 OS (C:)	<b>₹</b> ₹							
🏭 OS (C:) 👝 DATA (D:)								-
		ASM_Tutorial.X_T		•	All Files (*.X_1	D)		•

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:

👺 Surface Info				
Surface Type :	Cylinder			
Feature Type :	boss			
Radius:	0.474744:			
Height:	1.835716			
C Angle:	-0			
B Angle:	180			
Coordinate Sy Part	•			
Center 1	Center 2			
X1: 0	X2: 0			
Y1: 0	Y2: 0			
Z1: -2.615896	Z2: -0.780179			
<ul> <li>Transfer Geom</li> </ul>	etry			
Set Face Pl	ane as Selected			
Set Boundaries				
Transfer Unwrapped Geometry				
Proje	ct Outline			
	Close			

- 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:

👺 Setup			×
Spindle	Stock	Machining I	Function:
Main Spindle Sub-Spindle Sett	Bar 🔻	Tum	-
Orientation	Boundaries	List of Face	Windows
		1 Front	
	Length(L): 2.6159 OD: 1	Rename to:	
	ID: 0	1 Front	
Tool Change X(Xc): 2.8	Excess Stock (E): 0.01	New	Delete
Tool Change Z(Zc): 2	Guide Bushing Length(): 1	Duplicate	User Data
Face Options	Guide Bushing Diameter(d): 1.5	Apply	Close
N O T E S			

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**:

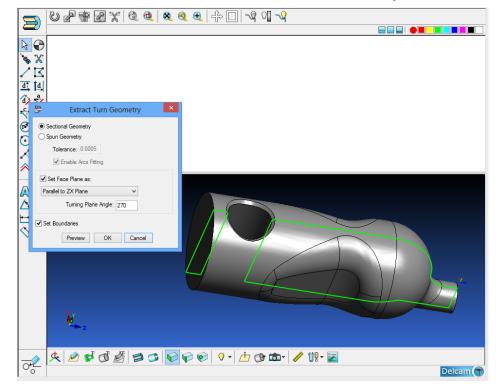
- 1 Choose Setup from the View menu
- 2 Check the Use Bar Stock box
- 3 Enter 2.6159 in the Length (L) field in the Setup dialog
- 4 The part will be made out of 1inch bar stock. Enter 1 in the OD field of the Setup dialog.
- 5 Enter .01 in the Excess Stock (E) field of the Setup dialog.
- 6 Enter 3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

Spindle Main Spindle Sub-Spindle Setti	Stock Slug 💌	Machining Function:
Orientation	Boundaries	List of Face Windows Front
	Length(L): 2.61589€ OD: 1 ID: 0	Rename to: 1 Front
Tool Change X(Xc): 5 Tool Change Z(Zc): 2 Face Options	Excess Stock (E): 0.01 Work Shift(): 3 Work Offset: G54	New     Delete       Duplicate     User Data       Apply     Close
N 0 T E S		

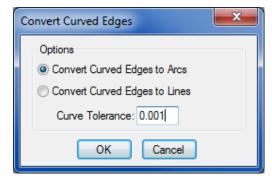
8 Click the **<Close>** button to close the **Setup** dialog.

# **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.

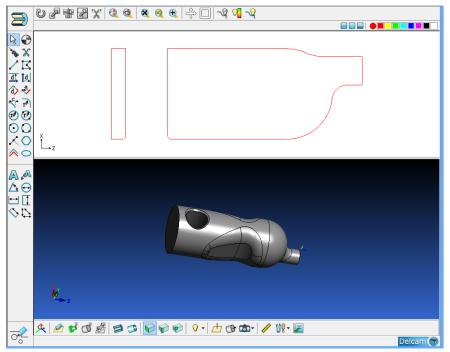


- 3 Set the "Turning Plan Angle" to 270 and click <OK>
- 4 Click the <Convert Curves to Arcs> button and click <OK> as shown below:





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>

lect Tool		Operation: Tool Type: Tum	Finishing	•				
ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T001	2/Gang	OD Tum 80-Right	Carbide	2	0.5	80	94	0.2
T003	44/Back	OD Tum-Sub	Carbide	2	0.5	80	94	0.2
Add	New Tool			(	Select	<b>C</b>	ancel	

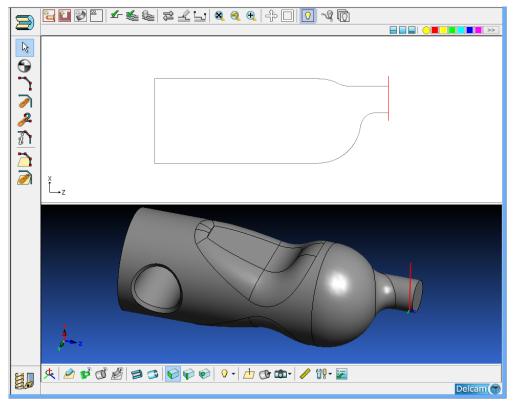
7 In the **Group Name** field, type "Face" and click the **<Apply>** button. Your **Profile Group Parameters** dialog should appear as shown below:

<del>č</del>	Profile Group Parameters,	Turn ×
Strategy: Dontourin		
Tool Location: Face X Finish (F Depth of Cut (c Initial Stock (c Diam Clearance (C Face Clearance (C Surface Roughne	x): 0 ): 0 ): 0 ): 0 ): 0 0 ): 0.05 ): 0.05	X     -Q.⇒     Cf       FX     -Q.⇒     Cf       FX     -Q.⇒     Cd       FX     -Q.⇒     Cd       Group Name:     -Q.⇒
Operations  Roughing  Finishing	Tool ID         Leads           T001	Select Tools Close Cancel Apply
Cutting Point (P): Machining Side Defined By: © Cutting Point O Tool Location	Pinch Tuming	

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:



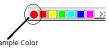
# **Creating the First OD Turning Operation**

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 TOO1 from the Select Tool dialog and click <OK>

ect Tool								
Þ		Operation: Tool Type: Tum	Finishing	•				
ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T001	2/Gang	OD Turn 80-Right	Carbide	2	0.5	80	94	0.2
T003	44/Back	OD Tum-Sub	Carbide	2	0.5	80	94	0.2
	New Tool				Select		ancel	

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

8 <del>.</del>		Profile G	roup Parameters, Turn	×
Strategy:	Contouring	~		
Tool Location:	Out	~		
Tool Orientation:	Right	~		X Is gal Alfred
	X Finish (Fx):	0		
	Z Finish (Fz):	0		
	Depth of Cut (d):			
	Initial Stock (q):	0		u lizza ja
				Fx +
	n Clearance (Cd):			→ Fz → Z
	e Clearance (Cf):			
Sur	face Roughness	32		Group Name: 1st Turn
				Select Tools
Operations		Tool ID	Leads	
	-			Close Cancel
Finish	hing	T001	<ul> <li></li></ul>	Apply
Cutting Point (P)		_	Pinch Turning	
Machining Side D			_ man roning	
<ul> <li>Cutting Point</li> </ul>				
O Tool Location				

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

## Create the Profile for OD Turning

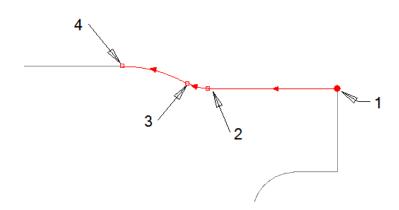
Now that you have created the part feature, you are ready to assign it to your part geometry to create a tool path. To do so:



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.

- ì
- 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>

Þ		Operation: Tool Type: Tum	Finishing	•				
ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T001	2/Gang	OD Tum 80-Right	Carbide	2	0.5	80	94	0.2
T003	44/Back	OD Tum-Sub	Carbide	2	0.5	80	94	0.2

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

		Profile Group	o Parameters, Tu	irn 🖻
Strategy:	Distance (	~		
Tool Location:	Out	~		
Tool Orientation:	Right	~		X Local Store
	X Finish (Fx):	0		
	Z Finish (Fz):	0		
	Depth of Cut (d):			
	Initial Stock (q):	0		J L J J J J J J J J J J J J J J J J J J
				Fx +
	Clearance (Cd): Clearance (Cf):			→ ← Fz → Z
	face Roughness			
Sur	race noughness	32		Group Name: 2nd Tum
Operations		Tool ID	Leads	Select Tools
	hina	100/10	20000	Close Cancel
Finish	-	тоо1	<· ->	Apply
• • • •	m ig		<>	тұру
Cutting Point (P)	:	Pine	ch Turning	
Machining Side De	efined By:			
Cutting Point				
O Tool Location				

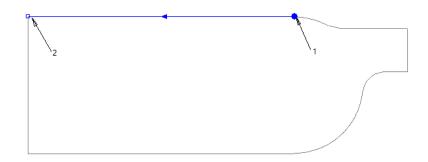
7 Click the <Close> button

## 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
- $\times$
- 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.



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

# **Creating a Cutoff Work Group**

1 Choose a New Color.

- ت
- 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>

		Tool Type: Gro	ove	•				
ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Dept
T002	4/Gang	Back Turning	Carbide	1	0.5	55	90	0
T004	1/Gang	Cut-off	Carbide	2	0.5	55	90	0
1004	17Gang	Cut-on	Carbide	2	0.5	55	30	U

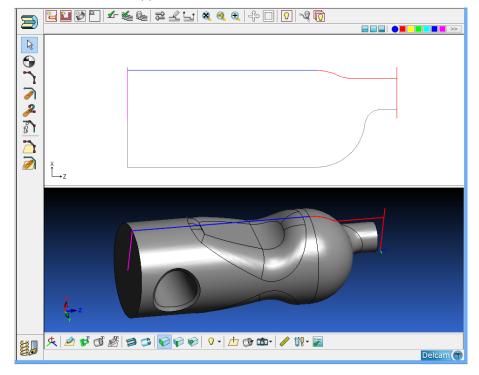
6 In the Cycle Name field, type "Cut-Off" and click the **<Apply>** button. Your **Profile Group Parameters dialog** should appear as shown below:

8	Profile Group	Parameters, Tur	m 📕
Strategy: JO Cut Off	~		
		Toolpath	Options: Insert Chamfer V
			× +
Cut-Off Distance	(D): 2.6159		
Chamfer OR Radius	; (q): 0		
Start X Point	(Xs): 0.5		• ×
End X Point	(Xe): 0		
Diam Clearance	(Cd): 0.05		
Axial S	itep: 0		→D€
			Group Name:
			Cutoff
Operations	Tool ID	Leads	Select Tools
Roughing	тоо2	<>	Close Cancel
			Apply
Groove Options			
Cutting Point (P):	Opt	ional Path 1->2->1	

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

H

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.

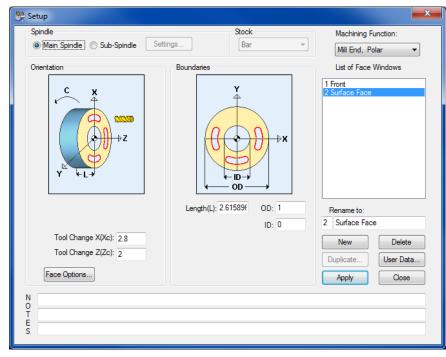


#### Create a New Face Window for Surfacing on the Face

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.

2

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:

<b>E</b>			Surface Group Parameters	x
Operations  Roughing  Finishing  Remachining	Diam (d)	Tool ID	Strategy  Model Area Clearance  Options  Raster Finishing  Options  Options  Options  Options	
Projection	0.3	T004	V Pattern Finishing	
Toolpath Settings Method O Machine All Si Machine Select			Surface Selection     Surface Type:     Machining Surface     No. of moves in Toolpath : 203       Surface Selection     Group Name:       Surface Type:     Machining Surface     Pattern Finishing       Selected Surfaces:     27     Select Tools	
✓ Lock Toolpath			Clear Selection Cancel Cancel	

#### 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

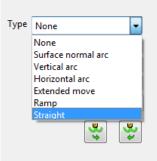
Pattern Finishing  Pattern Finishing  Multiple cuts  Summary Point distribution  Cut Start point  Cut Start	Pattern finishing
	Tolerance DOOL Thickness D
	OK Cancel

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.

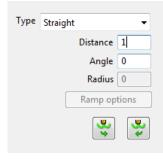
2 Click on **Lead in** the tree to see the lead-in options.

👺 Pattern Finishing	
Pattern finishing — Multiple cuts — 100 Limit	Lead in
Point distribution □ U Leads and links	Type None -
	Distance 0
Lead out	Angle 0 Radius 0
Links	Ramp options
End point	<b>Š</b>
V	OK Cancel

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

Tolerance	
0.005	
Thickness	

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

Lii	mit		
	Boundary		
		Trimming	Keep inside 🔻
	Block	Limit tool	Past block 🔹
	Z limits	- 🗖 M	laximum 0
		M	linimum 0

- 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:

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

Toolpath Settings
Method
Machine All Surfaces
Machine Selected Surfaces

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:

Surface Selection	
Surface Type:	Machining Surface
Selected Surfaces:	Machining Surface Collision Surface Ignore Surface
	Hole Surface

- **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:

Surface Selection	
Surface Type:	Machining Surface 🔹
Selected Surfaces:	0
	Clear Selection

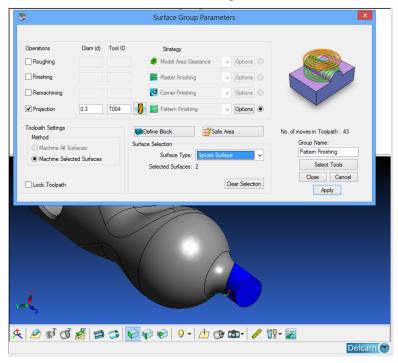
5 In the **Solid Model File** window, click on the base of the boss and the surface adjacent to it as indicated in the picture below:

	2 🛛 📎 🎦 🖌 🌜 💺	≇≟⊒'≋ 🧶 🕀 🗋 🔽 °	Q 10
	Box		
	<b>*</b>	Surface Group Parameters	×
$\odot$			
C-1	Operations Diam (d) Too	0.00035	
	Roughing	State Model Area Clearance V Options	
2	Finishing	Raster Finishing V Options	
<i>8</i> -	Remachining	Comer Finishing V Options	
	Projection 0.3 T00	🛛 🚺 🔀 Pattern Finishing 🗸 Options 💿	$\checkmark$
	Toolpath Settings	Define Block	No. of moves in Toolpath ; 43
1	Method	Surface Selection	Group Name:
_	Machine All Surfaces     Machine Selected Surfaces	Surface Type: Machining Surface v	Pattern Finishing
		Selected Surfaces: 2	Select Tools Close Cancel
	Lock Toolpath	Clear Selection	Apply
	J J		
	×		
	- al		
8.	冬 🖉 🖉 🖉 🖉	🚱 🏟 🐑   🖓 -   📥 😘 🎰   🥒 😘	
			Delcam (🌱

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.

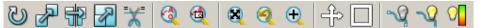
Double click on the face of the solid model as indicated below to open the

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

- 00
- Click the CAD/CAM switch to enter the CAD mode.
- 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.

👺 Surface Ir	nfo ×				
Surface Type :	Plane				
Angle with XY Plane:	0				
C Angle:	-0				
B Angle:	0				
X Offset:	0				
Y Offset:	0				
Z Offset:	0				
Transfer Geometry					
✓ Set Face Plane as Selected					
Set Boundaries					
Transfer Planar Geometry					
Project Outline					
Close					

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.

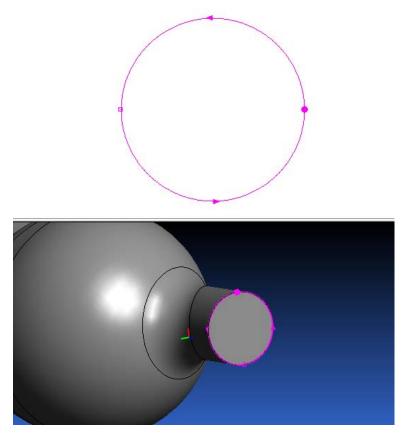


- 5 Switch to the CAM mode.
- 6 Choose the Chain Geometry icon
- 7 Click your cursor at the 9 o'clock position on the circle as indicated below to apply the pattern curve to the geometry.



1

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:

Curve Properties	×
Curve Type Boundary Pattern Top Swarfing Pattern Bottom Swarfing Pattern	Used in Operation Roughing Remachining Projection
Style: Tool Position: Boundary Type: OK	2D Curve

**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

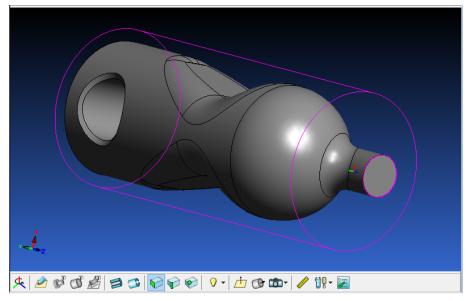
Operations	Diam (d) Tool ID	Strategy		
Roughing		🦸 Model Area Clearance	✓ Options ○	
Finishing		Raster Finishing	✓ Options ○	
Remachining		Comer Finishing	✓ Options ○	
<ul> <li>Projection</li> </ul>	0.3 T004	🎲 🔀 Pattem Finishing	✓ Options ●	
Foolpath Settings Method		Define Block	afe Area	No. of moves in Toolpath : 49
Machine All Si	urfaces	Surface Selection		Group Name:
Machine Sele		Surface Type: Machini	ing Surface 🗸 🗸	Pattern Finishing
0		Selected Surfaces: 2		Select Tools

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:

Define Block		×
Block Limits*		
Defined by: Cylinder - Z 🔹		
Diameter (D): 1.125		¥
Min Z (z1): -2.7408: 😭 Max Z (z2): 0.125 😭		Ĵ,,₂
Calculate	z1	1 z2
Expansion: 0.15		
From Selection From Stock		
Use Custom Shape	OK Apply	Cancel
*Block Limits are Defined in the Part Coordinate System		

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

2

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

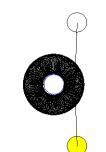
In the **Tool Path Verification Options** click the **Do not show tool** radio button as shown below.

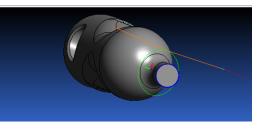
3 Enter a Verification Delay of 0.

**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.

Tool Path Verification Options
Verification on Face Window
Enable Verification on Face Window
Show Tool as:
<ul> <li>Hollow</li> </ul>
Solid Stock Motion Simulation
O Do not show tool
Verification on Solid Model
Enable Verification on Solid Model
Show Tool as:
Solid
O not show tool
Show Toolpath
Prompt for Separate Verification of each Operation
Verification Delay (0-9)
OK Cancel

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
- 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:

8 <del>0</del>			Surface Group Parameters	×
Operations	Diam (d)	Tool ID	Strategy  Model Area Clearance  Options	
Finishing     Remachining     Projection	0.3	T004	Image: State Finishing     Options       Image: Comer Finishing     Options       Image: Comer Finishing     Options       Image: Comer Finishing     Options	
Toolpath Settings Method Machine All Su Machine Select			Surface Selection Surface Type: Machining Surface  Selected Surfaces: 29 Clear Selection	No. of moves in Toolpath : 2170 Group Name: Offset Select Tools Close Cancel Apply

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

<ul> <li>3D Offset Finishing</li> <li>3D offset finishing</li> <li>Wind Listibution</li> <li>Unit Leads and links</li> <li>Start point</li> <li>End point</li> </ul>	3D offset finishing Spiral Smoothing Reverse toolpath Tolerance 0.001 Thickness 0 Stepover 0.05	segment order Cut direction Climb
		OK Cancel

- 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

Block			
	Limit tool	Past 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**

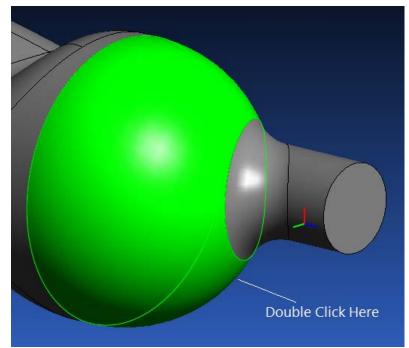
Boundary curves are created the same way as 2D profile curves. They are used as tool path boundaries to restrict the machining area. In this example, you will create two boundary curves.

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.

👺 Surface	Info ×
Surface Type :	Sphere
Feature Type :	boss
Radius:	0.474744
Coordinate System	n
Center X:	0
Center Y:	-0
Center Z:	-0.780179
Project Ou Close	

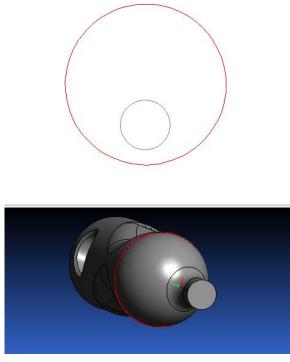
4 Click the Convert Curved Edges to Arcs radio button

Convert Curved Edges	×
Options Convert Curved Edges to Arcs Convert Curved Edges to Lines Curve Tolerance: 0.0005	
OK Cancel	



- 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.

Curve Properties	×
Curve Type Description: Pattern Top Swarfing Pattern Bottom Swarfing Pattern	Used in Operation          Image: Constraint of the second secon
Style: Tool Position:	3D Curve
Boundary Type:	Contact Point   Cancel

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:

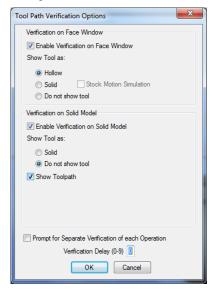


1

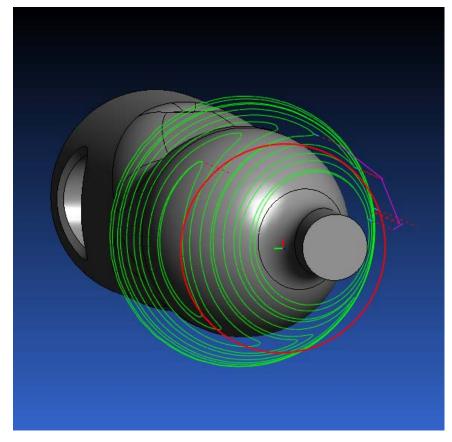
2

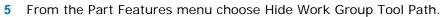
From the Part Features menu choose Verify Work Group Tool Path.

- 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:



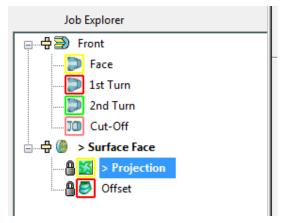


Note: You should always hide your tool path after verifying it.

**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:



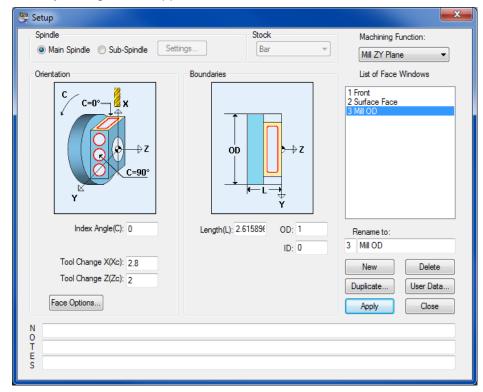
# Creating Surface Finishing Tool Paths on the Diameter of the Part

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

		)	Ľ	000
--	--	---	---	-----

- 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.
- Enter a tool diameter of 0.25 (6.0) in the Diam (d) field and click the 3 <Select Tools> button.
- From the Strategy drop down menu choose Raster Finishing. 4
- 5 In the Group Name field enter "Mill Sides" and click < Apply>. Your completed dialog should appear as shown below:

Operations	Diam (d)	Tool ID	Strategy	(Theory)	
Roughing	0.25		Dodel Area Clearance	Options	
Finishing	0.25	T019	🚺 📕 Raster Finishing 🔹	▼ Options	
Remachining			Comer Finishing	Options	
Projection			Pattern Finishing	Options	
Toolpath Settings Method			Define Block	Area No. of moves in Toolpath : 0	
Machine All Su	faces		Surface Selection	Group Name:	
Machine Select			Surface Type: Machining S	Surface - Mill Sides	
	tea sanaces		Selected Surfaces: 30	Select Tools	

- 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:

Raster Finishing	Raster finishing
Imit         Imit      <	Angle Start corner Lower left  Perpendicular pass Perpendicular pass Shallow angle 30 Optimize parallel pass
	Ordering Style Two way Tolerance 0.001 Thickness
	0 Stepover 0.05
	OK Cancel

- 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:

e 🖉	Raster Finishing
□-■     Raster finishing       □-□     □       □     □       □     □	Links
Control Contro Control Control Control Control Control Control Control Control Co	Short / Long threshold 1 Short On surface Long Skim Retract and approach moves Along Tool axis Automatically extend Maximum length 10 Retract distance 0 Approach distance 0
	OK Cancel

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



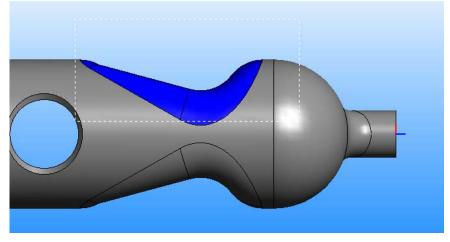
4 To assure you select the correct surfaces, click the Set View icon from the solids icon toolbar at the bottom of the screen and choose the ZX View as shown below:

Ctrl+Alt+D
Ctrl+Alt+X
Ctrl+Alt+Y
Ctrl+Alt+Z
Ctrl+Alt+I

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:

Toolpath Type:	3+1 Axis		•		
Remachining	Diam (d)	Tool ID T003	Strategy Model Area Clearance  Raster Finishing  Comer Finishing	Options     Options     Options     Options	
Projection Toolpath Settings Method Machine All Surfar Machine Selected Lock Toolpath		Sur	face Selection	Coptions	No. of moves in Toolpath : 541 Group Name: Mill Sides Select Tools Close Cancel Apply

**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

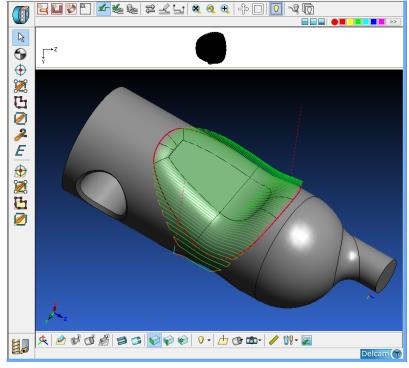
Once you have created the **Part Feature** selected the surfaces to machine,m you can visually verify the tool path PartMaker has calculated. To do so:



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:





Δ

- 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).

👺 Setup		<b>— X</b> —
Spindle Main Spindle   Sub-Spindle   Se	ttings Stock	Machining Function:
Orientation	Boundaries	List of Face Windows
C C - 0° X X C - 90° V Index Angle(C): 0	Length(L): 26158% OD: 1	1 Front 2 Surface Face 8 Mit (0) Rename to:
	ID: 0	3 Mil OD
Tool Change X(Xc): 2.8		New Delete
Tool Change Z(Zc): 2		Duplicate User Data
Face Options		Apply Close
N O T E S		

- 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:

Duplicate Face	<b>X</b>
Parameters Number of Copies (N): 1 Index Angle Increment (A): 180 Insert Into List of Faces IDisplay Parameters Next to Face Names ** OK Cancel * Enabled only when Groups of Features are defined on this Face ** Parameters will be displayed next to the Face Name as (N,A)	

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:

Spindle Main Spindle Sub-Spindle	Settings Stock		Machining Fu	inction:
Main spindle O Sub-spindle	Dai	· ·	Mill ZY Plane	
Orientation	Boundaries		List of Face V	Vindows
C C=0° X Y C=90°		≻→ z	2 Surface Face 3 Mill OD(1,180)	
Index Angle(C): 0		D: 1	Rename to:	
T 101 - 201 - 20		ID: 0	3 Mill OD	
Tool Change X(Xc): 2.8			New	Delete
Tool Change Z(Zc): 2			Duplicate	User Data
Face Options			Apply	Close
N				
0 T				
E S				

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**

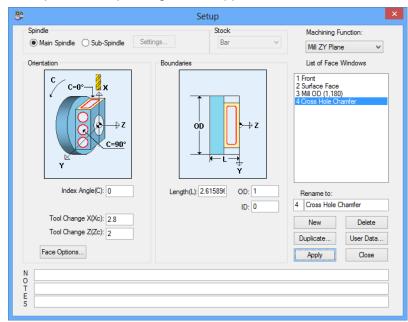
### Method 1: When the Solid Model already has a Chamfer

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**

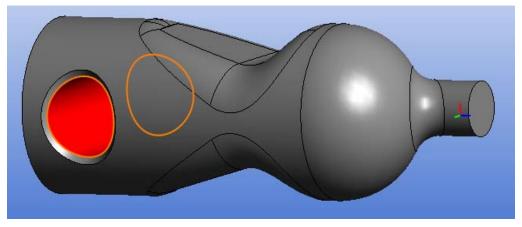


1

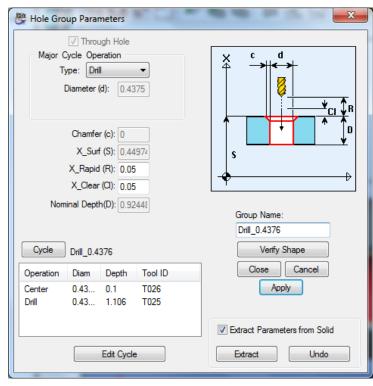
Choose New Hole Group from the Part Features menu to display the Hole Group Parameters dialog.

Here you will be making a through hole, so make sure to leave the  $\ensuremath{\text{Through}}$  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.

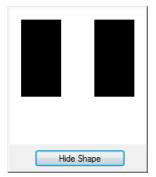


- 5 Once the hole is selected, click the <Extract> button
- 6 From the Select Cycle dialog, choose the Drill\_0.4376 Cycle and click the <Select> button. Your completed Hole Group Parameters dialog should appear as shown below.



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.

Select Tool	Projection				Display	Tools hing Diameter
Tool Type:			Tool Diamete	r: 0.25	-	ameters
	Drill	Diameter	Material	Max. Depth	Angle	Comer Radius
T019 20/Turr	Ream Bore End Mill Face Mill Slot Mill	0.25	HSS	2		0.125
Add New Tool	Dove Tail Comer Rounding Thread Mill ew Tool Diameter: 0.25	Add			(	Select Cancel

- 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:

Operations	Diam (d)	Tool ID	Strategy		
Roughing			🐠 Model Area Clearance	▼ Options ○	
Finishing			Raster Finishing	▼ Options ○	
Remachining			Comer Finishing	- Options	
Projection	0.25	T027	Pattern Finishing	- Options	$\checkmark$
Toolpath Settings Method Machine All Surfaces Machine Selected Surfaces			Surface Selection		No. of moves in Toolpath : 0 Group Name:
			Surface Type: Machinin	Select Tools	
Lock Toolpath			Selected Surfaces: 30	Clear Selection	Close Cancel

## **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.

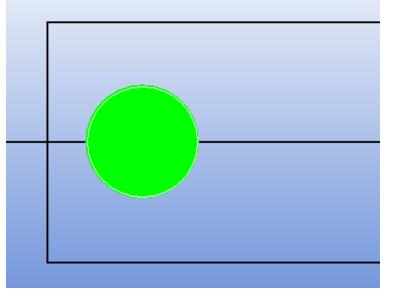
Click the **<Close>** button to return to the Face Window.

#### **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.

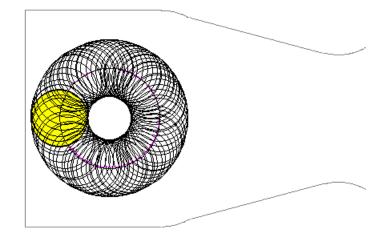
# Verify the Tool Path

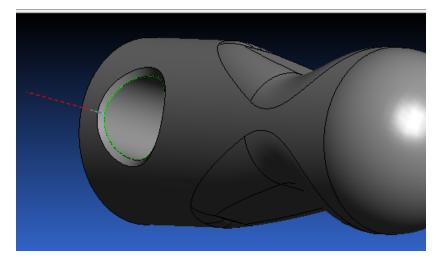
Once you have created the **Part Feature** and established a **Projection Curve** you can visually verify the tool path PartMaker has calculated. To do so:



- 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.



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

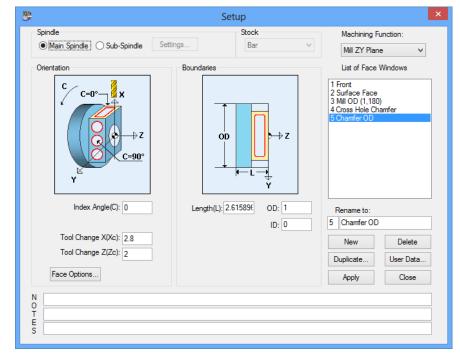
# Method 2: When the Solid Model Does Not Have a Chamfer

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.

Select Tool						X
Operation:	Projection				Display	Tools hing Diameter
Tool Type:	Center		Fool Diameter	0.25	© All Dia	ameters
ID Tool No		Diameter	Material	Max. Depth	Angle	Comer Radius
T019 20/Turr	Ream Bore End Mill Face Mill Slot Mill	0.25	HSS	2		0.125
Add New Tool	Dove Tail Comer Rounding Thread Mill ew Tool Diameter: 0.25	Add			(	Select Cancel

- 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:

Operations	Diam (d)	Tool ID	Strategy	
Roughing	0.25		Model Area Clearance 🚽 Options 🔿	
Finishing			Raster Finishing	
Remachining			Comer Finishing	
Projection	0.25	T027	Vettern Finishing Vettors	
Toolpath Settings			Define Block	No. of moves in Toolpath : 0
Method Machine All Surfaces Machine Selected Surfaces			Surface Selection	Group Name:
			Surface Type: Machining Surface 👻	Chamfer OD
	otod Sandoos		Selected Surfaces: 30	Select Tools

.....

**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:

👺 Pattern Finishing		<b>×</b>
Pattern finishing Multiple cuts - Multiple cuts - Mult	Pattern finishing	
	Tolerance 0.001 Thickness Thickness	0.025
		OK Cancel

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:

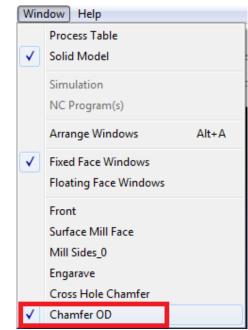
Win	dow Help	
	Process Table	
✓	Solid Model	
	Simulation	
	NC Program(s)	
	Arrange Windows	Alt+A
$\checkmark$	Fixed Face Windows	
	Floating Face Windows	
	Front	
	Surface Mill Face	
	Mill Sides_0	
	Engarave	
	Cross Hole Chamfer	
$\checkmark$	Chamfer OD	



12

- 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**



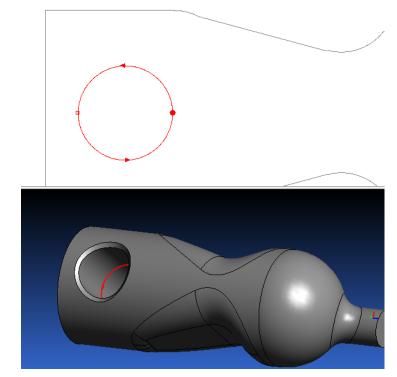
- 1 Choose the Chain Geometry icon
- 2 Click your cursor on the circle you just pasted into the face window. Doing so will create the projection curve. Your window should now appear as shown below:

# 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.

Process Table Options						
Retain Process Table Modifications						
V Move Cut-Off Process to the End of Process Table						
☑ Eject Part After Last Sub-Spindle Process						
Cancel OK						

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

№       P01       T001       2/Gang       OD Tum 80-Right       Face       Front       0.0010µr       351fpm       0.23       ♥       M150       ∰*         №       P02       T001       2/Gang       OD Tum 80-Right       1st Tum       Front       0.0010µr       351fpm       0.37       ♥       M150       ∰*         №       P02       T001       2/Gang       OD Tum 80-Right       1st Tum       Front       0.0010µr       351fpm       0.57       ♥       M150       ∰*         №       P05       T004       Ø/Gang       End Mill,0.15       3d Offset Finishing       Surface Mill Face       4.60µm       573µm       0.55       ♥       M150       ∰*         №       P05       T003       7/Gang       End Mill,0.25       Raster Finishing       Mill Soles_0       3.60µm       573µm       0.55       ₱       M150       ∰*         №       P03       T003       Z/Gang       End Mill,0.25       3d Offset Finishing       Mill Soles_100       3.60µm       573µm       0.51       ₱       M150       ∰*         №       P03       T003       Z/Gang       End Mill,0.25       3d Offset Finishing       Mill Soles_100       3.60µm       573µm <t< th=""><th>Pro</th><th>oc ID</th><th>Tool ID</th><th>Tool No.</th><th>Tool Name</th><th>Group</th><th>Face</th><th>Feed</th><th>Speed</th><th>Time(min)</th><th>Status</th><th>N</th><th>ode</th><th>Sync Group</th></t<>	Pro	oc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Status	N	ode	Sync Group
Image: Point	h	P01	T001	2/Gang	OD Turn 80-Right	Face	Front	0.0010upr	351fpm	0.23	ঘ	M150	4.	
P05         T004         b/Gang         End Mill_0.15         Jd Offset Finishing         Surface Mill Face         4.6upm         ST3pm         16.03         P         M150         ∰*           P05         T004         b/Gang         End Mill_0.15         Pattern Finishing         Surface Mill Face         4.6upm         ST3pm         16.03         P         M150         ∰*           P07         T003         7/Gang         End Mill_0.25         Raterr Finishing         Mill Sole_0         3.8upm         ST3pm         17.32         P         M150         ∰*           P07         T003         7/Gang         End Mill_0.25         Batter Finishing         Mill Sole_100         3.8upm         ST3pm         17.32         P         M150         ∰*           P03         T001         2/Gang         C0         Tum 80 Hight         And Tum         Frent         0.0010upr         311pm         1.48         P         M150         ∰*           P03 <t001< td="">         2/Gang         C0         Tum 80 Hight         2.nd Tum         Frent         0.0010upr         311pm         1.48         P         M150         ∰*           P03<t001< td="">         2/Gang         Dirit_0.4376         Dirit_0.43105         Side_hole_500</t001<></t001<>	bl	P02	T001	2/Gang	OD Turn 80-Right	1st Turn	Front	0.0010upr	351fpm	0.57	4	M150	₿*	
PO5         T004         8/Gang         End Mill_0.13         Pattern Finishing         Surface Mill Face         4.6upm         573pm         0.95         P         M150         ∰*           Ø         P07         T003         7/Gang         End Mill_0.25         Ratter Finishing         Mill Sides_0         3.8upm         573pm         17.32         P         M150         ∰*           Ø         P08         T003         7/Gang         End Mill_0.25         3d Offset Finishing         Mill Sides_180         3.8upm         573pm         6.21         P         M150         ∰*           Ø         P08         T003         7/Gang         End Mill_0.25         3d Offset Finishing         Mill Sides_180         3.8upm         573pm         6.21         P         M150         ∰*           Ø         P08         T001         2/Gang         C0 Tum 80-Right         2 net Tum         Prefet         0.0010µr         331fpm         1.48         P         M150         ∰*           Ø         P03 T007         7/Gang         Deli[0.4376         Deli[0.4376         1.8upm         384pm         0.76         P         M150         ∰*           Ø         P101         T008         Kriang         End Mili[0.4376 <td></td> <td>P05</td> <td>T004</td> <td>8/Gang</td> <td>End Mill_0.15</td> <td>3d Offset Finishing</td> <td>Surface Mill Face</td> <td>4.6upm</td> <td>573rpm</td> <td>16.03</td> <td>4</td> <td>M150</td> <td>₫.</td> <td></td>		P05	T004	8/Gang	End Mill_0.15	3d Offset Finishing	Surface Mill Face	4.6upm	573rpm	16.03	4	M150	₫.	
P07         T003         7/Gang         End Mill,0.25         Raster Finishing         Mill Sides,0         3.8upm         573pm         17.52         IP         M150         ∰*           P08         T003         7/Gang         End Mill,0.25         3.00 Offset Finishing         Mill Sides,180         3.8upm         573pm         8.21         IP         M150         ∰*           P08         T001         2/Gang         OD Tum 80-Hight         2.nd Tum         Front         0.0010µr         331pm         1.48         IP         M150         ∰*           P09         T007         7/Gang         Dnil,0.4178         Dnil,0.4178         Side_hole,50         1.8upm         938pm         0.76         IP         M150         ∰*           P10         T008         K/Gang         End Mill,0.4176         Pattern Finishing         Mill side_hole,50         0.5upm         4.30         Ø*         M150         ∰*           P101         KriGang         End Mill,0.4176         Pattern Finishing         Mill side_hole,50         0.5upm         4.30         Ø*         M150         ∰*           P111008         KriGang         End Mill,0.4176         Pattern Finishing         Mill side,500         0.5upm         4.30pm         3.30	-	P06	T004	8/Gang	End Mill_0.15	Pattern Finishing	Surface Mill Face	4.6upm	573rpm	0.95	9	M150	¢.	
Øp         POS         TOG3         7/Gang         End Mill,0.25         3d Offset Finishing         Mill Sole,180         3.8upm         573pm         8.21         P         M150         ∰*           IP         POS         TOG1         2/Gang         OD Tum 80-Right         2.nd Tum         Frent         0.0010µr         331fpm         1.48         P         M150         ∰*           IP         POS         TOG7         7/Gang         Dnill,0.4376         Dnill,0.4376         side_hole,50         1.8upm         932pm         0.76         P         M150         ∰*           IP         POS         TOG7         7/Gang         End Mill,0.4376         Pattern Finishing         Mill side_hole,50         0.5upm         4.10 pm         3.38         P         M150         ∰*           IP         POS         End Mill,0.4376         Pattern Finishing         did_hole,50         0.5upm         4.10 pm         3.38         P         M150         ∰*           IP         TOS8         EvGang         End Mill,0.4776         Pattern 2         chamfer 270         0.5upm         4.10 pm         3.38         P         M150         ∰*	0	P07	T003	7/Gang	End Mill_0.25	Raster Finishing	Mill Sides_0	3.8upm	573rpm	17.52	되	M150	4	
PO3 T001         2/Gang         0.01 um 80-Hight         2.nd Tum         Pront.         0.0010upr         3318pm         1.48         Pr         M150         ∰**           PO9 T007         7/Gang         Dnil_0.4376         Dnil_0.4376         side_hole_50         1.8upm         993pm         0.76         Pr         M150         ∰**           P10 T008         8/Gang         End Mil_0.4376         Pattern Finishing         side_hole_50         0.5upm         4J0rpm         3.38         Pr         M150         ∰**           P10 T008         8/Gang         End Mil_0.4376         Pattern Finishing         side_hole_20         0.5upm         4J0rpm         3.38         Pr         M150         ∰**           P11 T008         8/Gang         End Mil_0.4376         Pattern 2         charmfer;270         0.5upm         4J0rpm         3.27         Pr         M150         ∰**	0	POS	T003	7/Gang	End Mill_0.25	3d Offset Finishing	Mill Sides_180	3.8upm	573rpm	8.21	4	M150	₽.	
● PO9 T007 7/Gang Dnil[0.4376 Dnil[0.4376 side_hole_50 1.8upm 393ppm 0.76 F M150 骨* ▶ P10 T008 8/Gang End Mil[0.4376 Pattern Finishing side_hole_50 0.5upm 4J0ppm 3.38 F M150 骨* ▶ P11 T008 8/Gang End Mil[0.4376 Pattern 2 chamfer_270 0.5upm 4J0ppm 3.27 F M150 骨*	21	P03	T001	2/Gang	OD Turn 80-Right	2nd Turn	Front	0.0010upr	351fpm	1.48	4	M150	-B*	
후 P10 T008 8/Gang End Mill 0.4376 Pattern Finishing side_hole,90 0.5upm 430pm 3.38 F M150 뮾" • P11 T008 8/Gang End Mill 0.4376 Pattern 2 chamfer 270 0.5upm 430pm 3.27 F M150 뮾"		P09	T007	7/Gang	Dnil_0.4376	Drill_0.4376	side_hole_90	1.8upm	393rpm	0.76	9	M150	-B-	
🔁 P11 T008 8/Gang End Mill_0.4376 Pattern 2 chamfer_270 0.5upm 430rpm 3.27 🏴 M150 🛱	>	P10	T008	8/Gang	End Mill_0.4376	Pattern Finishing	side_hole_90	0.5upm	430rpm	3.38	ম	M150	÷	
부 P04 1002 1/Gang Cut-off Cutoff Front, 0.0016upr 1921pm 0.28 F M150/E ఉ가	*	P11	T008	8/Gang	End Mill_0.4376	Pattern 2	chamfer,270	0.5upm	430rpm	3.27	<b>V</b>	M150	÷	
	2	P04	T002	1/Gang	Cut-off	Cutoff	Front	0.0016upr	192fpm	0.28	ম	M1S0/E	<b>₽</b> *	

# **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:

- PartMaker Process Table - - -12 😪 Simulation 🍟 Tool Assembly 🚔 Insert 🎹 View 🕶 🧔 Process Status 🕶 🛞 Time Chart Synchronize Proc ID Tool ID Tool No. Tool Name Group Feed Speed Time(min) Status Mode Sync Group Face 
   P01
   T001
   2/Gang
   OD Turn 80-Right
   Face

   P02
   T001
   2/Gang
   OD Turn 80-Right
   1st Turn
   ⊽ ₽. Front 0.0010upr 351fpm 0.23 M150 2 2/Gang OD Turn 80-Right 0.0010upr 351fpm 0.57 ₽, Front M150 P03 T001 2/Gang OD Turn 80-Right 2nd Turn Front 5 0.0010upr 351fpm 1.48 M150 5 8/Gang End Mill\_0.15 1 8 P05 T004 M150 5 3d Offset Finishing Surface Mill Face 4.6upm 573rpm 16.03 
   P06
   T004

   P07
   T003

   P08
   T003

   P09
   T007

   P10
   T008

   P11
   T008

   P04
   T002
   6 8/Gang End Mill\_0.15 Pattern Finishing Surface Mill Face 4.6upm 573rpm 0.95 ⊽ M150 ₽. 0 • ₿, 7/Gang End Mill\_0.25 Raster Finishing Mill Sides\_0 3.8upm 573rpm 17.52 M1S0 0 ~ ₿, 7/Gang Drill\_0.4376 7/Gang End Mill\_0.25 3d Offset Finishing Mill Sides\_180 3.8upm 573rpm 8.21 M150 1 Drill\_0.4376  $\overline{\mathbf{v}}$ M1S0 ₿, side\_hole\_90 1.8upm 393rpm 0.76 ⇒ 8/Gang End Mill\_0.4376 Pattern Finishing side\_hole\_90 3.38 M150 8 0.5upm 430rpm + ₩ M1S0 8/Gang End Mill\_0.4376 Pattern 2 chamfer\_270 0.5upm 430rpm 3.27 4 ۲ M1S0/E 1/Gang Cut-off Cutoff Front 0.0016upr 192fpm 0.28 Material File: St\_toolw.mdb Main Spindle Time: 52.68 min. Sub Spindle Time: 0.00 min. Total Time: 52.68 min.
- 1 Click on process #3 (PO3) 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:

Proc	ID Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Status	M	ode	Sync Grou
		1001110.	roomanic	ologh	lace	i ccu	opeed	rinc(nin)	Status	1		- Sync Groc
J P	01 T001	2/Gang	OD Turn 80-Right	Face	Front	0.0010upr	351fpm	0.23	<b>V</b>	M1S0	<b></b>	
Цр	02 T001	2/Gang	OD Turn 80-Right	1st Turn	Front	0.0010upr	351fpm	0.57	শ	M1S0	<b></b>	
S P	05 T004	8/Gang	End Mill_0.15	3d Offset Finishing	Surface Mill Face	4.6upm	573rpm	15.11	5	M1S0	<b></b>	1
S P	06 T004	8/Gang	End Mill_0.15	Pattern Finishing	Surface Mill Face	4.6upm	573rpm	0.95	5	M1S0	₽	1
2 p	03 T001	2/Gang	OD Turn 80-Right	2nd Turn	Front	0.0010upr	351fpm	1.48	<b>N</b>	M150	<b></b>	1
Р	07 T003	7/Gang	End Mill_0.25	Raster Finishing	Mill Sides_0	3.8upm	573rpm	16.14	ঘ	M1S0	<b></b>	1
P	08 T003	7/Gang	End Mill_0.25	3d Offset Finishing	Mill Sides_180	3.8upm	573rpm	8.21	ম	M150		1
P	09 T007	7/Gang	Drill_0.4376	Drill_0.4376	side_hole_90	1.8upm	393rpm	0.76	ম	M1S0	-	1
	10 T008	8/Gang	End Mill_0.4376	Pattern Finishing	side_hole_90	0.5upm	430rpm	3.38	ম	M150	<b></b>	1
P	11 T008	8/Gang	End Mill_0.4376	Pattern 2	chamfer_270	0.5upm	430rpm	3.27	5	M1S0	<b></b> •	1
2 P	04 T002	1/Gang	Cut-off	Cutoff	Front	0.0016upr	192fpm	0.28	<b>v</b>	M1S0/E	<b></b>	ī.

# **Simulate the Cutting Process**

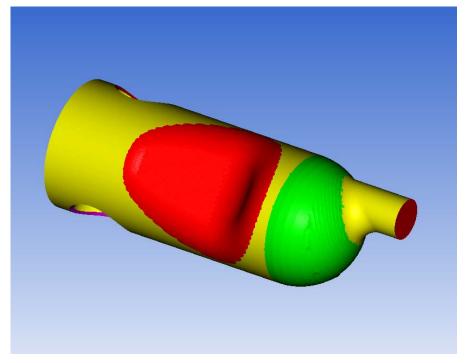
Once you are satisfied with the appearance of your Process Table you can simulate cutting. To do so:



- 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:

Warning	
<u> </u>	No eject operation programmed. Continue?
	Yes <u>N</u> o

- 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.

👺 Open Post Configuration File									x
🕞 🕞 🗸 🕌 « PartMaker 🕨 pm-swi	ss 🕨	surface_machining	swiss_tutorial_asm	<ul> <li>Inch-SMW_tutorial_swiss</li> </ul>	; • • <sub>?</sub>	Search Inch-SN	1W_tut	orial_sw	iss 🔎
Organize 🔻 New folder						8	≣ •		0
☆ Favorites	<b>^</b>	Name	A	Date modified	Туре	Size			
🧮 Desktop		😟 Swiss-demo.pst		10/14/2008 5:01 PM	Microsoft Office	367 KB			
Downloads									
💹 Recent Places									
🔚 Libraries									
Documents	=								
👌 Music									
Pictures									
Videos									
🖳 Computer									
🌉 OS (C:)									
🚽 Server_F (\\PMDATA\Root) (F:)									
🖵 c (\\Development-pc) (G:)									
🚽 Server_D (\\PMDATA\Root) (H:)									
🖙 Server Sales (\\PMDATA\ROOT)	*								
File name: Swiss-o	demo	o.pst			•	All Files (*.pst)			-
						Open 🗸		Cancel	5



4

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

Warning	x
<u> </u>	No eject operation programmed. Continue?
	Yes <u>N</u> o

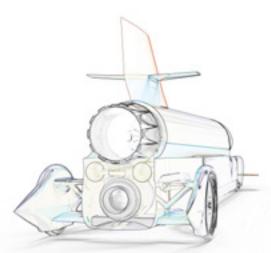
5 Click the **<Yes>** button to continue.

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

Post Options	×				
Program No. Program #1: 12 Bar Load: * Program #2:	Axis Support     B-Axis Output       Image: White Spindle C-axis     Local Coordinates       Image: Sub Spindle C-axis     Image: Global Coordinates				
Wait/Queue Commands Start: 2 Increment: 2	Machine Options           Air Blast           Cut-off Detection				
Job Settings Phase Angle: ** 0 Sub Spindle Collet Nose Extension: *** 0 Sub Spindle Feed onto Part (UPM): 50 Part Release Data Station No.: 20 Release-X: 5	Spindle Speed Limits Min RPM Main Spindle: 2000 Max RPM Main Spindle: 8000 Min RPM Sub Spindle: 2000 Max RPM Sub Spindle: 6000				
Parts Catcher Basket Release-Z: 0	Customer PIN:				
* Leave Blank if not using Bar Loader ** Leave Blank if not Phasing	Stop To Include User Input Auto re-load Post Config File Output Control				
Cancel	ОК				

- 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:

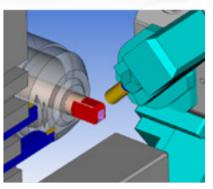
😂 PartMaker - N	Program
🔀 Post Options 📸 Generate NC Program 🖓 Find 👌 Print 🕀 Zoom In 😋 Zoor	Out 🔛 Align 💭 Launch NC Program Editor
ASM_TUTORIAL.H1	ASM_TUTORIAL.H2
%           07028 (FASM_TUTORIAL 1/6/2014) (CCULET RECHUCK)           M200           M94           G97 5705 T0101 M13           G1 699 X-0235 F.002           G0 M21           G4 U0.5           G302 023.659           G50 Z0.01           M22           G4 U0.5           G0 Z.716           T0           G0 X.74           M205           M1           (PROCESS 1 FACE OD TURN 80-RIGHT)           M205           M1           (PROCESS 1 FACE OD TURN 80-RIGHT)           M205           M1           (PROCESS 1 FACE OD TURN 80-RIGHT)           M210           T0           G28 U0           G95 G18.00 X1.036 Z0.           G95 G18.00 X1.036 Z0.           G95 G18.00 X1.036 Z0.           G95 G18.00 Q0           G65 S350           G1 X.0396 FL0.01           X-0.044           (PROCESS 2 1ST TURN)           M215           G3 X0.822 Z0.544 R0.148           G3 X0.822 Z0.544 R0.148           G3 X0.822 Z0.544 R0.148           G2 X0.4945 Z0.524 R0.767           G1 Z0.8022	ASM_TUTORIAL.H2           75%         70728 (FASM_TUTORIAL 1/6/2014) (COLLET RECHUCK)           70         70
(PROCESS 5 3D OFFSET FINISHING END MILL_0.15)	



# 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





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



# 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.

© Copyright Delcam Ltd 2014. All trademarks are the property of their respective owners.