

Advanced
Manufacturing
Solutions

PartMaker 2015

PartMaker Turn
Version 2015 or Higher



User Guide

Sequence of Programming Steps for PartMaker Turn

1. Specify part boundaries and the type of machining to be performed in the **Setup** dialog. It appears on the screen right after **PartMaker** logo when launching **PartMaker**. Determine whether you will be working on the Main Spindle or Sub Spindle in this **Face Window**.
2. Choose the **Tools** command from the **ToolMinder** menu to add new tools.
3. Choose the **Cycles** command from the **ToolMinder** menu to add new cycles.
4. Choose the **Open Material File** command from the **File** menu to open a material file from the Material directory.
5. Create geometry in CAD mode or choose the **Import** command from the File menu if you want to import geometry from a CAD system.
6. Switch back to CAM Mode by clicking the CAD/CAM switch.

The CAD/CAM switch appears as

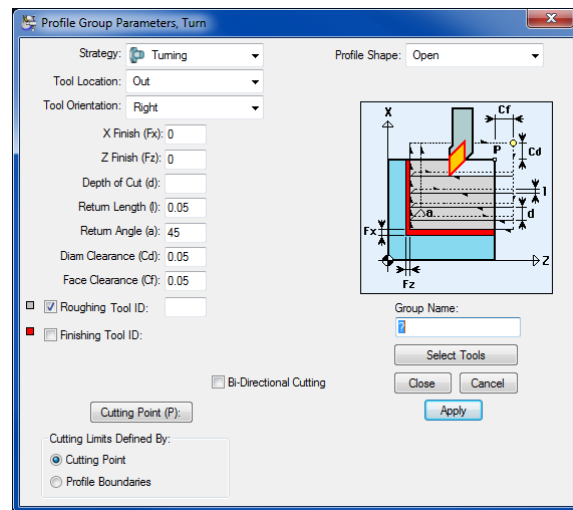
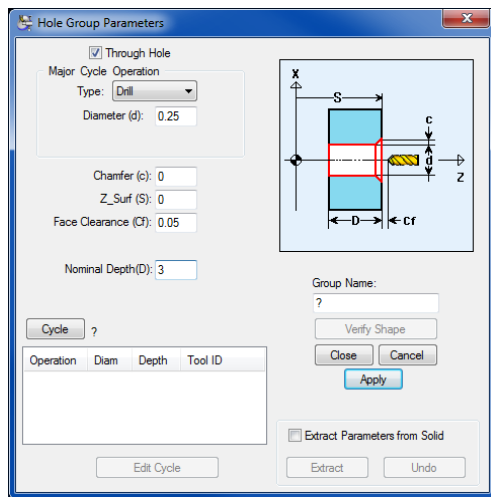


in CAM Mode and as



in CAD Mode

7. Choose the **New Hole Group** command from the **Part Features** menu to create an axial hole. You should see a dialog box as below:
8. Choose the **New Profile Group** command from the **Part Features** menu to define a new profile group. You should see a dialog box as below:



- 7.1. Enter your desired settings into the dialog.
- 7.2. Click the **Cycle** button.
- 8.1. Enter your desired settings into the dialog.
- 8.2. Click the **Select Tools** button.
9. Create tool paths using the Graphics Icons in the CAM mode
10. Repeat steps 7 through 9 to create groups of all part features and machining function face windows required for the job.
11. Choose the **Open Post Configuration File** command from the **Job Optimizer** menu and select the desired post configuration file from the PostLib Directory.
12. Choose the **Generate Process Table** command from the **Job Optimizer** menu to generate a **Process Table**.
13. Optimize and Synchronize your operations by using the **Set Modes** dialog
14. Simulate the generated program using **PartMaker's 3D Simulation** software
15. Choose the **Generate NC Program** command from the **Job Optimizer** menu to generate an NC Program

PartMaker 2015

User Manual

User Guide/PartMaker Turn



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	1
Using CAM Software to Program NC Machines	1
How PartMaker Improves Productivity	1
Intuitive Windows GUI Assures Quick Learning.....	1
Tools Data Base Keeps Track of Tool Inventory.....	2
Cycles Data Base Stores Sequences of Repetitive Operations	2
Materials Data Base Facilitates Automatic Feeds and Speeds Calculations.....	2
Patented Visual Programming Approach Simplifies Machining in Different Planes..	2
Optimized Process Plan Generation Saves Programming Time	3
Graphic Process Synchronization Simplifies Cycle Time Optimization	3
User Customizable Output Format Eliminates Manual Editing.....	3
Verification by Cutting Simulation Catches Programming Errors	3
About This Guide	4
Chapter 1: Getting Started With PartMaker	1
In Your Package	1
Technical Support.....	1
Before Using PartMaker... ..	2
System Requirements	2
Starting PartMaker	3
Exiting PartMaker	3
Using On-Line Help	4
Working With Databases	6
Working with Files.....	6
Chapter 2: PartMaker® Fundamentals	1
Introduction	1
Machining Function Selection.....	1
Getting Started: The Setup Dialog	2
Machining Function: Turning	4
PartMaker Programming Convention	5
Working with Files.....	5

Working With Databases	6
Material Database	6
Cycles Database	9
Using the CAD/CAM Switch	12
Using the Color Bar	12
Using the Icon Toolbars.....	13
Main Toolbar.....	13
Face Window Toolbar – CAM Mode	13
Face Window Toolbar – CAD Mode	13
Creating Groups of Part Features	14
Using the Graphics Icons in PartMaker CAD mode	14
Geometry Drawing Icons	15
Dimensioning Icons	16
Snap Modes	17
Using the Graphics Icons in PartMaker CAM mode.....	18
Graphics Icons – Machining Function Turn.....	18
Material Control Processes	19
Bar Feed.....	19
Pull with Side Gripper	20
Pull with Front Gripper	21
Tailstock Forward	22
Tailstock Retract.....	23
Remove from Main Spindle	24

Chapter 3: Turn Tutorial Part 1

Introduction	1
How You'll Create Turndemo	1
Starting PartMaker Turn	3
Setup Part Boundaries	3
View the Tool Information.....	5
View the Cycle Information.....	7
Select a Material for the Part	8
Setting Defaults	10
Turning.....	11
Setting Up the Face Window	11
Creating Geometry	12
Method 1: Import Geometry from a CAD system via DXF file	12
Method 2: Create Geometry in PartMaker CAD.....	14
Alternate Method 2: Create Geometry in PartMaker CAD.....	20

Creating TURNDEMO in PartMaker CAM Mode	27
Define Group 1: Profile for Face Turning	28
Define Group 2: Hole for Drilling Cycle	31
Define Group 3: OD Turning	32
Define Group 4: ID Turning	34
Define Group 5: ID Grooving	35
Define Group 6: ID Threading	37
Define Group 7: Cutoff	40
Choose a Post Processor	42
Generate a Process Table	42
Simulating the Part	43
Generate NC Code	44
Chapter 4: PartMaker® Simulation	1
Introduction	1
The Simulation Toolbar	2
Simulation Features	3
Reset	3
Show Travel Limits	3
Show Finished Part	3
Save Finished Part As STL file	4
Compare Finished Part to Solid Model	4
Comparison Options	4
View Setting Features	5
View Manipulation Features: Rotate, Move and Zoom	6
Sectioning Features	7
Simulation Options	8
Pause On	8
Settings	8
Enable Collision Checking/Report Collision Types	9
Display Control	9
Display Quality	10
Colors and Textures	10
Using PartMaker Basic Simulation	11
Using PartMaker Advanced Simulation	13
Machine File for Advanced Simulation	15
When should you use Basic vs. Advanced Simulation?	16
Appendix A: Materials	1
PartMaker Material Library	1

Appendix B: Family of Parts Programming Tutorial..... 1

Introduction	1
Sequence of Steps for Family of Parts Programming	1
Getting Started.....	2
Create Part Geometry	3
Setup PartMaker Databases	5
Assign Part Features	6
Creating Parameter (*.prm) files.....	12
Setup PartMaker for Parametric Input.....	14
Parameterize Your Part	15
Set Part Boundaries Parametrically	19
Update Each Part in the Family.....	20
Regenerate NC Code	20

Appendix C: Programming Directly on a Solid Model..... 1

Introduction	1
Key Definitions.....	1
Solids Programming Tutorial	3
How you will create the Sample part.....	3
Getting Started.....	4
Starting PartMaker.....	5
View the Finished Sample	5
Select a Material for the Part.....	6
Importing a Solid Model.....	7
Setting up the First Face Window	8
Programming a Face Operation.....	10
Programming a Tapping Cycle Operation.....	12
Programming a Turning Operation	14
Programming a Grooving Operation	17
Programming a Threading Operation	20
Programming a Cut Off Operation	23
Programming a Milling Operation	25
Duplicating the Face Window	29
Generating the Process Table.....	29
Simulating Machining in 3D	30
Other Samples.....	31
Turn Function Face Window Programming.....	31
Creating a Hole Group.....	36
Milling Functions Face Window Programming	38
Creating a New Profile Group (Mill ZY window shown).....	38
Creating a Hole Group (Mill Diameter Index Face window shown)	44

Introduction

Using CAM Software to Program NC Machines

The invention of NC (Numerically Controlled) machines has revolutionized the metal cutting industry. Computer controlled machine tools are faster, have a higher degree of accuracy and repeatability.

NC machines can be programmed manually, which means typing machine "language" motion and other instructions into a computer text file. Such a file, called a *part program*, is then loaded into the NC machine memory for execution.

The following are the major deficiencies of manual programming:

- The languages of NC machines are cumbersome for humans and require a long learning curve.
- Geometric calculations must be performed to determine coordinates of points comprising the tool path.
- Typing is a time consuming process.
- The absence of adequate methods to prove out the program results in tools breakage and material waste.

Since the earliest days of computer technology a significant effort has gone into automation of the programming process by utilizing CAM (Computer Aided Manufacturing) Software. CAM systems accept user input in an interactive manner and generate a part program file automatically. CAM Systems provide assistance in tool path calculations and verification. The overall productivity improvement resulting from a use of a particular CAM System depends on how long it takes to learn the software, how easy it is to use it and how much information a user has to enter during each programming session.

How PartMaker Improves Productivity

PartMaker relieves you from the tedious process of reentering the same tooling and process information over and over again in every program. It captures information about how to machine individual part features, namely, holes and profiles, and makes it available for future use. It allows shop personnel to benefit from the expertise of the most skilled programmer, who enters his or her knowledge into the software. Using **PartMaker** your parts can now be programmed faster, with higher precision and less cost.

Intuitive Windows GUI Assures Quick Learning

PartMaker features an intuitive user interface that complies fully with Windows GUI (Graphics User Interface) guidelines. The benefits of the Windows environment for a **PartMaker** user include Context Sensitive Help, pictorial explanation of parameter entry, the ability to Cut, Copy and Paste part features and/or geometry, and, most importantly, shorter learning time.

Tools Data Base Keeps Track of Tool Inventory

A tools database allows the user to keep track of his current tool inventory. Geometric characteristics saved with each tool are used for the automatic determination of cutting conditions. The software performs automatic hole depth calculations by maintaining through hole clearance and blind hole relief distances for each tool. A pictorial representation of each tool in a separate window simplifies the entry of tool parameters. Corrective coefficients compensate for tool wear. Special fields are reserved for the customer's data that includes part numbers and reordering information among others. The tools descriptions are kept in a tool file. The tools used on different machines may be kept in multiple tool files.

Cycles Data Base Stores Sequences of Repetitive Operations

A cycles database allows the user to create and store cycles, sequences of repetitive operations such as center drilling, drilling, tapping, boring, reaming, chamfering and circular hole milling. The cycles database links directly into the tools database. A tool for each operation is selected automatically based on the hole diameter. Windows GUI makes creation and modification of cycles a simple task. A full description of the tools and operations included in a cycle is shown in one window, while the corresponding hole cross-section is shown in another window. Studies have indicated that over 70 percent of machine cycle time for an average CNC machining center is spent making holes. Therefore, **PartMaker** pays significant attention to holmaking in order to achieve the greatest gain in productivity. Often, the same or similar features belong to a group of products that are ordered by the same shop's customer. Since cycles may be customer specific the user has a choice of whether or not to store them in multiple cycle files.

Materials Data Base Facilitates Automatic Feeds and Speeds Calculations

A materials database allows the user to store materials data that is used in the automatic calculation of feed rates and spindle speeds. The software comes with an extensive materials database, recommending average cutting parameters. Feed rate and spindle speed are computed based on tool geometry (size, number of flutes, etc.) and machinability data. This data is presented in the form of flexible computer charts giving the user full control over the performance of his machine. The charts can be quickly modified via a drag and drop screen interaction. Machinability charts can be further optimized by automatic curve fitting through points of best performance.

Patented Visual Programming Approach Simplifies Machining in Different Planes

PartMaker's Patented Visual Programming approach greatly simplifies programming parts involving both turning and milling operations in a single set-up, where such operations are being performed on multiple spindles. With this approach machining functions such as turning, plane milling and cylinder milling are carried out in separate 2D planes, allowing you to break down a part into its most basic elements when developing a part program. You can program machining operations in different faces (planes) and specify up to twenty-four different faces per part. A separate face window is associated with each face of a part.

Optimized Process Plan Generation Saves Programming Time

Rather than creating a job plan in a conventional sequential manner, the programmer describes the placement of groups of part features on the surfaces of a machined part. Once the part features description is entered into the computer, PartMaker automatically generates an optimized job plan. The results are summarized in a Process Table Window that can be reviewed on the screen or printed out. The Process Table lists tools and cutting conditions for each process. Feed rates, spindle speeds and cycle time are automatically calculated and displayed. The cycle time allows the programmer to quickly estimate the cost of machining a part. While reviewing the Process Table, the user can manually modify the sequence of processes as well as cutting conditions.

Graphic Process Synchronization Simplifies Cycle Time Optimization

Once you have laid out your various tool paths, PartMaker allows you to visually synchronize processes being performed on separate spindles or by different tool posts. This eases the process of optimizing cycle time to assure your Turn-Mill Center is being used most productively. This visual synchronization is accomplished via the Set Modes dialog on the Process Table. A unique mode has been specified for each of the various types of synchronization today's most advanced Turn-Mill Centers are capable of performing. A picture accompanies each type of synchronization allowing the user to better understand the type of synchronization being performed.

User Customizable Output Format Eliminates Manual Editing

When the user is satisfied with the views of the part and its job plan, he proceeds with the next step (known as postprocessing) to automatically generate an NC program for a designated machine. The part program is optimized, using subroutines, to occupy minimum space in the machine tool memory and shorten transmission time. **PartMaker** eliminates the need for manual editing of the generated NC program - a well-known stumbling block of most CAM systems.

The ability to generate editing-free programs results in an additional productivity gain. It is achieved by giving the user full control over the generated NC program format by means of a flexible ConfigPost application that allows the user to create and save custom postprocessor configuration files for virtually any Turn-Mill control. ConfigPost uses the familiar Windows interactive environment that enables the user to do output customization quickly and easily.

Verification by Cutting Simulation Catches Programming Errors

PartMaker comes with an integrated simulation module that graphically simulates the entire cutting process-utilizing solid modeling techniques. This allows the programmer to catch errors before expending machining time.

About This Guide

Please read this section to determine which chapters in the **PartMaker User Guide** address your needs.

- Chapter 1** **“Getting Started With PartMaker”** explains how to get started with **PartMaker**. This chapter includes a description of the hardware and software you need to run **PartMaker** and tells you how to install **PartMaker** and related files. It also shows you how to access **PartMaker** on-line context sensitive help.
- Chapter 2** **“PartMaker Fundamentals”** discusses the unique programming approach **PartMaker** employs for programming Turning Centers. Concepts covered in this chapter include selecting the appropriate Face Window, setting up and working with **PartMaker**’s databases, PartMaker Synchronization modes and material control processes, using various PartMaker icons and Snap Modes and synchronizing processes being executed simultaneously on the machine.
- Chapter 3** **“Programming the Turn Tutorial Part”** shows you how to create a sample part using some of the most popular features of **PartMaker**. We recommend you go through this chapter to familiarize yourself with the **PartMaker** application.
- Chapter 4** **“PartMaker Simulation”** discusses how to use and the functionality of PartMaker’s 3-D Simulation module.

What you will not find in this manual is an explanation of every **PartMaker** dialog and parameters. For information on specific **PartMaker** dialogs, use the context sensitive help that comes with **PartMaker**. This context sensitive help can be found by pressing the F1 key on your keyboard.

Chapter 1: Getting Started With PartMaker

In Your Package

- PartMaker CD-ROMs
- PartMaker *Manuals*
- Hardware lock

You will receive a license file that works with the hardware lock. For standalone licensing the file is called a Product Authorization File (PAF). For network licensing the file is an RMS license file. Obtaining and installing the PAF file will be discussed later in this chapter in the section titled PAF Installation Instructions. The PartMaker Installation guide explains the RMS license file in detail.

Technical Support

Please use the following information when you wish to get Technical Support or more information about PartMaker's products:

Phone: 215-643-5077

Fax: 215-653-0105

E-mail: support@partmaker.com

Web Site: <http://www.PartMaker.com>

Before Using PartMaker...

Before you begin working with **PartMaker**, you need a basic knowledge of Microsoft Windows operations. You should understand mouse techniques such as pointing, clicking, double clicking, dragging, choosing menu commands, and making dialog selections. If you are unfamiliar with basic Microsoft Windows terms or techniques, see your Microsoft Windows documentation for details.

System Requirements

Please read the following sections to determine what you'll need to get started with **PartMaker**. To use **PartMaker**, you need a Windows-based PC with the following specifications:

	PartMaker Standard	Parasolid Import Module (Solids) *	Full Machine Simulation (FMS)	Advanced Surface Machining (ASM)
Processor	Pentium 3 800 MHz or higher	Pentium 3 800 MHz or higher	Pentium 4 3 GHz or higher	Intel Core Duo processor or higher
Memory	512 MB or more	1 GB or more	2 GB or more	4 GB or more
Hard drive	40 GB	40 GB	40 GB	80 GB
Independent video card	128 MB NVIDIA	512 MB NVIDIA	512 MB NVIDIA	1 GB NVIDIA

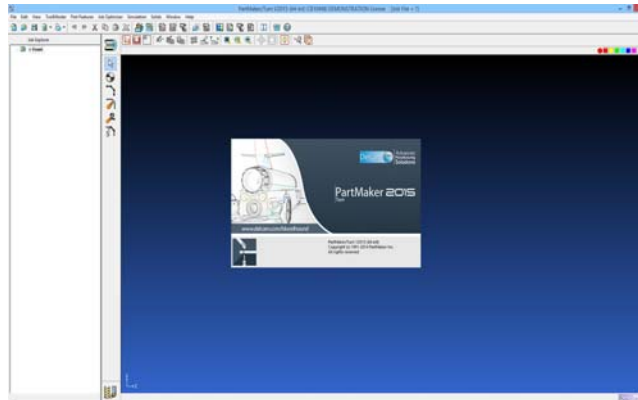
64-bit and 32-bit versions of PartMaker are available and are supported on Windows 7. Only the 64-bit version of PartMaker is compatible with Windows 8 PCs.

Please take note that Microsoft has discontinued support for the Windows XP and Vista operating systems.

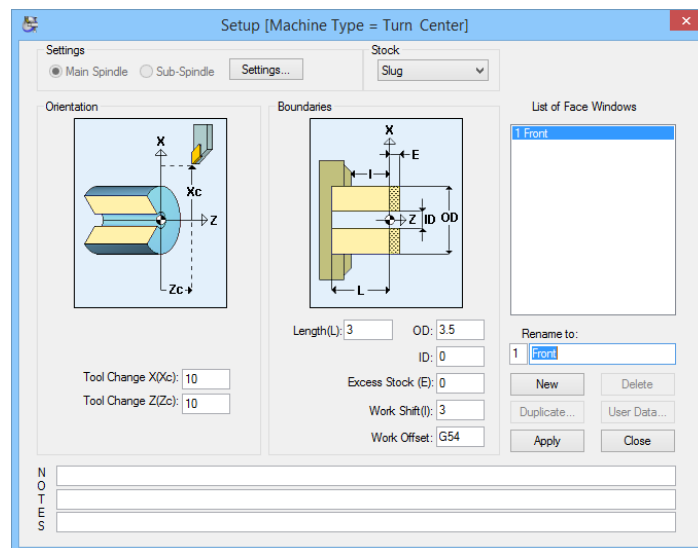
Starting PartMaker

From your desktop, double-click the PartMaker Mill icon.

A window containing information about **PartMaker** appears for a few moments.



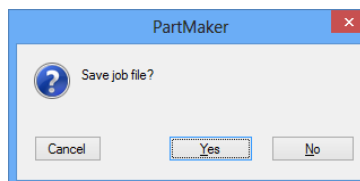
The **PartMaker** application window and the Setup dialog are displayed in the **Face Window**.



Exiting PartMaker

- 1 To exit **PartMaker**, choose **Exit** from the **File** menu.

If you changed an open job file, a dialog prompts you to save the current job file before exiting.



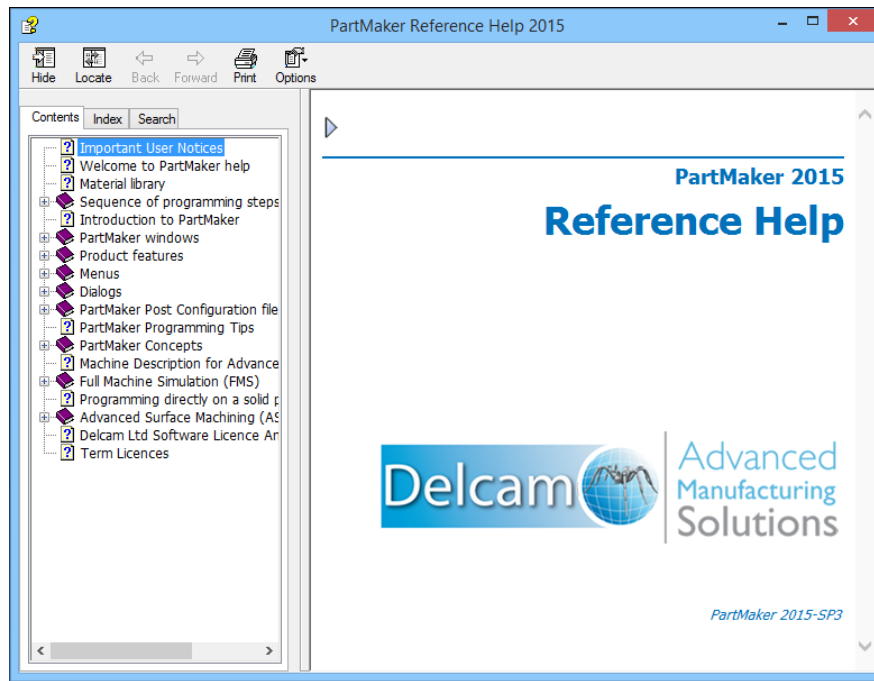
- 2 Do one of the following:
 - a Click **<Yes>** to save the changes.
 - b If the file is untitled, select a location and enter a file name in the dialog that appears, then click **<OK>**.
 - c Click **<No>** to discard the changes and exit.
 - d Click **<Cancel>** to return to the presentation window without exiting.

Using On-Line Help

On-line help provides fast access to information about the application's tools, commands, dialogs, and program features. Help commands are located in the Help menu in **PartMaker**:

- 1 Choose **Help Topics** from the **PartMaker Help** menu or press <F1>.

The Help window appears, displaying the **Main Index**.



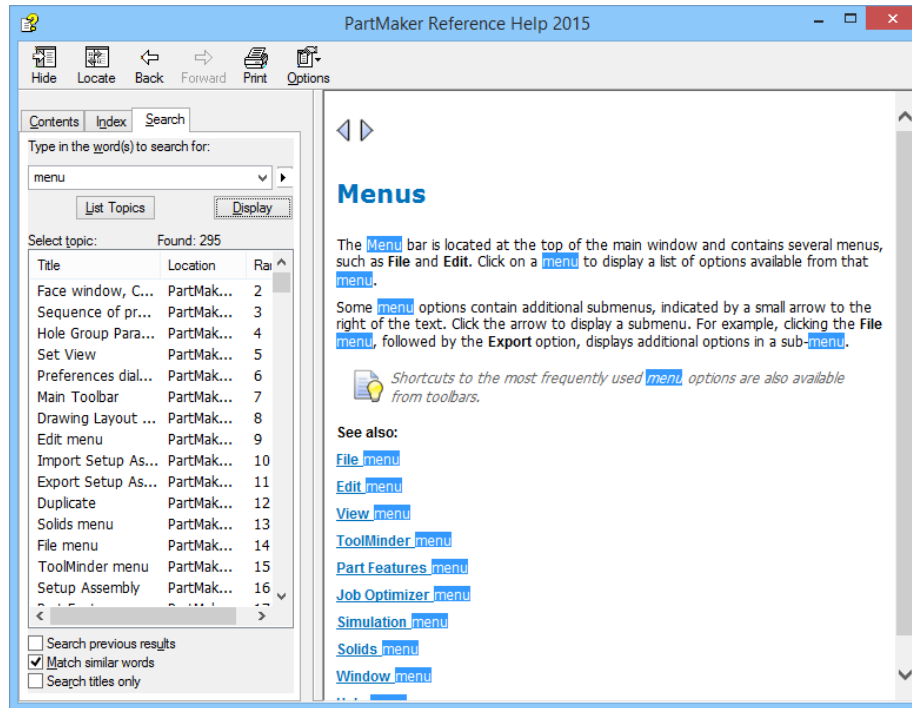
- 2 Click an index to view a list of help topics.
- 3 Click the topic you want to read about.

Information about the selected topic appears in the window.

Note: Some topic lists include sub-lists from which you can select a help topic.

- 4 To return to the last screen you viewed, click the <**Back**> button.
- 5 To search for information, click the <**Search**> tab.
- 6 Type in a key word to choose a topic and click the <**List Topics**> button.

- Click the **<Display>** button to view the selected topic. Your help screen will appear as below:



- When you are finished using help, choose **Close** from the help window's Control menu.

Using Context-Sensitive Help

To view context-sensitive help while viewing any **PartMaker** dialog:

- Display any **PartMaker** dialog.
- Press **<F1>** to display the help topic for the dialog.

Installing Updates

PartMaker Version 2010 and higher allows you to update to more recent versions and service packs of the software automatically from the PartMaker environment. You can check for and download the most recent version of PartMaker by selecting **Check for PartMaker Updates** from the **Help** menu.

Working With Databases

Databases are where you store tool, material, and cycle information for **PartMaker** jobs. **PartMaker** uses material, tools, and cycles databases to store the information needed for each job.

Material Database

A Materials Database allows you to store your shop's accumulated materials data. Materials data is used by PartMaker in the automatic calculation of feedrates and spindle speeds. In addition, you can utilize material data in the extensive material library provided with PartMaker.

Turn to Appendix A for a complete list of materials.

Tools Database

A Tools Database allows you to keep track of your current tool inventory. PartMaker uses geometric and cutting tool characteristics saved with each tool in the database for automatic cutting conditions and automatic depth calculation.

Cycles Database

PartMaker lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a Cycle. All cycles are saved in a Cycles Database. For each cycle operation, you can designate a tool from the Tools Database. Once a cycle is created, you can recall it at any time when you need to use it again.

Working with Files

PartMaker uses several kinds of files for storing tools, cycles, and material information as well as your jobs and the postprocessor files used to create NC programs for machining.

- Information about tools in the tool crib is saved in **Tools** files that have the file extension **.TDB**.
- Information about material characteristics needed to calculate feedrates and spindle speeds is saved in **Material** files that have the file extension **.MDB**.
- Information about user-created machining cycles is saved in **Cycles** files that have the file extension **.CDB**.
- Part geometry is saved in **Job** files that have the file extension **.JOB**.
- Postprocessor configuration information is saved in **Post Configuration** files that have the file extension **.PST**.

Chapter 2: PartMaker® Fundamentals

Introduction

This chapter discusses the various components of **PartMaker's** programming approach for Turn Centers. This chapter introduces the following topics:

- Choosing the proper Machining Function Face Windows
- Programming convention
- Setting up and working with **PartMaker's** Tools, Materials and Cycles databases
- Using Icons
- Process Synchronization Rules and Techniques
- Material Control Processes

Machining Function Selection

CNC laths are capable of performing a full set of turning operations: facing, turning, grooving and threading.

PartMaker CAM Software from PartMaker, Inc. applies a Patented 'divide and conquer' programming strategy to simplify programming of Turn-Mill Centers.

PartMaker helps you separate a complicated part surface into a manageable set of faces that may be either planar or rotational in nature. All features are represented by a surface relative to which a set of part features is referenced.

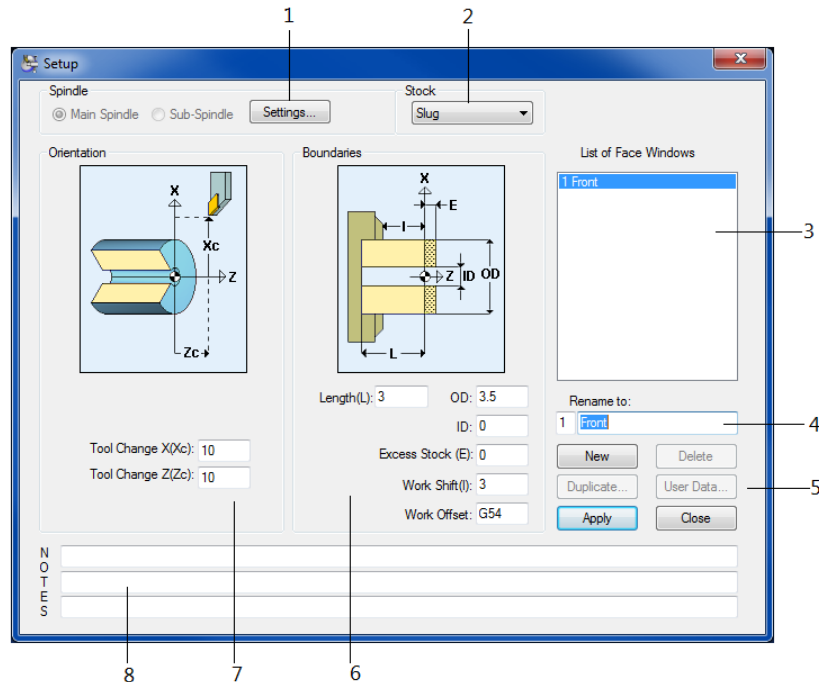
On each face, a set of features may be machined using a variety of cutting tools. The software takes advantage of the Microsoft Windows operating system, which allows multiple windows to be displayed and accessed simultaneously.

Getting Started: The Setup Dialog

PartMaker employs a programming methodology whereby each machine motion type can be separated into a different window dedicated to a specific machining function. Every one of these windows is known as a **Face Window**. Once created, each **Face Window** can be accessed from the Window menu.



Selection of the various machining functions is carried out in the **Setup** dialog. The **Setup** dialog can be accessed from the **View** menu or by clicking on the shortcut to the **Setup** dialog located in the upper left hand corner of your **Face Window**. On Startup, or when accessing the Setup dialog for the first time, you will see the following dialog. Some of the important components of this dialog are explained below:



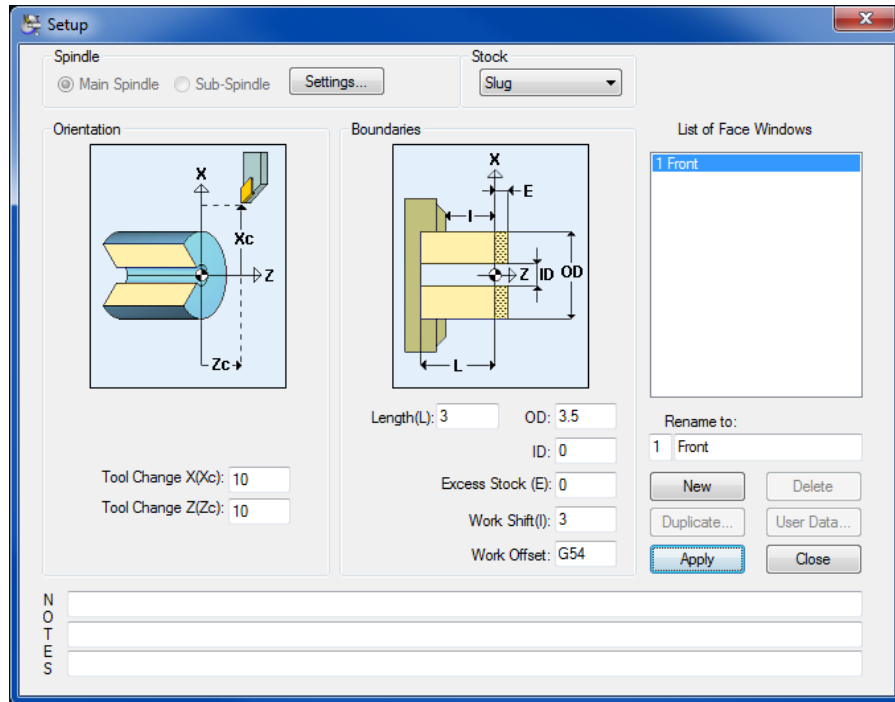
- Settings:** Allows you to access the **Spindle Settings** dialog. The Spindle Settings dialog allows you determine if you wish to use a chuck or a collet. The **Main Spindle** and **Sub Spindle** radio buttons are disabled in PartMaker Turn as this module assumes you are working on a single spindle lathe. If performing work on a twin spindle lathe, please use the PartMaker Turn-Mill module.
- Stock:** Determines the type of stock being machined, either a prismatic **Slug** or **Bar** stock. Once the setup dialog has been completed, you can use PartMaker's **Define Custom Stock Shape** command to create a custom, user defined stock.
- List of Face Windows:** Presents the list of Face Windows created for a particular job file.
- Rename To:** States the name of a given Face Window. You should give each Face Window you are working with an appropriate name based on the machining occurring within that Face Window. The number on the left of this field determines the order of the Face Window when you have a number of Face Windows in a particular job file. Generally speaking, you will only need one machining Face Window in PartMaker Turn.

- 5 Function Buttons:** These buttons allow you to perform the following operations in the **Setup** dialog:
- New:** Creates a new Face Window.
 - Delete:** Deletes an existing Face Window. This button will only be active when you have more than one Face Window and DO NOT have a Process Table generated.
 - Duplicate:** Not enabled in PartMaker Turn.
 - Apply:** Applies any changes made within the **Setup** dialog and leaves the dialog open.
 - Close:** Applies any changes made within the **Setup** dialog and closes the dialog and places you into a Face Window.
- 6 Boundaries:** Used to set the stock boundaries of your part. The parameters used to do so are explained here:
- Length (L):** Length of the finished stock
 - OD:** Outside diameter of the stock
 - ID:** The starting inside diameter when using tube stock
 - Excess Stock (E):** The amount of material to be removed from the stock to create the desired finished length. PartMaker takes excess stock into account only for facing operations.
 - Work Shift (I):** Specifies distance of the stock face from the chuck or collet face.
 - Work Offset:** If used by the post processor, inserts a work offset code into the part program. This option may be used with Fanuc and similar controls that make use of work offset codes. Consult the documentation for your control for specific information about work offset codes. See the ConfigPost User Guide for more information on configuring your post processor output.
- 7 Orientation:** Used to set the tool change position in X and Z for the tools in the job. These fields may or may not be used by the post processor, depending on the tool change NC code requirements of your particular machine tool. See the ConfigPost User Guide for more information on configuring your post processor output.
- Tool Change X (Xc):** The tool change position along the X axis.
 - Tool Change Z (Zc):** The tool change position along the Z axis.
- 8 Notes:** Allows you to specify notes about a particular Job or Face Window. Depending on the configuration of the post processor you choose, the contents of these fields can be output into your NC program. See the ConfigPost User Guide that came with your PartMaker Software for more information on post processor configuration.

Machining Function: Turning

The Turn machining function uses conventional turning tools moving in the X and Z-axes. Motion is programmed using X and Z coordinates. This machining function includes on centerline drilling where a drill is held stationary while the spindle holding the part rotates. All drilled holes must therefore be on the rotational centerline of the part.

The picture below shows the appearance of the **Setup** dialog when performing Turning on the Main Spindle.



Length (L)

Length of the finished part

OD

Starting outside diameter of bar stock before machining

ID

Starting inside diameter of bar stock before machining

Excess Stock (E)

Rough stock left for facing on the front of the part

Work Shift(I)

Specifies amount of the part protruding from the chuck during machining

Work Offset

Specifies the work offset for the current face window

PartMaker Programming Convention

When using PartMaker Turn for creating part geometry and NC programs, it is important to understand that all work is done moving from right to left in the Z-axis, this means all drawing and tool path generation is created by moving to the left in the Z negative direction. To summarize, PartMaker always assumes the collet or chuck to be to the left.

It is important to keep the right to left programming convention in mind when drawing, creating tool paths, and when setting up tools. Tool orientation should always be assigned assuming programming in the same direction.

Working with Files

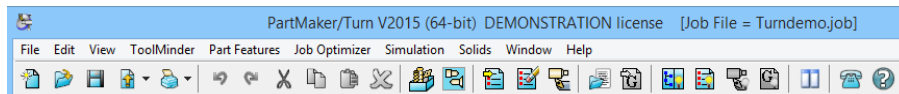
PartMaker uses several kinds of files for storing tools, cycles, and material information. Your jobs and post processor files used to create NC programs for machining can also be stored in their own files.

- Part geometry, tool path, and process information is saved in **Job** files that have the file extension **.JOB**.
- Information about tools in the tool crib is saved in **Tools** files that have the file extension **.TDB**.
- Information about material characteristics needed to calculate feedrates and spindle speeds is saved in **Material** files that have the file extension **.MDB**.
- Information about user-created machining cycles is saved in **Cycles** files that have the file extension **.CDB**.
- Postprocessor configuration information is saved in **Post Configuration** files that have the file extension **.PST**.

The Job File

The Job file is the main file type used in **PartMaker**. Every job file must have a tools and cycle file associated with it. However, particular tools and cycles files can be (and should be) associated with many if not all of your job files. Job files have the extension **.job**.

The name of the current job file you are working with is displayed at the top of the **PartMaker** screen as shown below:



Working With Databases

Databases are where you store tool, material, and cycle information for **PartMaker** jobs. **PartMaker** uses tools, materials, and cycles databases to store the information needed for each job.

Material Database

A Materials Database allows you to store your shop's accumulated materials data. Materials data is used by **PartMaker** in the automatic calculation of feed rates and spindle speeds. In addition, you can utilize material data in the extensive material library provided with **PartMaker**.

Turn to Appendix A for a complete list of materials.

Tools Database

A Tools Database allows you to keep track of your current tool inventory. **PartMaker** uses geometric and cutting tool characteristics saved with each tool in the database for automatic cutting conditions and automatic depth calculation.

Cycles Database

PartMaker lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a Cycle. All cycles are saved in a Cycles Database. For each cycle operation, you can designate a tool from the Tools Database. Once a cycle is created, you can recall it at any time when you need to use it again.

Material Database

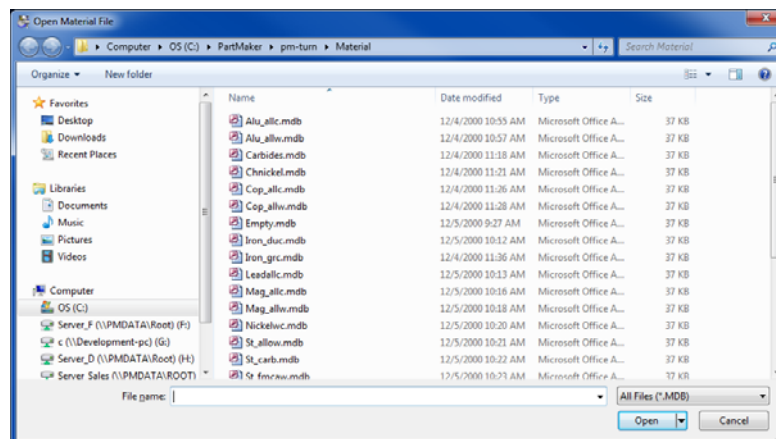
The **Material Database** allows you to store your shop's accumulated materials data. **PartMaker** uses materials data in the automatic calculation of feedrates and spindle speeds. The Materials Database allows you to choose from a variety of different materials and even adjust them and store your changes in order to customize feeds and speeds to your shops requirements.

Turn to Appendix A for a complete list of materials. You can also use the On-line Context Sensitive Help that comes with your **PartMaker** system to view a complete list of included material files, inclusive with various material grades for each.

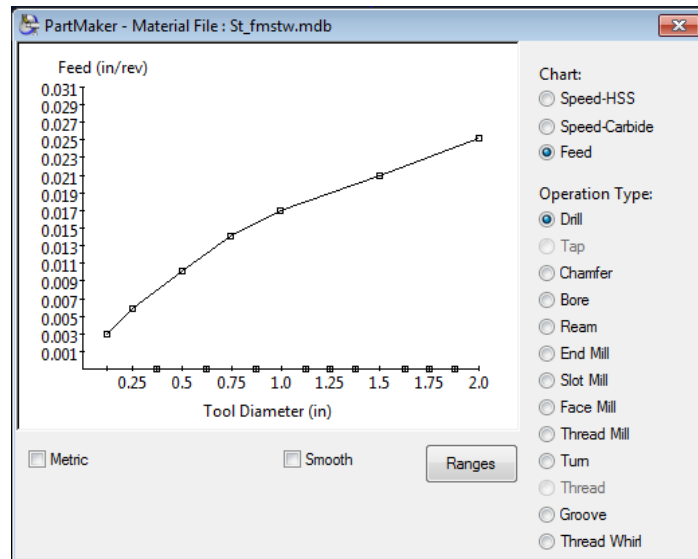
Selecting a Material File

When creating a part, it is important to select a material. To do so:

- 1 Choose Open Material File from the **File** menu to display the **Open Material File** dialog shown below:



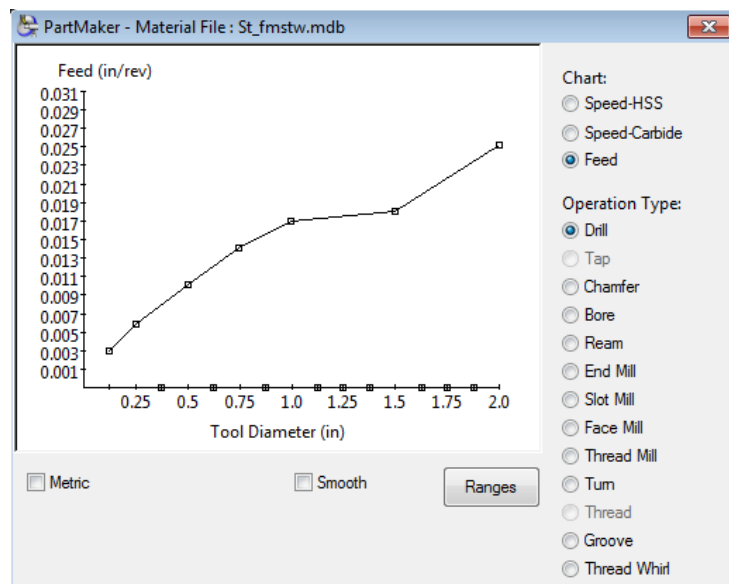
- 2 Locate and select ST_FMSTW (Free Machining Stainless Steel Wrought) from the list and click <Open>. You are returned to the **Face Window**.
- 3 To view the cutting characteristics of the selected material, choose **Material** from the **ToolMinder** menu and you will see the dialog shown below:



Customizing a Material File

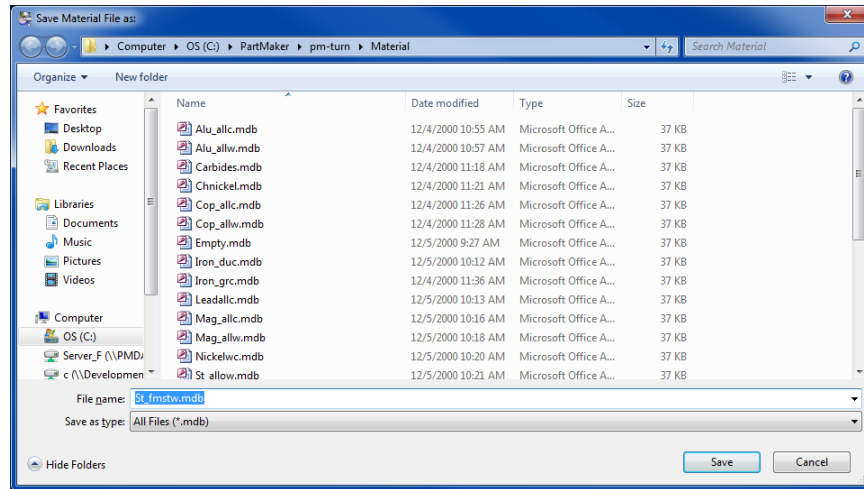
You can customize materials files in **PartMaker** by adjusting the feed and speed charts displayed in the **Material** dialog. To do so:

- 1 Choose **Material** from the **ToolMinder** menu to display the **Material File** dialog shown below:

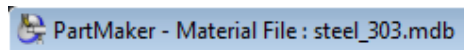


- 2 Click the third radio button on the right hand side to display the feed chart. By clicking on and dragging the node points lying on the chart or the axes, you can customize feed rates for different sized drill. The same technique can be employed for spindle speeds as well for all the tools listed.

- 3 Once you are satisfied with the changes you have made, you can save your changes by choosing **Save Material File As...** from the **File** menu. In the file name section, enter “steel_303” and click **<Save>** to save the material as shown below:

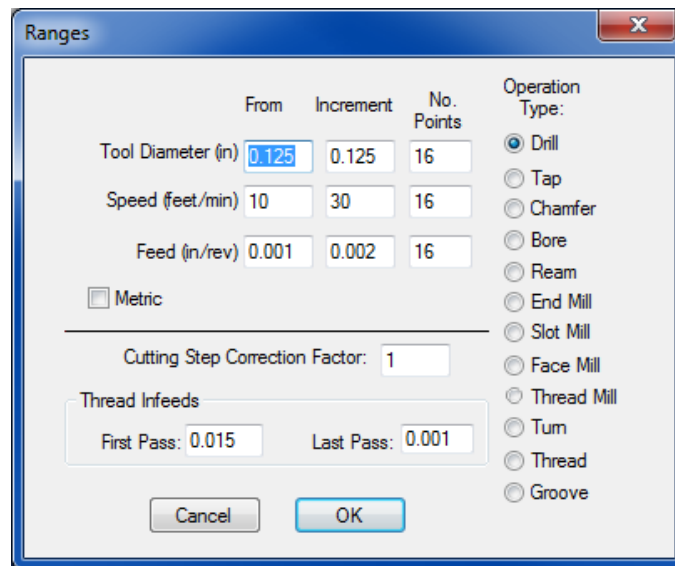


You will now notice the new name of the material file at the top of the **Material File** dialog as shown below:



Note: When saving changes to material files, you must give them a different name than the initial **PartMaker** material used. The material files that come with **PartMaker** are read-only files and cannot be overwritten.

- 4 Click on the **Ranges** button to see how the chart is calculated. Notice how with each tool you can change parameters on which speeds and feeds are based as well as the speeds and feeds themselves. Click **<OK>** to close this dialog.



- 5 Click the close box in the upper, right-hand corner of the **Save Material File as:** dialog box to return to the **Face Window**.

Cycles Database

PartMaker lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a Cycle. All cycles are saved in a **Cycles Database**. The Cycles Database can store up to 1,800 cycles. For each cycle operation, you can designate a tool from the Tools Database. Once a cycle is created, you can recall it at any time when you need to use it again. Cycles can be setup for both cross holes and holes on the face of the part, as well as for holes involving both live and dead tooling.

Creating and Storing Cycles

In this exercise, you will create a cycle for on-center hole on the face of a part using existing tools in the tools database. This cycle will contain a center drill and a drill.



You can access **PartMaker**'s Cycles Database by choosing **Cycles** from the **ToolMinder** menu. Once you are in the Cycle Data dialog, to create a new cycle:

- 1 Click the **<New>** button to create a new cycle. A blank cycle such as the one below will be displayed.

The screenshot shows the 'Cycles Data' dialog box with the file path 'C:\PartMaker\pm-turn\Cycles.cdb'. The 'List of Cycles' on the right contains 'Untitled2' and 'Drill_1'. The main table has columns: Major Operation, Operation Type, Canned Cycle, Hole Diam, Depth, and Tool ID. The first row is selected with 'Drill' in Major Operation, 'Drill' in Operation Type, 'DRILL' in Canned Cycle, and '0' in Hole Diam. The 'Tool ID' column has a checkmark. At the bottom right, there are buttons: 'New', 'Delete', 'Duplicate', 'Close', 'Select Tools', and 'Apply'. An 'Insert Operation' button is at the bottom center.

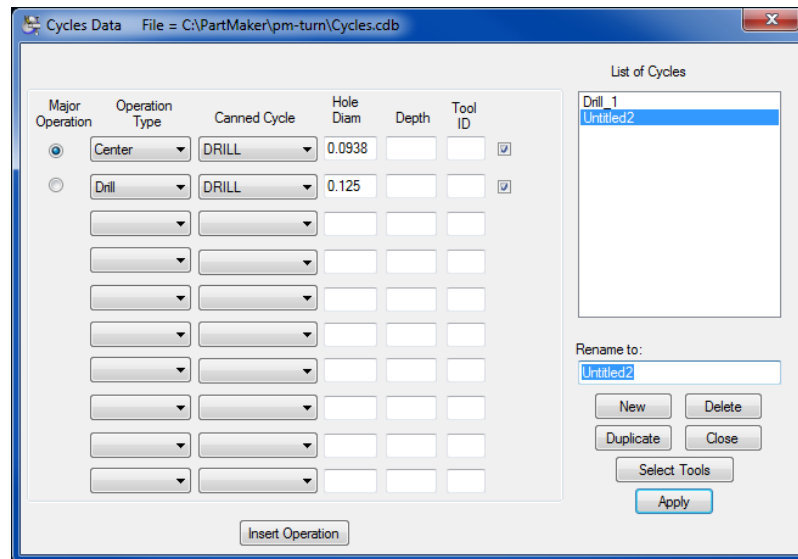
- 2 Next, choose the operations that will comprise the cycle. To do so, choose center drill and drill, respectively, from the Operation Type drop-down menu. After doing so, your dialog should appear as shown below:

The screenshot shows the 'Cycles Data' dialog box with the file path 'C:\PartMaker\pm-turn\Cycles.cdb'. The 'List of Cycles' on the right contains 'Drill_1' and 'Untitled2'. The main table has columns: Major Operation, Operation Type, Canned Cycle, Hole Diam, Depth, and Tool ID. The first row is selected with 'Center' in Major Operation, 'Center' in Operation Type, 'DRILL' in Canned Cycle, and '0' in Hole Diam. The 'Tool ID' column has a checkmark. The second row is 'Drill' in Major Operation, 'Drill' in Operation Type, 'DRILL' in Canned Cycle, and '0' in Hole Diam. The 'Tool ID' column has a checkmark. At the bottom right, there are buttons: 'New', 'Delete', 'Duplicate', 'Close', 'Select Tools', and 'Apply'. An 'Insert Operation' button is at the bottom center.

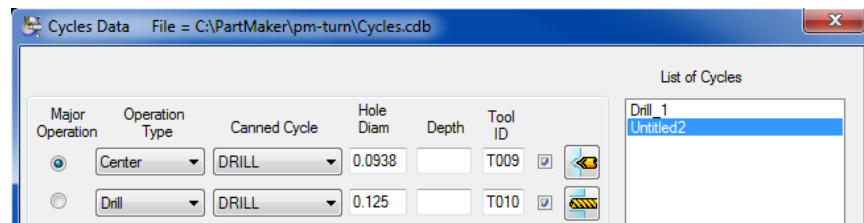
- Next, choose the corresponding Canned Cycles from the Canned Cycles drop menu for each of these operations, which will be DRILL and DRILL, respectively.

Finally, specify the hole diameter for each tool. In this case, the Center Drill should be 0.0938 and the Drill 0.125.

After entering the parameters as specified above, your dialog should appear as shown below:

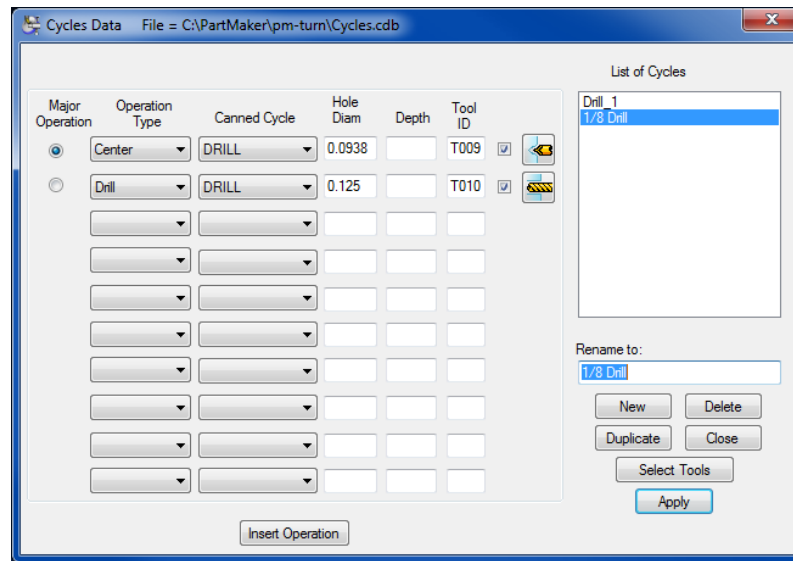


- Click the **<Select Tools>** button to select or add tools for the corresponding operations as shown below:



- In the Rename To: field, rename the cycle Drill 1/8 and click the **<Apply>** button.

- 6 The completed Cycle Data dialog should appear as shown below:



Notice that the column for depth has not been filled out. It has been intentionally left blank in order to allow **PartMaker** to calculate the depths of cut for each tool automatically when applied to the part. This automatic calculation is based on the nominal hole depth and specified geometric characteristics of the tools in the cycle.

One cycle may be applied to many different jobs, and allowing **PartMaker** to calculate the depth of cut makes each cycle more flexible. It may at times be desirable to enter a value into the depth field. When doing so, be advised that this value will be "hard coded" and that whenever using the cycle, this tool will always travel to the depth specified in the Cycles Data dialog.

- 7 Once you are satisfied that your dialog appears as the one above, click the **<Close>** button.

Using the CAD/CAM Switch



This icon represents CAM mode. Clicking it will switch the program to the CAD mode.



All tool path creation and process development is done in the CAM mode. All geometry creation and alteration is done in the CAD mode.

Using the Color Bar



You use the color bar in the upper-right corner of a **Face Window** to select a color for group symbols. You can also use it to change the color of geometry. The group symbol or geometric element will have the color of the currently chosen **Sample Color**.



Note: If you want to change the color of a Group Symbol, select the Group Symbol in the **Job Explorer**, click a color on the color bar, and then click the Sample Color on the left side of the color bar.

To change the color of a geometric element, select the geometry in the **Face Window**, click a color on the color bar, and then click the Sample Color on the left side of the color bar.

Using the Icon Toolbars

PartMaker Version 9 and higher provides Toolbars to speed up commonly used functions throughout the software. PartMaker supports four types of icon toolbars, a **Main Toolbar** which is available throughout a job file, a **Face Window Toolbar** which is associated with each **Face Window**, a **Solids Window Toolbar** which appears when working with an imported Solid Model and a **Simulation Toolbar** which appears when working in PartMaker's 3D simulation. Here, the **Main Toolbar** and **Face Window Toolbar** are explained. Chapter 4 of this guide will explain the use of the Simulation Toolbar while Appendix C to this manual will explain the use of the **Solids Window Toolbar**.

Main Toolbar

The **Main Toolbar** lets you perform many of the functions in PartMaker's menus with the click of a button. The **Main Toolbar** will look the same throughout your PartMaker session and appears as below:



Note: When holding your mouse over any of these icons, a "Tool Tip" will appear showing the function performed by that icon.

Face Window Toolbar – CAM Mode

The **Face Window Toolbar** lets you perform many of the functions in PartMaker's menu's with the click of a button. The **Face Window Toolbar** applies only to the active **Face Window** and may look differently depending on the **Machining Function** Face Window being used.



Face Window Toolbar – CAD Mode



Displaying Toolbar Icons

The display of the Toolbars can be manipulated by choosing **Toolbars** from the **View** menu. When a toolbar is checked, it will appear. When unchecked, it will not be visible.

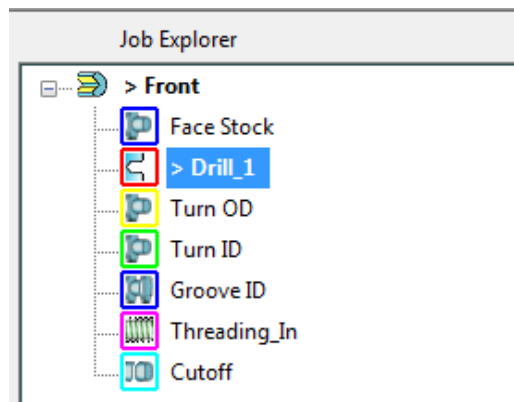
Creating Groups of Part Features

Groups of part features such as holes and profiles are created in **PartMaker**. The currently selected group icon or **Group Symbol** in a **Face Window** is the work group from which tool paths are being added.

To create a new Group Symbol:

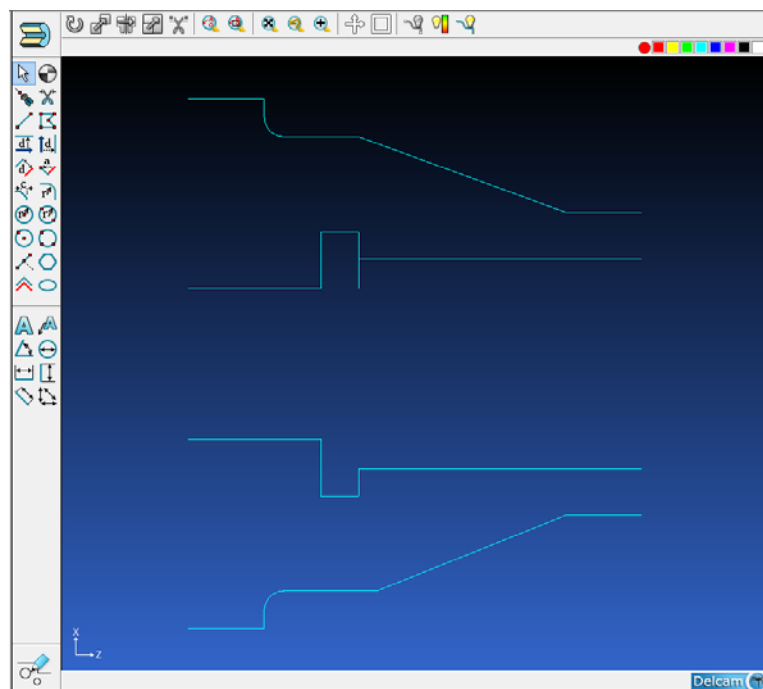
- 1 Click the desired color on the color bar.
- 2 Choose New Profile or New Hole Group (depending on the type of Part Feature being created) from the Part Features menu to display the Group Parameters dialog.

When you exit the dialog, the new group is created using the color you specified as shown in the image below.



Using the Graphics Icons in PartMaker CAD mode

You can create your part drawings in **PartMaker** in the CAD mode. Alternatively, you also import drawings from other CAD systems into **PartMaker** by using the **Import** command from the **File** menu. Imported geometry can be altered in **PartMaker**'s CAD mode.



Geometry is created in **PartMaker** CAD using the Graphics Icons that include Drawing Icons and Snap Modes.

Geometry Drawing Icons

Geometry Drawing icons allow you to create different types of geometric constructions such as lines and arcs. **PartMaker's** icons are defined below.



Selection Icon indicates that the system is in the selection mode. In this mode you can select lines, arcs and circles in the Face Window either by clicking on them or dragging a selection rectangle around them.



Zero Icon allows you to define a new location for the zero point using Snap Modes.



Trim/Extend Icon allows you to stretch or shorten an existing line or arc.



Remove Icon allows you to remove portions of lines or arcs between intersection points.



Line Thru Two Points Icon allows you to create a line between two points specified using Snap Modes.



Connected Lines Icon allows you to create connected lines with end points specified using Snap Modes.



Line Parallel Horizontal axis Icon allows you to create a line parallel to the Horizontal axis by specifying positive or negative offset.



Line Parallel Vertical axis Icon allows you to create a line parallel to the Vertical axis by specifying positive or negative offset.



Parallel Element Icon allows you to create a line, arc, or circle parallel to the cursor selected entity specifying an unsigned distance.



Line on an Angle Icon allows you to create a line on an angle to a cursor-selected line passing through a point specified using Snap Modes.



Fillet Icon allows you to insert a fillet between two cursor-selected lines or arcs.



Chamfer Icon allows you to insert a chamfer between two cursor-selected lines or arcs.



Circle with a Known Radius and Center Icon allows you to define a circle with a center specified using Snap Modes.



Circle with a Known Radius and Two Points Icon allows you to define a circle with two points on the circumference specified using Snap Modes.



Circle with a Known Center and a Point Icon allows you to define a circle with a center and a point on the circumference specified using Snap Modes.



Circle Through Three Points Icon allows you to define a circle with three points specified using Snap Modes.



Polygon Icon allows you to create a various rectangles by entering a flat-to-flat distance.



Divide/Append Icon aids you in the "breaking" of segments at specific point within the geometry.



Multiple Offset Icon allows you to offset the selected geometry chain through a user-defined distance.



Ellipse Icon allows you to create an approximate ellipse in the face window, using tangential circular arcs.

Dimensioning Icons

The Dimensioning Icons appear in the PartMaker CAD mode. These icons allow you to create dimension and annotations on your drawings in PartMaker.



Note Icon allows you to create a note on a dimensioned drawing.



Note with Leader Icon allows you create a note with an arrow leader to point to a specific item on your drawing you wish to annotate.



Angular Dimension allows you to insert an Angular Dimension between two lines..



Circular Dimension Icon allows you to insert a Circular Dimension on arc or circle.



Horizontal Dimension Icon allows you to insert a Horizontal Dimension between two points that are either end points of arcs and lines or centers of circles.



Vertical Dimension Icon allows you to insert a Vertical Dimension between two points that are either end points of arcs and lines or centers of circles.



Linear Dimension Icon allows you to insert a Linear Dimension between two points that are either end points of arcs and lines or centers of circles.



Flexible Dimension Icon allows you to insert a “Flexible Dimension” that is automatically set by the software to either Horizontal Dimension or Vertical Dimension or Linear Dimension depending on the cursor position when the text location is specified.

Snap Modes

Snap modes are used in conjunction with geometry icons to define point location. **PartMaker's** Snap Modes are defined below. Snap drawing modes are also used in the CAM mode to define tool path location.



ZX Coordinates: Used for entering Cartesian coordinates.

For **Face Windows** of various machining functions, this icon will change accordingly.

ZX for Mill ZX Plane and Turn

XY for Mill XY Plane, Mill Polygon, Mill End Index, Mill End Polar and Mill Diam Polar

ZC for Mill Diam Index and Mill Cylinder

ZY for Mill ZY Plane



Polar Coordinates: Used for entering Polar coordinates



Point on a Circle: Clicking a point on an existing arc or circle and entering an Angle



Screen: Clicking a cursor location anywhere within the face window



Grid: Clicking a grid location (grid can be turned on from the View Menu by choosing Show Grid. Grid size can be set in the Preferences dialog accessed from the View menu).



Circle Center: Clicking the center an existing circle



Closest Intersection: Clicking the closest intersection of geometric elements



End of an Element: Clicking the end of an existing line or arc



Middle of an Element: Clicking the middle of an existing line or arc



Tangency: Clicking near the tangency point that will be calculated by the system



Horizontal Constraint allows you to specify a point whose vertical coordinate remains constant, allowing changes in horizontal coordinate only.



Vertical Constraint allows you to specify a point whose horizontal coordinate remains constant, allowing changes in vertical coordinate only.

Using the Graphics Icons in PartMaker CAM mode

The Graphics Icons on the left side of a CAM **Face Window** allow you to define and manipulate tool paths. Certain Graphics Icons for use in assigning a particular part feature to your part (i.e. a Group Symbol) will have a different appearance and function depending on the Machining Function **Face Window** being used.

The two icons below will have the same appearance and usage regardless of which machining function **Face Window** you are using.



[Selection Icon](#) indicates that the system is in the selection mode. In this mode, you can select part features and tool paths in the Face Window either by clicking or dragging a selection rectangle around them.

Selected profiles in a **Face Window** are highlighted; unselected profiles are not highlighted. Selected holes in a **Face Window** are highlighted; unselected holes are hollow. You can delete selected elements by pressing the <Backspace> key on the keyboard or by choosing the Delete command from the **Edit** menu.



[Zero Icon](#) allows you to define a new location for the zero point or origin using Snap Modes.

Graphics Icons – Machining Function Turn

The Machining Function Turn **Face Window** uses two icons to allow you to assign part features location in order to create tool paths.



[Profile Icon](#) enables you to specify that the currently active group symbol is to be applied to the profile described by either coordinate entry or by clicking on geometric elements.



[Chain Icon](#) enables you to specify that the currently active group symbol is to be applied to a profile that is created when PartMaker automatically connects all the elements that form an unbroken chain when an initial line or arc is clicked.



[2-Point Chain Icon](#) The 2-Point Chain icon allows you to define a profile by the start point in the chain, and the end point of the chain.



[Profile Info](#) allows you to select an element (line or arc) on the profile curve and to display a [Profile Info Dialog](#). This icon is available only if a profile group already exists.

Material Control Processes

Material Control Processes (MCPs) are turning processes that do not involve material removal but do require NC programming. MCPs can be inserted into the Process Table via the **<Insert Material Control Process>** command under the **Job Optimizer** menu. Upon insertion into the Process Table, MCPs will be placed at the top of the Process Table by default. MCP actual place in the process will be determined by clicking on the arrow beneath the cutting after which the MCP should be activated.

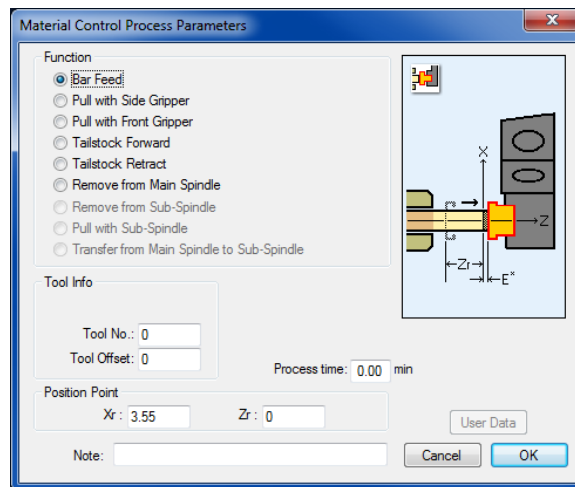
The applications of each MCP available in PartMaker Turn are explained below along with an explanation of the parameters needed to activate it.



Note: Individual post processors may support certain MCPs but not others. Please consult your machine-specific addendum or PartMaker Technical Support to determine which MCP is supported by your post processors.

Bar Feed

The Bar Feed MCP allows the user to insert code for feeding the bar stock out to a certain Z-coordinate and stopping with it a bar stop mounted in the turret.



Tool No.: Enter the tool station number at which the Bar Stop is mounted. The number must be a one or two digit integer.

Tool Offset: Enter the offset number for the Bar Stop. The number must be a one or two digit integer.

Process Time: Enter a time estimation from when the Bar Stop moves from its safe index position at the beginning of the process to when the Bar Stop moves back to its safe index position at the end of the process. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Xr: Enter an absolute x coordinate (measured from the center of the spindle) to position the Bar Stop before stock is pushed out. This number would typically be zero.

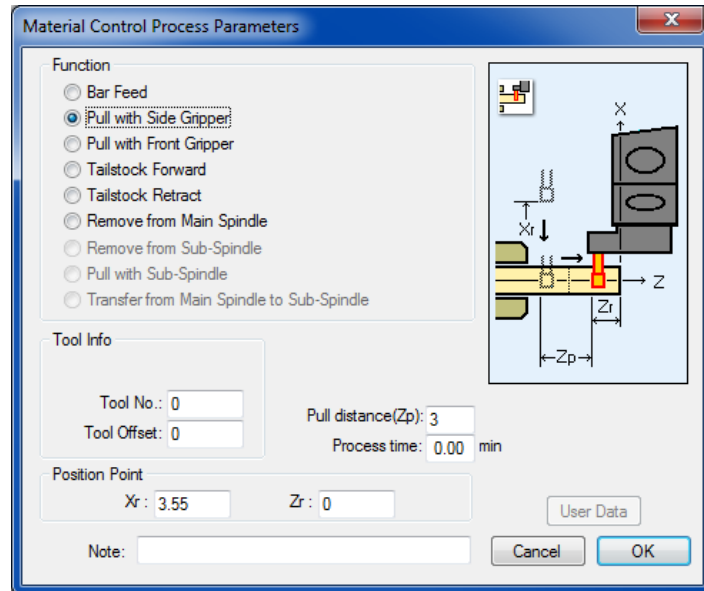


Note: If using the cutoff tool as the Bar Stop, enter a negative x coordinate in order to move the cutoff tool insert below the stock diameter.

Zr: Enter an absolute z coordinate (measured from the face of the part) to position the bar stop before the stock is pushed out. For short parts, this number can be set to zero. For longer parts, it is preferred to set this number equal to the length of the part so that the stock does not ram into the Bar Stop after the collet opens.

Pull with Side Gripper

The Pull with Side Gripper Material Control Process allows the user to insert code for pulling the bar stock out to a certain Z-coordinate with a part gripper mounted in the turret which supports the part from the side.



The dialog box titled "Material Control Process Parameters" contains the following sections:

- Function:** A list of radio buttons with "Pull with Side Gripper" selected. Other options include Bar Feed, Pull with Front Gripper, Tailstock Forward, Tailstock Retract, Remove from Main Spindle, Remove from Sub-Spindle, Pull with Sub-Spindle, and Transfer from Main Spindle to Sub-Spindle.
- Tool Info:** Fields for Tool No.: 0, Tool Offset: 0, Pull distance(Zp): 3, and Process time: 0.00 min.
- Position Point:** Fields for Xr: 3.55 and Zr: 0.
- Note:** A text area for additional notes.
- Buttons:** User Data, Cancel, and OK.

A schematic diagram on the right shows a side gripper mechanism clamping a bar stock. It labels the X-axis (horizontal), Z-axis (vertical), and the pull distance Zp.

Tool No.: Enter the tool station number at which the Side Gripper is mounted. The number must be a one or two digit integer.

Tool Offset: Enter the offset number for the Side Gripper. The number must be a one or two digit integer.

Pull Distance (Zp): Enter an incremental z distance from where the Side Gripper has clamped onto the bar to where the stock is to be pulled.

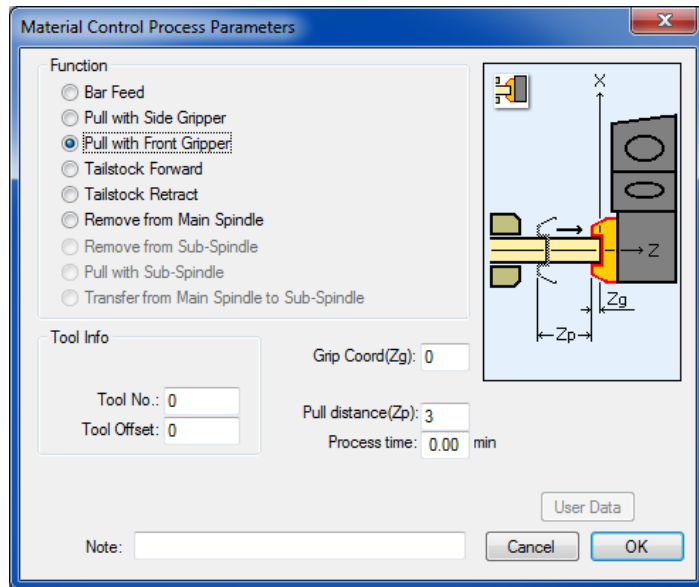
Process Time: Enter a time estimation from when the Side Gripper moves from its safe index position at the beginning of the process to when the Side Gripper moves back to its safe index position at the end of the process. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Xr: Enter an absolute x coordinate (measured from the center of the spindle) to position the Side Gripper above the bar stock before it moves to X0.0 to clamp the bar. This number is usually greater than the bar stock diameter.

Zr: Enter an incremental z distance (measured from the final cutoff z coordinate) to position the Side Gripper above the bar stock before it moves in to clamp the bar.

Pull with Front Gripper

The Pull with Front Gripper Material Control Process allows the user to insert code for pulling the bar stock out to a certain Z-coordinate with a part gripper mounted in the turret which supports the part from the front.



The dialog box titled "Material Control Process Parameters" contains the following elements:

- Function:** A list of radio buttons with "Pull with Front Gripper" selected.
 - Bar Feed
 - Pull with Side Gripper
 - Pull with Front Gripper**
 - Tailstock Forward
 - Tailstock Retract
 - Remove from Main Spindle
 - Remove from Sub-Spindle
 - Pull with Sub-Spindle
 - Transfer from Main Spindle to Sub-Spindle
- Tool Info:** Input fields for "Tool No.:" (0) and "Tool Offset:" (0).
- Grip Coord(Zg):** Input field with value 0.
- Pull distance(Zp):** Input field with value 3.
- Process time:** Input field with value 0.00 min.
- Note:** A text area for additional notes.
- Buttons:** "User Data", "Cancel", and "OK".
- Diagram:** A schematic on the right showing a bar stock being pulled by a gripper. It includes coordinate axes (X, Z) and labels for Zg (gripper coordinate) and Zp (pull distance).

Grip Coord (Zg): Enter an incremental z distance (measured from the face of the bar stock) to where the Front Gripper is to clamp on to the bar.

Tool No.: Enter the tool station number at which the Front Gripper is mounted. The number must be a one or two digit integer.

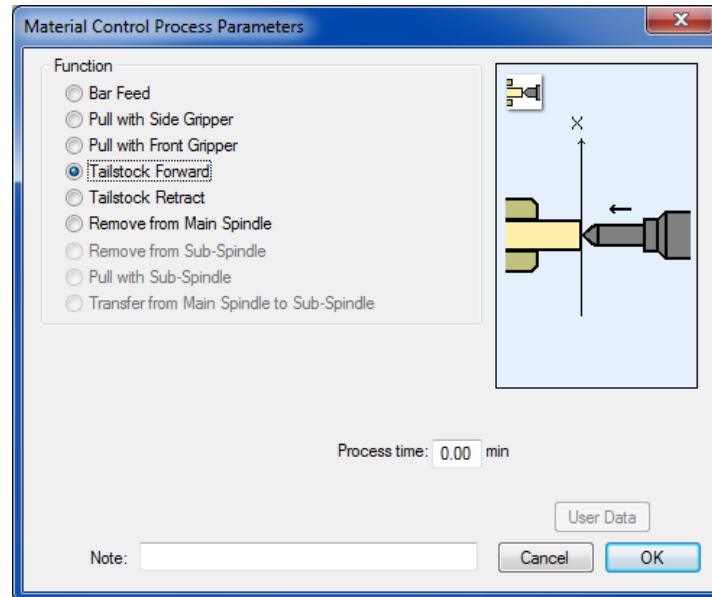
Tool Offset: Enter the offset number for the Front Gripper. The number must be a one or two digit integer.

Pull Distance (Zp): Enter an incremental z distance from where the Front Gripper has clamped onto the bar to where the stock is to be pulled.

Process Time: Enter a time estimation from when the Front Gripper moves from its safe index position at the beginning of the process to when the Front Gripper moves back to its safe index position at the end of the process. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Tailstock Forward

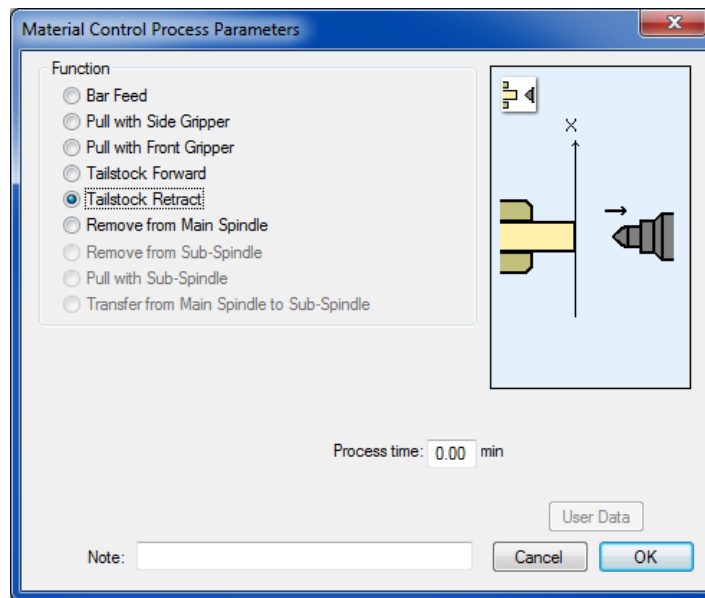
The Tail Stock Forward Material Control Process allows the user to insert code for bringing a tailstock forward for support as a steady rest. This MCP is used for machines equipped with a hydraulic Tailstock. The tailstock will be in the advanced position until the Tailstock Retract MCP is activated.



Process Time: Enter a time estimation during Tailstock advancement. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Tailstock Retract

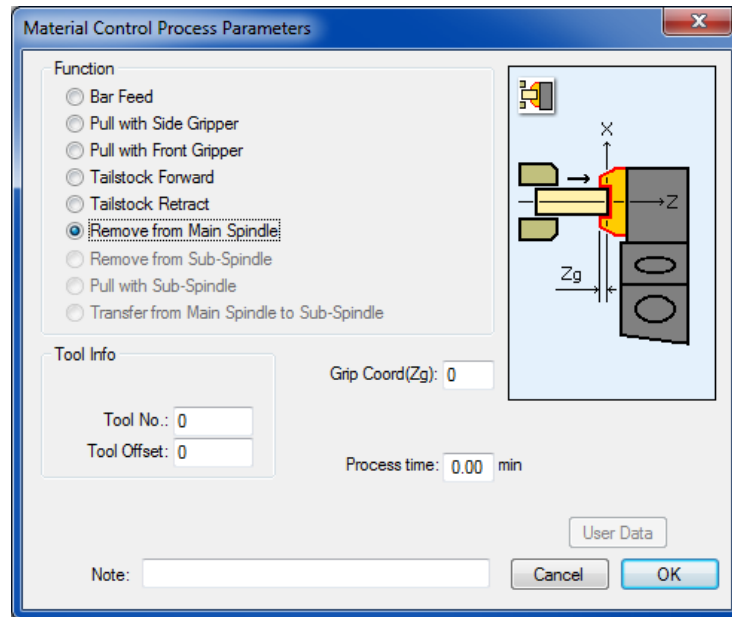
This Tailstock Retract Material Control process is used for machines equipped with a hydraulic tailstock. If selected, the tailstock retracts away from the face of the part. The tailstock will be in the retracted position until **Tailstock Advance** is activated.



Process Time: Enter a time estimation during Tailstock retraction. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Remove from Main Spindle

The Remove from Main Spindle Material Control Process allows the user to insert code for removing a work piece from the main spindle using a part gripper mounted in the turret.



Tool No.: Enter the tool station number at which the Part Ejector is mounted. The number must be a one or two digit integer.

Tool Offset: Enter the offset number for the Part Ejector. The number must be a one or two digit integer.

Grip Coord (Zg): Enter an incremental z distance (measured from the face of the work piece on the Main Spindle) to where the Part Ejector is to clamp on to the work piece.

Process Time: Enter a time estimation from when the Part Ejector moves from its safe index position at the beginning of the process to when the Part Ejector moves back to its safe index position after it has released the work piece. Time can either be set to seconds or minutes – see Preferences on how to set time units.

Chapter 3: Turn Tutorial Part

Introduction

This tutorial is designed to help you learn some of the most common **PartMaker** commands and features you'll use every day. The illustration on the following page represents a typical task you might perform. The name of this part is Turndemo. You'll create the Turndemo part in this chapter using **PartMaker**.

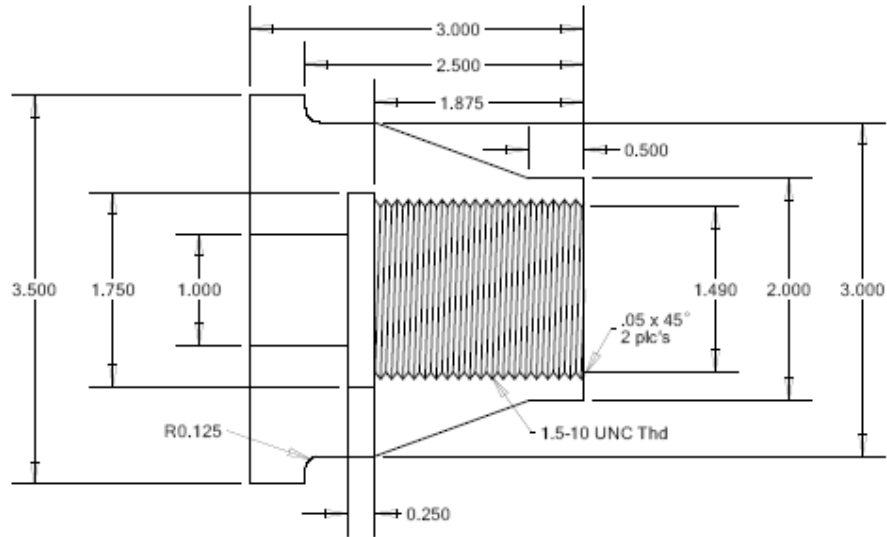
This tutorial is presented to cover both inch and metric data entry. Inch units are presented as the primary unit to enter and metric values are presented in parentheses.

How You'll Create Turndemo

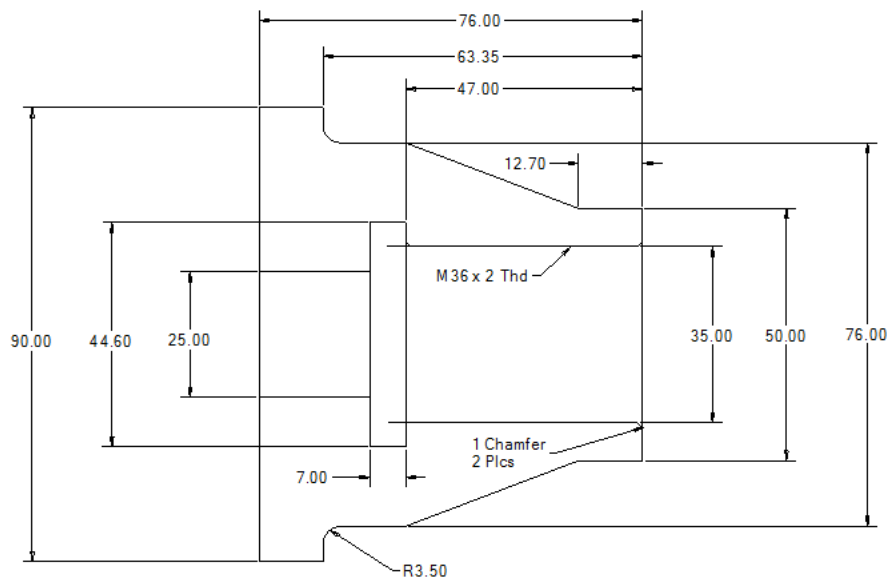
Here are the major steps you will follow to create the Turn tutorial part:

- √ Start **PartMaker**
- √ Define the stock boundaries for the face window
- √ View tool information for the part
- √ View cycle information for the part
- √ Select a material for the part
- √ Setup defaults for Turn
- √ Create Geometry
- √ Perform Turning operations
- √ Face the front of the part to finished length
- √ Define the hole for a drilling cycle
- √ Define the profile for an external turning cycle
- √ Define a groove for an internal grooving cycle
- √ Define an internal threading cycle
- √ Create a cut-off operation
- √ Generate the Process Table
- √ Simulate the cutting of the part
- √ Select a post processor
- √ Generate an NC Program

Inch Units



Metric Units



The major operations involved in producing the part shown in TURNDEMO are as follows:

- Face the front of the part using Face Turning
- Center drill and drill the 1.00-inch diameter hole in the center using a Drilling Cycle.
- Rough and finish the OD profile using OD Turning
- Bore the ID profile and create the chamfer using ID Turning
- Groove the thread relief using an ID Grooving cycle
- Cut the ID Threads using ID Threading
- Cut-off the part

You may find it useful to refer to the above illustration as you program Turndemo.

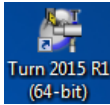
Starting PartMaker Turn

You will now look in detail at each of the steps necessary to program each of the components of the finished part. The components of the completed tutorial Turn part can be found in your PartMaker Turn directory under the following file names:

Completed Job File: C:\PartMaker\PM-Turn\Turndemo.job

Completed Tools Data Base: C:\PartMaker\PM-Turn\Tools.tdb

Completed Cycles Data Base: C:\PartMaker\PM-Turn\Cycles.cdb

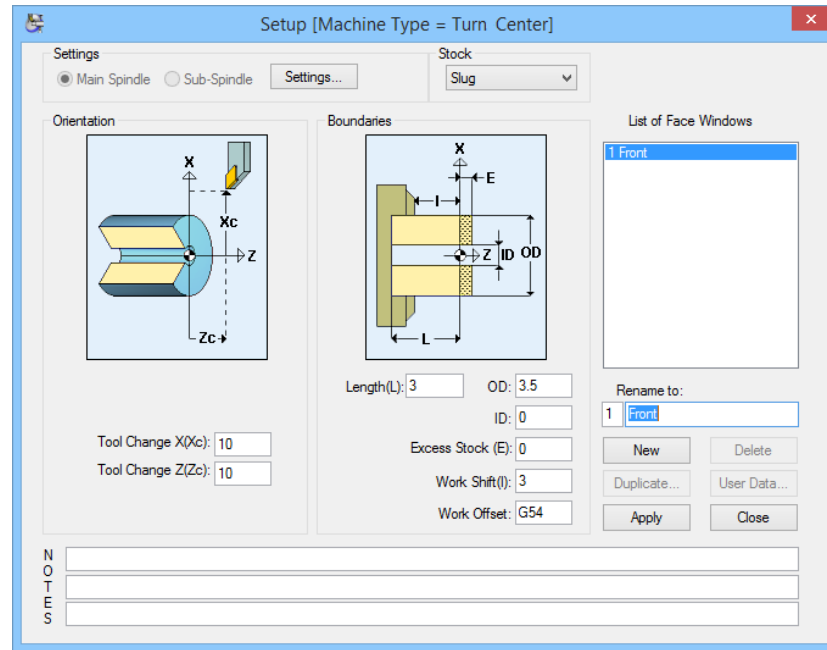


- 1 Double click the desktop icon to start the **PartMaker** application.
A window containing the **PartMaker** logo appears.
- 2 After a few seconds, you will see the Setup definition dialog box open as seen in the next section.

Setup Part Boundaries

In this section you'll specify the part boundaries for the **Face Window**.

The first dialog you see when you start **PartMaker** is the **Setup** dialog, shown below.



To manually open this dialog, choose **Setup** from the **View** menu.

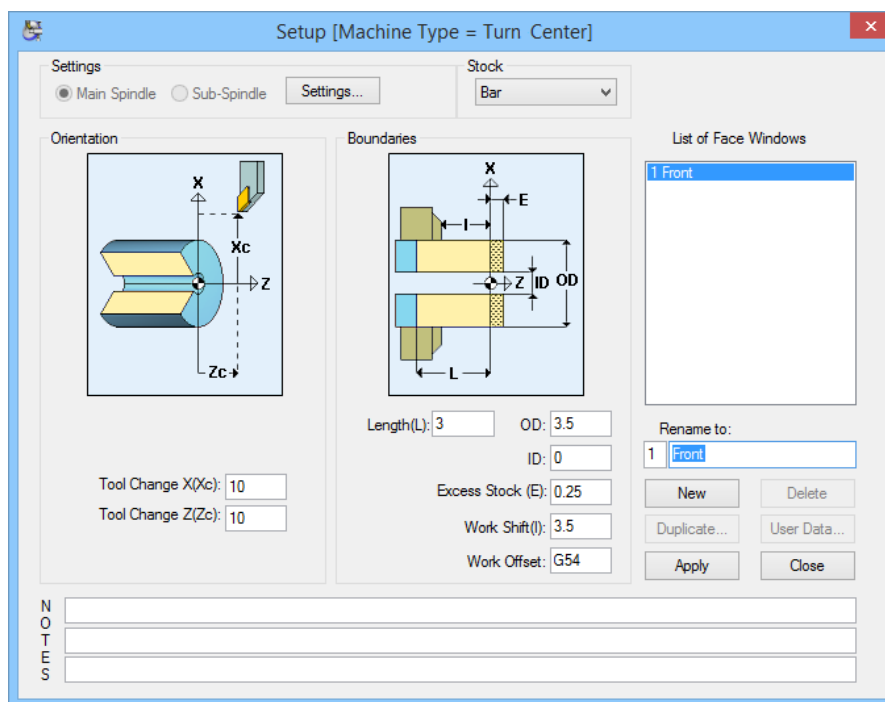


Tip: To advance from one numeric entry field to another in a dialog, press the **<Tab>** key. You can also press the **<Tab>** key to advance from the Z coordinate field to the X coordinate field when a numeric prompt appears at the bottom of a **Face Window**.

The first step in creating any part and/or part program is to complete the **Setup** dialog. To do so for Turndemo part:

- 1 From the **Stock** drop down menu, choose **Bar** to indicate you wish to work from.
- 2 Next, define the size of the stock for Turndemo.job which is 3.0" (76.0) long, 3.5" (90.0) diameter by making the following entries:

Length (L):	Length of the finished stock	3	(76.0)
OD:	Outside diameter of the stock	3.5	(90.0)
ID:	The unmachined stock has no center hole	0	(0.0)
Excess Stock (E):	The amount of material to be removed from the stock to create the desired finished length	0.3	(0.5)
Work Shift (I):	The amount of stock exposed from the work holding (chuck for slug work, collet for bar work). Since you are working from Bar Stock, this should be greater than the overall length of the part to accommodate for width of the cut-off tool.	3.5	(90.0)
Tool Change X (Xc):	Default tool change position X-axis coordinate	10	(100.0)
Tool Change Z (Zc):	Default tool change position Z-axis coordinate	10	(100.0)



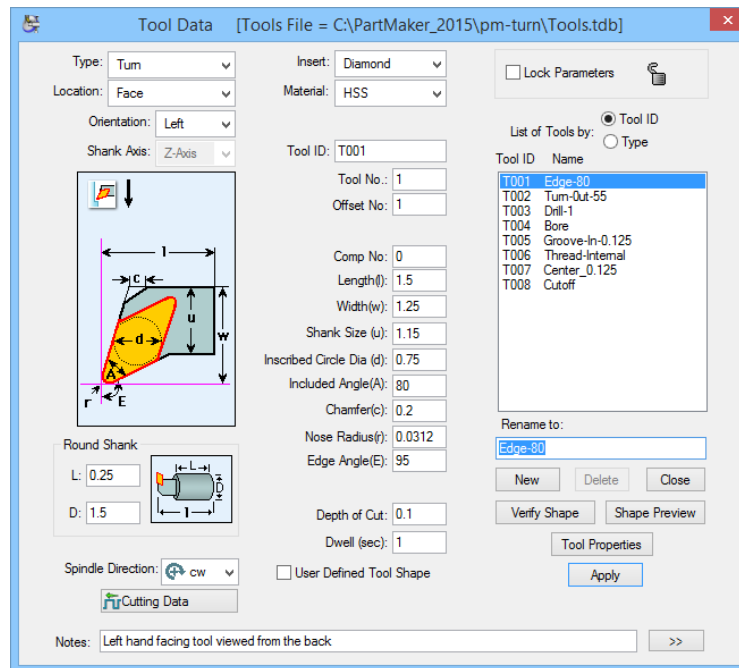
After you are sure all your entries are as shown above, click **<Close>**.

View the Tool Information

The tool library for the Turn tutorial has been created.



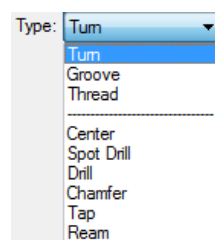
- 1 Choose **Open Tools File** from the **File** menu.
- 2 Open the **Tools.tdb** from the **C:\PartMaker\PM-Turn** directory as shown below:
- 3 Choose **Tools** from the **ToolMinder** menu to display the Tool Data dialog.
- 4 Select the tool ID T001 from the list shown on the right side of the dialog to view a graphic display of the tool and its characteristics.



Notice on the color graphic that there is a pair of intersecting pink lines. The intersection point of these lines is the default tool programming point used by **PartMaker**.

Each tool has many parameters that need to be fully defined so that **PartMaker** can accurately program your parts.

In the upper left-hand corner of the dialog is a pull down menu called **Type**. This menu is used to select the basic tool types. For turning operations, these types are typically: **Turn, Groove, Thread, Center, Spot Drill, Drill, Chamfer, Tap and Ream**.



The **Location** menu describes the tool locations of **In, Out** and **Face**.

The **Orientation** menu describes the tool orientations of **Left** and **Right**.

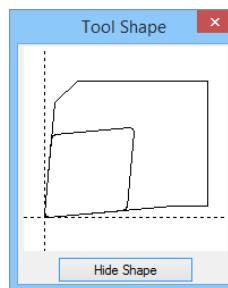
In the center, top of the dialog, is a pull down menu that defines the basic insert shape. Open this list to see the available shapes. The defined shape for T001 is **Diamond**, shown on the following page.

In a vertical column in the center of the dialog is a list of the parameters that are used to define the exact shape of the tool and its holder.

The color graphic to the left clearly shows what each of the parameters describes, except for the following, which will be given special attention:

- **Material:** defines the cutting insert material, typically either **HSS** or **Carbide**.
- **Tool ID:** A unique ID number that is used by **PartMaker** to keep track of the tools you have defined.
- **Tool No.:** The number as it appears in the actual part program.
- **Comp No.:** The tool nose radius compensation number used for the tool.
- **Offset No.:** The offset register number used to define the length and diameter offset values for the defined tool.
- **Depth of Cut:** The value used as the default depth of cut at any time this tool is used. The programmer may change this value at any time during the programming process as needed.

Below is a button labeled **Verify Shape** this button is used to display the exact shape of the tool defined by the user parameters. Press this button now to see the exact shape of the **Edge-80** tool.



- 5 Click on the **Hide Shape** button to close the dialog.

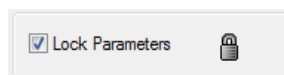
There are three other items of interest on this dialog for you to note before proceeding:

- **Speed Factor:** User adjustment for the spindle speed calculated by the software.
- **Feed Factor:** User adjustment for the feed rates calculated by the software.
- **Dwell (sec):** Dwell time for the defined tool.



Note: The tools used in this chapter are for the *Turndemo* tutorial only. As you begin to use **PartMaker**, you will create your own tools database.

Once you are finished editing the tool information select the **Lock Parameters** check box to disable all parameters for that particular tool. Each tool can be individually locked to prevent modification.



- 6 When you are finished viewing tool information, click the **<Close>** button.

View the Cycle Information

PartMaker lets you combine a number of repetitive operations (center drilling & drilling for example) into a single entity called a **cycle**. The Cycle information you create is saved in a *Cycle Database* for easy retrieval.

The cycles library for the Turn-Mill tutorial has also been created.

- 1 Choose **Open Cycles File** from the **File** menu.
- 2 Open the **Cycles.cdb** from the **C:\PartMaker\PM-Turn** directory as shown below:
- 3 To view the cycles, choose **Cycles** from the **ToolMinder** menu.
- 4 Select **DRILL_1 (Drill_25)** on the right side of the dialog to view the information relating to this cycle that will be used in Turndeno.JOB tutorial.



Note: If left blank, the Depth for each operation is calculated automatically by **PartMaker**.

The screenshot shows the 'Cycles Data' dialog box with the title bar '[Cycles File = C:\PartMaker_2015\pm-turn\Cycles.cdb]'. The dialog is divided into two main sections. The left section contains a table for defining cycle operations, and the right section contains a 'List of Cycles' and control buttons.

Major Operation	Operation Type	Hole Canned Cycle	Hole Diam	Depth	Tool ID
<input type="radio"/>	Center	DRILL	0.625	0.5	T007
<input checked="" type="radio"/>	Drill	DEEP HOLE	1		T003
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					
<input type="radio"/>					

Below the table is an 'Insert Operation' button. On the right, the 'List of Cycles' shows 'Drill_1' selected. Below this list are buttons for 'Rename to:', 'New', 'Delete', 'Duplicate', 'Close', 'Select Tools', and 'Apply'.

Note that the **DRILL_1** cycle consists of two operations: Center Drilling and Drilling.

Notice that there is a specific depth for the center drill 0.125 (3.0) and no depth specified for the deep hole drill. This is because **PartMaker** will automatically determine the depth of the deep hole when the cycle is applied in the program.

- 5 When you are finished looking at **Drill_1 (Drill_25)** cycle, close the dialog box.

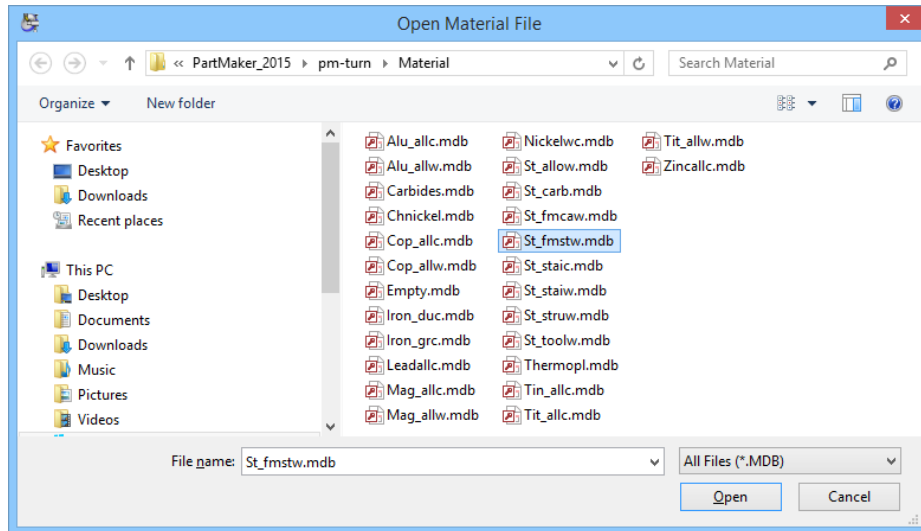
Select a Material for the Part

Before you create a part program for Turndemo, you must select the type of material to be machined in the Open Material File dialog. Turndemo is to be cut from 303 Stainless steel, which is classified as **Free Machining Stainless Steel**.

PartMaker has a materials database from which you can select a variety of material types with different characteristics. A materials database allows you to store your shop's accumulated materials data. **PartMaker** uses this data to automatically calculate feed rates and spindle speeds.

To tell **PartMaker** that you want to cut a part from for this material:

- 1 Choose **Open Material File** from the **File** menu to display the Open Material File dialog, shown below:



2. Locate and select ST_FMSTW (Free Machining Stainless Steel Wrought) from the list on the left side of the dialog, then click **<OK>**. You are returned to the **Face Window**.

If you wish to learn more about material files, do the following:

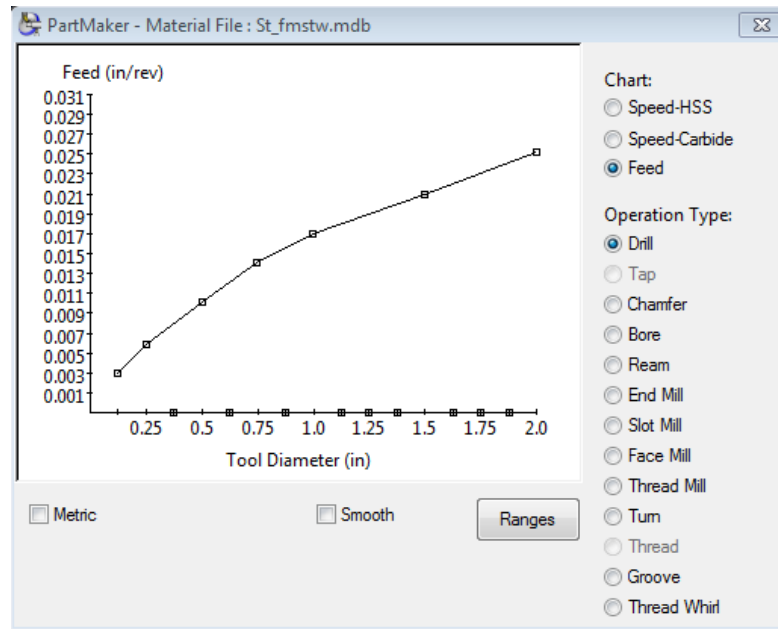
- 1 Go to the **Help** menu and select **Index**
- 2 Select Material Library
Notice the abbreviated and complete descriptive names for each material library.
- 3 Click on **ST_FMSTW.MDB** and you will see a list of the different grades of stainless steel to which the database applies. In this case, you will see 303 as one of the grades.

View the Material Characteristics

In this section, you will view the characteristics of the material you selected with respect to feed rate and/or speed, and the operation type.

The Material dialog lets you view and/or change surface speed and feed information for each operation based on the currently open material file. For each material and operation, you enter data by dragging points on the graph. (Points are initially positioned on the X-axis of the graph.)

- 1 Choose **Material** from the **ToolMinder** menu to display the Material dialog.



- 2 Click the **<Feed>** button to see how the feed varies for the currently selected operation.
- 3 Click an Operation Type to see the feed for that operation.
- 4 Drag a point on the graph in the dialog to see how you can change the feed for the selected operation.
- 5 Click the **<Ranges>** button to see how the chart is calculated. Notice at the bottom of the **Ranges** dialog how the **First Pass** and **Last Pass** threading infeed defaults are defined.
- 6 Click the **<Cancel>** button to close the **Ranges** dialog.
- 7 Double-click the close box in the upper-left corner of the dialog to return to the **Face Window**.

Setting Defaults

Before beginning the Turn tutorial, you will have to set certain turning defaults to facilitate the programming process:

- 1 To access the **Defaults** dialog, select **Defaults** from the **Job Optimizer** menu. After doing so, you will see the dialog below:

The screenshot shows the 'Defaults for Turning' dialog box. It contains several sections for configuring turning parameters. The 'Hole Group Parameters' section includes a checked 'Through Hole' option and fields for Diameter (0.25), Chamfer (0), Clearance (0.05), and Nominal Depth (0). The 'Profile Group Parameters' section includes fields for X Finish (0), Z Finish (0), Return Length (0.05), Return Angle (45), Surface Roughness (63), Chamfer Length (0), and # Spring Passes (1). It also has checked options for 'Start at Cutting Point' and 'Return to Cutting Point', and a 'Thread Height' dropdown set to '% Pitch'. The 'Process Parameters' section includes a checked 'Apply Comp in PartMaker' option, an unchecked 'Output Canned Cycle for Turning' option, a 'Coolant' dropdown set to 'Standard', 'Feed Units' set to 'upr', a checked 'Constant Surface Speed' option, 'Default Feed' (0.01 upr), and 'Default Speed' (100 fpm). The 'Machining Data' section includes 'Min Feed' (0.1 upm), 'Max Feed' (100 upm), 'Rapid Feed' (500 upm), 'Max Speed' (10000 rpm), and 'Tool Change Time' (0.1 min). The 'Retract from Groove Options' section has a 'Groove Options Defaults' button. The 'Input Options' section has a checked 'Diameter Programming' option. The 'Tool Path Options' section has checked options for 'Corner Rounding' and 'Remaining Stock Detection'. The 'Leads' section includes 'Arc Radius' (0), 'Line Length' (0.05), and 'Lead Angle' (0). There are 'OK' and 'Cancel' buttons at the bottom right.

- 2 Make sure to check the **Diameter Programming** box. Checking this box will allow you to define all geometry and feature programming in terms of diameter, alleviating the need to divide measurements in half when programming or drawing on the X-axis in Turning Face Windows.
- 3 Select **Thread Height**. The following options are available:
 - %Pitch**. The user will enter the height as a percent and resulting height will be $\text{Pitch} * \text{Thread Height} / 100$.
 - Part Units**. The user will enter the actual measured height of the thread in the current part units.
- 4 Set a default **Line Length** for your programmed **Leads**. For the Turn tutorial this value should be 0.100 (2.5)



Note: Setting a generally applicable lead as a default will save you a great deal of time when programming because you will not have to reenter the same data from feature to feature.

Turning

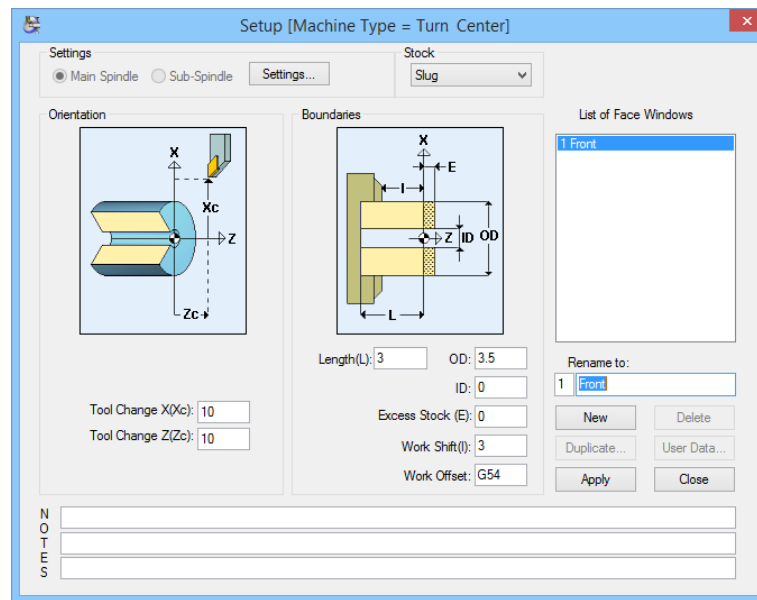
This section of the tutorial will instruct you in the steps necessary to perform the turning operations.

Setting Up the Face Window

In this section, you will set up the **Face Window** for the turning operations. To do so:

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box. In this dialog, you will define the machining function and part boundaries and location of the stock with respect to the work holding.

The default **Setup** screen will appear as shown below:



- 2 Rename the current face window by clicking on **Front** in the **List of Face Windows**: field and type **Turning** in the **Rename To**: field and click the **<Apply>** button.
- 3 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current face window. Click the **<Close>** button to close the **Setup** dialog.
- 4 In the CAM face window, choose **Show Axes** from the **View** menu. This will display the horizontal and vertical axis lines which will help you create the geometry in the next section of the tutorial.



Creating Geometry

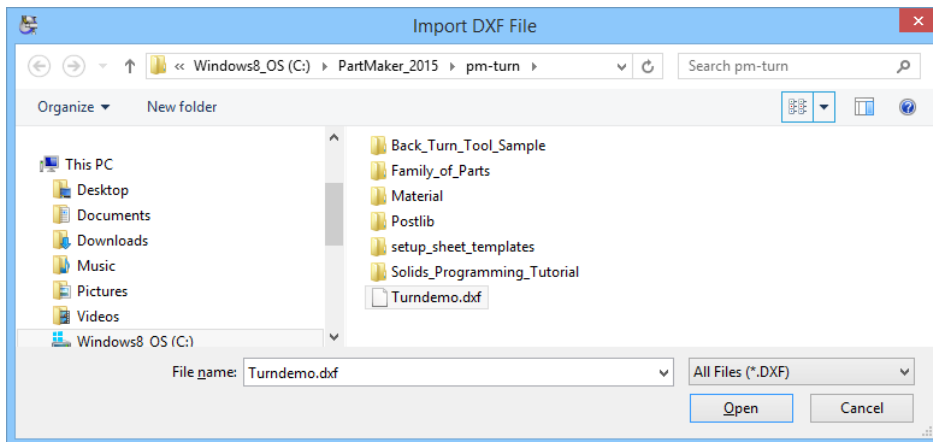
There are two methods to create Geometry in the PartMaker. You can either import Geometry from a CAD file (method 1) or define Geometry using PartMaker's own CAD mode (method 2). Here we will show how to define geometry in both ways. Additionally, at the end of this chapter we will demonstrate another way of creating the geometry for our tutorial part in PartMaker CAD mode.

Method 1: Import Geometry from a CAD system via DXF file

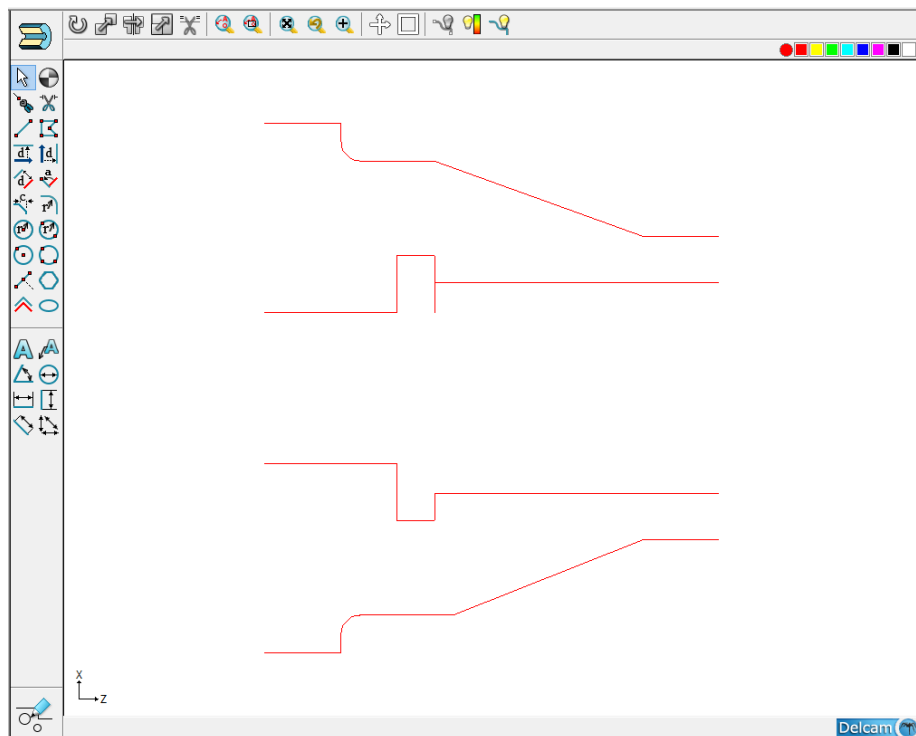
Import the geometry as follows:



- 1 Choose **Import DXF file** from the **File** menu.
- 2 Select **Turndemo.dxf** from the **C:\PartMaker\PM-Turn** directory as shown here:



Your screen should now appear as shown below:



Now that you have a drawing in the **Face Window**, take a few minutes to explore some of the basic features of **PartMaker** that you will be using.

From the **View** menu, you have the following **Zoom** options available: (Not all options have a corresponding icon.)



Select **Full View** to expand the drawing to fill the screen.



Select **Previous View** to return the screen display to the previous setting.



Select **Zoom In** and use any mouse button to drag opposite corners of a rectangle that you want to magnify.

Select **Zoom Out** to shrink the drawing on the screen

Select **Continuous Zoom** to magnify the drawing at a steady rate. You can do this by pressing **<Ctrl>** and the **<+>** or **<->** key.

Select **Continuous Pan** to move the drawing to the left or right at a steady rate. You can do this by pressing **<Ctrl>** and the directional keys.

Select **Show Axes** to show the current X0 Z0 location. Select **Hide Axes** to turn them off again.



Show and **Hide Grid** is used to display a user definable grid that can be used to locate convenient points on the screen.



Show and **Hide Boundaries** is used to show the boundaries of the rough stock being turned.

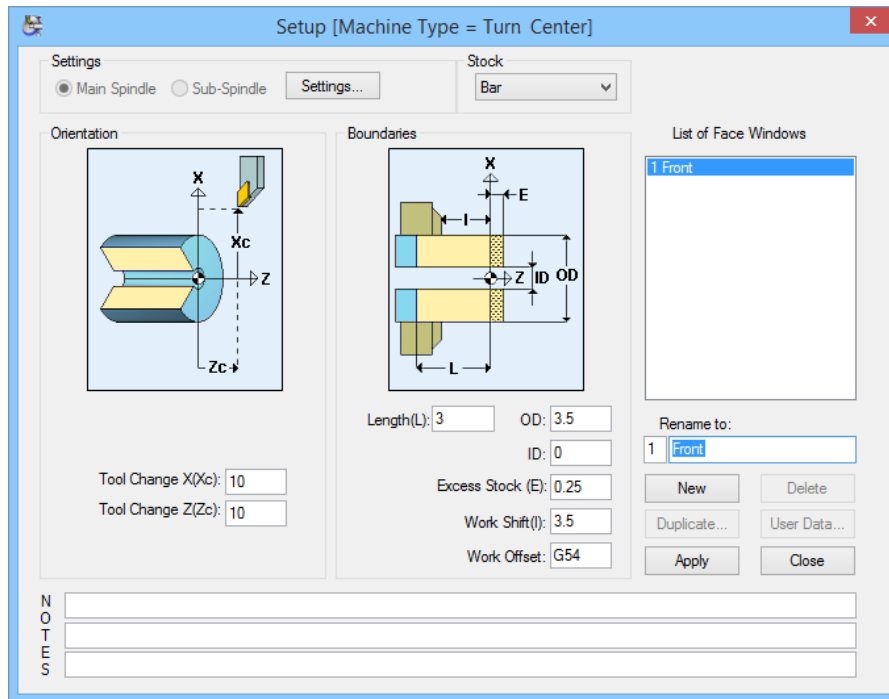
Depending on your needs, any or all of these display options may be activated at any time during the drawing (CAD) or programming (CAM) process.

Method 2: Create Geometry in PartMaker CAD

Now that you are familiar with geometry imported via a DXF file, we will now show you how to create the geometry using **PartMaker** CAD functions.

The method that we will demonstrate in the following pages will demonstrate how geometry can quickly and easily be created in **PartMaker** CAD. At the end of the chapter, we demonstrate another series of steps for drawing the same part. This second exercise will give you an overview of as many CAD features as possible with the sample part. It is recommended you try drawing the part with both exercises to gain a fuller understanding of the capabilities of **PartMaker** CAD.

- 1 First, create a new job by selecting **New Job** from the **File Menu**.
- 2 Make sure you have setup your part defaults correctly as shown below. From the **View** menu, select **Setup**.
- 3 Make sure the Setup dialog box is set exactly as you did in the previous section of the Turndemo sample. A copy of the finished dialog is again provided for your reference:



- 4 To create the profile of the TURNDEMO part in **PartMaker** CAD, first make sure you are in CAD mode. You can switch between CAM and CAD mode by clicking on the button in the lower left-hand corner of the screen.

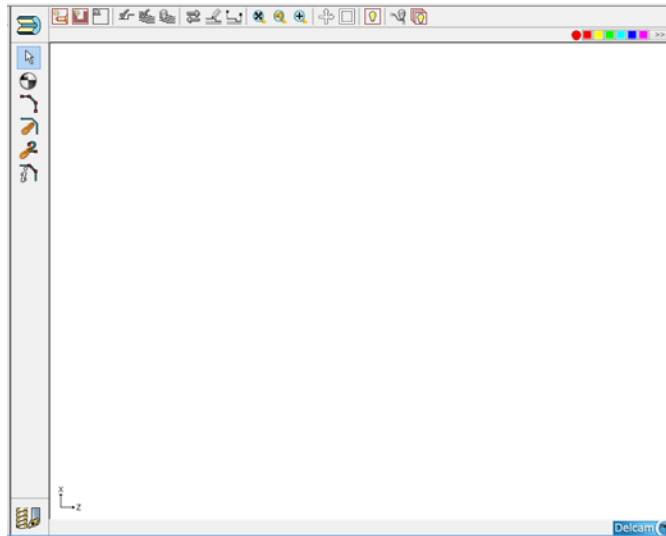


The button looks like this in CAM mode.



The button looks like this in CAD mode.

The window you are in should be empty and look like this:

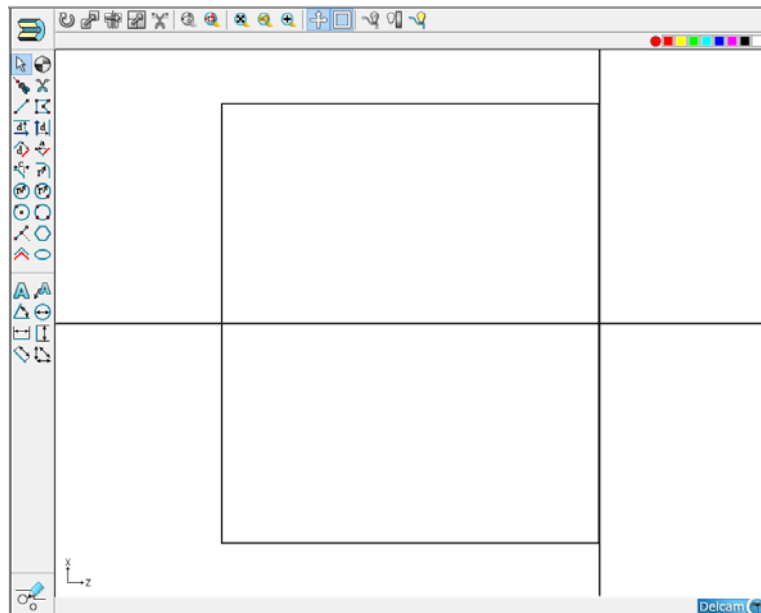


Now that you are in CAD mode you will want to show the X and Z axes as well as the boundaries of your part. To do so,



- 5 Select the **Show boundaries** icon and the **Show Axis** icon from the Toolbar.

After doing so, your window should show the Z and X axes as well as the boundaries of your part as shown below.



Once you begin drawing you may choose to hide the boundaries and axes.



- 6 Now we are ready to begin drawing the profile of our part. To begin drawing we select the Connected Lines Icon from the drawing tools on the left of the **Face Window**.

ZX

We will use the coordinate snap mode to begin drawing the profile. When prompted enter the following Z, X coordinates:

0, 1.5 (0, 35)

0, 2.0 (0, 76)



Note: To switch between diameter and radius programming in CAD, choose the **Defaults** command from the **Job Optimizer** menu and check the **Diameter Programming** box.



- 7 We will now draw a line to the next point relative to the last point we have entered. Staying with the **Connected Lines Icon**, enter **<@-0.5>** (**<@-12.7>**) in the Z coordinate prompt when it appears at the bottom of the screen. Here you do not need to enter anything for the X. Entering an '@' sign before a coordinate in the coordinate snap mode will plot a point relative to the last point plotted and not relative to the absolute axes.



Note: You can make the '@' symbol by pressing **<Shift+2>**.



Note: If you are plotting coordinates in **PartMaker** you do not need to reenter a coordinate if it is the same as the last one entered. You can simply leave the space blank and **PartMaker** will default to the last coordinate entered.

Once you have performed this step, enter the following ZX coordinates

-1.875, 3	(-47.0,76.0)
-2.5, <default>	(-63.35, <default>)
<default>,3.5	(<default>, 90.0)
-3.0, <default>	(-76.0, <default>)



Note: To quickly switch between entry prompts in **PartMaker**, use the **<Tab>** button instead of pointing and clicking with the mouse.

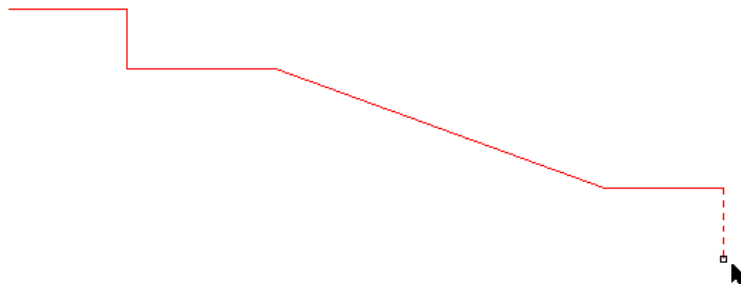
After entering these coordinates your drawing should like the one below:



- 8 Once you have drawn the geometry shown above, choose the **Selection Icon** to break the continuity of the point-to-point drawing mode and move your cursor about the screen freely.



- 9 To draw the under side of the profile, we again select the **Connected Lines Icon**. Then select the **End of an Element snap mode** icon and clicking on the bottom right hand corner of the drawing as depicted below:



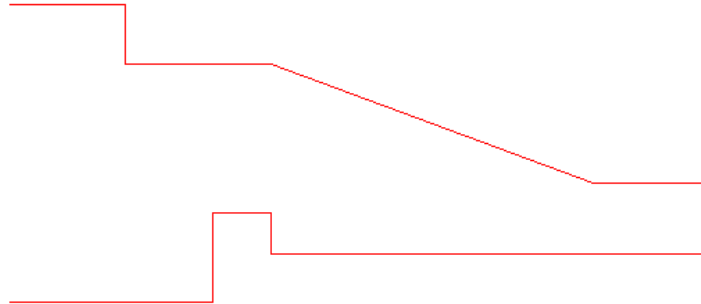


- 10 Next select the coordinate snap mode again and enter the following coordinates to complete the file:

ZX

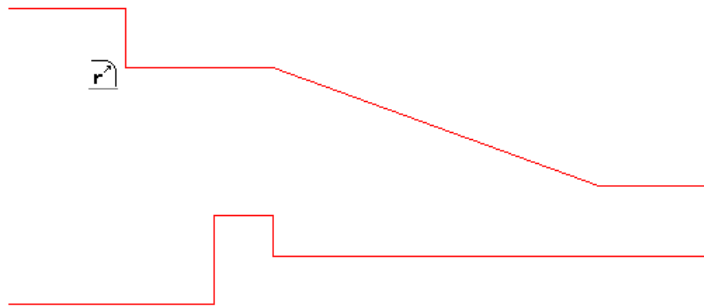
-1.875, 1.5	(-47.0,35.0)
<default>,1.75	(<default>, 44.6)
@-2.5, <default>	(@-63.35, <default>)
<default>,1.0	(<default>, 25.0)
-3.0, <default>	(-76.0, <default>)

After doing so, your profile should look like the drawing below:

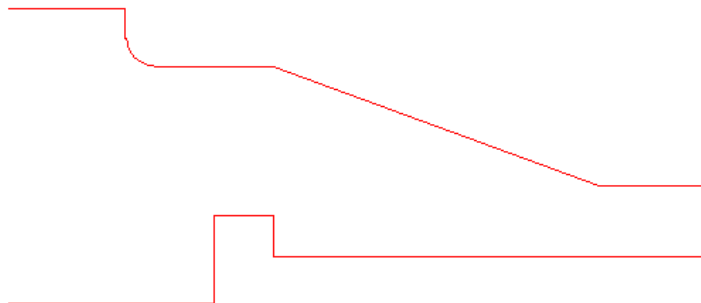


We will now enter a 0.125 (3.5) radius fillet as specified by the drawing. To do so:

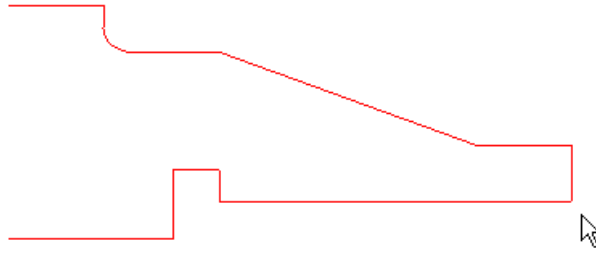
- 11 Select the Fillet icon, enter 0.125 (3.5) for the radius and drag the cursor to the area where you want to insert the fillet as shown below.



- 12 Click the cursor over this area to add the fillet. Your drawing should now look like this:



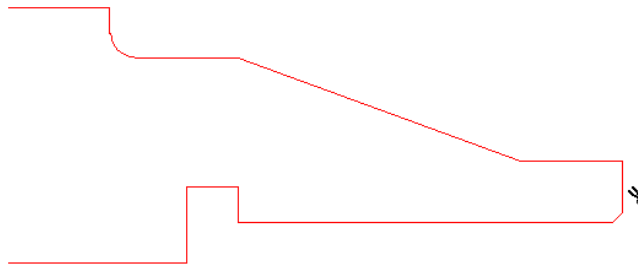
- 13 We will now add a small chamfer to the end of our geometry. To do so, select the **Chamfer** icon. Enter a chamfer of 0.05 (1.0) when prompted and press **<Enter>**. Move the cursor over the area where the chamfer is to be inserted as shown below.



- 14 Once you have moved the cursor into this position, just click your left mouse button to add the chamfer.



- 15 At this point our drawing is nearly complete. We now need only remove the excess geometry. To do so, we select the **Scissors Icon** and click on the unwanted geometry as shown below:



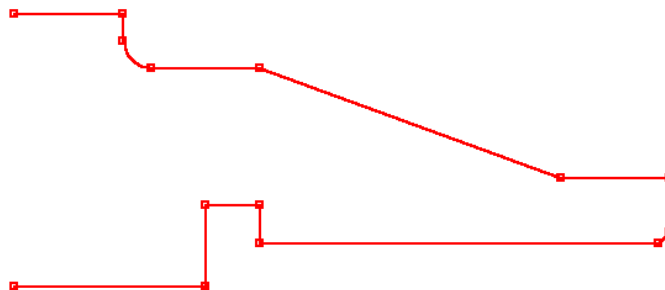
We will now be able to quickly create the lower half of the profile. To do so:

- 16 You will first need to select the geometry you have drawn. In **PartMaker** CAD there are many ways to select geometry. Here we will demonstrate two techniques of selection:

Technique 1:



Choose the **Selection** icon and then while holding down the left mouse button, drag a box around the entire picture and release the mouse button. You will have selected the entire picture when your part now looks like the picture below:

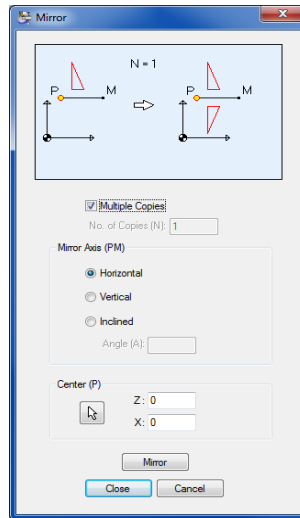


Technique 2:

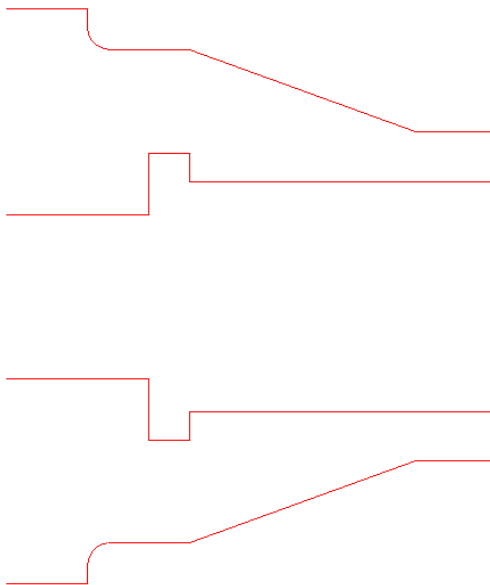
Click anywhere in the **Face Window** to deselect the geometry. Again, choose the **Selection Icon**. This time, under the **Edit** menu, choose **<Select All>** or press **<Ctrl+A>** and you will achieve the same results as in Technique 1.



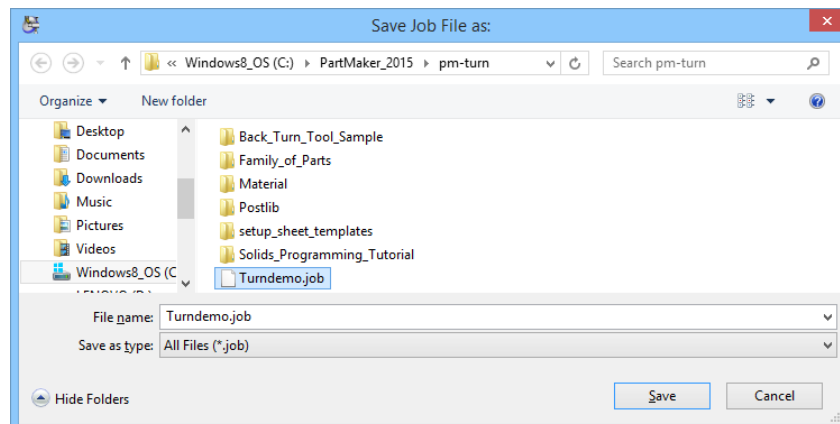
- 17 With the geometry selected, you can make a mirror image, as it will appear below the centerline. From the **Edit** menu, choose **Mirror**.
- 18 Enter the **Mirror** dialog values as shown below:



PartMaker will then create the following drawing:



Now that you have completed the drawing, save your work by choosing **<Save>** from the file menu. When doing so, you will be prompted with the dialog below:

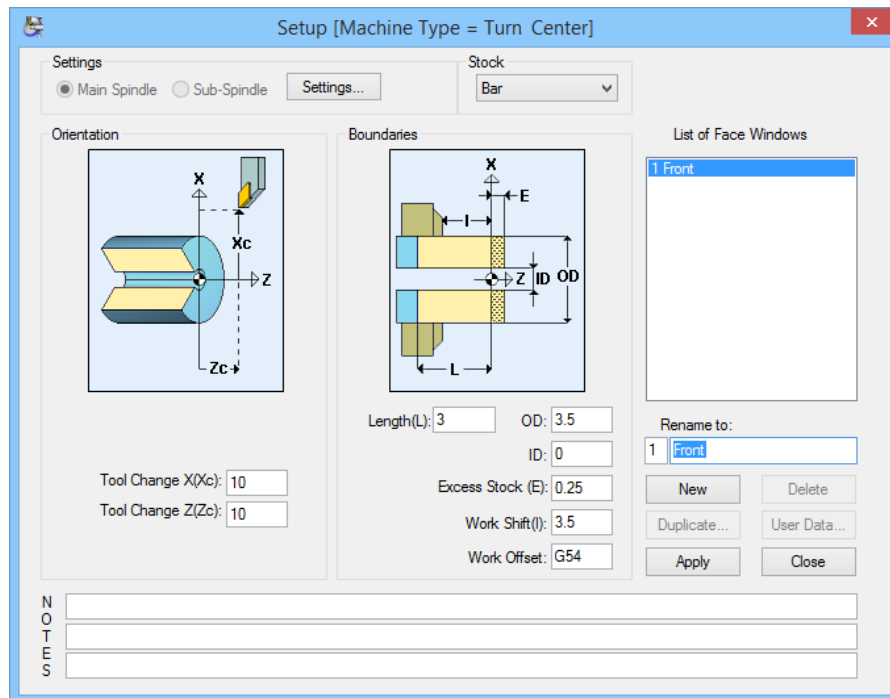


Alternate Method 2: Create Geometry in PartMaker CAD

Now that you are familiar with creating a program using **PartMaker** and geometry imported via a DXF file, we will now show you another way to create the geometry using PartMaker CAD functions.

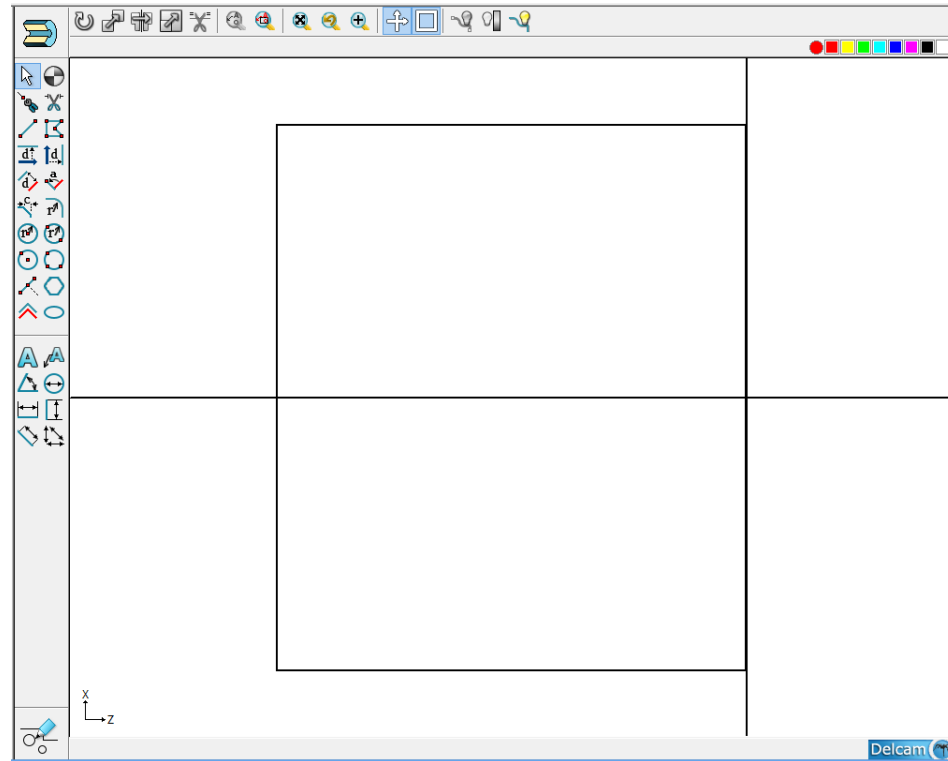
The methods that we will demonstrate to you in this section will give you an overview of as many CAD features as possible with the sample part.

- 1 First, you need to start a new Job. From the **File** menu select **New Job**.
- 2 From the **View** menu, select **Setup**.
- 3 Set up the dialog exactly as you did in the previous section of the TURNDEMO sample. A copy of the finished dialog is again provided for your reference:



- 4 Now that you are in CAD mode you will want to show the X and Z axes as well as the boundaries of your part. To do so, select the **Show boundaries** icon and the **Show Axis** icon from the Toolbar.

After doing so, your window should show the Z and X axes as well as the boundaries of your part as shown below.



The **boundaries** are the rectangular area. The **axes** are the heavy black vertical and horizontal lines on the right and center of the diagram respectively.

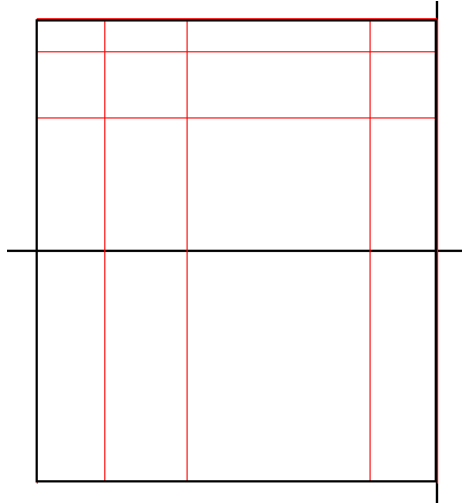
Creating the Outside Profile of the Sample Part





- 1 To create the profile of the outside, begin by using the **Line Parallel to Horizontal Axis** icon.
- 2 Select the icon and type the number 0 in the box and press <Enter>. **PartMaker** will draw a line along the centerline of the part.
- 3 Next, enter the number 3.5 (90.00).
*Note how **PartMaker** will draw the OD of the part without you having to divide the diameter by two.*
- 4 Enter the remaining two dimensions for the OD profile of the part: 3" (76.00) and 2" (50.00).
- 5 Now, click on the Line Parallel to Vertical Axis icon.
- 6 Make the following entries when prompted: 0 <enter>, -0.5 (-12.7) <enter>, -1.875 (-47.00) <enter>, -2.5 (-63.35) <enter>, -3.0 (-76.00) <enter>.

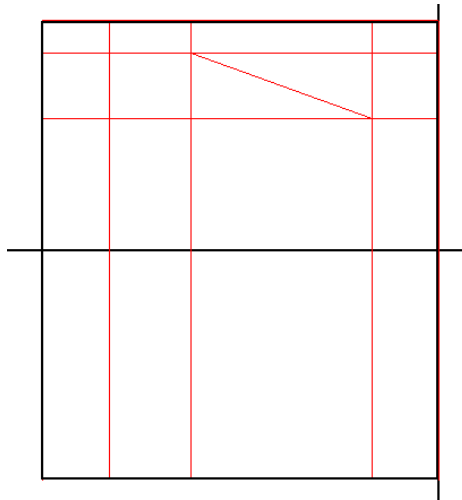



Your graphics window should look like this:



 **7** Next, draw the angled line representing the taper. Choose the **Line Thru 2 Points** icon.

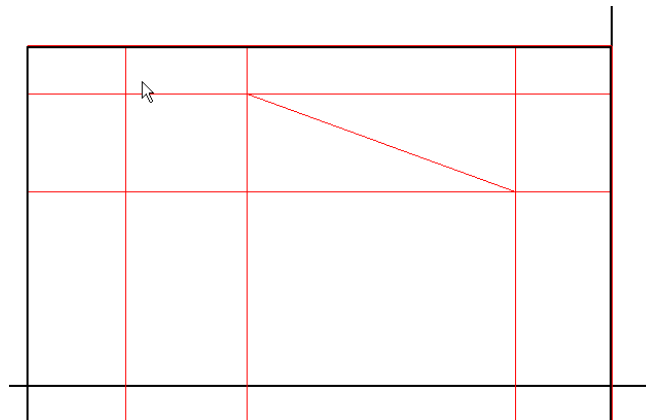
 **8** Choose the **Closest Intersection** snap mode and select the two points shown below to draw the taper:



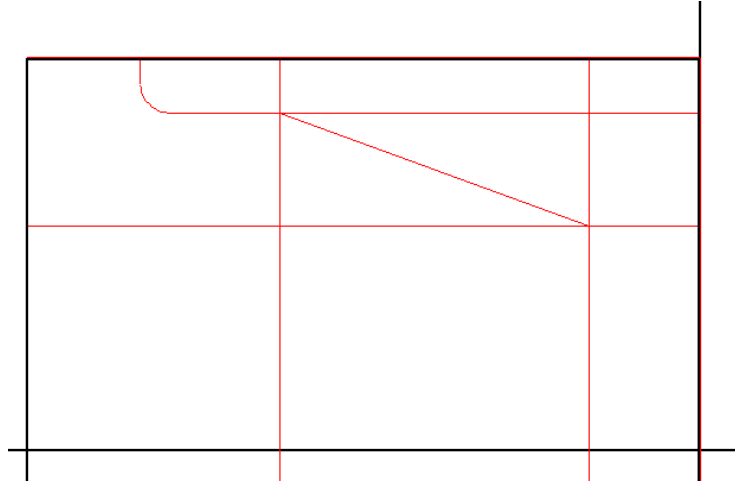
 **9** Draw the fillet using the **Fillet** icon.

10 Type 0.125 (3.5) in the box for the fillet radius and press **<Enter>**.

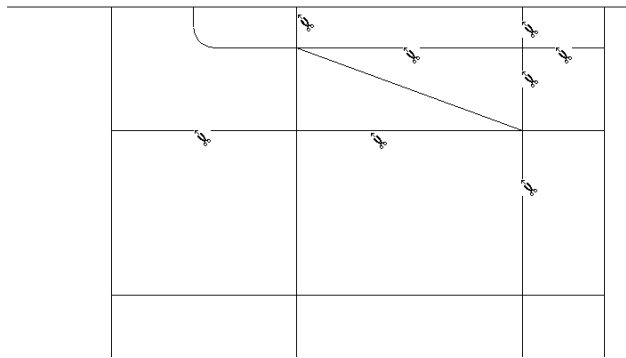
11 Click on the part just above and to the right of the intersection where the fillet is to be drawn, as shown here:



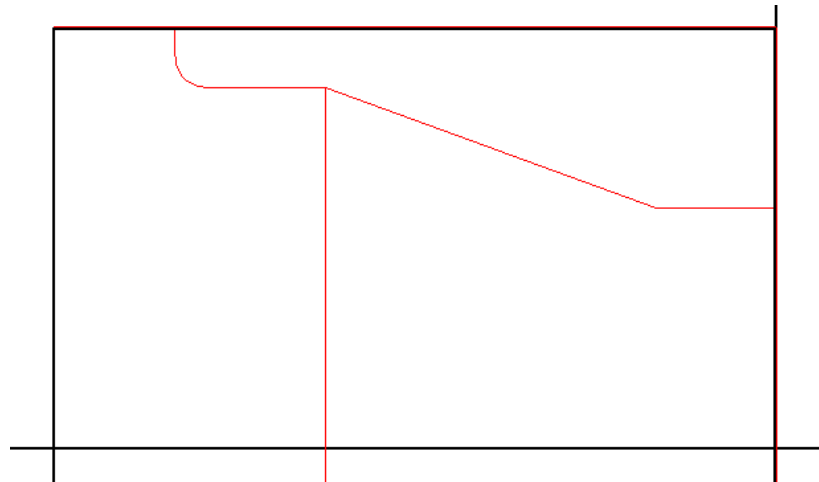
The result will be:



- 12 Next, choose the **Scissors Icon** and trim away the lines and parts of lines that are no longer needed. Note that some of the vertical line at the left hand end of the taper will remain.



The resulting profile should appear as shown below:



You are now finished creating the OD profile of the part.



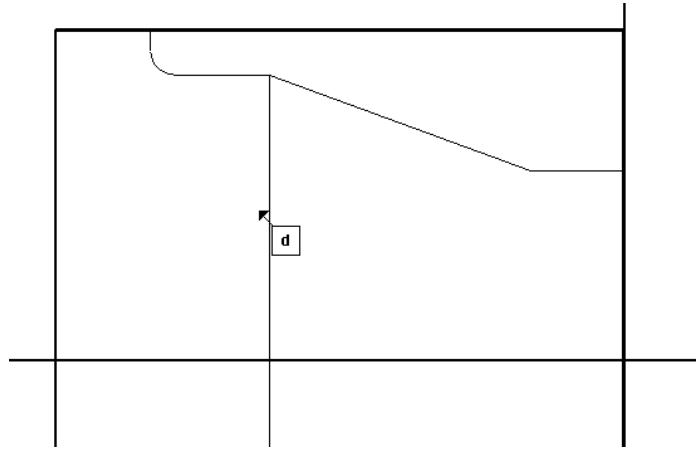
Now is a good time to **save** your work. Choose a new name that you have not previously used.

Creating the ID Profile of the Sample Part

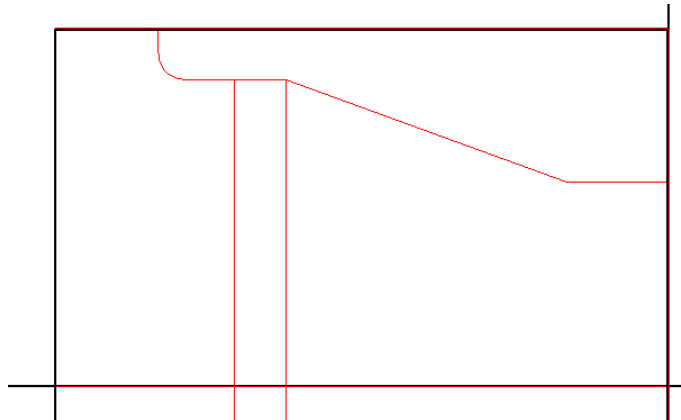
Before drawing the ID geometry of the part, it is a good idea to select a new color. For this example we will use **red**. Choose the color red from the **color bar** at the upper right hand corner of the screen.



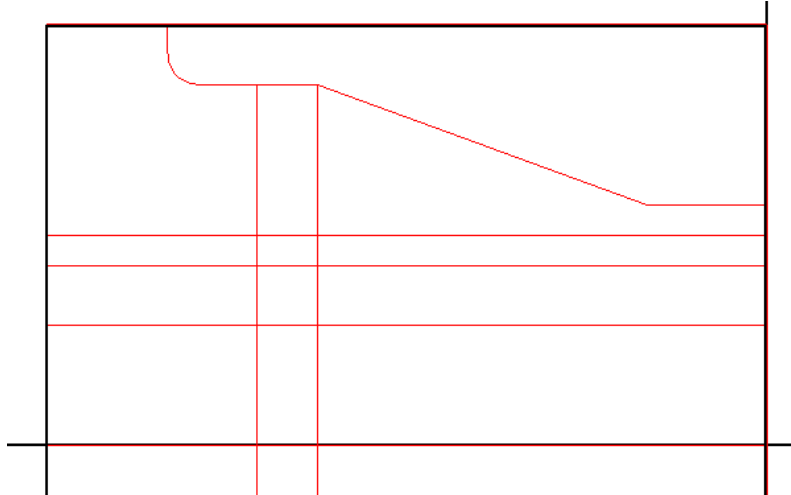
- 1 First we will create the only remaining vertical line needed for the ID. Select the **Parallel Element Icon**.
- 2 Type 0.25 (7.00) in the box and press **<Enter>**. Click with the mouse just to the left of the vertical line as shown here.



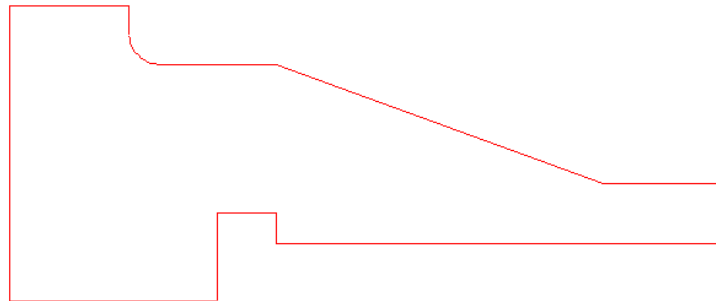
The result will be:



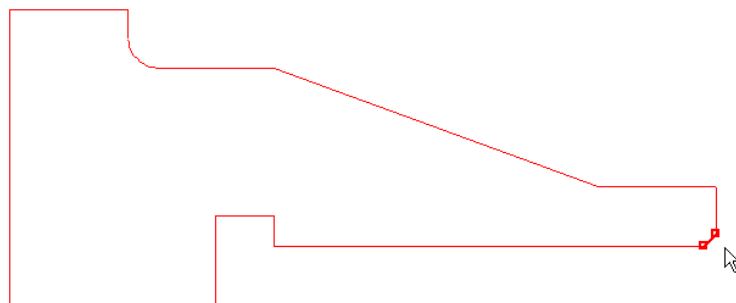
- 3 Now, using the **Line Parallel to Z Axis Icon** make the following entries:
1.00 (25.00) <Enter>, 1.750 (44.600) <Enter>, 1.490 (35.00) <Enter>.
The result will be:



- 4 Choose **Hide Boundaries** from the **View** menu and use the **Scissors** to trim away all unneeded lines that still remain. This includes the vertical lines at either end of the part.:
- 5 Using the **Scissors**, cut away the lines that are not needed until the drawing appears as shown here:



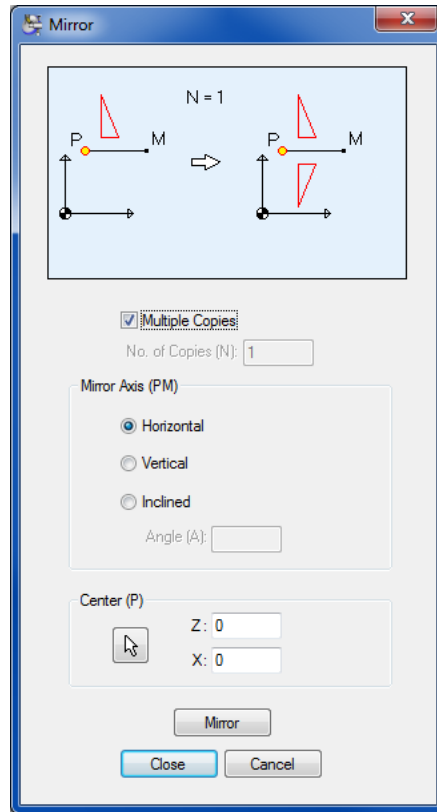
- 6 The next operation is to draw the **chamfer**. Choose the **Chamfer Icon** and enter 0.05 in the box and press <Enter>.
- 7 Click inside the corner shown here:



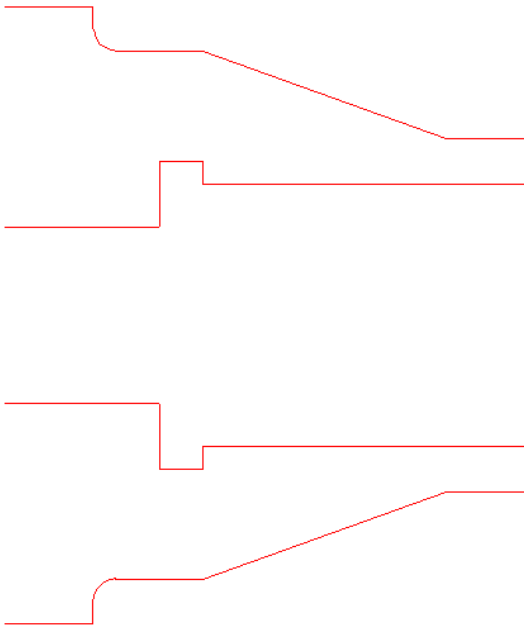
Note: Always remove vertical connecting lines between the OD and ID of part profiles. This action is necessary to prevent any closed loops from confusing the profile-chaining tool.

The next operation is to **Mirror** the drawn profiles so that they appear below the centerline. From the **Edit** menu, choose **Select All** then **Mirror**.

The **Mirror** dialog values are:



PartMaker will then create:



Now you have completed the drawing, save your work by choosing **<Save>** from the **File** menu.

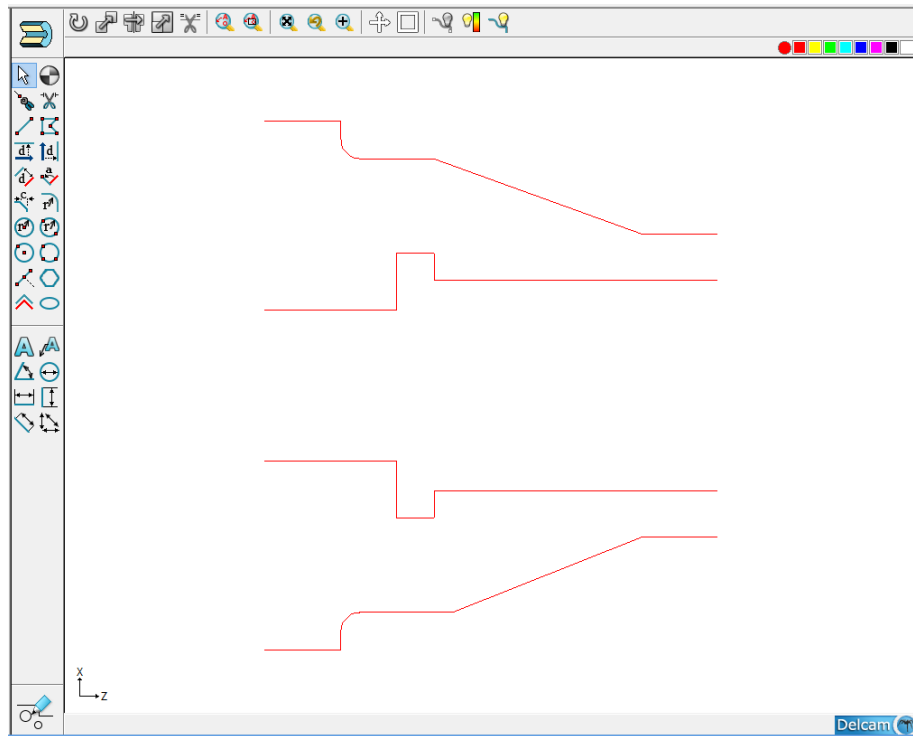
Creating TURNDEMO in PartMaker CAM Mode



Now that you have created the geometry for TURNDEMO, you are ready to begin working in **PartMaker** CAM mode. To do so, you must first switch to the CAM mode. You are in CAM mode when the CAD/CAM switch in the lower left-hand corner of the screen appears as the one to the left.

For those of you who chose to create geometry via **Method 2**, you should select **New Job** from the **File Menu**. Having done so, choose **Import DXF file** from the **File Menu** and choose **Turndemo.dxf** as in **Method 1** to insure you are working with same drawing which will be shown in the pages to come.

Before proceeding, your **Face Window** should appear as the one below, with your geometry appearing as a gray outline on the screen:



Define Group 1: Profile for Face Turning

The first operation in **TURNDEMO** is to face the stock to the desired finished length of 3.0 inches.

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

- 2 Click the blue square in the **Color Bar**



- 3 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.

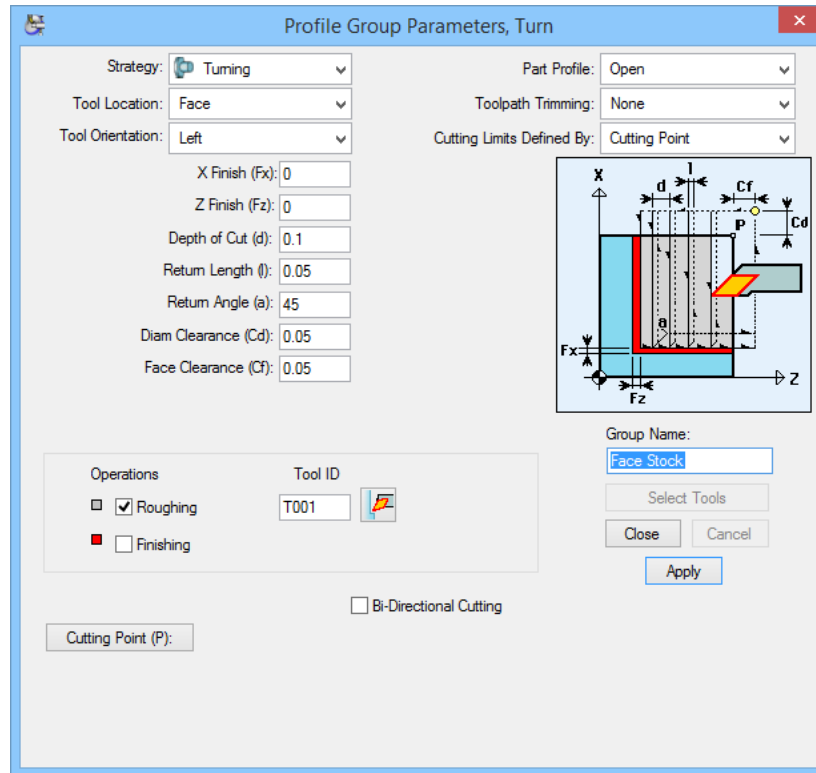
The screenshot shows the 'Profile Group Parameters, Turn' dialog box. It has a blue title bar and a close button. The dialog is divided into several sections. On the left, there are dropdown menus for 'Strategy' (set to 'Turning'), 'Tool Location' (set to 'Out'), and 'Tool Orientation' (set to 'Right'). Below these are input fields for 'X Finish (Fx): 0', 'Z Finish (Fz): 0', 'Depth of Cut (d):', 'Return Length (l): 0.05', 'Return Angle (a): 45', 'Diam Clearance (Cd): 0.05', and 'Face Clearance (Cf): 0.05'. On the right, there are dropdowns for 'Part Profile' (set to 'Open'), 'Toolpath Trimming' (set to 'None'), and 'Cutting Limits Defined By' (set to 'Cutting Point'). In the center-right, there is a 3D diagram of a lathe tool cutting a workpiece, with labels for X, Z, Fx, Fz, Cd, Cf, d, l, and a cutting point P. At the bottom left, there is a section for 'Operations' with checkboxes for 'Roughing' (checked) and 'Finishing' (unchecked), and a 'Tool ID' field. At the bottom right, there is a 'Group Name' field, a 'Select Tools' button, and 'Close', 'Cancel', and 'Apply' buttons. A 'Cutting Point (P):' button is at the bottom left. A 'Bi-Directional Cutting' checkbox is at the bottom center.

- 4 From the **Strategy** menu, select **Turning**
- 5 From the **Tool Location** menu, select **Face**
- 6 Click on the **Select Tools** button and allow **PartMaker** to select the appropriate tool. In this case, T001.



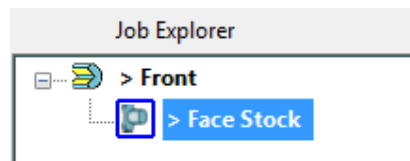
Note: In the color graphic, there are areas in gray representing roughing and areas in red representing finishing. For this operation, we will only be roughing, so do NOT check the **Finishing Tool** box.

In the field called **Group Name**, enter a short description such as **Face Stock**. When you are finished, click on **Close**. This description is for your reference only, and has no specific meaning to **PartMaker**. When you are finished, the dialog should appear as shown here:



- 7 Click <**Close**> after viewing the group parameters to return to the **Face Window**.

A square displaying the cycle name **Face Stock** appears in **Job Explorer** as shown below. The square is a *group symbol*. Each part you create in **PartMaker** will display one or more group icons in the **Job Explorer** to indicate the various features.



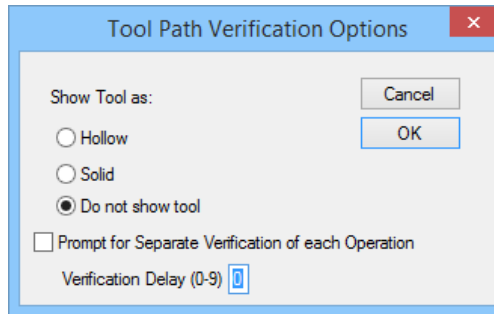
- 8 Double click on the group symbol to return to the dialog.
- 9 Click the <**Close**> button.

Tool Path Verification for Face Stock Group

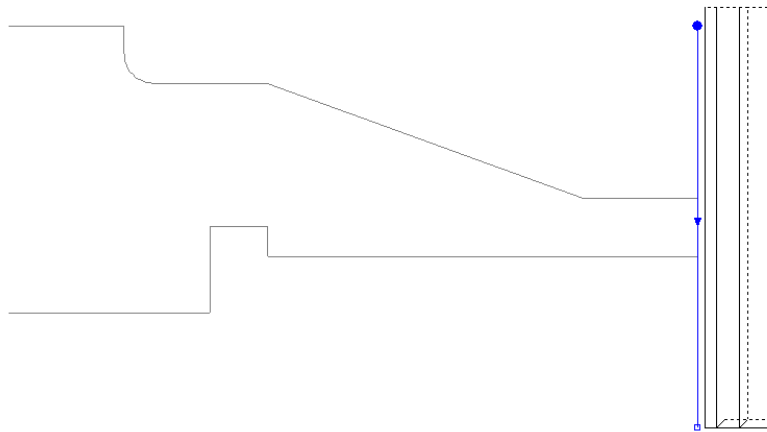
Because you have chosen a tool location of **Face** in the profile group parameters dialog, PartMaker has automatically assigned your tool path to the part geometry, creating the tool path for you.



To check your work, go to the **Part Features** menu and select **Verify Work Group Tool Path** and watch as **PartMaker** displays the operations to be performed. When you do so, you will be prompted with the dialog below:



Click **<OK>** and your screen will appear as shown below:



Choose **Hide Every Tool Path** from the **Part Features** menu to clear the programmed tool paths from the screen.



Verification options include showing the tool as either **Hollow**, **Solid** or **Do Not Show Tool**. You can also insert **Delay Times** into each program step. Delay times are relative to each computer and will require some user experimentation to determine which is best for each particular computer system.

Dotted lines are used to display rapid moves while solid lines are used to display feed moves.

The next step is to drill the 1-inch hole in the center of the part.

Define Group 2: Hole for Drilling Cycle

This section of the TURNDEMO tutorial will define the cycle and size of the hole to be drilled in the stock. From the drawing, you can see that the hole is 1.0" (25.00) in diameter and extends completely through the part.

To properly instruct **PartMaker** how to perform this operation, use the following steps:



- 1 Choose the color **red** from the **Color Bar**
- 2 From the Part Features menu, select New Hole Group.
- 3 Check Through Hole.
- 4 Enter the desired diameter of 1.0 (25.00)
- 5 In Major Cycle Operation select **<Drill>**.
- 6 Other parameters should be set as show here:

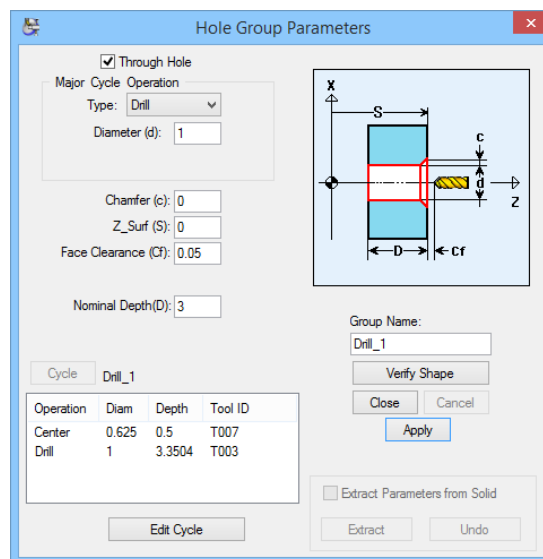


Chamfer	0.0	
Z_Surf	0.0	
Face Clearance (Cf)	0.05	(2.0)
Nominal Depth (D)	3.0	(76.0)

PartMaker will calculate the necessary programmed depth from the desired nominal depth taking into account the diameter and point angle of the drill.

- 7 Press the **Cycle** button and **PartMaker** will select the appropriate cycle to create the desired hole. This selected cycle should be **Drill_1 (Drill_25.00)**
- 8 Notice in the box in the lower left-hand corner of the dialog that **PartMaker** has displayed each tool to be used along with the appropriate depths required.
- 9 To verify the shape of the hole, click the **Verify Shape** button. When you are finished viewing the hole shape, click on the **Hide Shape** button.

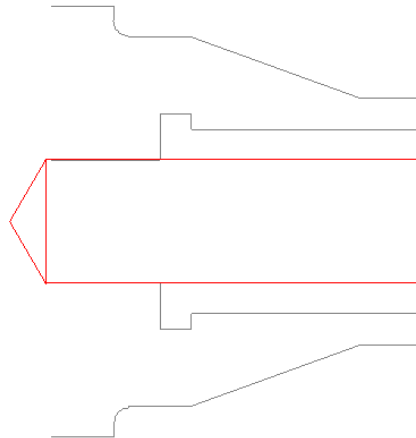
Your dialog should appear as shown below:





- 10 Click on the Edit Cycle button in Hole Group Parameters dialog to access the selected cycle. Only the selected cycle will be available for modification, any changes that are made will automatically update the Hole Group Parameters
- 11 When you are satisfied that you have correctly defined the Group Parameters, **Close** the dialog.

PartMaker should now display the drilled hole in the **Face Window** as shown here:



At this time, it is a good idea to save your work. To do so, select the **<Save>** command from the **File** menu.



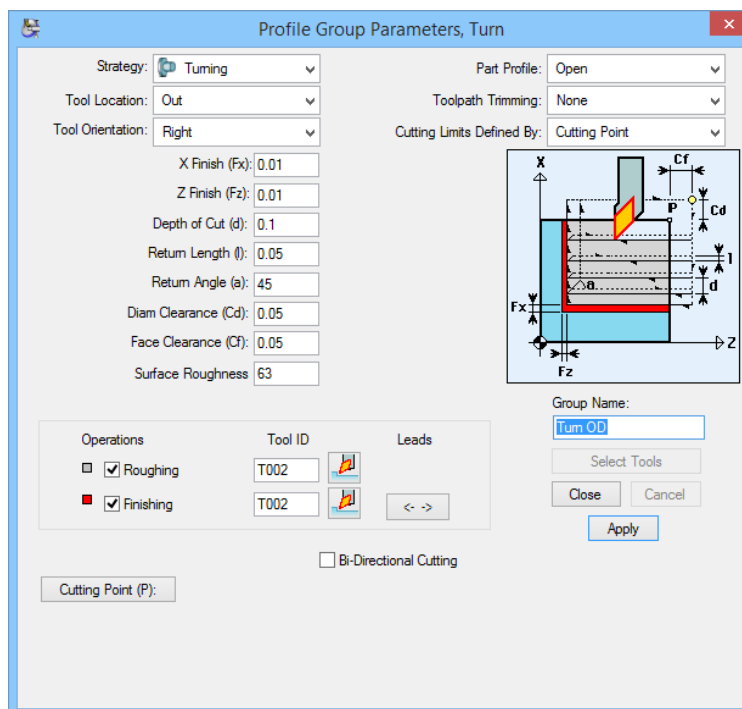
Define Group 3: OD Turning

In this section, you will learn how to cut the OD profile of the sample part.

- 1 Click on the green square in the **Color Bar**.



- 2 Select New Profile Group from the Part Features menu to display the Profile Group Parameters Dialog.



You will define the profile to be turned using the turning cycle

- 1 You will be finishing the part with the same tool, so click on the **Finishing Tool ID** check box.
- 2 Click on the **Select Tools** button and allow **PartMaker** to select the appropriate tools from the library. For this example, **PartMaker** will select T002 for both operations.

If you want to see detailed information about the selected tool, you can do so by clicking on the small, colored graphic of the tool. **PartMaker** will display the tool details.

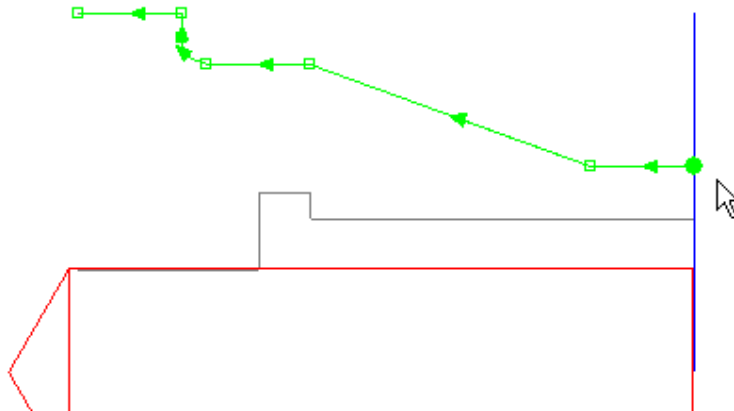


Note: After you clicked the **Select Tools** button, **PartMaker** has automatically set the **Depth of Cut** to **<0.1> inches (.250)**. This value comes from the tool database.

- 3 In the **Surface Roughness** field, enter **63 (1.6)**. This will set the appropriate feed rate for the finishing operation.
- 4 Set the **Finish X (Fx)** and **Finish Z (Fz)** values to **0.01 (.1)** respectively.
- 5 In the **Group Name**, enter **Turn OD** and click **<Close>** button to return to the Face Window.



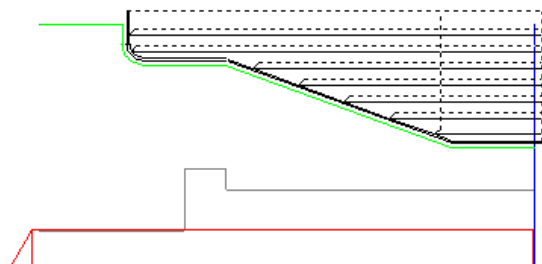
- 6 Click on the **Chain Profile** icon on the left side of the screen.
- 7 Using the cursor click on the OD profile that you wish to turn. Your selection point should be on the profile, all the way to the right, close to Z=0 as shown here:



PartMaker will highlight the profile in Green and create arrows showing the direction of cut.



- 8 Verify your work by selecting **Verify Work Group Tool Path** from the **Part Features** menu. **PartMaker** will show you each and every move required to create the profile.



- 9 When you are finished, select **Hide Work Group Tool Path**. The next operation is to bore the ID in preparation for threading.

Define Group 4: ID Turning

In this section we will define the cycle and select the profile to be bored on the ID of the part.

- 1 First, you will want to again change the working color on the screen. Click on the **cyan** (light blue) color square in the **Color Bar**.



- 2 Select New Profile Group from the Part Features menu.
- 3 Set the Strategy to Turning.
- 4 Set the Tool Location to In.
- 5 Click on the **Select Tool** button.
- 6 **PartMaker** should pick T004 from the tool database.
- 7 Type Turn_ID in the Group Name field.
- 8 Once you are satisfied that you have set up the Group parameters as shown below, press <Close>.



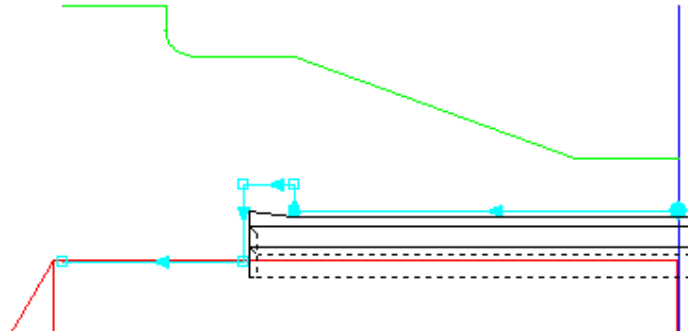
Note: Since we are only preparing the bore for threading, it is not necessary to finish the bore, so we will **not** use a finishing tool.



- 9 Choose the **Chain Profile** icon.
- 10 Select the ID geometry by clicking at the face of the part:
As before, **PartMaker** will highlight the profile as you see above.



- 11 To verify your work, select **Verify Work Group Tool Path** under the **Part Features** menu. If you have selected **Do Not Show Tools** under **Verification Options**, your screen should appear as below:



Notice how **PartMaker** has not only turned the bore, but also removed as much material as possible from the groove area without violating the finished part.



- 12 When you are finished, select **Hide Work Group Tool Path**.

The next operation will be to cut the thread clearance groove.

Now is a good time to save your work. From the **File** menu, select **<Save>** or simply press **<Ctrl S>**.

Define Group 5: ID Grooving

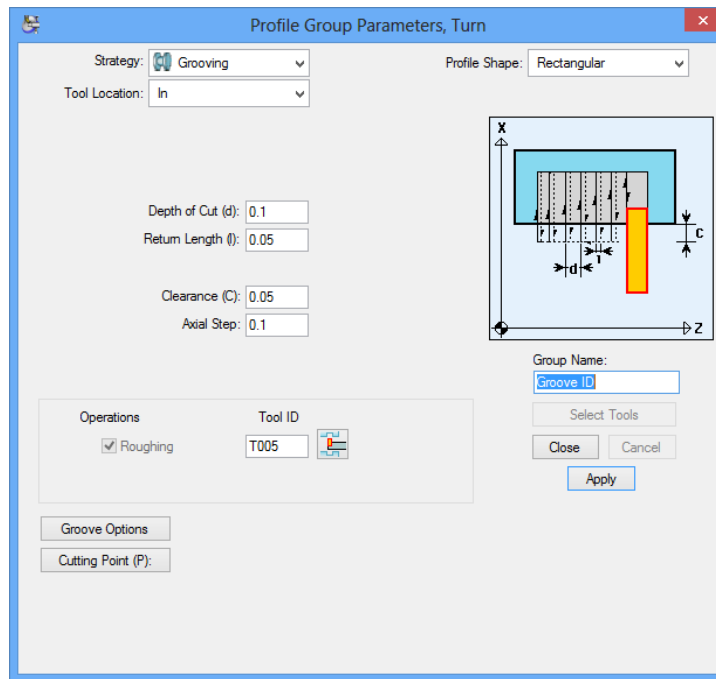
The next step in programming TURNDemo part is to clean out the ID groove at the bottom of the threads.

- 1 As with previous operations, begin this one by selecting a new color. Select **Green** for this operation.

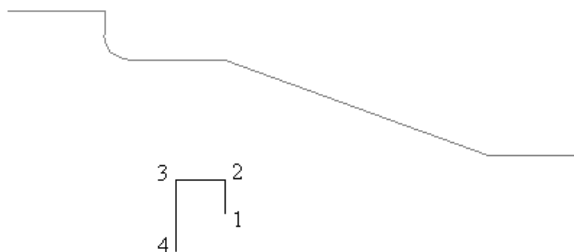


- 2 Select **New Profile Group** from the **Part Features** menu.
- 3 From the **Strategy** menu, select **Grooving**
- 4 From the **Tool Location** menu, select **In**
- 5 Notice that you now have a new pair of definitions: **Rectangle** or **General**. A **rectangular** groove is what we are cutting. A **general** groove is any profile shape that you want to machine using grooving operations. For this example, leave the selection as **Rectangular**.
- 6 Click on **Select Tools** and **PartMaker** will select the ID grooving tool defined in the tool database: T005
- 7 Type Groove ID in the Group Name box.

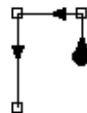
- 8 Take a minute and familiarize yourself with the parameters required for this cycle. For now, leave them at the defaults. Your dialog should appear as shown:



- 9 When you are satisfied that you have defined the Group Parameters correctly, **Close** the dialog.
- 10 Choose the **Define Profile** icon to select the desired elements defining the groove. Use of this icon allows you to individually select the lines and arcs to be included in the groove shape.
- 11 Select the **End of an Element** snap mode on the top of the screen.
- 12 In the **Face Window**, begin selecting the lines that define the groove, by Click on each endpoint (1, 2, 3, and 4) in succession until you have defined the groove by selecting 4 endpoints as shown here:

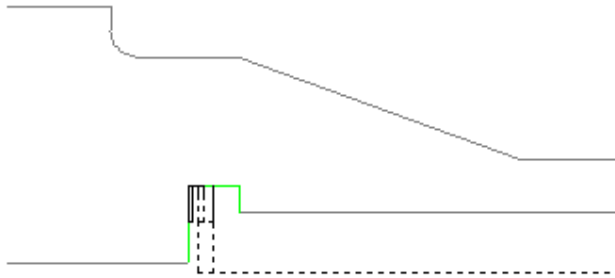


Your drawing should appear as below:





- 13 Select **Verify Work Group Tool Path** command from the **Part Features** menu. You should see what is shown below:



- 14 Choose **Hide Every Tool Path** to clear the tool path from the **Face Window**.

Define Group 6: ID Threading

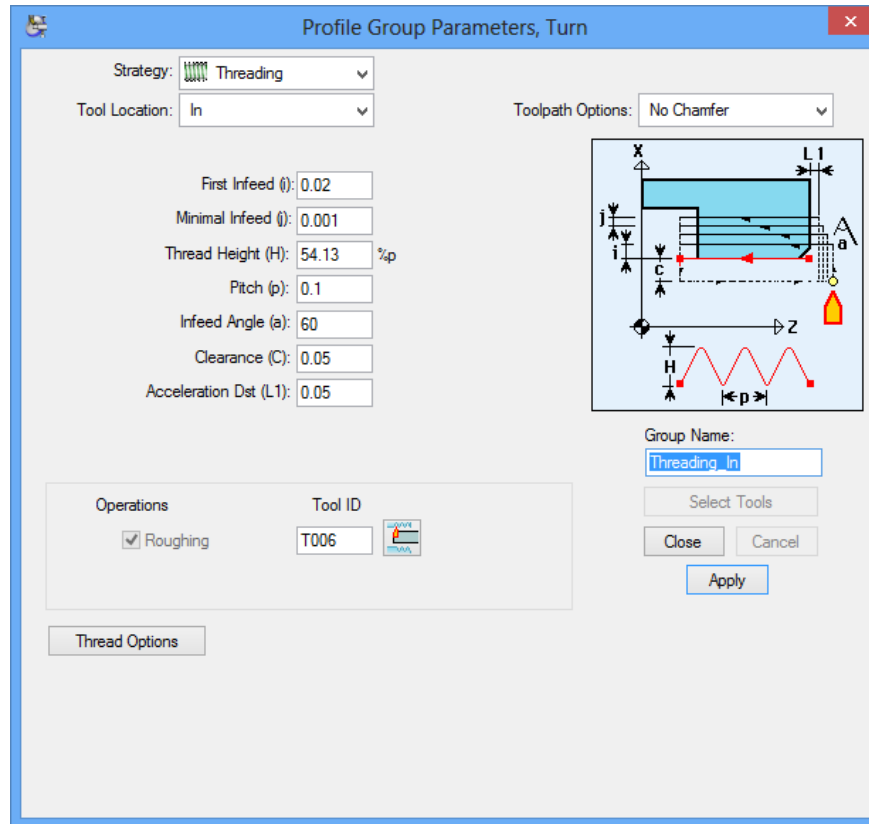
The next operation to be performed is to cut the ID thread.

- 1 Select the color **magenta** from the **color bar**.



- 2 Select **New Profile Group** from the **Part Features** menu.
- 3 From the **Strategy** menu, select **Threading**.
- 4 From the **Tool Location** menu, select **In**.
- 5 Look at the graphic. Notice that there are two options: **No Chamfer** and **Chamfer**. No Chamfer would be used in our case where there is a relief groove at the end of the thread. **Chamfer** would be used if there is no relief groove at the end of the thread. Take a minute and look closely at both graphics and the user parameters for each. When you are finished, leave this setting at **No Chamfer**.
- 6 Click the **Select Tools** button. **PartMaker** will pick the appropriate ID threading tool from the tool database. In this case, the tool is T006.
- 7 Type **Thread ID** next to Group Name.
- 8 Note the default **Pitch** of **0.125" (3)**. The desired pitch for this operation is 10 threads / inch (2/mm). Change the **pitch** to **0.1" (2)**.

- 9 The final Profile Group Parameters should appear as shown below:



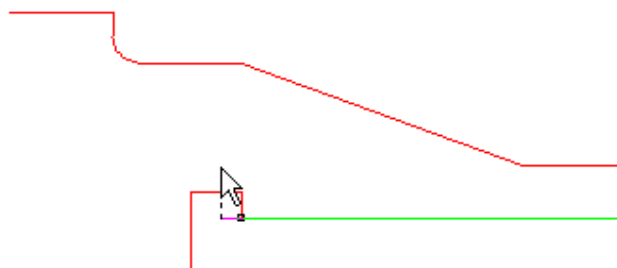
- 10 Click on **<Close>** to close the **Profile Group Parameters** dialog when you are sure that all settings are correct.

Defining the geometry for the threading cycle is a bit more involved than previous operations, so follow along carefully.

The geometry for the threading cycle is not currently defined on the screen, so we will have to define it first. While there are many ways to define the geometry, we will use a new method to help familiarize you with as many features of **PartMaker** as possible.



- 11 In the lower left-hand corner of the screen click on the CAM mode icon. It will change to the CAD icon as shown to the left.
- 12 Click on the **Trim / Extend** icon.
- 13 Click on the horizontal line that represents the minor diameter of the thread. Click near the left-hand end of the line.



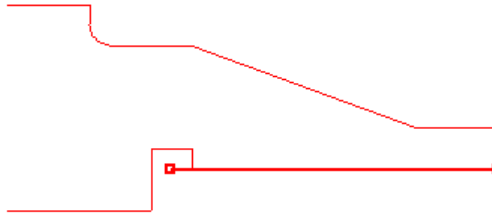
Notice that as you move the cursor left and right you change the length of the selected line, which is now shown in pink.



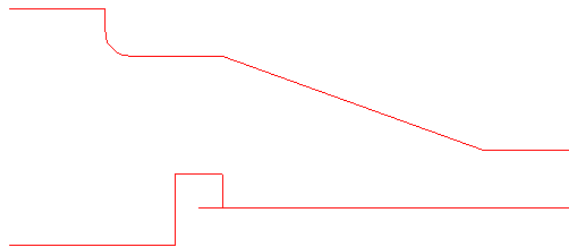
- 14 To extend the line horizontally to the center of the groove, **PartMaker** has a snap tool that can find the exact point required. Click on the **Midpoint** snap mode.

The goal of this exercise is to extend the selected line so that it ends in the center of the groove along the Z-axis. While it is quite possible to 'eyeball' a point that is close enough for some cases, it is better to learn to use exact coordinates.

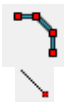
- 15 Use the cursor to select the line representing the bottom of the groove as shown here:



The resulting face window will appear as shown below:

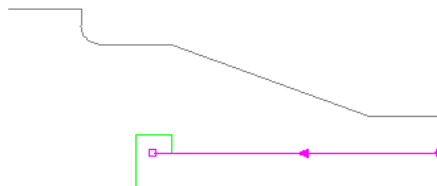


- 16 You are now finished with CAD mode. Click on the CAD icon in the lower left-hand corner of the screen to return to the CAM mode.

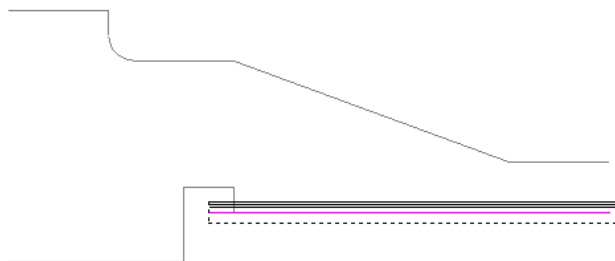


- 17 Choose the **Define Profile** icon and the **End of an Element** snap mode. First click on the right end, then the left end of the line, which you just modified.

PartMaker will highlight the line as shown:



- 18 Choose Verify Work Group Tool Path from the Part Features menu.



If you are satisfied with what you see, you are finished with the definition of all part features for this job. **Hide Every Tool Path** to complete this phase.

Choose **Save** from the **File** menu or press **<Ctrl + S>** to save your work.

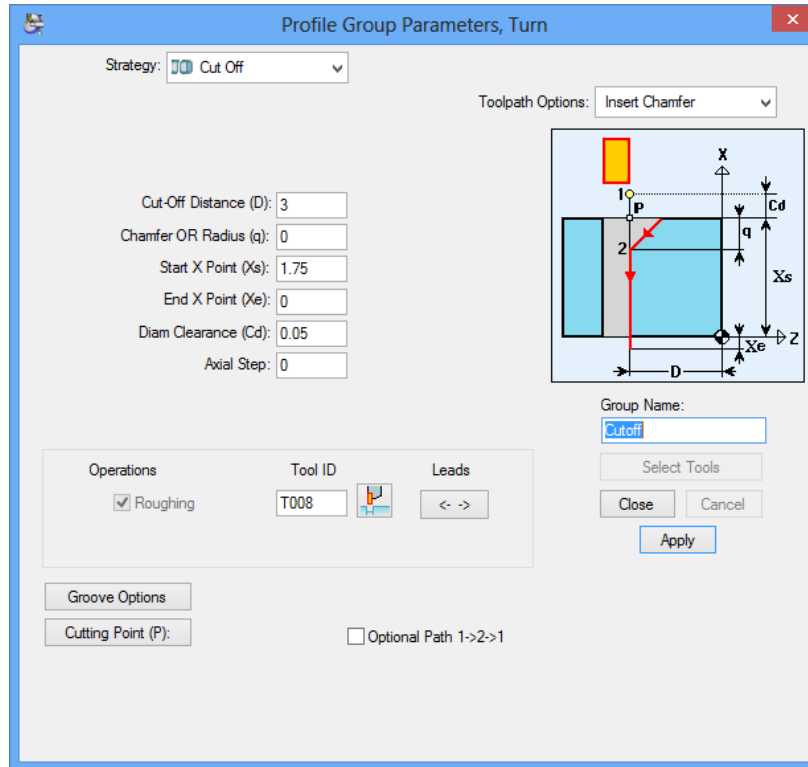
Define Group 7: Cutoff

The final operation to be performed is to cut the part off.

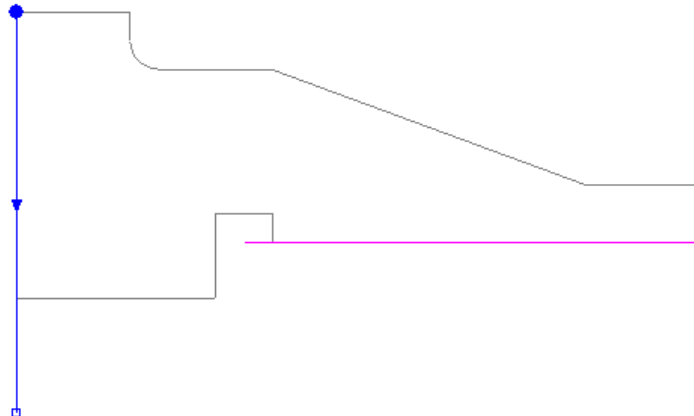


- 1 Select the blue square from the color bar.
- 2 Select New Profile Group from the Part Features menu.
- 3 From the **Strategy** menu, select **Cut-Off**
- 4 Click the Select Tools button. PartMaker will pick the appropriate Cut-off tool from the tool database. In this case, the tool is T008.
- 5 Type **Cut-Off** next to Group Name.

Your **Profile Group Parameters** dialog should now appear as shown below:



- 6 Click on **<Close>** to close the Profile Group Parameters dialog when you are sure that all settings are correct.
- 7 As with the facing operation, PartMaker will automatically apply the cut-off tool path to your geometry.



Now that all the programming is complete, it would be a good idea to see exactly what you have programmed.

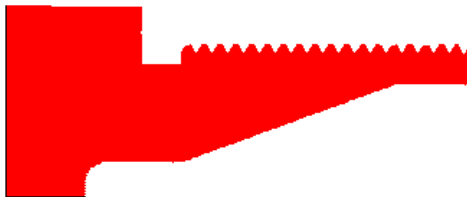
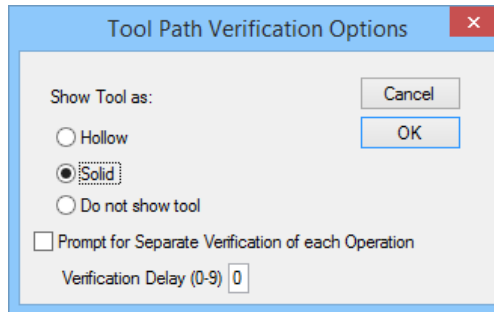


1 From the **View** menu, select **Show All**.



2 From the **Part Features** menu, select **Verify Every Tool Path**. Partaker will show you exactly what you have programmed in a graphical manner that is easy to recognize and check for errors.

3 Set the Show Tool As option to Solid.

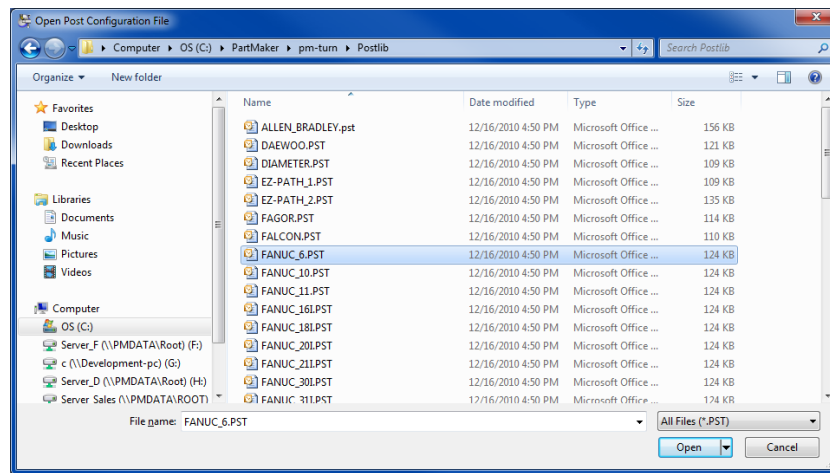


4 When you are finished, select **Hide Every Tool Path** from the **Part Features** menu.

Choose a Post Processor

The next step is to choose a **Post Configuration File** or Post Processor. To do so:

- 1 In the Job Optimizer menu, select Post Config File=?
- 2 From the **postlib** directory, select **FANUC_6.pst** as shown here:



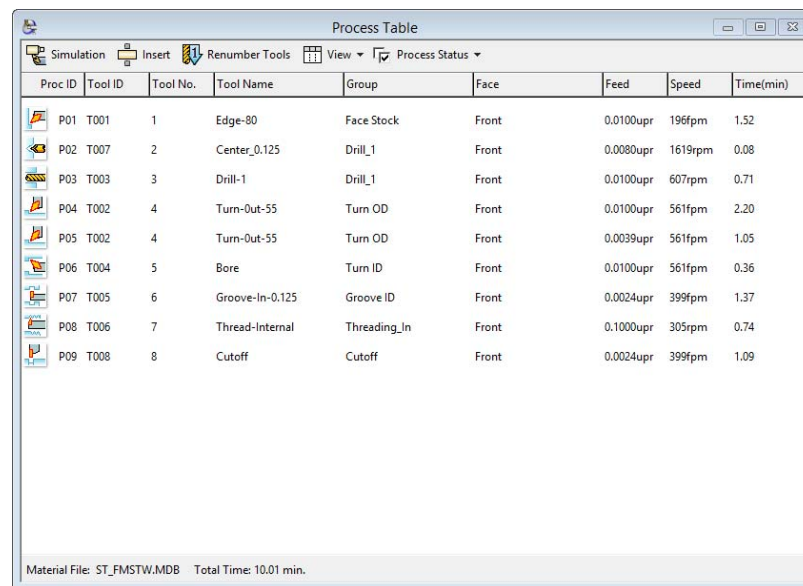
Generate a Process Table

The next step is to generate a **Process Table** of all the operations you have just defined.



- 1 Select **Generate Process Table** from the **Job Optimizer** menu. Use the defaults shown and click on **OK**.

PartMaker will generate a table showing each process to be programmed.



Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)
P01	T001	1	Edge-80	Face Stock	Front	0.0100upr	196rpm	1.52
P02	T007	2	Center_0.125	Drill_1	Front	0.0080upr	1619rpm	0.08
P03	T003	3	Drill-1	Drill_1	Front	0.0100upr	607rpm	0.71
P04	T002	4	Turn-Out-55	Turn OD	Front	0.0100upr	561rpm	2.20
P05	T002	4	Turn-Out-55	Turn OD	Front	0.0039upr	561rpm	1.05
P06	T004	5	Bore	Turn ID	Front	0.0100upr	561rpm	0.36
P07	T005	6	Groove-In-0.125	Groove ID	Front	0.0024upr	399rpm	1.37
P08	T006	7	Thread-Internal	Threading_In	Front	0.1000upr	305rpm	0.74
P09	T008	8	Cutoff	Cutoff	Front	0.0024upr	399rpm	1.09

Material File: ST_FMSTW.MDB Total Time: 10.01 min.

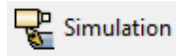
Take a minute and look carefully at all the information shown in the **Process Table**. Shown on each line, in order of operation, is the tool information, turret position, process description, group number, face name, feed rates, spindle speed, and total turning time.

If you wish to observe more information related to the tool, click on the colored tool icon. All details about the tool will be shown to you in the **Tool Data** dialog.

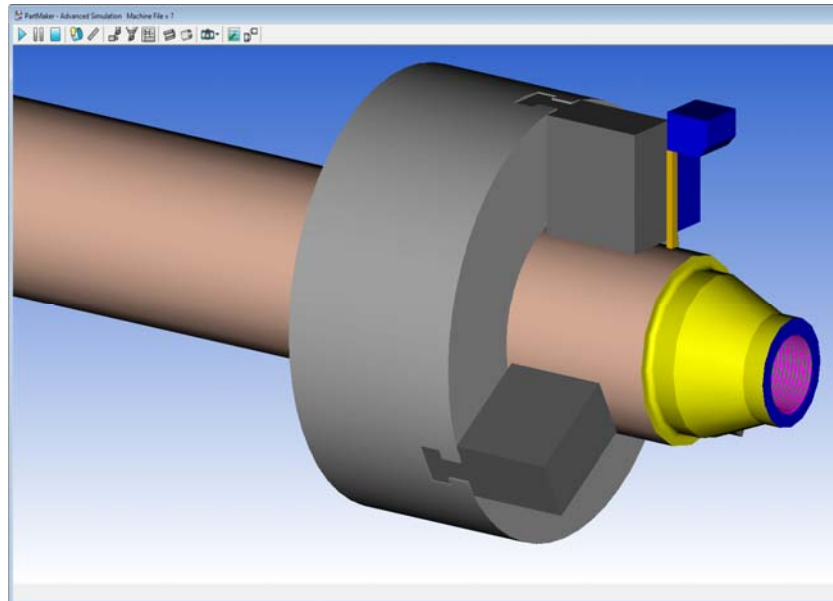
If you want more information about each process, double click on the text, anywhere in the line. The details of the process are displayed in an open dialog box.

Simulating the Part

Once you are satisfied with the order of operations on the Process Table, you can simulate your part. This can be done by clicking the **Simulation** button in the upper left hand corner of the Process Table or by selecting the Simulation button on the main tool bar.



- 1 Once in the Simulation window, choose **Start Simulation** from the tool bar menu or just press the **<Space Bar>**. Your simulated part should appear as shown below:



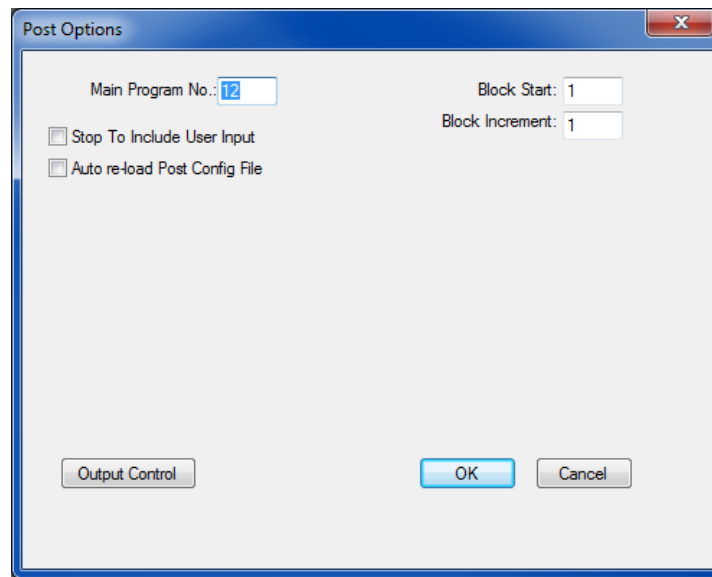
You can learn more about using PartMaker's simulation module in Chapter 4.

Generate NC Code

It is now time to **Generate NC Code** for the TURNDEMO sample.

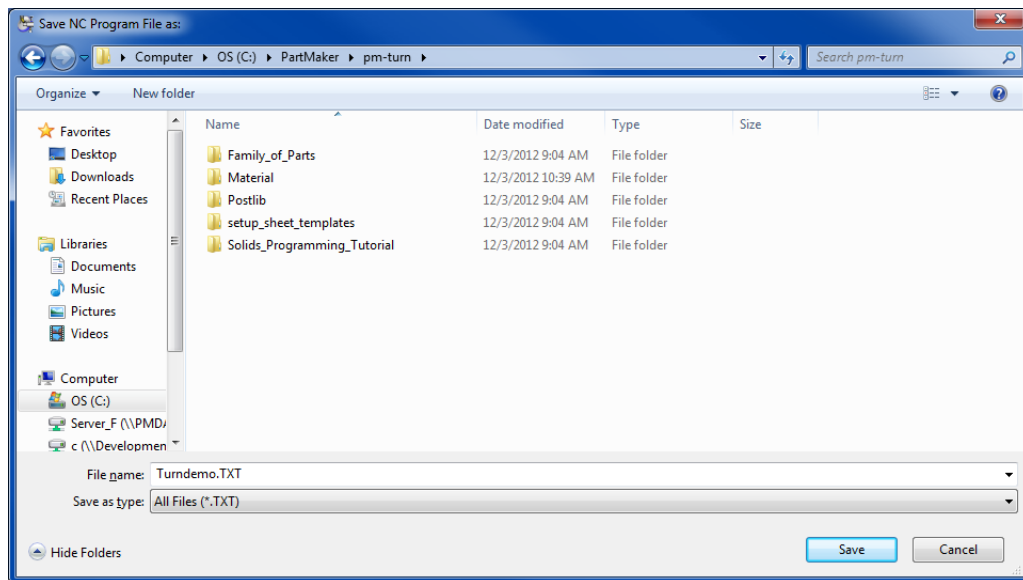


- 1 From the Job Optimizer menu, select Generate NC Program
- 2 Next dialog to appear is **Post Options** as seen below:



Look at the options - program number and line numbering variables, and click **<OK>** when you are finished.

- 3 You will next be prompted for a name for the NC code file. Enter the name **TURNDEMO**. The file extension **.TXT** will automatically be added to the file:



PartMaker will generate your NC file and display it on the screen. An example is shown below:

```
PartMaker - Turndemo.TXT
;12
N2 (TURNDemo)
N3 G97 S104 M3
N4 G0 X20.0 Z10.0
N5 T0101 (EDGE-40 LEFT HAND FACING TOOL VIEWED FROM THE BACK)
N6 G50 S1000
N7 G96 S100
N8 M48
N9 G99
N10 G0 X3.6 Z0.3
N11 Z0.15
N12 G1 X-0.0624 F0.01
N13 Z0.3
N14 G0 X3.6
N15 Z0.05
N16 G1 X-0.0624
N17 Z0.15
N18 X0.0083 Z0.1854
N19 G0 X3.6
N20 Z0.
N21 G1 X-0.0624
N22 Z0.05
N23 X0.0083 Z0.0854
N24 G0 Z0.3
N25 X3.6
N26 G40 G0 X20.0 Z10.0 T0100

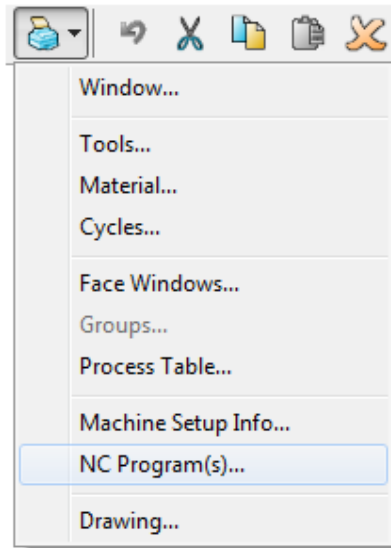
N28 G97 S1017 M3
N29 G0 X20.0 Z10.0
N30 T0202 (CENTER_0.125 )
N31 G96
N32 X0. Z0.05
N33 G1 Z-0.5 F0.01
N34 G0 Z0.05
N35 G40 G0 X20.0 Z10.0 T0200

N37 G97 S381 M3
N38 G0 X20.0 Z10.0
N39 T0303 (DRILL-1 )
N40 G99
N41 X0. Z0.05
N42 G1 Z-1.0 F0.01
N43 G0 Z0.05
N44 Z-0.95
N45 G1 Z-2.0
N46 G0 Z0.05
N47 Z-1.95
N48 G1 Z-3.0
N49 G0 Z0.05
N50 Z-3.95
N51 G1 Z-3.3504
N52 G0 Z0.05
N53 G40 G0 X20.0 Z10.0 T0300

N55 G97 S113 M3
N56 G0 X20.0 Z10.0
N57 T0404 (TURN-OUT-55 RIGHT HANDED TOOLS VIEWED FROM THE BACK (ACCORDING TO MAKER ))
N58 G50 S250
N59 G99
```

Printing the Results

The **Print** command in the **File Menu** allows you to print the active (front most) window, or print a specific window (a specific face window, for example).



Choose **NC Program** from the **Print** menu to print the part program so you can examine it more closely.

As you can surmise, there are many different ways to draw the same profiles, some easier, some more difficult. We strongly recommend that you use this example as a place to experiment with different methods of creating geometry – so that when you get into a production situation where you HAVE to use another method, you will know all of the available options.

Chapter 4: PartMaker® Simulation

Introduction

PartMaker Version 8 and higher comes with an integrated 3D solid modeling process simulation and verification module. This module is available in two levels: Basic Simulation and Advanced Simulation. Basic Simulation comes standard with PartMaker Turn while Advanced Simulation is available as an optional module with PartMaker Turn.

PartMaker's Basic Simulation Module allows you to:

- Simulate all machining processes as they appear on the Process Table
- Observe the results of any process synchronization
- Examine a solid model of the part through dynamic rotation and cross-sectioning

PartMaker's Advanced Simulation Module allows you to do all of the above and:

- View a 3D parametric machine model for your specific machine
- Automatically check that all tool numbers assigned in the tool database are assigned correctly for your specific machine model
- Customize your machine through the manipulation of an MCH file
- Detect crashes, collisions and errors between various machine components and the part
- Detect any machine over travels on screen

Users who have licensed the Advanced Simulation or Full Machine Simulation modules may still find it beneficial to use the Basic Simulation module to focus on only the part as it is being machined.

You may choose between Basic and Advanced Simulation by choosing either **Basic Simulation** or **Advanced Simulation** from the **Simulation** menu or by right mouse clicking on the **Process Table** and choosing either option.

Whichever simulation option has been selected will determine which type of simulation will be launched.

The Simulation Toolbar

Many of the common functions used in PartMaker's 3D simulation can be accessed via the Toolbar that appears at the top of the Simulation Window as shown here:



Each icon on the **Simulation Toolbar** is explained below. The features each of these icons enable will be explained in greater depth throughout this chapter.



Starts the simulation



Pauses the simulation



Resets the simulation



Shows the finished 3D model of the part



Allows the user to measure the completed part.



Sets simulation to operate in Basic Simulation mode



Sets simulation to operate in Advanced Simulation mode



Sets simulation to operate in Full Machine Simulation mode



Allows you to take cross sectional views of the completed part



Allows you to show a full view of a part after sectioning



Allows you to manipulate the view of machined part



Allows you to see the Simulation Options dialog



Allows you to see the Spindle Setting dialog



Shows the Machine Housing

Simulation Features

PartMaker Simulation has a number of process and part analysis features available both during and after process simulation is complete. These features are described here:

Reset

At any point you can reset the simulation to the initial screen by choosing **Reset** from the **Simulation** menu. Alternatively, if you have suspended **Simulation** by pressing <Esc>, you can press <Esc> again to reset the simulation.

The **Reset** command is also accessible by clicking the right mouse button when viewing the finished part (see below).

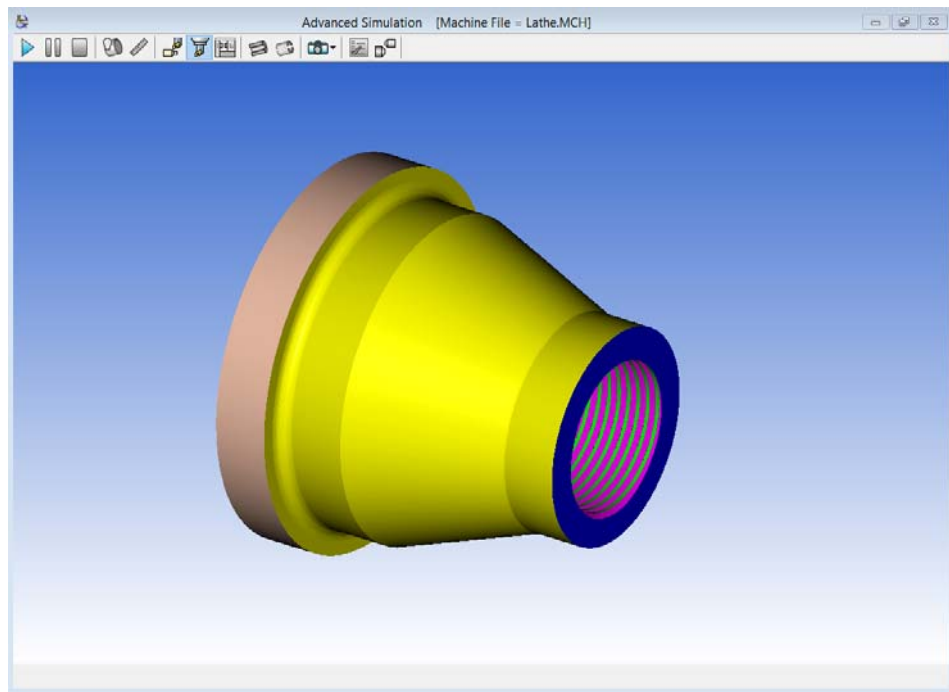
Show Travel Limits

Displays the travel limits for machine being simulated for each machine component depending on how the machine is equipped. Travel limits for each machine components including main spindle, sub spindle and tool posts are set in the **Machine Data File** dialog accessed from the **Toolminder** menu.

Show Finished Part

By choosing the **Show Finished Part** command from the **Simulation** menu, you can see a 3D model of your programmed part. You need not run the simulation prior to choosing **Show Finished Part**.

When choosing **Show Finished Part**, the verified part alone will appear in the **Simulation** window as shown here:

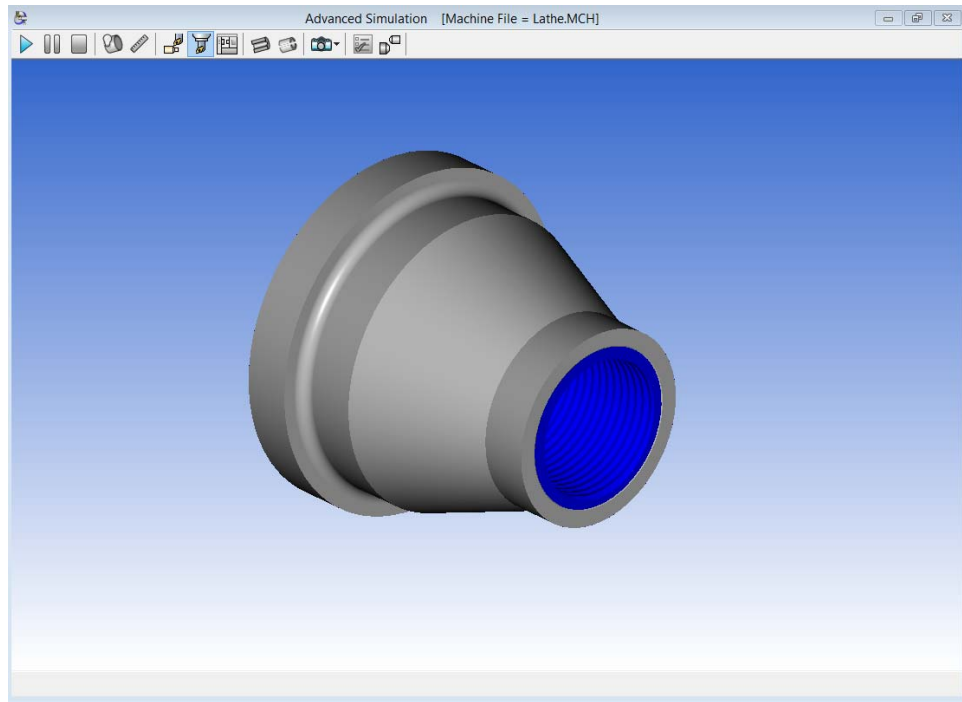


Save Finished Part As STL file

This will save the finished part to an STL file format. This can be then used again in PartMaker for Custom Stock or Surface Machining.

Compare Finished Part to Solid Model

This will use the solid model specified in the **Comparison Options Dialog** to show a comparison of the part to the solid model.



Comparison Options

This will open the **Comparison Options** dialog. This allows you to specify the comparison models as well as the over and undercut display colors.

View Setting Features

PartMaker Basic Simulation allows you to automatically set a variety of views when simulation has been halted by using the <Esc> key or when using the **Show Finished Part** command. Various view orientations are available from the **View** sub-menu under the **Solids** menu.

Default

Choosing **Default** sets the view in the simulation window to the default view when **Simulation** was launched.

Best Fit

Choosing **Best Fit** sets the view in the simulation window to the best fit relative to the **Simulation** window as determined automatically by PartMaker.

XY View

Choosing the **XY View** command sets the view in the simulation window normal to the cross sectional plane formed by XY axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

YZ View

Choosing the **YZ View** command sets the view in the simulation window normal to the cross sectional plane formed by YZ axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

ZX View

Choosing the **ZX View** command sets the view in the simulation window normal to the cross sectional plane formed by ZX axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

Isometric View

Choosing the **Isometric View** command sets the view in the simulation window aligned to the X, Y and Z axes. This view presents the part at a 45 degree angle.

Show Coordinate Axes

Checking this selection on allows the user to the axes symbol in lower left hand corner of PartMaker Simulation Window.

Show Finished part Origin

This will hide or show the 3-D axis on the finished part.

Show Coordinate axis

This will hide or show the 3-D coordinate system axes in the lower left hand side of the simulation window.

Background Color

Allows the user to choose the background color of the simulation. The background may be set to gradient by choosing different colors for the Top and Bottom Gradient Color respectively.

Datum Color

Sets the color of each axis of the datum, X, Y and Z respectively.

View Manipulation Features: Rotate, Move and Zoom

PartMaker simulation allows you to manipulate the view of either the entire process simulation or the finished part (when using **Show Finished Part**). When using PartMaker, one of these commands will be activated, as denoted by a check next to it under the **Change View** sub menu under the **Simulation** menu.

Whenever the **Rotate**, **Move** or **Zoom** command is checked, it can be activated by holding down the left mouse key and moving your mouse.

It is advisable to always leave the **Rotate** command checked as both **Move** and **Zoom** can always be accessed using keyboard shortcut keys or the mouse. For example, if the **Rotate** command is checked under the **Change View** sub menu under the **Simulation** menu, **Rotate** can be activated by clicking the left mouse button and moving the mouse, **Move** can be activated by holding down the mouse's scroll wheel and moving the mouse and **Zoom** can be activated by simply scrolling the mouse's scroll wheel.

PartMaker provides fully dynamic simulation such that the **Rotate**, **Move** and **Zoom** commands can be accessed at any time during simulation.

Rotate

This command allows you to dynamically rotate the entire process simulation within the simulation window. When using **Show Finished Part**, **Rotate** allows you to dynamically rotate the solid model of the completed part.

When **Rotate** is active, you can rotate the process simulation or finished part by clicking the left hand mouse button and dragging your mouse in the orientation you would like to see the view.

Rotate may only be used when it is activated from the **Solids** menu. You can activate **Rotate** by choosing it from the **Solids** menu. Its activation is denoted by a check next to it under the Simulation window.

Move

This command allows you to dynamically move the entire process simulation around the simulation window. When using **Show Finished Part**, **Rotate** allows you to dynamically rotate the solid model of the completed part.

When **Move** is active, you can move the process simulation or finished part by clicking the left hand mouse button and dragging your mouse in the direction you want to move the view.

You can move the view if **Move** is NOT active by holding down the **<Ctrl> + <Arrow>** keys on your keyboard, with arrows denoting the direction in which the view will move. These are the same keys used when performing panning in PartMaker.

Additionally, **Move** can always be activated using the mouse by holding down the mouse wheel (if your mouse is so equipped) and dragging your mouse in the direction you want to move the view.

You can activate **Move** by choosing it from the **Solids** menu. Its activation is denoted by a check next to it under the Simulation window.

Zoom

This command allows you to dynamically size the entire process simulation within the simulation window when it has been halted by using the **<Esc>** key. When using **Show Finished Part**, **Zoom** allows you to dynamically size the solid model of the completed part.

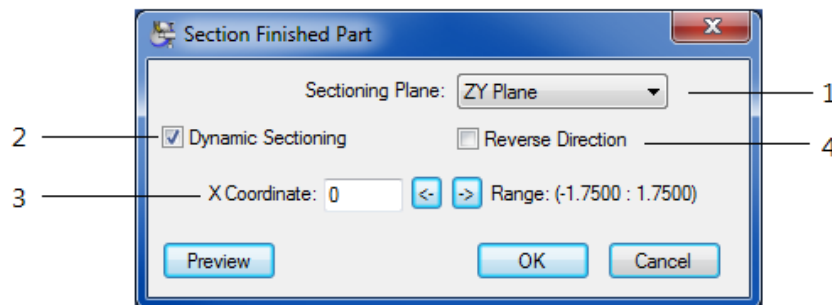
When **Zoom** is active, you can scale the process simulation or finished part by clicking the left hand mouse button and dragging it upward to increase the size of the part and dragging downward to decrease the size of the part.

You can scale the view if **Scale** is NOT active by pressing the <F4> key to decrease the size of the view or the <F5> key to increase the size of the view. These are the same keys used when performing **Continuous Zoom In** or **Zoom Out** in PartMaker.

Additionally, **Scale** can always be activated using the mouse by scrolling the mouse wheel (if your mouse is so equipped) upward to zoom in and downward to zoom out.

Sectioning Features

PartMaker simulation allows you to section the solid view to inspect the interior of the part. Various sectioning options are available by choosing **Sectional View** from the **Simulation** menu. Choosing **Sectional View** opens the **Section Finished Part** dialog as shown below:



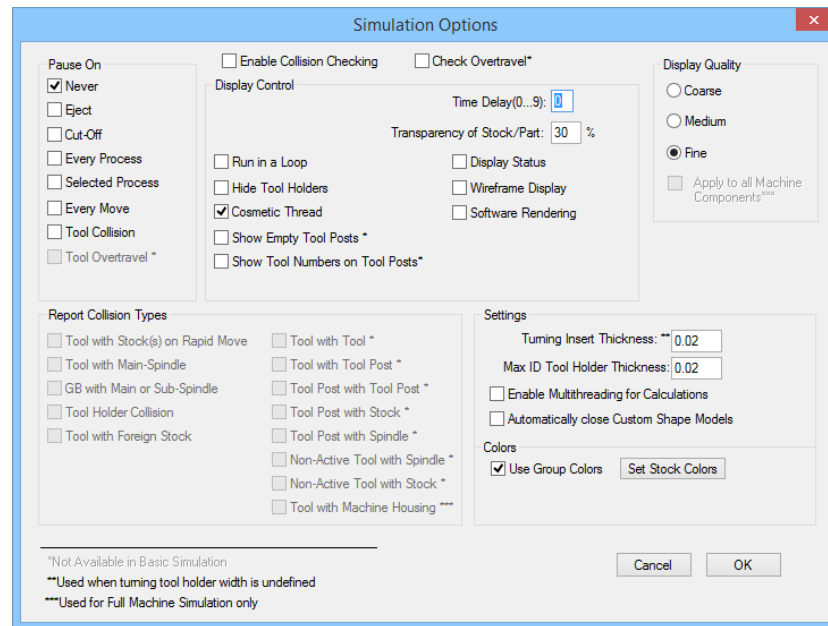
- 1 **Sectioning Plane** – The Sectioning Plane section of this dialog allows you to determine the plane across which sectioning will occur normal to. The user can section a finished part normal to the XY, ZY and ZX plane. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.
- 2 **Dynamic Sectioning** – Checking this box enables dynamic section, or the ability to see the part being sectioned as you adjust the location of the sectional plane. If Dynamic Sectioning is unchecked, you will be able to see the result of your sectioning by clicking the <Preview> button or by exiting this dialog by clicking <OK>
- 3 **Plane Coordinate** - This area of the dialog allows you to adjust the level of the sectioning plane.
- 4 **Reverse Direction** - This allows you to reverse the direction of sectional plane shown.

Simulation Options

You can manipulate the appearance of simulation by choosing **Options** from the **Simulation** menu or by clicking the right mouse button in the **Simulation Window** and choosing **Options**.

PartMaker Simulation allows you to change the appearance of the view in the simulation window when simulation has been halted by using the <Esc> key or when using the **Show Finished Part** command.

The **Options** dialog as it appears in Basic Simulation for PartMaker Turn is displayed below:



Pause On

Never: Specifies that simulation should never pause during simulation

Eject: Specifies that simulation should pause before part eject operation is performed

Cut-off: Specifies that simulation should pause after the cut-off operation is performed

Every Process: Specifies that simulation should pause at the beginning of every process in the current Process Table

Selected Process: Specifies that simulation should pause at the beginning of a selected process in the current Process Table

Every Move: Specifies that simulation should pause before every tool move

Tool Collision: Specifies that simulation should pause every time a collision occurs

Tool Overtravel: Specifies that simulation should pause every time tool post goes outside of its travel limits specified in the Machine File.

Settings

Turning Insert Thickness: Specifies the thickness of the turning insert

Max ID Tool Holder Thickness: Specifies the maximum thickness of a tool holder for inside diameter tool.

Enable Collision Checking/Report Collision Types

Checking the **Enable Collision Checking** box allows you to detect the following collision types in Simulation:

Tool with Stock(s) on Rapid Move: Specifies that PartMaker should check for collisions between the tool (or insert) and that stock in the main spindle as well as a part in sub-spindle

Tool with GB and/or Sub-Spindle: Specifies that PartMaker should check for collisions between the tool (or insert) and the Guide Bushing and/or Sub-Spindle

GB and Sub-Spindle: Specifies that PartMaker should check for collisions between the Guide Bushing and Sub-Spindle.

Tool Holder Collision: Specifies that PartMaker should check for Tool Holder collisions with any part/stock as well as with the Main Spindle and Sub-Spindle

Tool with Foreign Stock: Specifies that PartMaker should check for collisions between the tool (or insert) working on the Main Spindle with part in the Sub-Spindle or between the tool (or insert) working on Sub-Spindle with the stock or part in the Main Spindle

Tool with Tool: Specifies that PartMaker should check for collisions between two tools on the machine. This is for Advanced and Full Machine simulations only.

Tool with Toolpost: Specifies that PartMaker should check for collisions between a tool and another tool post on the machine. This is for Advanced and Full Machine simulations only.

Toolpost with Toolpost: Specifies that PartMaker should check for collisions between tool posts on the machine. This is for Advanced and Full Machine Simulations only.

Toolpost with Stock: Specifies that PartMaker should check for collisions between a tool post and the stock being machined. This is for Advanced and Full Machine Simulations only.

Toolpost with Spindle: Specifies that PartMaker should check for collisions between a tool post and the main or sub spindle. This is for Advanced and Full Machine Simulations only.

Non-Active Tool with Spindle: Specifies that PartMaker should check for collisions between a tool not in the cut and the main or sub spindle. This is for Advanced and Full Machine Simulations only.

Non-Active Tool with Stock: Specifies that PartMaker should check for collisions between a tool not in the cut and the stock. This is for Advanced and Full Machine Simulations only.

Tool with Machine Housing: Specifies that PartMaker should check for collisions between a tool (or insert) and the machine housing. This is for Full Machine Simulation only.

Display Control

Time Delay (0...9): Specifies a time delay following every screen update for every tool motion. Negative number -9 to -1 may also be used to speed simulation more than 0.

Transparency of GB/Sub-Spindle: Specifies how transparent the Main Spindle and/or Sub-Spindle should appear. 0% means solid appearance, 100% means 'fully transparent' appearance, i.e. the Guide Bushing and/or Sub-Spindle will become invisible.

Transparency of Stock/Part: Specifies how transparent the part and/or stock should appear. 0% means solid part, 100% means "fully transparent" part, i.e. an invisible part.

Display Status: Specifies that the current simulation status consisting of the Process ID and current Tool Point coordinates will be displayed on the top of the Simulation window

Run in a Loop: Specifies that Simulation will run in a continuous loop

Hide Tool Holders: Specifies that Tool Holders for Turning Tools will not be displayed during Simulation

Display Status: Allow the Simulation to display coordinate and movement information at the top of the simulation window.

Cosmetic Thread: Specifies that during Simulation every Multi-Pass helical thread will be replaced by a single pass in order to speed up simulation.

Software Rendering: Specifies that Open GL software graphics libraries should be used instead of the hardware implementation of the Open GL drivers that are dependent on the actual Graphics Video card used in every computer. If you use a high performance AGP video card you may achieve a significant simulation speed improvement if you do not use Software Rendering

Wireframe Display: Specifies that everything will be displayed using a wire frame representation instead of solid modeling.

Show Tool Numbers on Tool Posts: This will show all tool numbers available on a tool post on the solid for that tool post. This is for Advanced Simulation only.

Show Machine Housing: This will show the Machine Housing Components specified in the Machine File. This is for Full Machine Simulation only.

Display Quality

Coarse: Specifies that solids will be displayed using relatively large facets to assure the fastest simulation

Medium: Specifies that solids will be displayed using medium size facets that will result in lesser simulation speed

Fine: Specifies that solids will be displayed using very small size facets that will result in further decrease in simulation speed

Apply to All Machine Components: Specifies that solid models will be converted to simulation models with the specified Display Quality (Coarse, Medium, Fine), regardless of the quality specified in its model data. These solid models can include custom stocks and tool posts.

Colors and Textures

Use Group Colors: Specifies that user-defined Group Colors will be used as cut colors. If this option is not checked a metallic gray color will be used as cut color for all tools

Set Stock Color: Allows a user to choose the color of initial stock using a Windows standard dialog.

Set Stock Texture: Opens the **Texture Dialog** for various texture options for the stock.

Using PartMaker Basic Simulation

PartMaker Basic Simulation can be viewed at any point in the programming process. You need not complete programming a part to see the results of your work.

Important! To launch PartMaker Simulation you must have a Process Table generated. To generate the Process Table, choose **Generate Process Table** from the **Job Optimizer** menu (ALT + P) or use the **Generate Process Table** icon from the **Main Toolbar**. It is not necessary to complete synchronization of simultaneous operations before launching PartMaker Simulation.

Launching Basic Simulation

To launch PartMaker Basic Simulation:



- 1 Click the Start Simulation icon from the Main Toolbar

or

Press the **Simulation** button in lower left hand corner of the Process Table

or

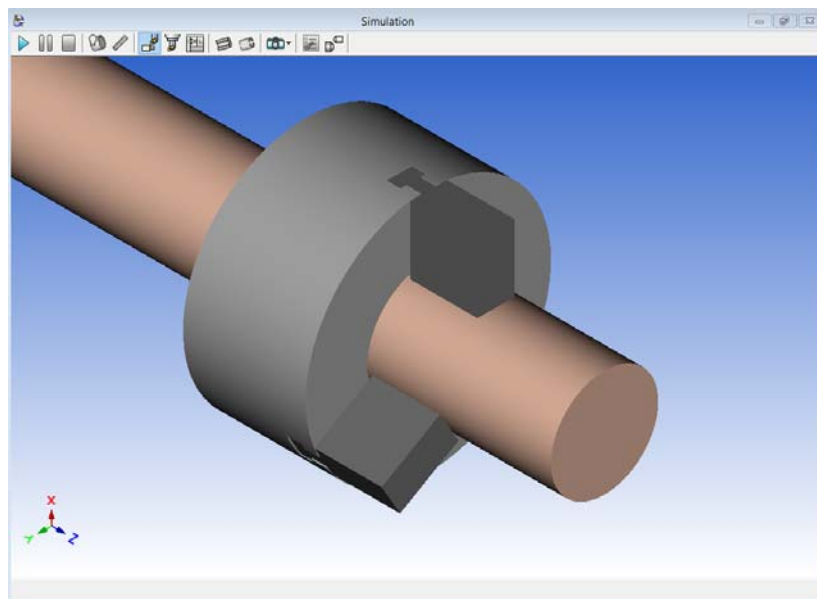
Choose **Simulation** from the **Simulation** menu



- 2 Make sure the **Basic Simulation** icon on the **Simulation Toolbar** is checked or Basic Simulation is checked under the **Simulation** menu

Upon launching simulation, you will see an uncut cylindrical work piece.

A newly launched simulation window is shown below:



Running Simulation

Once you have launched PartMaker Simulation, to start machining:



- 1 Press the Start Simulation button on the Simulation Toolbar

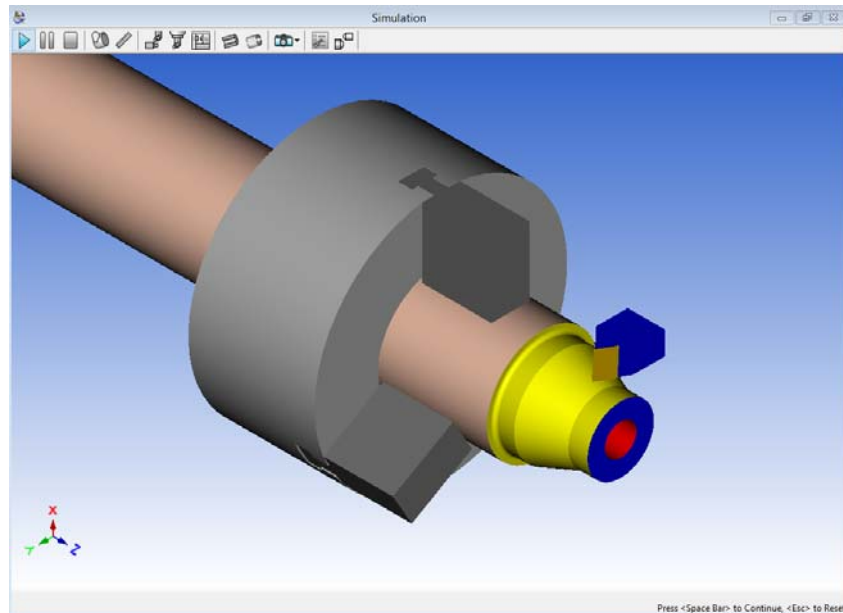
or

Press the **<Space Bar>** on your keyboard

or

Select Start Simulation from the Simulation Menu

- 2 Once simulation has started, you can suspend cutting by pressing the **<Esc>** button on your key board. Press the **<Space Bar>** to continue the simulation. Below, is an example of suspending simulation in process:



Using PartMaker Advanced Simulation

PartMaker Advanced Simulation can be viewed at any point in the programming process. You need not complete programming a part to see the results of your work.

None of the examples listed above is meant to represent a specific machine model. You will be able to view a more machine specific model for your machine when loading the *.MCH file specifically tailored to the machine you are programming.

Launching Advanced Simulation



Important! For purposes of this tutorial, please load the *.job, *.tdb and *.cdb files called *Turndemo.job*, *Tools.tdb* and *Cycles.cdb* located in the *C:\PartMaker\pm-turn* directory.

Important! To launch PartMaker Advanced Simulation you must have an *.MCH (machine configuration file) loaded. The MCH file contains a description of the machine you are using to machine the part.

You will only be able to launch PartMaker Simulation when the Process Table is generated. If you do not have a Process Table generated, choose **Generate Process Table** from the **Job Optimizer** menu.

To launch PartMaker Advanced Simulation:



- 3 Click the Start Simulation icon from the Main Toolbar

or

Press the **Simulation** button in lower left hand corner of the Process Table

or

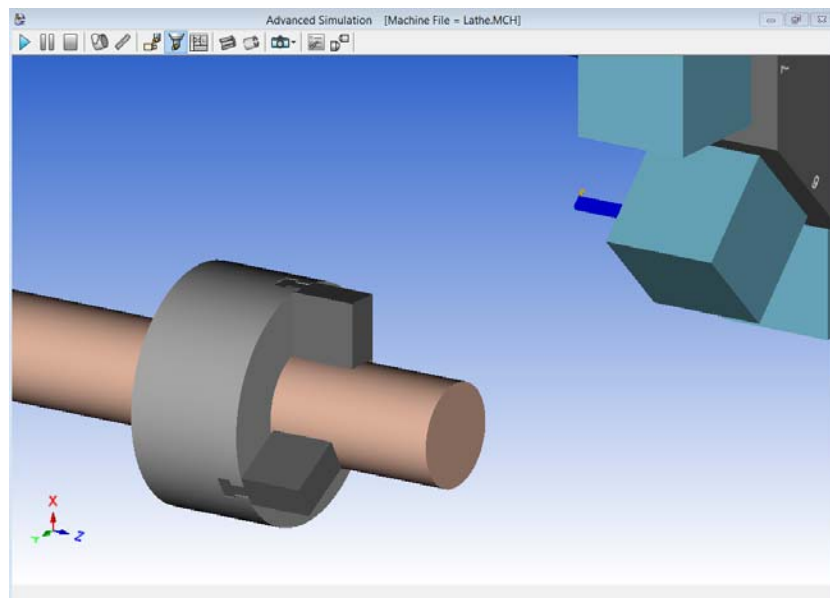
Choose **Simulation** from the **Simulation** menu



- 4 Make sure the **Advanced Simulation** icon on the **Simulation Toolbar** is checked or **Advanced Simulation** is checked under the **Simulation** menu

Upon launching simulation, you will see an uncut cylindrical work piece, tool posts with tools attached, and either one or two spindles depending on whether your part involves both main and sub spindle operations.

A newly launched simulation window is shown below:



Running Simulation

Once you have launched PartMaker Simulation, to start machining:



- 5 Press the Start Simulation button on the Simulation Toolbar

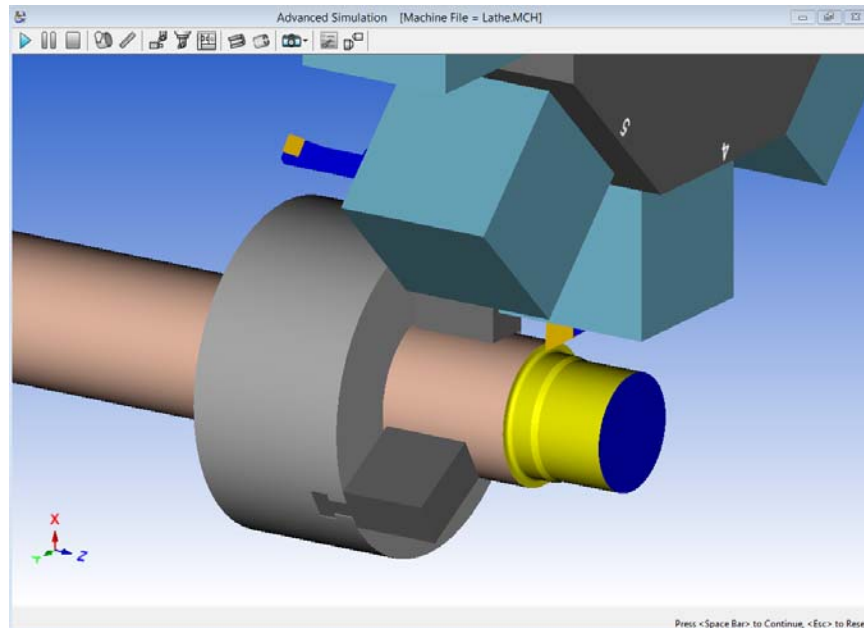
or

Press the **<Space Bar>** on your keyboard

or

Select Start Simulation from the Simulation Menu

- 6 Once simulation has started, you can suspend cutting by pressing the **<Esc>** button on your keyboard. Press the **<Space Bar>** to continue the simulation. Below, is an example of suspending simulation in process:



Machine File for Advanced Simulation

The machine you are seeing in PartMaker's Advanced Simulation is described by a **Machine Description** stored in a *.MCH file. Each machine architecture will have a separate Machine Description. The **Machine Description** may be viewed graphically by choosing **Machine...** from the **ToolMinder** menu. When doing so, the following dialog appears:

The **Machine Data** dialog box (Machine File = C:\PartMaker\pm-turn\Lathe.MCH) contains the following sections:

- Main Spindle:** Includes a **Properties** button.
- Machine Coordinate System:** Includes **Main Spindle Location** (Left/Right) and **X-Axis Orientation** (Vertical/Horizontal).
- Sub Spindle:** Includes **Stand Alone** (selected), **Attached To End-Working Slide**, **Not Available**, **Motion Axes** (X, Y, Z, R), and **Rotation Options** and **Properties** buttons.
- Machine Housing:** Includes a **Components** button.
- Tool Holders:** Includes a **Holder Data** button.
- File:** Includes **Open Machine File...**, **Save Machine File...**, and **Create FMS Reference** buttons.
- Units:** Includes **Metric** (checked), **Lock Units**, and **Convert from Metric** button.
- Supported Simulation Types:** Includes **Basic Simulation** (checked), **Advanced Simulation** (checked), and **Full Machine Simulation** (unchecked).
- Tool Posts:** A table with columns for **Tool Range** and **Motion Axes** (On Main Spindle, On Sub-Spindle). Each row has a **Properties** button.

	Tool Range	Motion Axes							
		On Main Spindle				On Sub-Spindle			
<input checked="" type="checkbox"/> Turret 1	1-8	<input checked="" type="checkbox"/> X	<input type="checkbox"/> Y	<input checked="" type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Turret 2	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Turret 3	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Turret 4	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Tool Head	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Gang 1	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Gang 2	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	<input type="checkbox"/> B
<input type="checkbox"/> Back	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z		<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	
<input type="checkbox"/> End	1-9	<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z		<input type="checkbox"/> X	<input type="checkbox"/> Y	<input type="checkbox"/> Z	

See PartMaker Help for a more in depth explanation of each of the dialogs and parameters in the **Machine Data File** dialog.

When should you use Basic vs. Advanced Simulation?

Basic and Advanced Machining Simulation are not mutually exclusive technologies.

Basic Simulation may be preferable to use in certain circumstances over Advanced Simulation because it:

- Provides more focus on just the part and cutting process because space is not taken up on screen with multiple machine components.
- Can run faster than Advanced Simulation because it is less graphically intensive.

Advanced Simulation will be preferable to use in certain circumstances over Basic Simulation because it:

- Provides a machine specific simulation, i.e. supports right and left-handed machines, specific machine components, etc.
- Provides additional crash and collision avoidance over Basic Simulation
- Gives the user verification that all tool numbers are correct for a given machine

As a rule of thumb, you may find Basic Simulation more beneficial for simulating your “first pass” at programming a part, while Advanced Simulation may be better when completing the programming of the entire component at which time you wish to see how the part will look as well as how it will run in its machine environment.

Advanced Simulation Limitations

The machine model you are seeing in PartMaker’s Advanced Simulation is a parametrically defined model of the machine you are programming. As it is based on parametric approximations, it may not simulate your machine exactly the way it looks. The limitations you may find at this point are:

Multiple Tool Stations

Multiple Tool Stations only applicable to the tools on the Turret. A Multiple Tool Station is the station on the turret that has more than one tool with the same Tool Number defined in Tool Data Dialog. During Advanced Simulation only one tool on the tool stations is shown. Therefore the tools that belong to the multiple tool station will be switched on the Turret based on the sequence in the Process Table.

Non-Standard Machine Architectures

PartMaker Advanced Simulation may not support certain elements of machines with irregular architectures at this time.

Please consult PartMaker Technical Support for more information on any limitations you may encounter with respect to Advanced 3D Machining Simulation.

Appendix A: Materials

PartMaker Material Library

This appendix contains a list of all material files and their abbreviations.

File	Description
ALU_ALLC.MDB	Aluminum Alloy, Cast
ALU_ALLW.MDB	Aluminum Alloy, Wrought
CARBIDES.MDB	Machinable Carbides
CHNICKEL.MDB	Chromium-Nickel, Alloy
COP_ALLC.MDB	Copper Alloy, Cast
COP_ALLW.MDB	Copper Alloy, Wrought
IRON_DUC.MDB	Ductile Iron, Cast
IRON_GRC.MDB	Gray Iron, Cast
LEADALLC.MDB	Lead Alloys, Cast
MAG_ALLC.MDB	Magnesium Alloys Cast
MAG_ALLW.MDB	Magnesium Alloys Wrought
NICKELWC.MDB	Nickel Alloys, Wrought and Cast
ST_ALLOW.MDB	Alloy Steels, Wrought
ST_CARB.MDB	Carbon Steels, Wrought

File	Description
ST_FMCAW.MDB	Free Machining Carbon Steel, Wrought
ST_FMSTW.MDB	Free Machining Stainless Steel, Wrought
ST_STAIC.MDB	Stainless Steel, Cast
ST_STAIW.MDB	Stainless Steel, Wrought
ST_STRUW.MDB	Structural Steels, Wrought
ST_TOOLW.MDB	Tool Steel, Wrought
THERMOPL.MDB	Thermoplastics
TIN_ALLC.MDB	Tin Alloys, Cast
TIT_ALLC.MDB	Titanium Alloys, Cast
TIT_ALLW.MDB	Titanium Alloys, Wrought
ZINCALLC.MDB	Zinc Alloy Cast

Appendix B: Family of Parts Programming Tutorial

Introduction

This tutorial will teach you how to use PartMaker's Family of Parts Programming Feature.

PartMaker Version 4.6 and higher features a unique Family of Parts Programming (FOP) capability, which greatly speeds CNC programming for parts created from tabulated drawings. This feature allows a programmer to program one part in a family and then automatically substitute any variable parameters by manipulating a separate file containing parameter definitions. A virtually unlimited number of parameter files (*.prm files) can be defined by the user.

FOP is an available option with every PartMaker product, namely, PartMaker/Mill, PartMaker/Turn, PartMaker/Turn-Mill and PartMaker/SwissCAM. It applies to Turning as well as Milling faces.



The steps presented in this tutorial are carried out in PartMaker SwissCAM but can be used in any PartMaker application.

Sequence of Steps for Family of Parts Programming

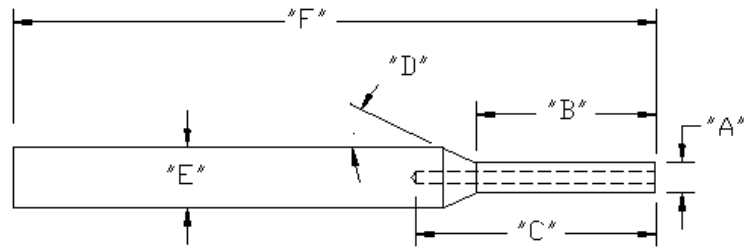
The steps outlined below describe the sequence in which you should approach programming Families of Parts from tabulated drawings. This programming tutorial follows the sequence of steps below in creating a typical family of parts.

You can use this page as desk reference when programming a Family of Parts.

- 1 **Create Part Geometry** for one part in the family using actual numbers. It is not critical which part number in the family you choose.
- 2 **Setup PartMaker Databases** appropriately, i.e. make sure all tools, materials and cycles files for the part have been set appropriately
- 3 **Assign part features** to your geometry. When doing so, make sure to assign toolpaths by snapping to existing geometry. If you do not snap tool paths to existing and parameterized geometry, they will not be updated when you regenerate other parts in the family.
- 4 **Create the appropriate Parameter Files (*.prm)** for each part in the family. Each Parameter File contains the variable definitions for each part in the family.
- 5 **Setup PartMaker for Parametric Input** by clicking the Parametric Input box in the Family of Parts section of the Preferences dialog which can be found under the View menu.
- 6 **Parameterize your part** by assigning variables to parametric geometric elements and part features such as thread pitch and hole depth.
- 7 **Set Part Boundaries Parametrically** by choosing Set Parametric Boundaries from Family of Parts Programming accessed from the Edit Menu.
- 8 **Update Each Part in the Family** by opening the respective Parameter File by choosing Open Parameter File from Family of Parts Programming, then choosing Apply Parametric Dimensions.
- 9 **Regenerate NC Code** for each part in the family by choosing Generate Process Table from the Job Optimizer menu and then choosing Generate NC Program from the Job Optimizer menu.

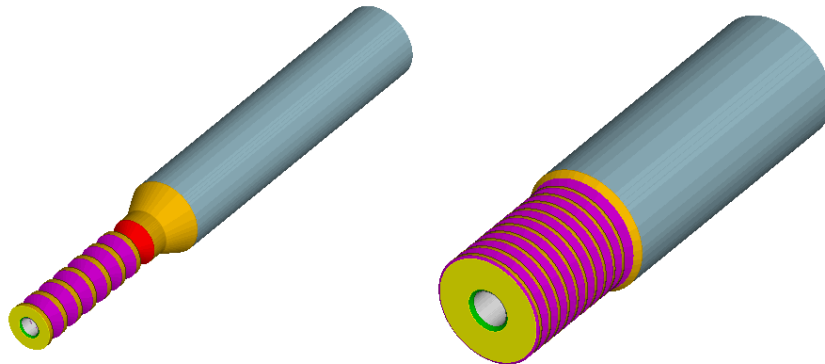
Getting Started

The blue print below represents a typical parametric part.



Part #	A	B	C	D	E	F	P/Pitch
50-100	0.25	1.0	1.5	25	0.4	3.0	0.05 (20)
50-200	0.25	0.8	1.0	15	0.38	2.5	0.03 (30)
50-300	0.1875	0.7	0.75	25	0.3	2.0	0.05 (20)
50-400	0.2	0.5	0.6	30	0.3	1.5	0.1 (10)
50-500	0.3125	0.4	0.5	35	0.35	1.25	0.04 (25)

When you complete this tutorial, you will have parts that looks like the picture below:



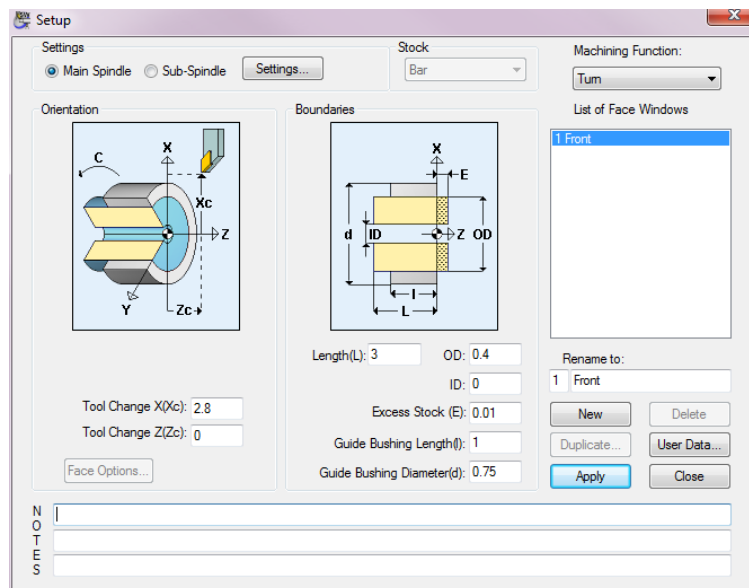
Create Part Geometry

The first step in programming a part in a family of parts is to create the part geometry. When doing so, you should pick one part number from the family. In this case, the Part #50-100 in the table on page B-2 will be used.

- 1 Double click on the SwissCAM icon on your desktop or choose **SwissCAM** from the **PartMaker** Group accessed through Programs under the Windows Start button.
- 2 You will then see the **Setup** definition dialog box.
- 3 Enter the parameters listed below in the appropriate fields:

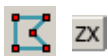
• Length (L):	3.0	• OD:	0.4
• Excess Stock (E):	0.0	• ID:	0.0
	1		
• GB Length (I):	1.0	• GB Dia (d):	0.75
• Tool Change X(Xc):	2.8	• Tool Change Z (Zc):	0.0

The setup dialog will then appear below.



- 4 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current **Face Window**. Click the **<Close>** button to close the **Setup** dialog.
- 5 In the CAM **Face Window**, choose **Show Axes** from the **View** menu. This will display the horizontal and vertical axis lines which will help you create the geometry.
- 6 Click on the **CAD/CAM** switch in the lower left-hand corner of the window. The icon will change its appearance from cutting tools to a pencil.
- 7 Select the **Connected Lines Icon** from the drawing icons on left side of the **Face Window**. Here, you will draw the part in the print on the second page of this chapter by plotting ZX coordinates, using the **ZX Snap Mode** from the **Snap Modes** icon bar at the top of the screen.

After selecting the **ZX Snap Mode** two data entry fields will appear in the lower left hand corner of the **Face Window**.



Now enter the ZX coordinates as shown below, pressing **<Enter>** after each:

default	0.25
1	default
default	0.4
3	default
default	0

After entering these coordinates, you should have a drawing which appears as the profile below:



- 8 You will now place the diagonal line represented by the “D” dimension on. To do so, select the **Line on an Angle** icon and select the **End of Line** snap mode.

When doing so, you will be prompted to **Select end point of line or arc**. Drag your cursor into place as shown below and click the left mouse button.



You will now be prompted to enter an angle for this line. At the prompt enter a value of 155 (i.e. $180 - 25$) and press **<Enter>**.

Move your cursor so the arrow is on the line as shown below and click the left mouse button.



Having done so, you will now have a drawing which appears as shown below.



Now select the **Remove** icon and click the scissors on lines 1 and 2 in the picture below.



After having done so, your geometry will appear as shown below:

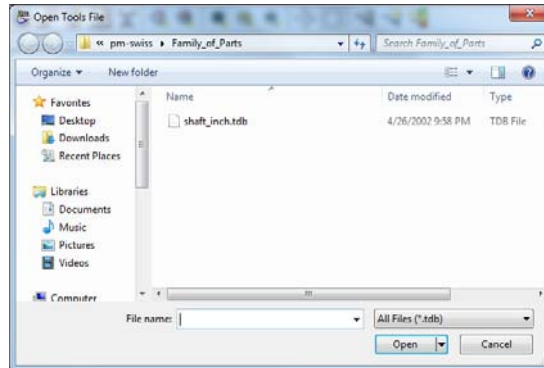


Note: When programming Families of Parts, you *SHOULD NOT* mirror geometry for turning features about the Z-axis. Doing so will cause the lower half of your part geometry to look deformed when you generate other parts in the family. Alternatively, you could also parameterize the lower half of your part geometry (though this would entail additional work). Parameterizing geometry will be discussed in Step 5.

Setup PartMaker Databases

The second step in programming a part in a family of parts is make sure the proper Tools, Materials and Cycles have been created and loaded in PartMaker.

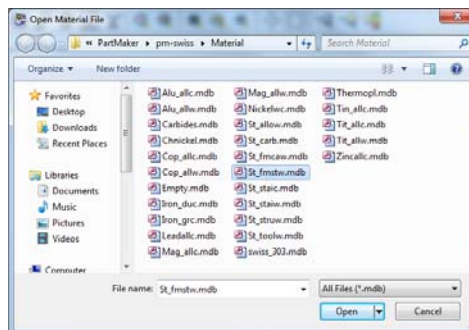
- 1 Open the Tools file by choosing **Open Tools File** from the **File** menu.
- 2 From the Family_of_Parts directory, choose shaft_inch.tdb and click **<OK>**.



- 3 Open the Cycles file by choosing **Open Cycles File** from the **File** menu.
- 4 From the Family_of_Parts directory, choose shaft_inch.cdb and click **<OK>**



- 5 Open the Materials file by choosing **Open Material File** from the **File** menu.
- 6 Choose st_fmstw.mdb and click **<Open>**



Assign Part Features

Once you have created your part geometry and loaded the appropriate tools, materials and cycles files, you are ready to assign part features to your geometry.



Note: When programming Families of Parts, it is important to always assign part features to your part by snapping to geometry instead of using the ZX Snap Mode. Tool paths not associated with geometry will not be rescaled when updated for other parts in a part family.



- 1 Before assigning the facing operation to the part, it is necessary to create the geometry to which the facing tool path will be assigned. To do so, while still in the CAD Mode, choose the **Line Through Two Points** and **ZX** Snap Mode.

Now enter the ZX coordinates as shown below, pressing **<Enter>** after each:

(0,.4)

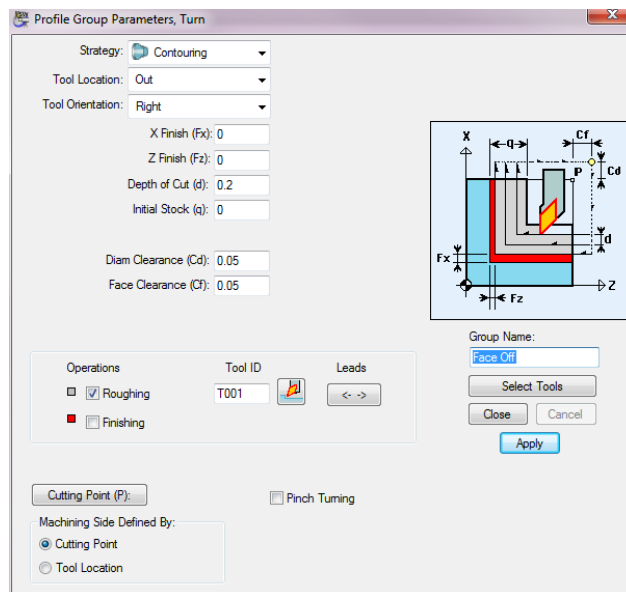
(<default>,0)

After having done so, your geometry will appear as shown below:



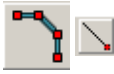
- 2 Click on the **CAD/CAM** switch to return to the CAM mode and create the part feature for the facing operation by choosing **New Profile Group** from the **Part Features** menu.

- a From the **Strategy** menu, choose **Contouring**.
- b From the **Tool Location** menu, select **Out**.
- c From the **Tool Orientation** menu, select **Right**.
- d Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.
- e In the field called **Group Name**, enter **Face Off**. Your dialog should now appear as shown below:

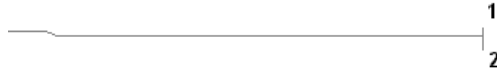


Click Apply and Close.

Now that you have defined the machining operation to be used, you must tell **PartMaker** where to place the tool path:



- 1 From the left-hand side of the screen select the **Define Profile** icon.
- 2 From the top of the screen, select the **End of an Element** snap mode.
- 3 Click your cursor at points 1 and 2 below:



Your completed profile should appear as below:

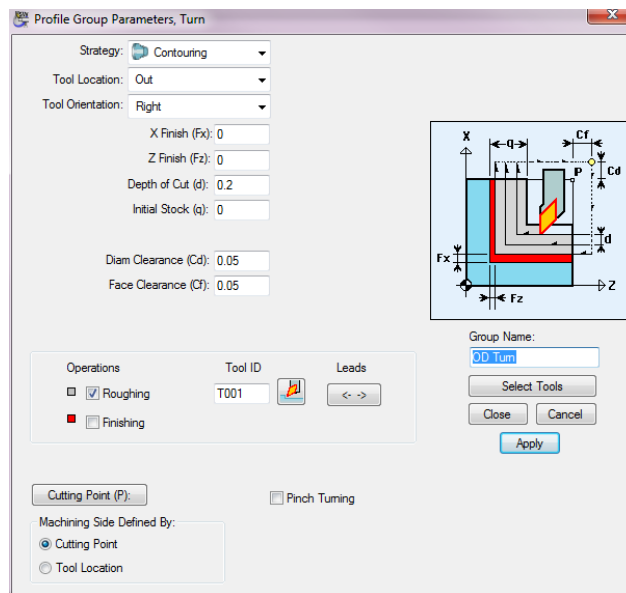


- 4 Click the **Selection** icon to bring yourself to a home or neutral position.

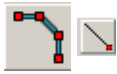
Before moving on, save your work! Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**. Give your job file a name you will remember. A good job file name for part families is **Master**.

Next, you will program the first outside diameter turning feature.

- 1 Create the first OD Turning tool path by choosing a new color and selecting **New Profile Group** from the **Part Features** menu.
 - a From the **Strategy** menu, choose **Contouring**.
 - b From the **Tool Location** menu, select **Out**.
 - c From the **Tool Orientation** menu, select **Right**.
 - d Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.
 - e In the field called **Group Name**, enter **OD Turn**. Your dialog should now appear as shown below:



Now that you have defined the machining operation to be used, you must tell **PartMaker** where to place the tool path:



- 2 From the left-hand side of the screen select the **Define Profile** icon.
- 3 From the top of the screen, select the **End of an Element** snap mode.
- 4 Click your cursor at points 1, 2 and 3 below:

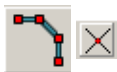
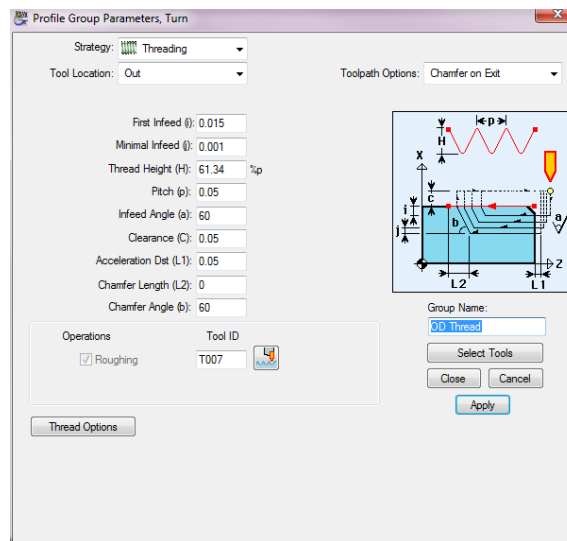


Your completed profile should appear as shown below:



Next, you will program the outside diameter threading feature.

- 1 Create the OD Threading tool path by choosing a new color and selecting **New Profile Group** from the **Part Features** menu.
 - a From the **Strategy** menu, choose **Threading**.
 - b From the **Tool Location** menu, select **Out..**
 - c Select Chamfer on Exit from the Toolpath Options drop down list.
 - d Specify the pitch as 0.05.
 - e Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.
 - f In the field called **Group Name**, enter **OD Thread**. Your dialog should now appear as shown below:



- 2 From the left-hand side of the screen select the **Define Profile** icon.
- 3 From the top of the screen, select the **Closest Intersection** snap mode.
- 4 Click your cursor at points 1 and 2 below:



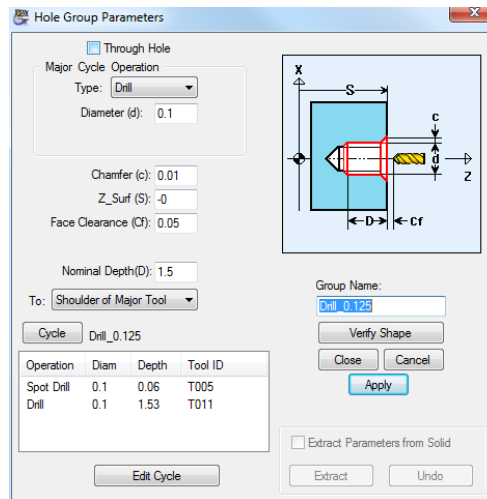
Your completed profile should appear as shown below:



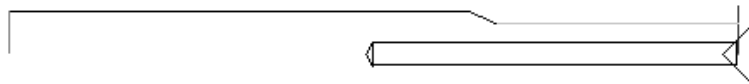
Next, you will program the drilled hole.

- 1 Create the drilled hole by choosing a new color and selecting **New Hole Group** from the **Part Features** menu.
 - a Uncheck the **Through Hole** box
 - b From the **Major Cycle Operation** drop down menu, choose **Drill**
 - c Enter .1 in the **Diameter** field
 - d Enter .01 in the **Chamfer** field
 - e Enter 1.5 in the **Nominal Depth** field
 - f Click the **Cycle** button and allow **PartMaker** to select the appropriate cycle.

Your completed Hole Group Parameters dialog should now appear as shown below:



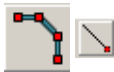
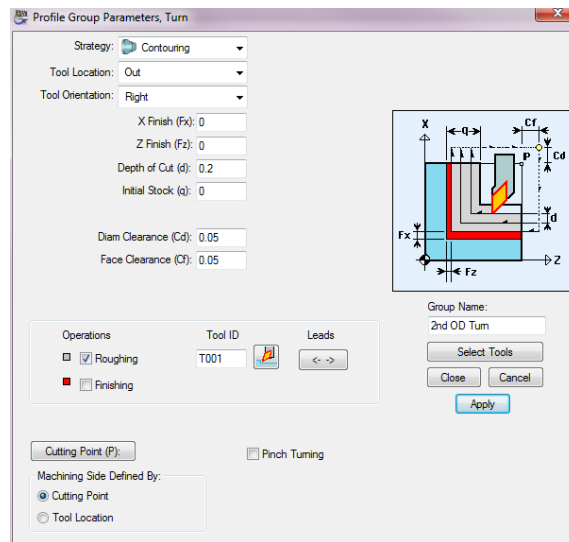
- 2 Your completed profile should appear as shown below:



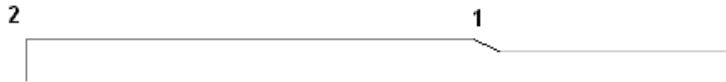
Next, you will program the second outside diameter turning feature.

- 1 Create the second OD Turning tool path by choosing a new color and selecting **New Profile Group** from the **Part Features** menu.
 - a From the **Strategy** menu, choose **Contouring**.
 - b From the **Tool Location** menu, select **Out**.
 - c From the **Tool Orientation** menu, select **Right**.
 - d Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.

- e In the field called **Group Name**, enter **2nd OD Turn**. Your dialog should now appear as shown below:



- 2 From the left-hand side of the screen select the **Define Profile** icon.
- 3 From the top of the screen, select the **End of an Element** snap mode.
- 4 Click your cursor at points 1 and 2 below:



Your completed profile should appear as shown below:

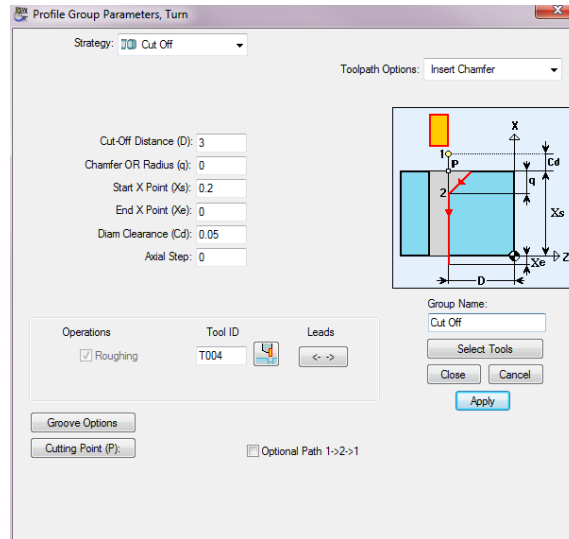


Finally, you will program the cut-off operation.

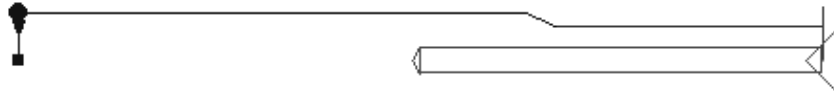
When creating a cut-off operation using PartMaker's Family of Parts programming feature, you will need to assign the part feature to the geometry instead of having PartMaker do so automatically for you as you would in conventional use of the software.

- 5 Create the Cut-off tool path by choosing a new color and selecting **New Profile Group** from the **Part Features** menu.
 - a From the **Strategy** menu, choose **Cut-Off**
 - b In the **Start X Point (Xs)** enter .4/2
 - c Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.
 - d In the field called **Group Name**, enter **Cut-off**.

Your dialog should now appear as shown below:



Your completed part should appear as shown below:



- 6 Before continuing, choose the Selection icon and delete the tool path that has been automatically applied to the geometry.
- 7 Using the Profile icon and End Of Element Snap Mode, reapply the tool path to the geometry by clicking at the start and end of the cut-off tool path. Your completed part will appear as shown below:



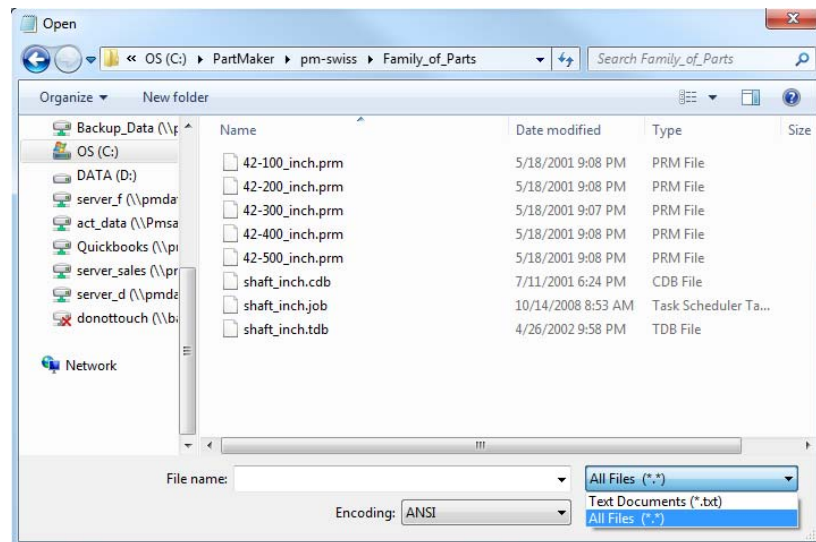
Creating Parameter (*.prm) files

Once you have completed programming the “master” part in the family, you must now take a step back and setup files containing parameter definitions for each part in the family. These files can be created in any word processing program and then saving files with *.prm extension. In this example, **Notepad** (which comes loaded on every Windows operating system) will be used

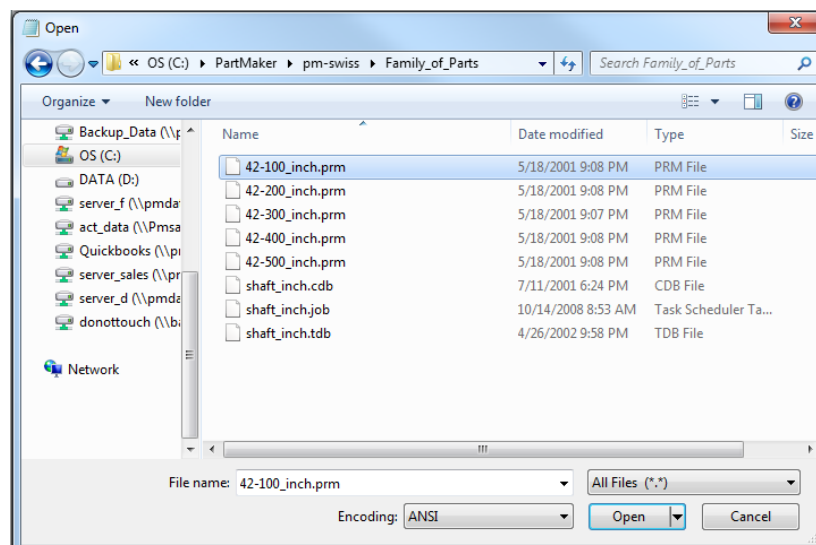
- 1 Create the first parameter for the “master” part that has already been programmed.

Note: To make creating the first parameter file faster, start by manipulating an existing file, either from a part family you have made before or from the parameter files which came standard with your PartMaker installation.

- 2 From the **Start** button on your Windows desktop, choose **Accessories** and launch the **Notepad** application.
- 3 In **Notepad**, from the File menu, choose **Open**. In the **Open** dialog, choose **All Files *.*** from **Files of Type** as shown below:



- 4 Choose an existing parameter file. In this example, open the file titled **42-100_inch.prm** that can be found in the **Family of Parts** directory (located in the root directory of the PartMaker module you are working with.)

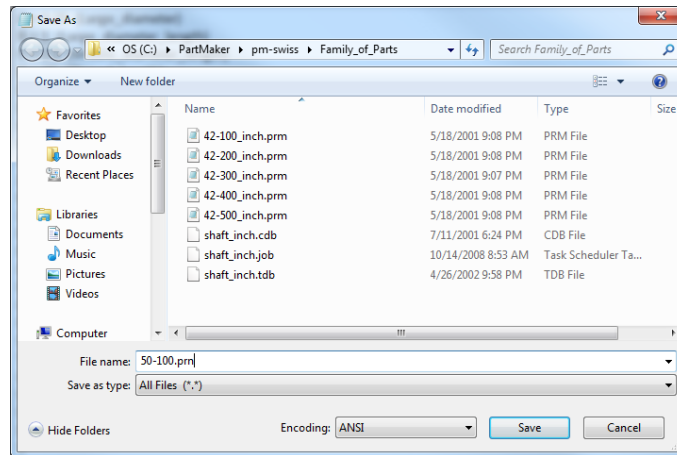


- 5 Edit the parameter file to reflect the parameters being used in the current Family of Parts. Replace the parameter values with those you have used in the creation of part 50-100 above.

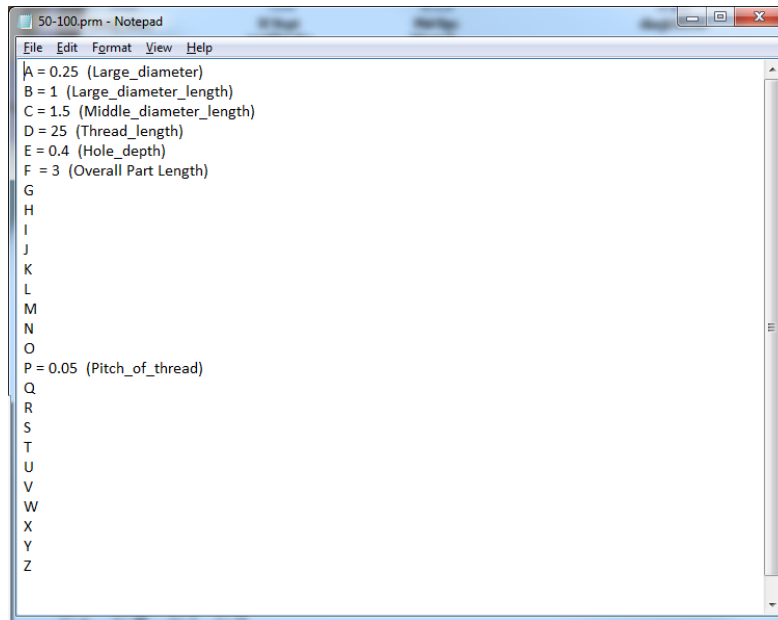


Note: All text and characters in parenthesis are comments and will not affect programming. Comments will help you remember which parameters correspond to various features of your part.

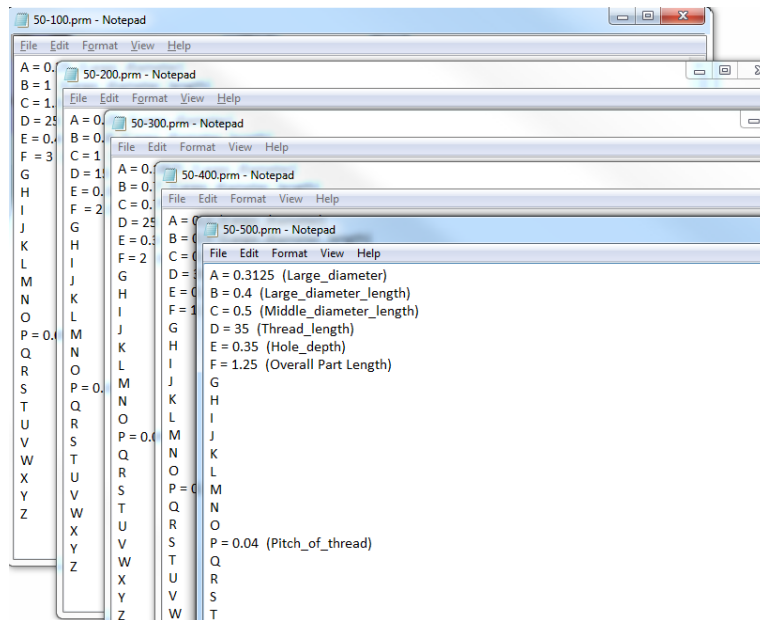
Once you have defined the parameters for the part, save your Parameter File as 50-100.prm by choosing **Save As** from the **File** menu as shown below:



- 6 Click **<Save>** to save your file. The saved parameter file for 50-100 should appear as shown below:



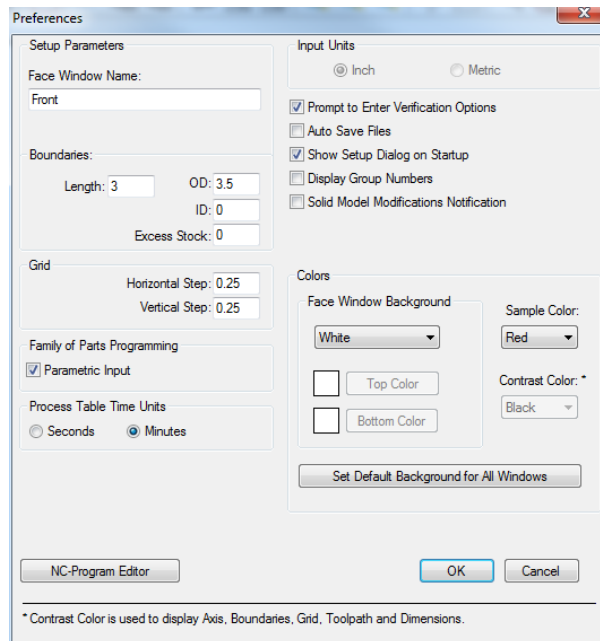
- Having created the first parameter file, create a file for all other parts in the family by simply replacing parameter values and saving a ***.prm** file for each part in the family. The parameter values for each part for this exercise can be found on page 2 of this appendix. When finished, you should have five parameter files as shown below:



Setup PartMaker for Parametric Input

In order to be able to apply parametric dimensions to the part, you must enable Parametric Programming.

- Choose **Preferences** from the **View** menu and check the **Parametric Input** box as shown below:

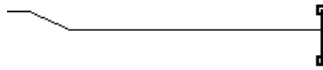


- Once you have checked **Parametric Input**, click **<OK>**.

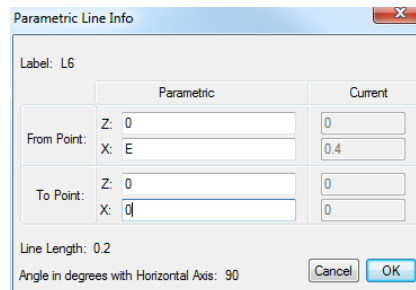
Parameterize Your Part

Now that all the appropriate parameter files have been created, you can assign parametric variables to the master part in the family. Before proceeding, make sure all parameter files have been saved and are closed.

- 1 Click on the CAD/CAM switch to return to the CAD mode. Any part parameterization exercise should always start in the CAD mode.
- 2 Double click on the line segment drawn vertically along the face of the part as shown below:



Doing so will allow you to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:

The dialog box is titled "Parametric Line Info". It has a "Label" field with the value "L6". Below this is a table with two columns: "Parametric" and "Current". The table has two rows: "From Point:" and "To Point:". The "From Point:" row has "Z: 0" and "X: E" in the "Parametric" column, and "0" and "0.4" in the "Current" column. The "To Point:" row has "Z: 0" and "X: 0" in the "Parametric" column, and "0" and "0" in the "Current" column. Below the table, there are fields for "Line Length: 0.2" and "Angle in degrees with Horizontal Axis: 90". At the bottom right are "Cancel" and "OK" buttons.

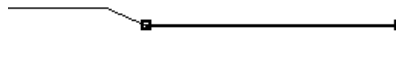
	Parametric	Current
From Point:	Z: 0 X: E	0 0.4
To Point:	Z: 0 X: 0	0 0

Line Length: 0.2
Angle in degrees with Horizontal Axis: 90

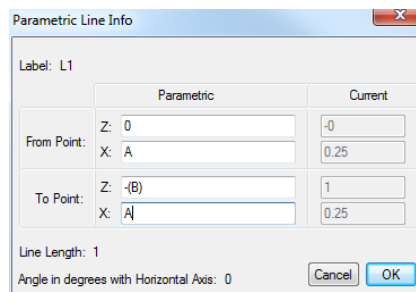
Cancel OK

- 3 Once the dialog appears as above, click <OK>.

The next geometric element to be parameterized will be the thread as shown below:



- 4 Double click on the horizontal line segment along which the thread is applied to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:

The dialog box is titled "Parametric Line Info". It has a "Label" field with the value "L1". Below this is a table with two columns: "Parametric" and "Current". The table has two rows: "From Point:" and "To Point:". The "From Point:" row has "Z: 0" and "X: A" in the "Parametric" column, and "-0" and "0.25" in the "Current" column. The "To Point:" row has "Z: -(B)" and "X: A" in the "Parametric" column, and "1" and "0.25" in the "Current" column. Below the table, there are fields for "Line Length: 1" and "Angle in degrees with Horizontal Axis: 0". At the bottom right are "Cancel" and "OK" buttons.

	Parametric	Current
From Point:	Z: 0 X: A	-0 0.25
To Point:	Z: -(B) X: A	1 0.25

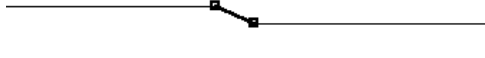
Line Length: 1
Angle in degrees with Horizontal Axis: 0

Cancel OK

- 5 Once your dialog appears as above, click <OK>.

Note: In PartMaker, all Z dimensions are programmed in negative. Whenever performing mathematical operations on parameters, always make sure to use parenthesis as shown here. PartMaker observes standard mathematical order of operations for multiplication, division, addition, subtraction and the application of trigonometric functions.

The next geometric element to be parameterized will be the chamfer as shown below:



- 6 Double click on the diagonal line segment of the chamfer to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:

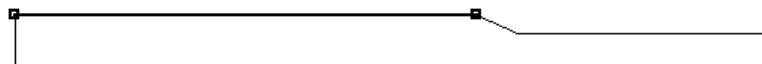
Parametric		Current
From Point:	Z: $-(B)$	1
	X: A	0.25
To Point:	Z: $-((E-A)/2)/(TAN(D))-(B)$	1.16083802
	X: E	0.4

Line Length: 0.17746512
Angle in degrees with Horizontal Axis: 155

Cancel OK

Notice the use of the use of trigonometric relationships in Family of Parts programming. PartMaker data entry fields can accept trigonometric functions. Once your dialog appears as above, click **<OK>**.

The next geometric element to be parameterized will be the largest horizontal diameter as shown below:



- 7 Double click on the horizontal line segment of the OD to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:

Parametric		Current
From Point:	Z: $-((E-A)/2)/(TAN(D))-(B)$	1.16083802
	X: E	0.4
To Point:	Z: $-(F)$	3
	X: E	0.4

Line Length: 1.83916198
Angle in degrees with Horizontal Axis: 0

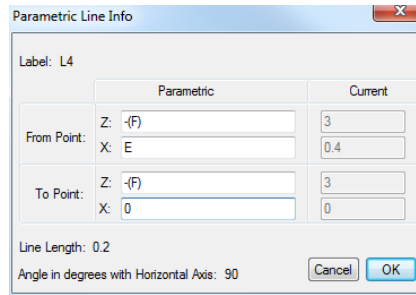
Cancel OK

- 8 Using the **Copy** and **Paste** functions from the **Edit** menu (standard windows accelerator keys **Ctrl + C** and **Ctrl + V**) you can copy and paste parametric relationships from one dialog to another. Once your dialog appears as above, click **<OK>**.

The next geometric element to be parameterized will be the geometry along which the cut-off operation is defined:



Double click on the diagonal line segment of the chamfer to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:



Parametric Line Info

Label: L4

	Parametric	Current
From Point:	Z: $-(F)$	3
	X: E	0.4
To Point:	Z: $-(F)$	3
	X: 0	0

Line Length: 0.2

Angle in degrees with Horizontal Axis: 90

Cancel OK

Once your dialog appears as above, click **<OK>**.

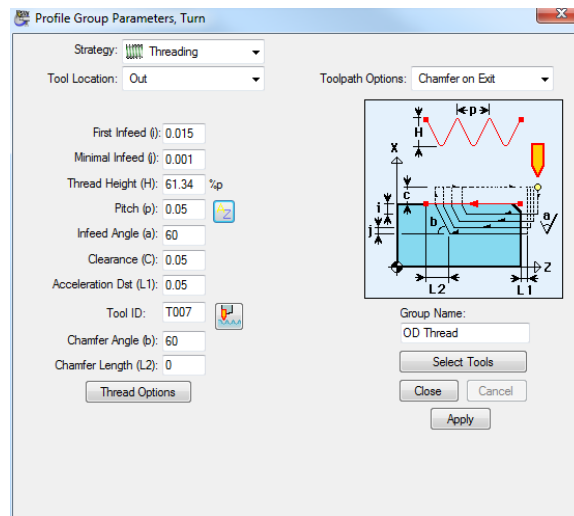


- Click on the CAD/CAM switch to return to the CAM mode. The parameterization exercise will be completed by applying parametric dimensions to programmed part features.

Note: In Machining Function Face Windows of Type Turn, in the **Profile Group Parameters** dialog, Strategy types **Threading** and **Cut-off** allow for parametric input. In Milling Machining Function Face Window, all **Profile Group Parameters** cycles accept parametric input. The **Hole Group Parameters** dialog also accepts parametric input.

When using Parametric Input you will notice the appearance of **Parametric Input** icons. Clicking on these icons allows you to enter parametric data for part features.

Double click on the **OD Thread** Group Symbol on the right hand side of the screen to enter the **Profile Group Parameters** dialog for threading as shown below:



Profile Group Parameters, Turn

Strategy: Threading

Tool Location: Out

Toolpath Options: Chamfer on Exit

First Infeed (i): 0.015

Minimal Infeed (i): 0.001

Thread Height (H): 61.34 %p

Pitch (p): 0.05

Infeed Angle (a): 60

Clearance (C): 0.05

Acceleration Dist (L1): 0.05

Tool ID: T007

Chamfer Angle (b): 60

Chamfer Length (L2): 0

Thread Options

Group Name: OD Thread

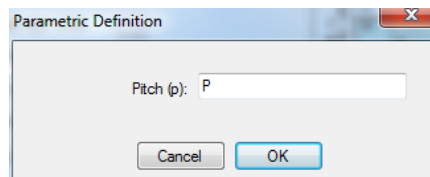
Select Tools

Close Cancel

Apply



- Click on the Set **Parametric Pitch** icon to insert a parametric value for Thread Pitch as shown below:



Parametric Definition

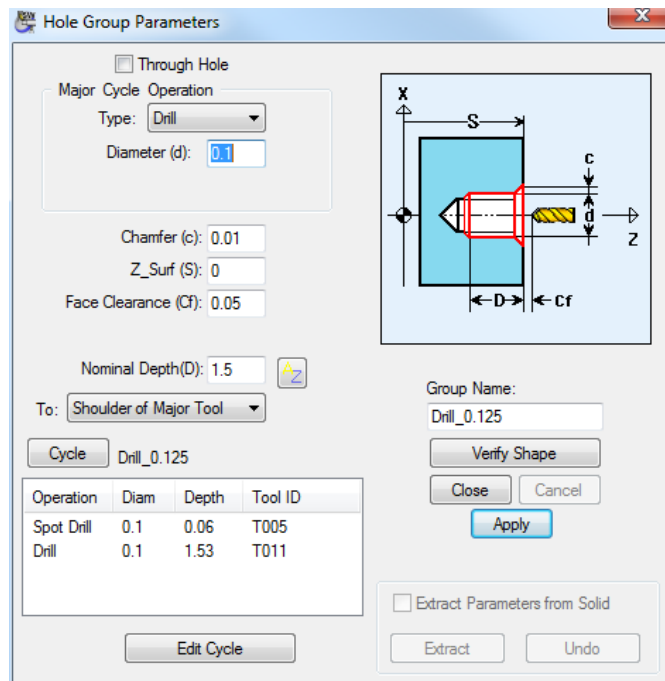
Pitch (p): P

Cancel OK

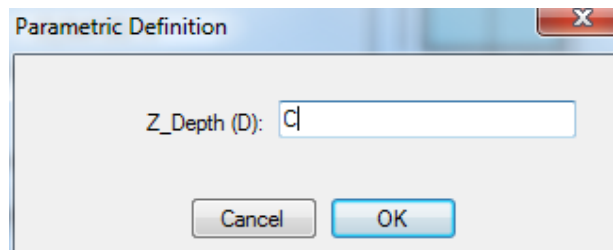
- Once your dialog appears as above, click **<OK>**. Click **<Close>** in the **Profile Group Parameters** dialog to return to the CAM Face Window.

Next, parameterize the depth of the hole programmed on centerline. To do so:

- 12 Double click on the group symbol for the on-centerline drilled hole.



- 13 Click on the Set **Parametric Pitch** icon to insert a parametric value for Z Depth as shown below:



- 14 Once your dialog appears as above, click <OK>. Click <Close> in the **Profile Group Parameters** dialog to return to the CAM Face Window.

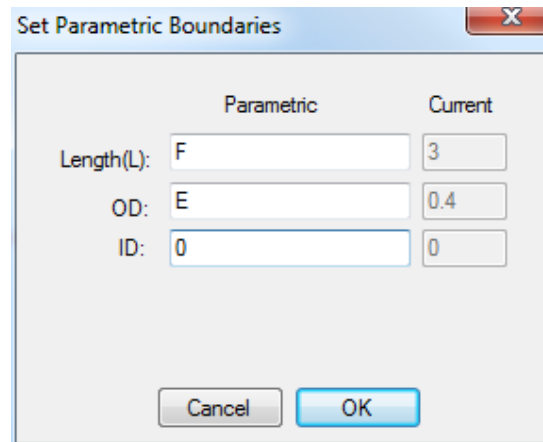
Set Part Boundaries Parametrically

The final step in setting up a parametric part is to set the part boundaries parametrically. To do so:

- 1 Choose Family of Parts Programming from the Edit menu.
- 2 Choose Set Parametric Boundaries... from the Family of Parts Programming menu as shown below:



- 3 Complete the **Set Parametric Boundaries** dialog as shown below:



Once the dialog appears as above, click **<OK>** to return to the CAM Face Window.

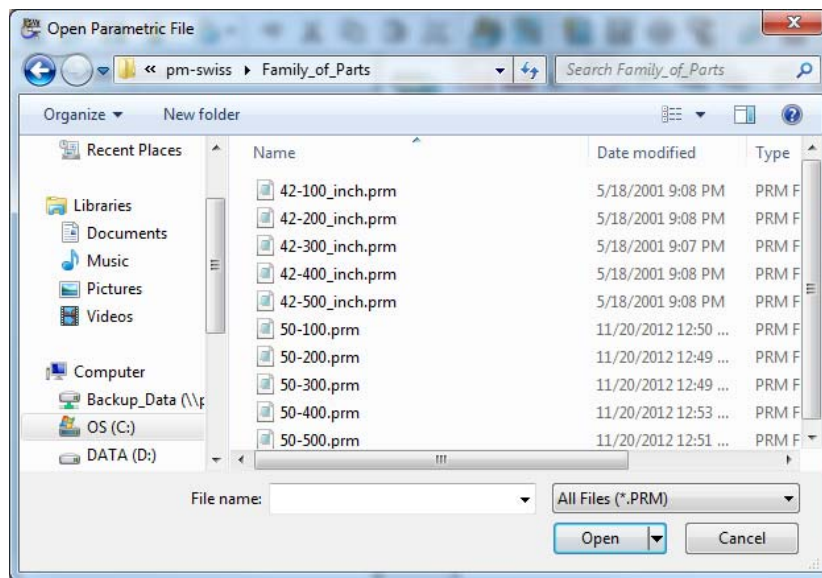
Update Each Part in the Family

You have now completed setting up your master part and are now ready to automatically program each other part in the family. To automatically reprogram other parts in the family:

- 1 Choose Family of Parts Programming from the Edit menu.
- 2 Choose Open Parameter File... from the Family of Parts Programming menu as shown below:



Select the parameter file for the part you would like to program. In this case, choose 50-500.prm as shown below:



Once the dialog appears as above, click **<Open>** to return to the CAM Face Window.

- 3 Choose **Apply Parametric Dimensions...** from the Family of Parts Programming menu as shown below:



You should now see your part change size on screen.

Regenerate NC Code

You can update your NC program for each part in the family by regenerating the **Process Table** and then choosing **Generate NC Program**.

Appendix C: Programming Directly on a Solid Model

Introduction

This document describes the process of programming directly on a Solid Model in PartMaker Turn-Mill, SwissCAM and Turn Version 9.1 and higher.

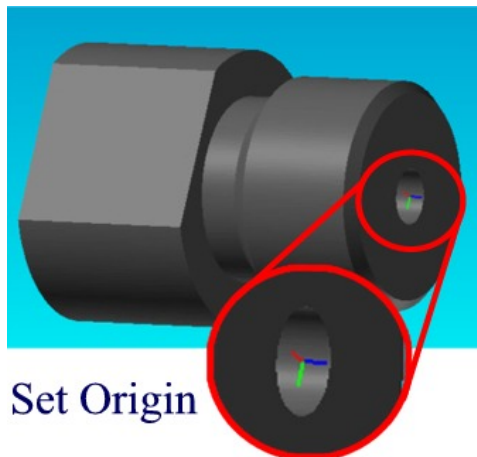


NOTE: The steps presented in this tutorial are carried out in **PartMaker SwissCAM** but can be used in any **PartMaker** application.

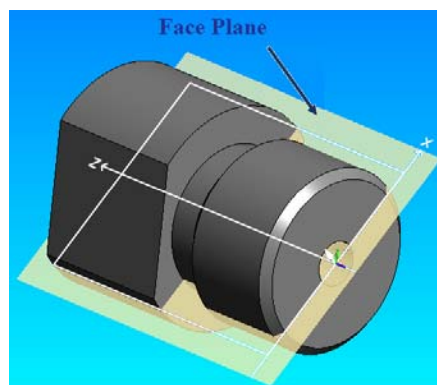
Key Definitions



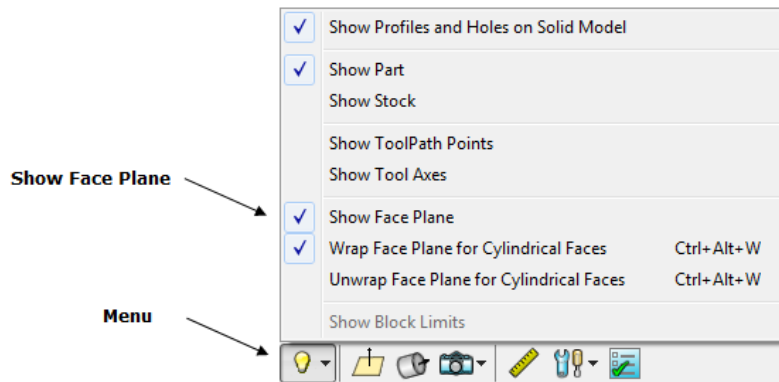
Part Coordinate System (PCS): A part coordinate system (PCS) is the coordinate system in which a solid model is oriented. Before any programming exercise begins, you should always set the PCS to be the same as the coordinate system of the machine you are programming. You can use the **Edit Coordinate System** dialog to place the origin on the front of the part and set the axes to be coincident with the machine's coordinate system. Being set initially the PCS should not change during the programming session.



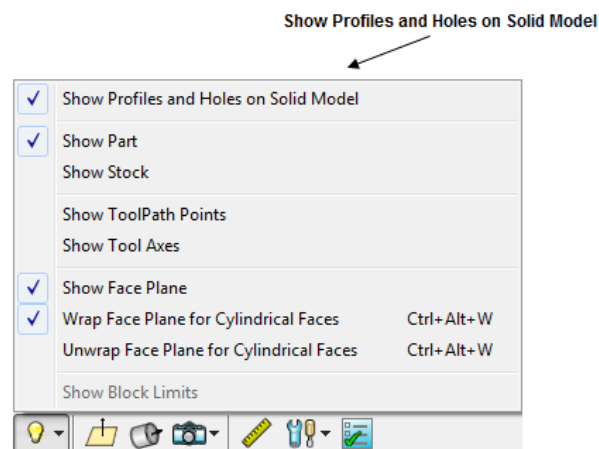
Face Coordinate System (FCS): This is a coordinate system with two axes in the Face Plane and the third axis perpendicular to the Face Plane. The Face Plane is a plane in which the tool paths in the active PartMaker Face Window are created. Each Face Window can have its own Face Plane. The Face Plane must be set in the correct place to properly program on the Solid Model. The Face Plane can be displayed using the **Show Face Plane** icon. The Face Plane can be set and manipulated using the **Define Face Plane** dialog.



Face Plane Icons for Solid Model:



Show Profiles and Holes on Solid Model: You will need to have this feature activated to program directly on the Solid Model. This will allow you to view the Profiles as they are created.



Solids Programming Tutorial

This tutorial is designed to help you learn the steps to follow for using **PartMaker SwissCAM**, **Turn-Mill** and **Turn** to program on a Solid Model.

The complete finished job files can be found in the following directories as indicated:

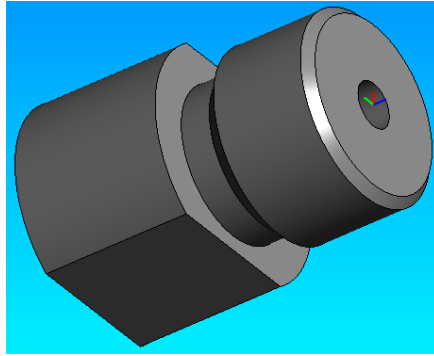
SwissCAM	C:\PartMaker \PM-Swiss\Solids_Programming_Tutorial
Turn-Mill	C:\PartMaker \PM-TM\Solids_Programming_Tutorial
Turn	C:\PartMaker \PM-Turn\Solids_Programming_Tutorial

How you will create the Sample part

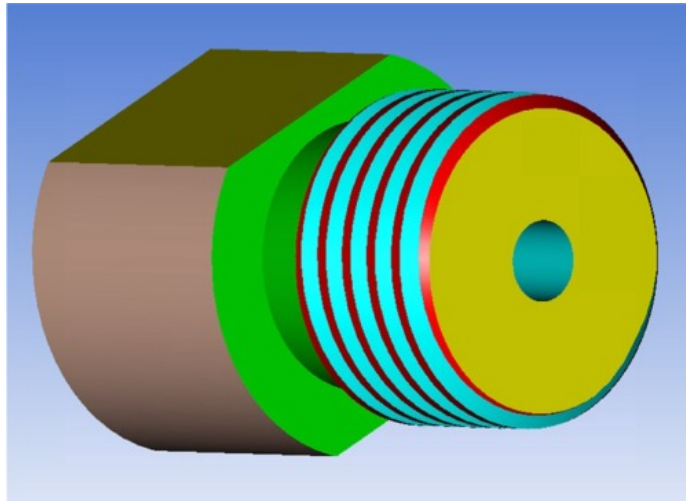
Here are the major steps you will follow to create the Swiss tutorial part:

- √ Start **PartMaker**
- √ Open Sample Program and Review
- √ View tool information for the part
- √ View cycle information for the part
- √ Select a material for the part
- √ Open a new Face Window
- √ Create Profile and Hole Groups
- √ Extract Information from the Solid Model
- √ Create a Process Table
- √ Simulate the Machining Process
- √ Generate an NC Program

The picture below shows the part to be programmed in this tutorial.



When you complete this tutorial, you will have a part that looks like the picture below.



Getting Started

The machining functions used in this tutorial are as follows:

Turn

Use turning tools to perform a variety of turning operations including Facing, Threading, Grooving, Drilling, Tapping and Pick-off.

Mill ZY

Use an end mill to create a set of flats.

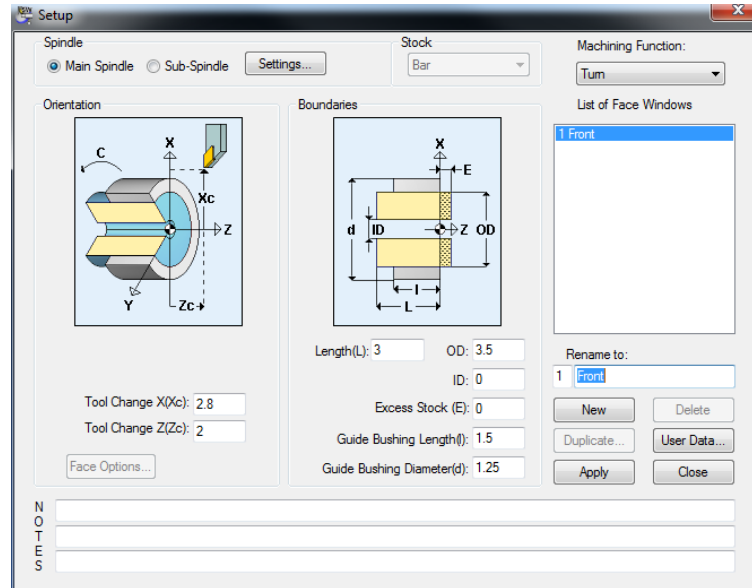
Programming on the Solid Model can be accomplished for almost every Machining Function needed. In most cases you will not need to work in the 2D Face window at all.

Now you will look in detail at the steps necessary to program a part.

Starting PartMaker

- 1 Double click on the PartMaker SwissCAM icon
- 2 You will see the **Setup** definition dialog box shown below. Click the <Close> button.

This dialog is explained in greater depth starting on page 8 of this guide.



View the Finished Sample

- 1 Use the **Open All Files** Command from the **File** Menu.
- 2 You will find the Job Files in the Following Directories.

SwissCAM C:\PartMaker\PM-Swiss\Solids_Programming_Tutorial

Turn-Mill C:\PartMaker\PM-TM\Solids_Programming_Tutorial

Turn C:\PartMaker\PM-Turn\Solids_Programming_Tutorial



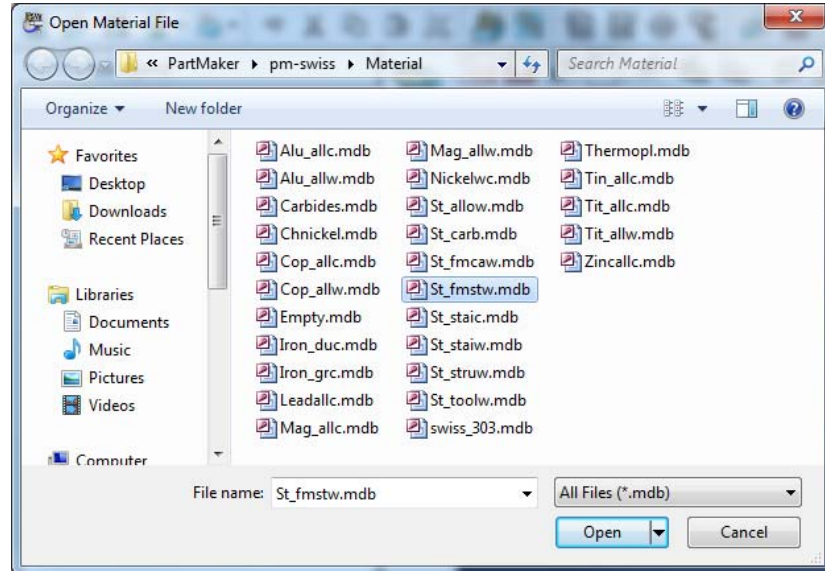
Note: This will also open the Sample Tools and Cycles files for you to use when you create your own Part.

- 3 Look through the Sample Job file and familize yourself with the Profiles and Hole Groups.
- 4 When Ready, choose **New Job** from the **File** Menu.

Select a Material for the Part

The work piece in this exercise will be machined from **Free Machining Stainless Steel, Wrought**. To load this material:

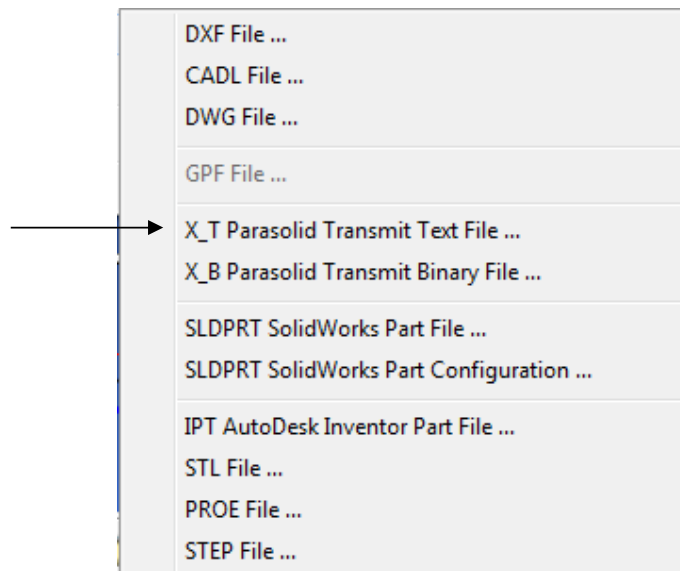
- 1 Choose **Open Material File** from the **File** menu.
- 2 Choose **st_fmstw.mdb** as shown below and click **<Open>**.



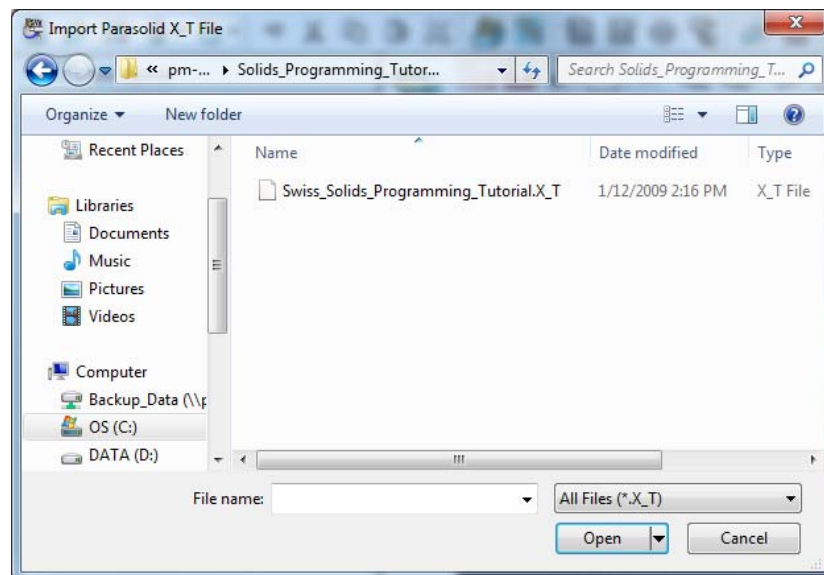
You have now loaded the tools, cycle, and material files that were previously developed for the Solid Model Programming Sample.

Importing a Solid Model

- 1 Select the **Import** icon and select the **X_T Parasolid Transmit Text File** option.



- 2 Choose the `Swiss_Solids_Programming_Tutorial.X_T` file and click <Open>

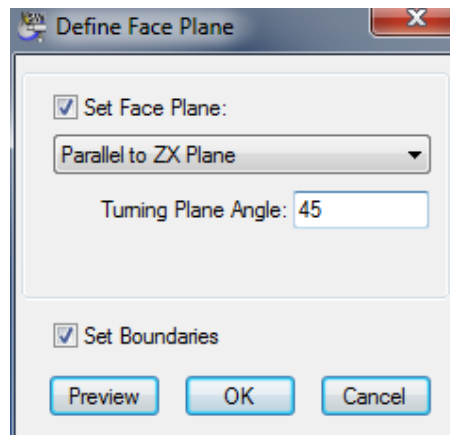


Setting up the First Face Window

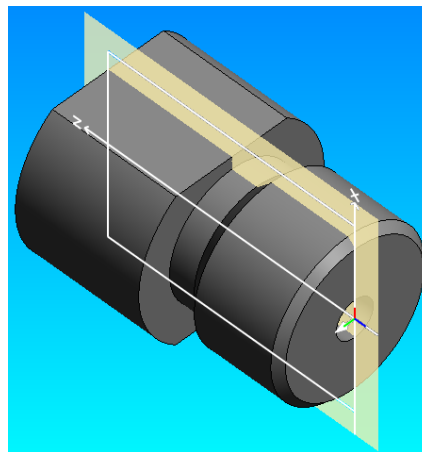
In this section, you will set up the **Face Window** for the turning operations on the main spindle. To do so:



- 1 When Programming with a Solid Model, many items will be set automatically from the information obtained from the Solid Model. To import settings into the Set-up Window, you will need to define a faceplane. Select the **Define Face Plane** icon from the Solids Tool Bar.
- 2 For Swiss and Turn-Mill parts, you should define your Face Plane at an Angle which will allow you to select the Points you will need for Turning in this first window. Check the **Set Face Plane** box. Choose **Parallel to ZX Plane** from the Drop Down Menu. Enter **45** into the **Turning Plane Angle** field and click the **Set Boundaries** checkbox.



- 3 Click the **<Preview>** button to see the Face Plane on the Solid Model update.

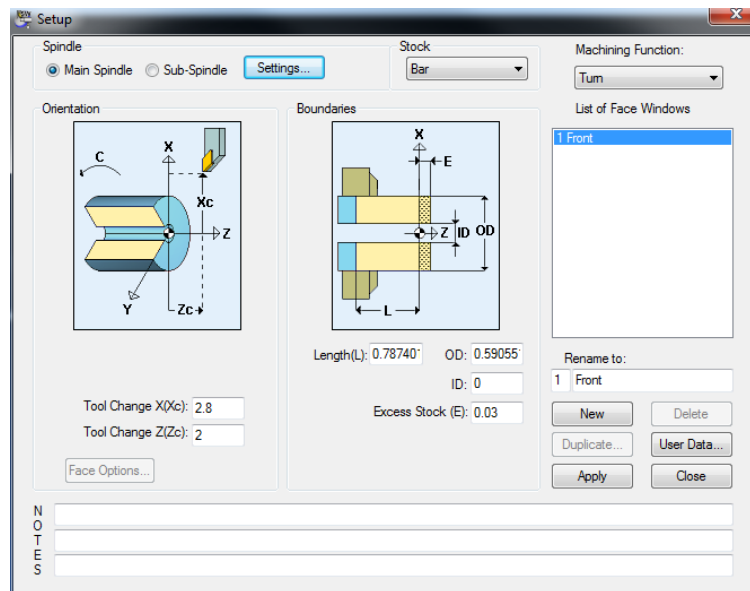


- 4 Click **<OK>**

- 5 Open the Set-up Window. Note the Part Length and OD were imported from the Solid Model. You should now make any changes required for your Setup dialog. Make the following changes to the Setup Dialog.

Spindle:	Main Spindle
Stock:	Bar
Machining Function:	Turn
OD:	0.59055118 (For Stock Size)
Excess Stock:	.03

Your Set-up Window should appear as below:



- 6 Click <Apply> and <Close>

Programming a Face Operation

- 1 Select the **New Profile Group** icon.
- 2 Change the Following settings in the Profile Group Parameters dialog and Leads Out, Finishing dialog

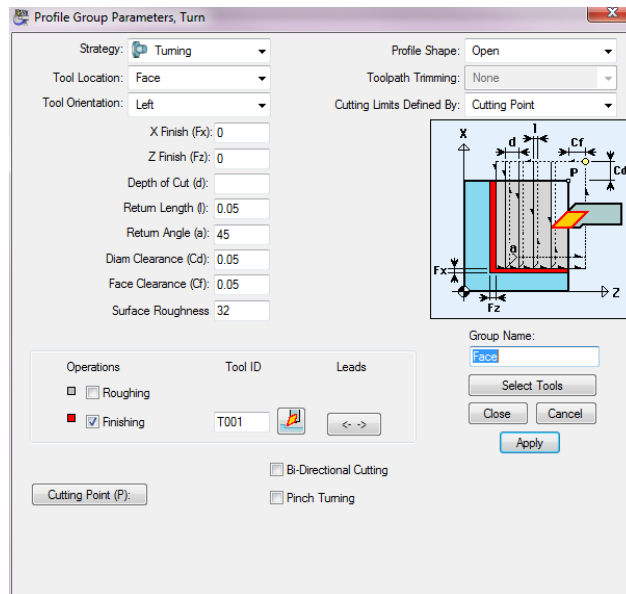


Strategy	Turning
Tool Location	Face
Tool Orientation	Left
Remove Check	Roughing
Check	Finishing
Lead Out Angle (A2)	90 Degrees (click Leads button)

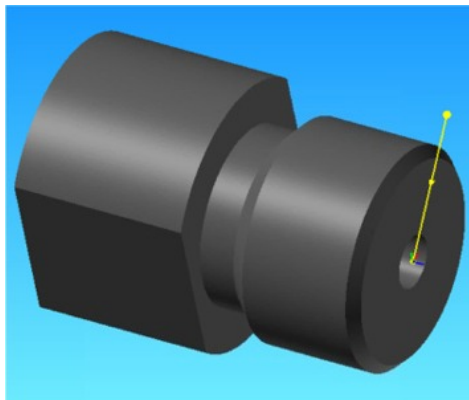
- 3 Click **<Select Tools>** and choose the OD Turn - 55 Turning Tool.

ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T001	1/Turn	OD Turn - 55	HSS	1	1.1	55	95	0.1

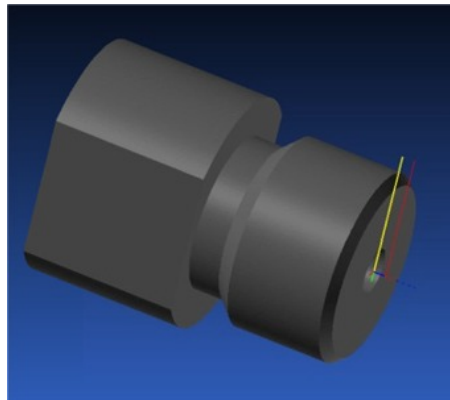
- 4 Name the Group **Face**. Your dialog should look like the following.



- 5 Click <Apply> and <Close>
- 6 You should now see a Facing Profile created on the Solid Model.



- 7 To verify the tool path for the Profile Group, Select the **Verify Work Group Toolpath** icon.
- 8 The Verify Options Dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**.

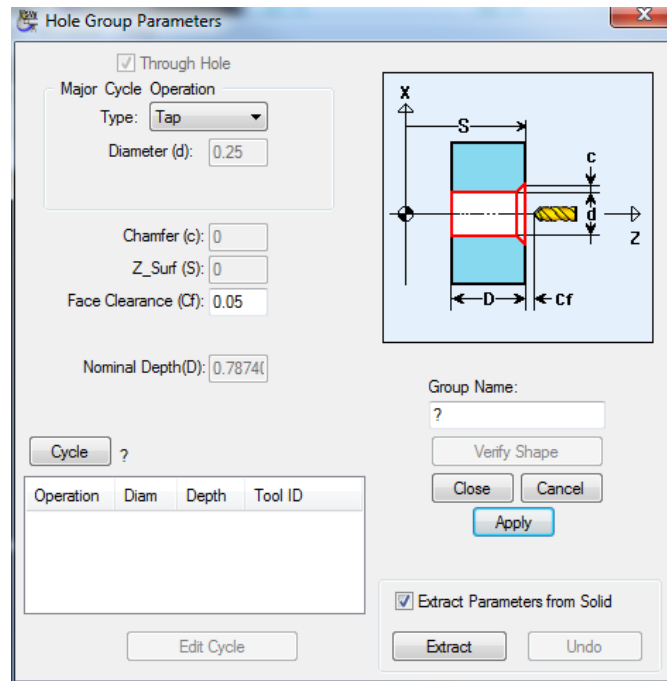


- 9 Once the tool path has been verified, you can hide by using the **Hide Work Group Toolpath** icon

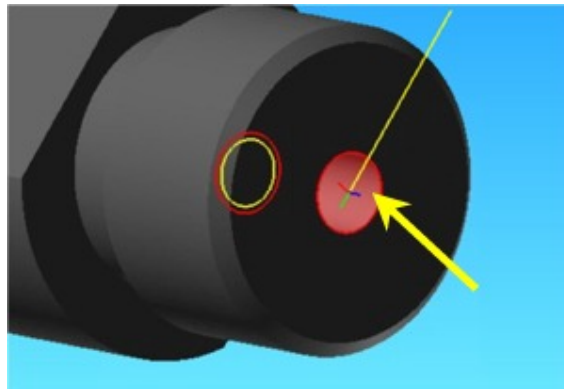


Programming a Tapping Cycle Operation

- 1 Choose a new color for this operation in the upper right corner of the screen
- 2 Select the **New Hole Group** icon.
- 3 Select **Tap** from the **Type** drop down menu.
- 4 Click the Extract Parameters from Solid checkbox.



- 5 Click on the Major Hole Diameter.



- 6 Click the **<Extract>** button to extract the feature data from the Solid Model:

- 7 Highlight and Select the **6-40 Tap** Cycle.

Select Cycle

Major Cycle Operation

Type: Tap

Diameter: 0.11811024

List of Matching Cycles

6-40 Tap

Add New Cycle Select

Cycle Preview

Operation	Diam	Depth	Tool ID
Center	0.094		T003
Drill	0.094		T004
Tap	0.118		T005

Cancel

- 8 Click <Apply> and <Close>



- 9 To verify the Profile Group, select the **Verify Work Group Toolpath** icon.

- 10 The Verify Options Dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**



- 11 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.



Programming a Turning Operation

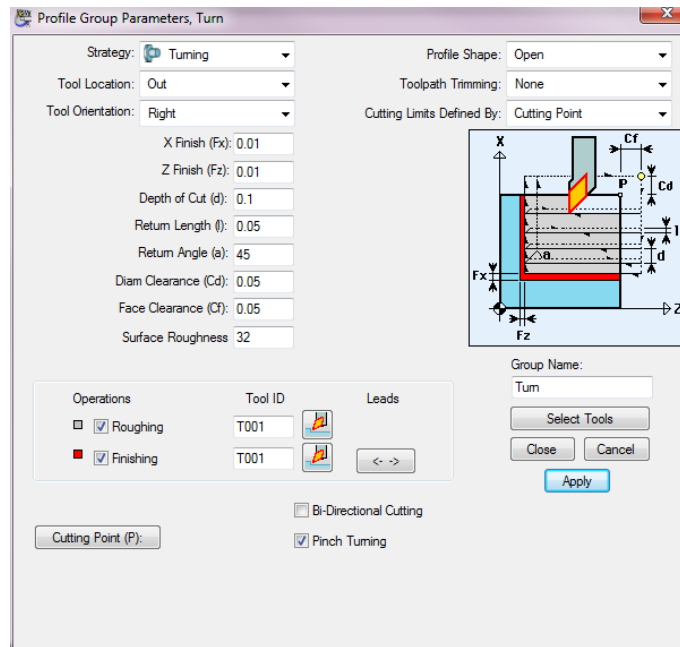
- 1 Choose a new color for this operation in the upper right corner of the screen.
- 2 Select the **New Profile Group** icon.
- 3 Change the following settings in the Profile Group.

Cycle Type	Turning
X Finish	.01
Z Finish	.01
Check	Finishing

- 4 Click the **<Select Tools>** button and choose the OD Turn - 55 Turning Tool for both the Roughing and Finishing Operations.

ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T001	1/Turn	OD Turn - 55	HSS	1	1.1	55	95	0.1

- 5 Name the Group **Turn**. Your dialog should look like the following.



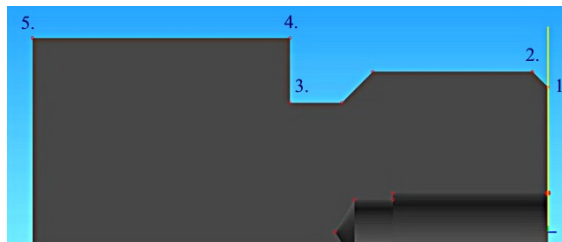
- 6 Click <Apply> and <Close>



- 7 Select **Define Profile on Solid Model** icon. The model will automatically section itself at the Angle chosen for the Face Plane.

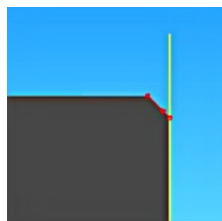


- 8 Next, choose the **Edge Endpoint Snap Mode**. This will show the selectable points on the Solid Model as shown here. The points are numbered for the following instructions.



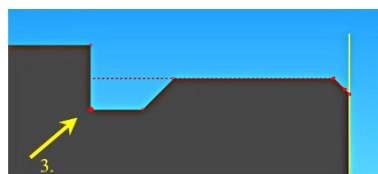
- 9 Select Point 1

- 10 Select Point 2



- 11 Select the Horizontal Constraint icon.

- 12 Select point 3



- 13 De-select the **Horizontal Constraint** icon if needed.

14 Select point 4

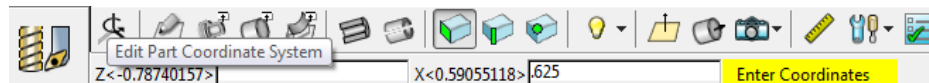
15 Select point 5



ZX

16 Select ZX Coordinates Snap Mode icon.

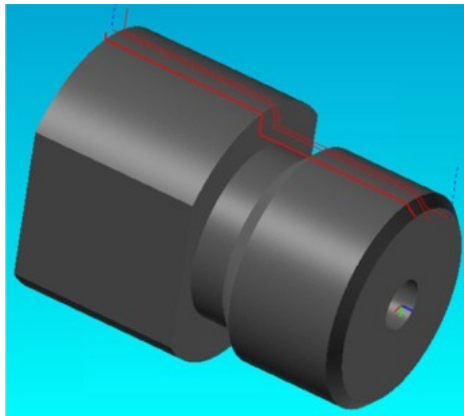
17 In the input field at the bottom of the screen, enter **.625** into the X and press the **<Enter>** key to accept.



18 Select the **Selection** icon.

19 To verify the Profile Group, Select the **Verify Work Group Tool path** icon

20 The Verify Options Dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**. The verification will appear as shown below:



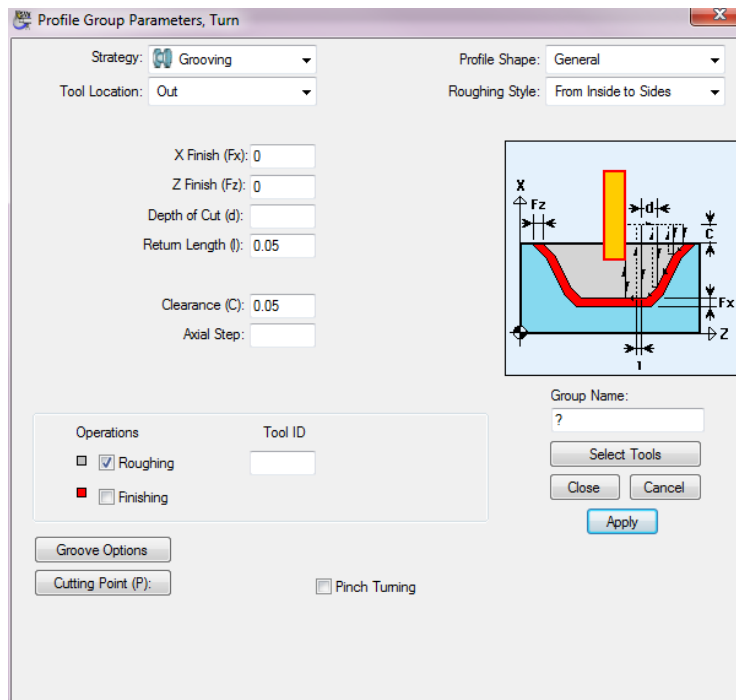
21 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

Programming a Grooving Operation

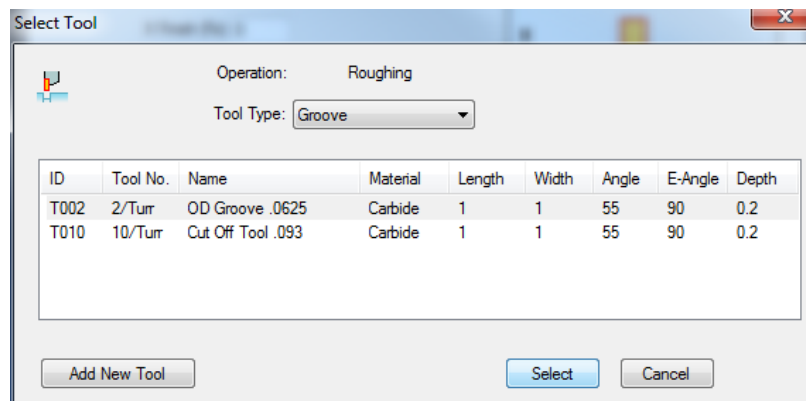


- 1 Choose a new color for this operation in the upper right corner of the screen.
- 2 Select the **New Profile Group** icon.
- 3 Change the following settings in the Profile Group.

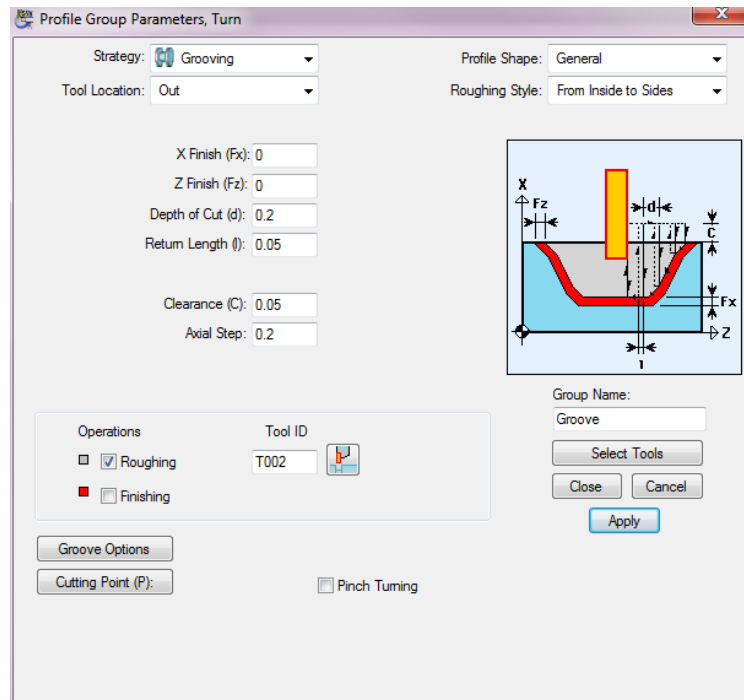
Strategy	Grooving
Profile Shape	General
Check	Roughing



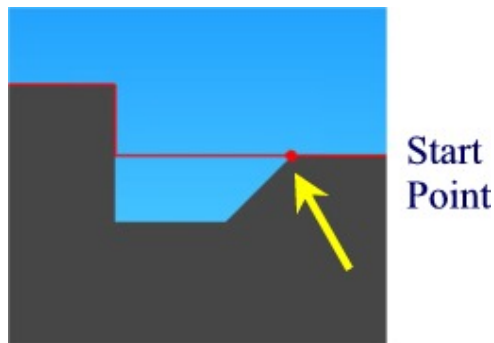
- 4 Click **<Select Tools>** and choose the OD Groove .0625 Grooving Tool



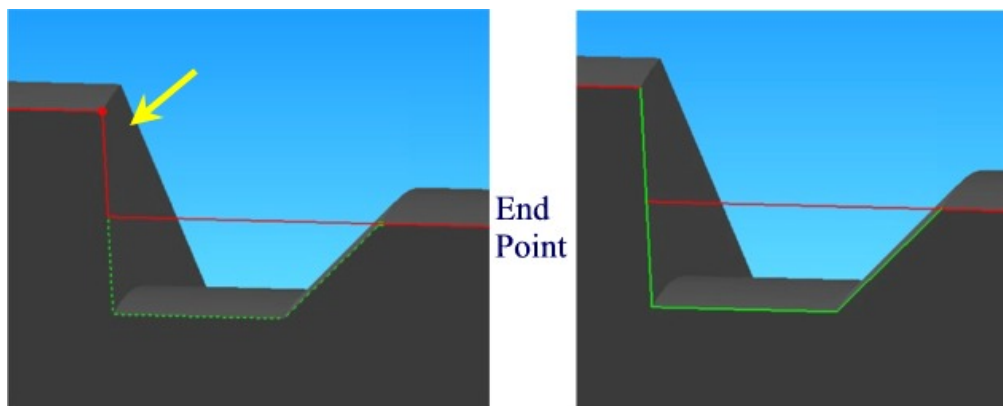
- 5 Name the Group **Groove**. Your dialog should appear as shown below:



- 6 Click the **<Apply>** and **<Close>** buttons
- 7 Select the Chain Profile on Solid Model icon.
- 8 Select the Start Point of the profile. When choosing start point, click on the side of the point that you want the Chain to follow. This will determine the direction the chain will follow.



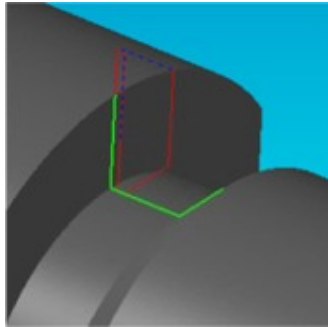
- 9 Select End Point of the profile.





10 To verify the Profile Group, Select the **Verify Work Group Toolpath** icon

11 The **Tool Path Verification Options** dialog will now display. Make sure to check the option to **Enable Verification on Solid Model**. The verification will appear as shown below:

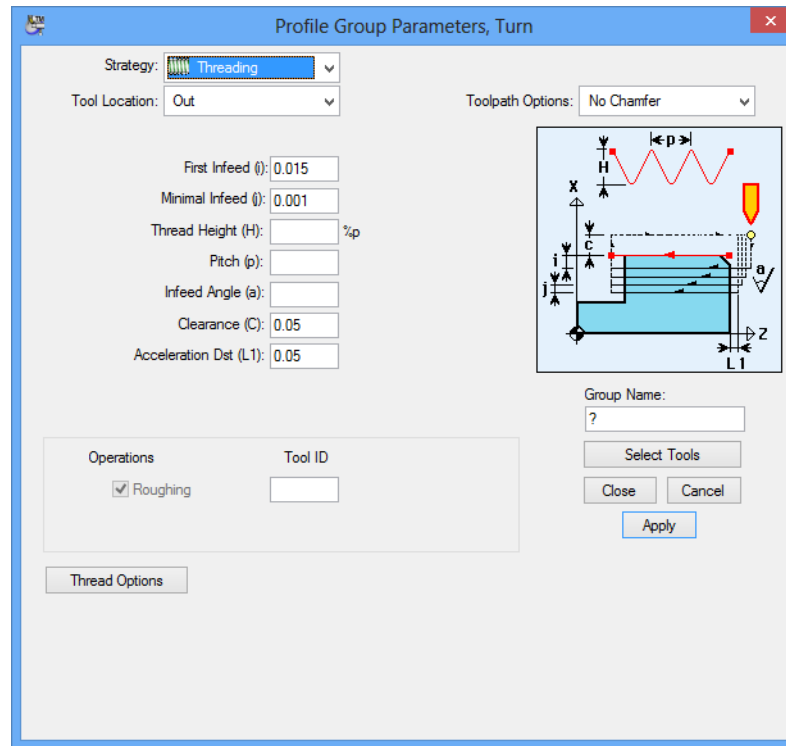


12 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

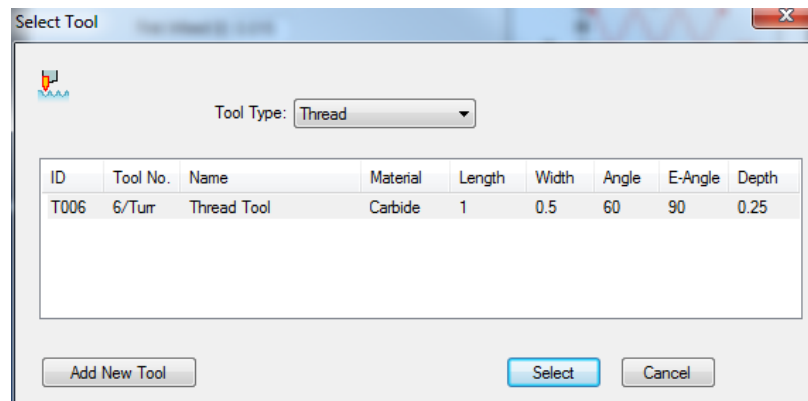
Programming a Threading Operation

- 1 Choose a new color for this operation in the upper right corner of the screen
- 2 Select the New Profile Group icon
- 3 Change the following settings in the Profile Group.

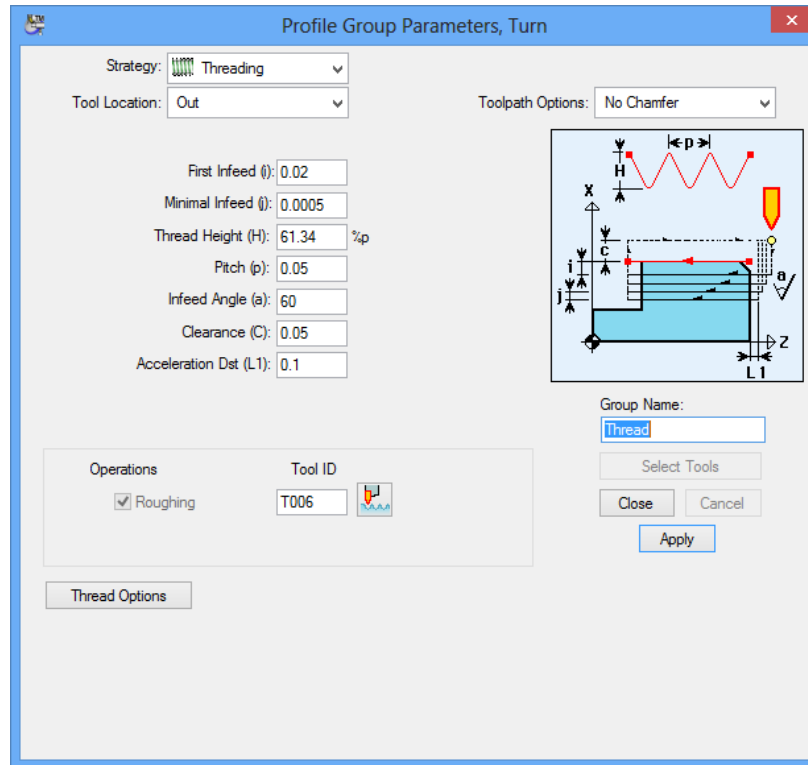
Strategy	Threading
Pitch for 1/20	.05
Acceleration Dst	.1



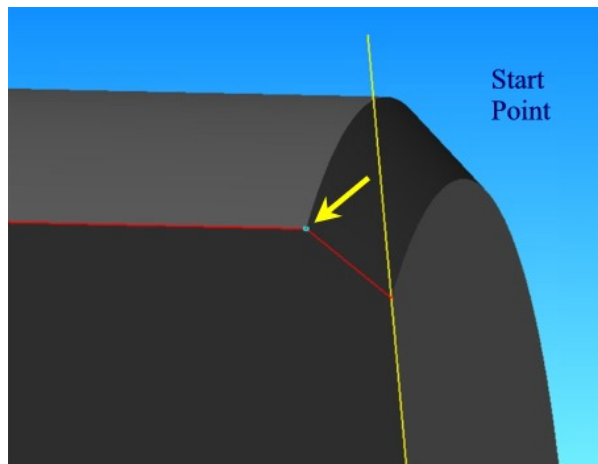
- 4 Click **<Select Tools>** and Choose the Thread Tool.



- 5 Name the Cycle **Thread**. Your dialog should look like the following:



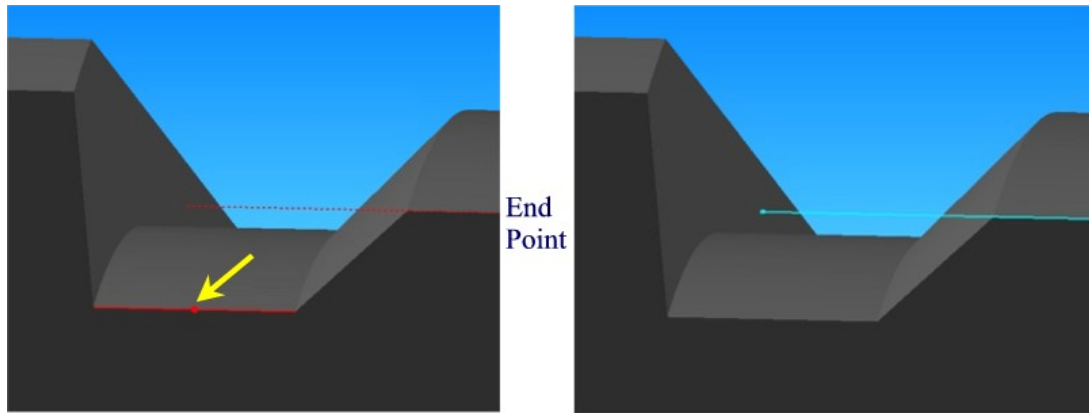
- 6 Click the **<Apply>** and **<Close>** button
- 7 Select Define Profile on Solid Model icon.
- 8 Choose the Edge End Point Snap Mode icon
- 9 Select the Start Point of thread



- 10 Select Edge Midpoint Snap Mode icon
- 11 Select the Horizontal Constraint icon



- 12 Select End Point from bottom of groove area.

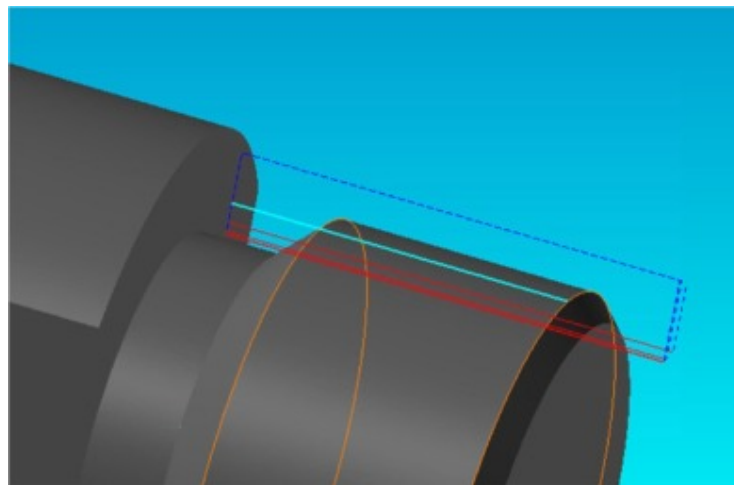


- 13 De-Select the Horizontal Constraint icon

- 14 Select the **Selection** icon

- 15 To verify the Profile Group, Select the **Verify Work Group Toolpath** icon

- 16 The **Tool Path Verifications Options** dialog will now display. Make sure to check the Option for **Enable Verification on Solid Model**. The verification will appear as shown below:



- 17 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.



Programming a Cut Off Operation



- 1 Choose a new color for this operation in the upper right corner of the screen
- 2 Select the New Profile Group icon
- 3 Change the following settings in the Profile Group dialog and Leads Out, Roughing.

Strategy	Cut Off
End Point X	.05
Lead Out Line Length	0

Profile Group Parameters, Turn

Strategy: **Cut Off**

Toolpath Options: **Insert Chamfer**

Cut-Off Distance (D): 0.787401

Chamfer OR Radius (q): 0

Start X Point (Xs): 0.295275

End X Point (Xe): 0.05

Diam Clearance (Cd): 0.05

Axial Step: 0

Group Name: ?

Operations: ☒ Roughing

Tool ID:

Leads: <- ->

Groove Options

Cutting Point (P):

☐ Optional Path 1->2->1

Select Tools

Close Cancel

Apply

- 4 Click **<Select Tools>** and choose the Cut Off Tool .093

Select Tool

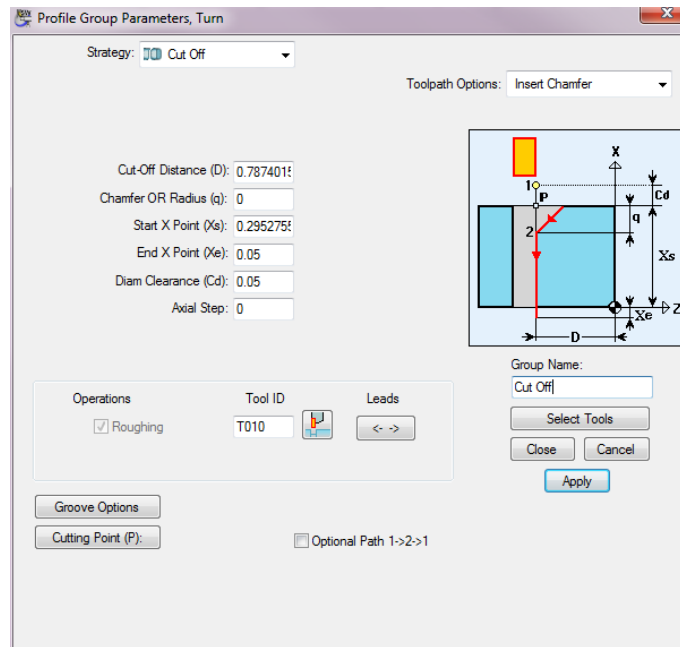
Tool Type: **Groove**

ID	Tool No.	Name	Material	Length	Width	Angle	E-Angle	Depth
T002	2/Turn	OD Groove .0625	Carbide	1	1	55	90	0.2
T010	10/Turn	Cut Off Tool .093	Carbide	1	1	55	90	0.2

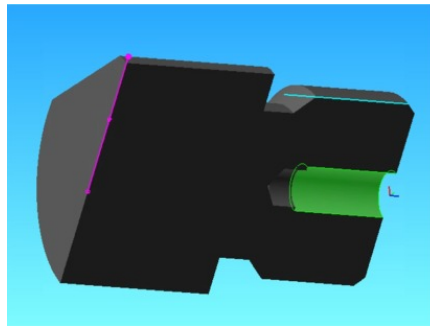
Add New Tool

Select Cancel

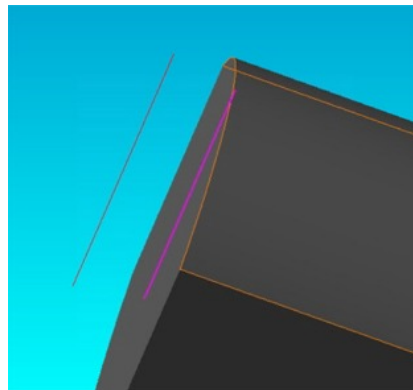
- 5 Name the Cycle **Cut off**. Your dialog should look like the following.



- 6 Click <Apply> and <Close>
7 The Profile will be created for you.



- 8 To verify the Profile Group, select the **Verify Work Group Tool path** icon
9 The **Tool Path Verifications Options** dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**.



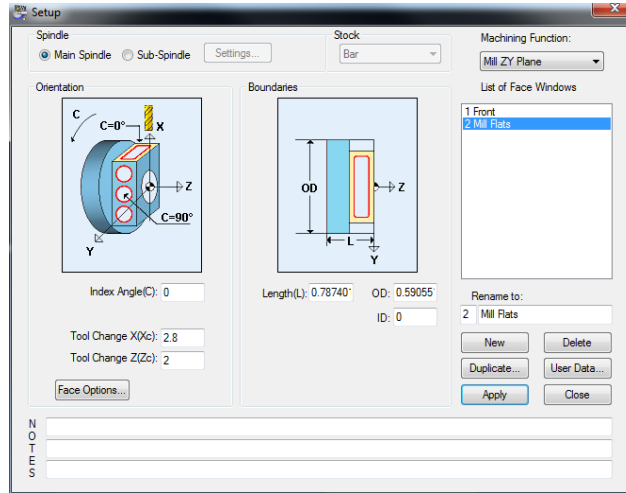
- 10 After Verifying the Tool Path, click the **Hide Every Tool Path** icon to hide all verifications.

Programming a Milling Operation

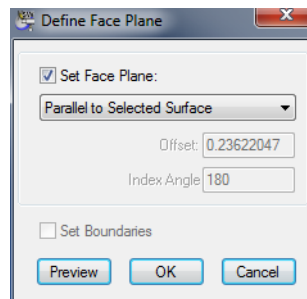
Note: This section only applies to **PartMaker Turn-Mill** and **SwissCAM** modules



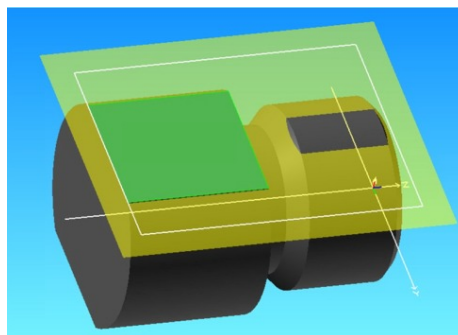
- 1 Select the **Setup Dialog** icon, then choose **<New>** to make a new Face Window.
- 2 Change the Machining Function to **Mill ZY Plane**. Name the window **Mill Flats**.



- 3 Click the **<Apply>** and **<Close>** button
- 4 If you have not done so already, choose the **Full Part View** icon from the **Solids** toolbar to make sure you have a complete solid model shown and not a sectional view
- 5 Select the **Define Face Plane** icon.
- 6 In the Drop down Menu, choose **Parallel to Selected Surface**.

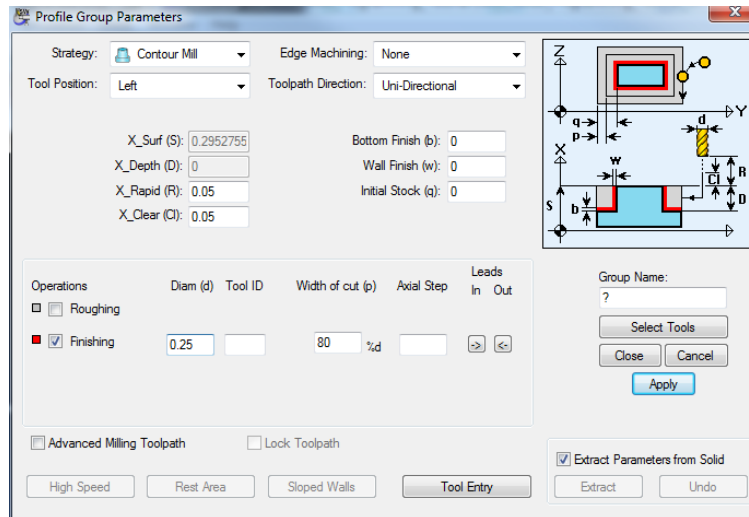


- 7 Click on one of the Flats. Note Face Plane is set coincident with the Plane of the chosen flat.

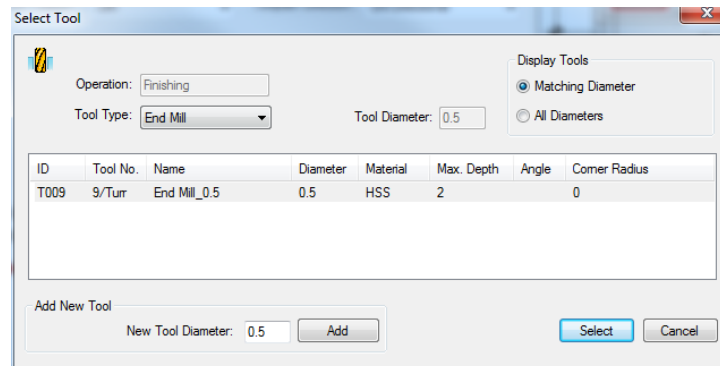


- 8 Click **<OK>**

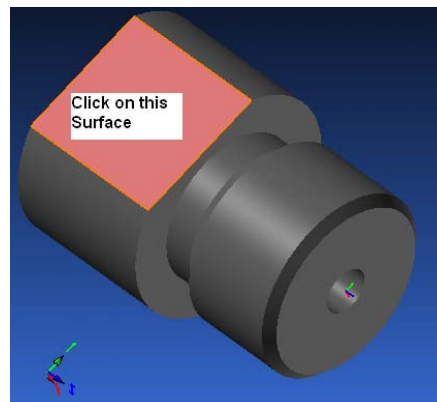
- 9 Choose a new color from the colors pallet in the upper right hand corner of the screen
- 10 Select the **New Profile Group** icon
- 11 Change the **Tool Position** to Left
- 12 Check the **Extract Parameters from Solid** check box as shown below:



- 13 Uncheck the **Roughing** box so that **Finishing** is checked
- 14 Enter 0.5 in the **Diam(d)** field and click **<Select Tools>**

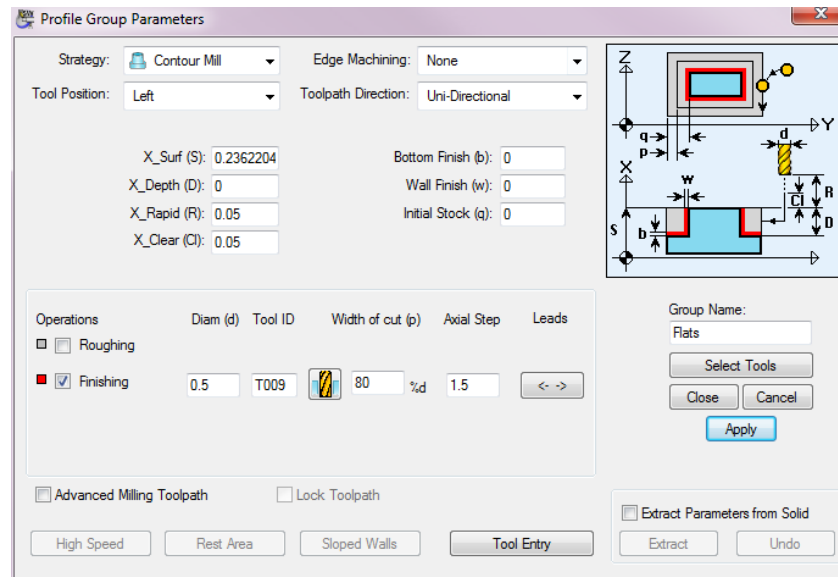


- 15 With the **Profile Group Parameters** dialog open, click on the plane as shown below:



- 16 Click the **<Extract>** button to extract the machining parameters directly from the model.
- 17 Rename the Group to **Flats**

18 The completed dialog will appear as shown below:

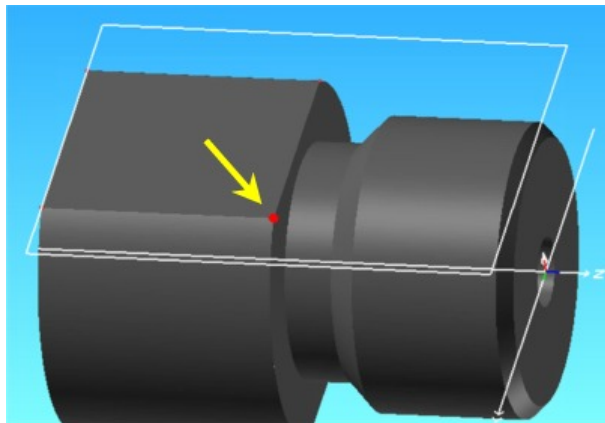


19 Click the **<Apply>** and **<Close>** button.

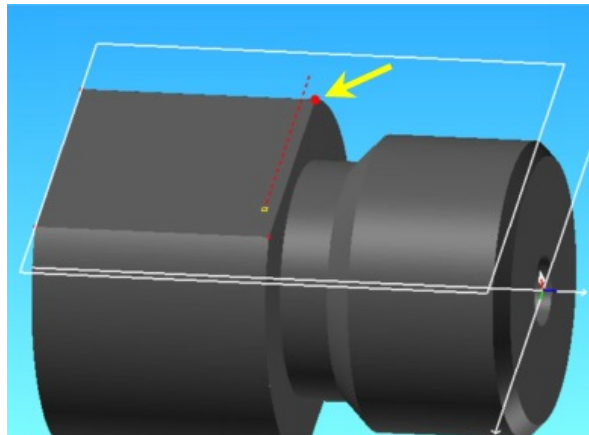
20 Choose the Define Profile on Solid Model icon

21 Select the Edge Endpoint Snap Point icon

22 Select the Start Point of the profile

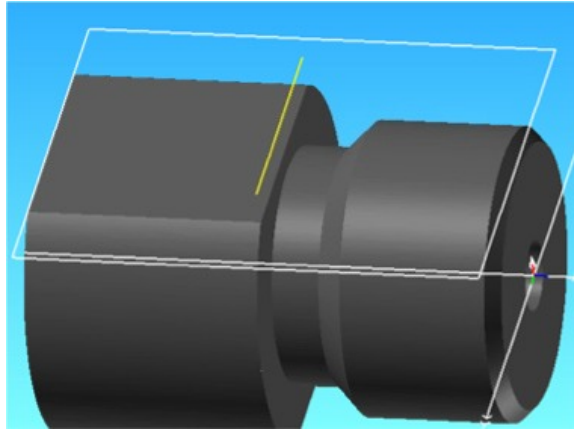


23 Select the End Point of the profile

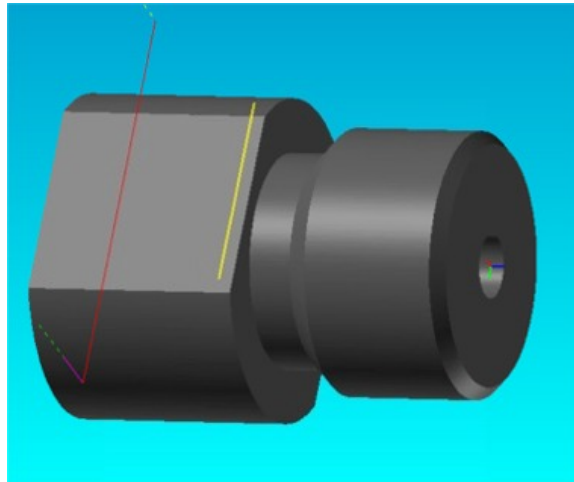




24 Choose the **Selection** icon to end the profile



25 To verify the Profile Group, select the **Verify Work Group Tool** path icon



26 After Verifying the Tool Path, click the **Hide Every Tool Path** icon to hide all verifications.

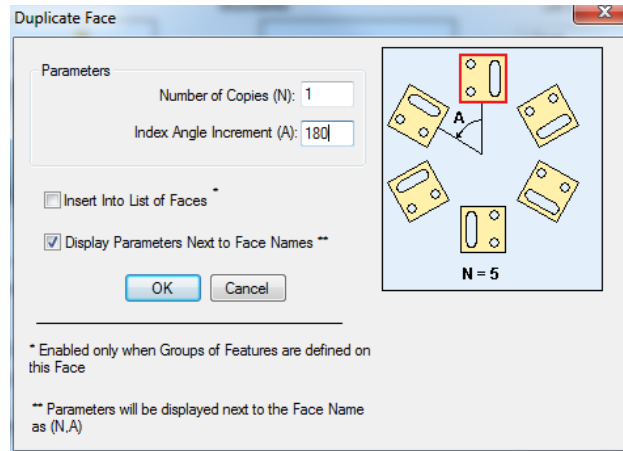
Duplicating the Face Window

Having programmed this milling path, you can now duplicate the window to create the second flat. This can be done as follows:

- 1 Open the **Setup Dialog** and click the **<Duplicate>** button
- 2 Enter the following into the **Duplicate Face** dialog.

Number of Copies	1
Index Angle Increment	180

The completed dialog should appear as shown below:

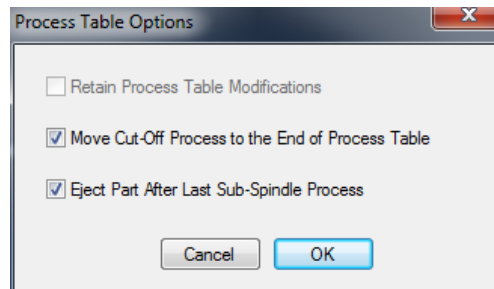


- 3 Click **<OK>**
- 4 Click the **<Close>** button to close the **Setup** dialog

Generating the Process Table



- 1 Choose the Generate Process Table icon
- 2 Click **<OK>** for the Process Table Options



3 Your Process Table should appear as shown below:

PartMaker - Process Table											
Simulation Tool Assembly Insert View Process Status Time Chart Synchronize											
Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group	
P01	T001	1/Turr	OD Turn - 55	Face	Front	0.0028upr	208fpm	0.07	M1S0		
P02	T003	3/Turr	Center Drill	6-40 Tap	Front	0.0022upr	4875rpm	0.03	M1S0		
P03	T004	4/Turr	Drill .098	6-40 Tap	Front	0.0023upr	4875rpm	0.05	M1S0		
P04	T005	5/Turr	Tap .118 6-40	6-40 Tap	Front	0.0250upr	1637rpm	0.02	M1S0		
P05	T001	1/Turr	OD Turn - 55	Turn	Front	0.0109upr	196fpm	0.06	M1S0		
P06	T001	1/Turr	OD Turn - 55	Turn	Front	0.0028upr	196fpm	0.15	M1S0		
P07	T002	2/Turr	OD Groove .06	Groove	Front	0.0015upr	399fpm	0.21	M1S0		
P08	T006	6/Turr	Thread Tool	Thread	Front	0.0500upr	896rpm	0.09	M1S0		
P10	T009	9/Turr	End Mill .05	Flats	Mill Flats	3.7upm	946rpm	0.70	M1S0		
P09	T010	10/Turr	Cut Off Tool J	Cut Off	Front	0.0019upr	399fpm	0.06	M1S0		

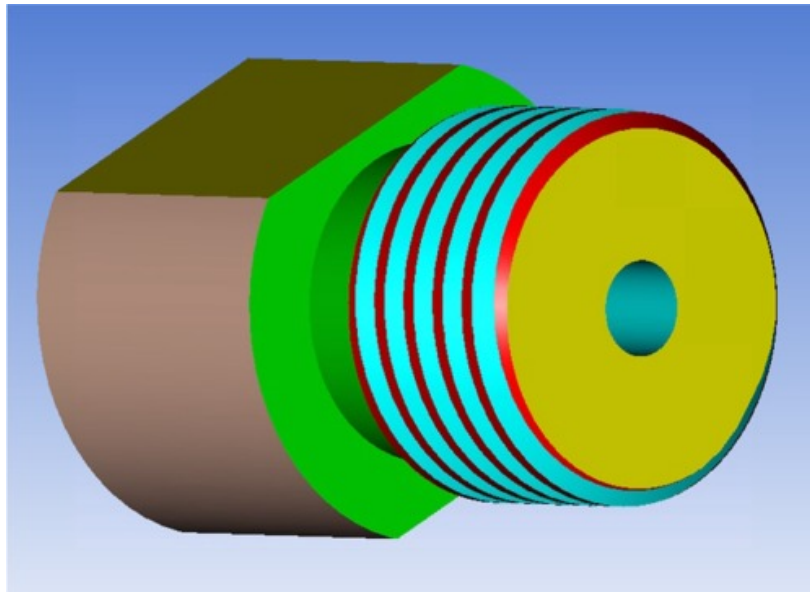
Material File: St_fmstwmdb Main Spindle Time: 1.43 min, Sub Spindle Time: 0.00 min. Total Time: 1.43 min.

Simulating Machining in 3D

Now that you have generated the Process Table, you can simulate the cutting of the part in 3D. To do so:



- 1 Choose the **Simulation** icon to launch PartMaker's 3D simulation
- 2 Press the **Start Simulation** button to watch the part being machined in 3D
- 3 Click the **Show Finished Part** icon to view the completed part:



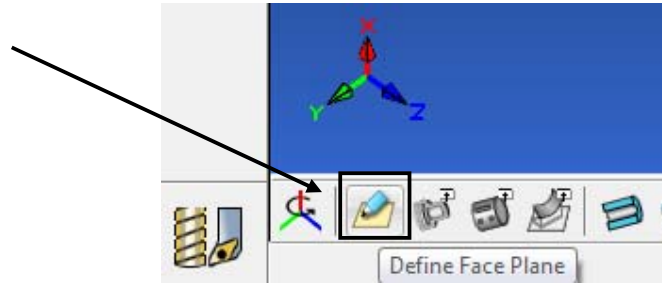
Other Samples

Turn Function Face Window Programming

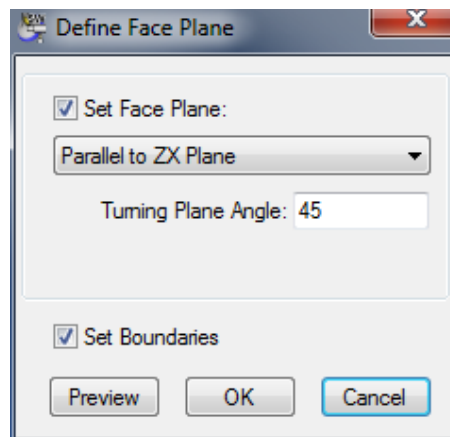
Describes Programming on a Solid Model in a Turn Function Face Window. This applies to PartMaker SwissCAM, TurnMill and Turn modules.

Setting the Face Plane

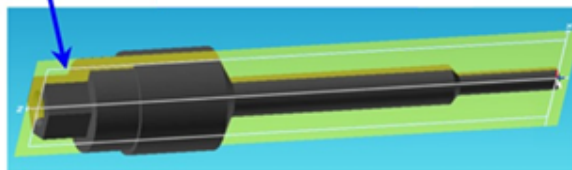
- 1 Open the Define Face Plane Dialog.



- 2 Set Face Plane as required to be able to Select Points on this plane for the Profile. Note the features on the Part when setting and use the Preview button to preview the changes.

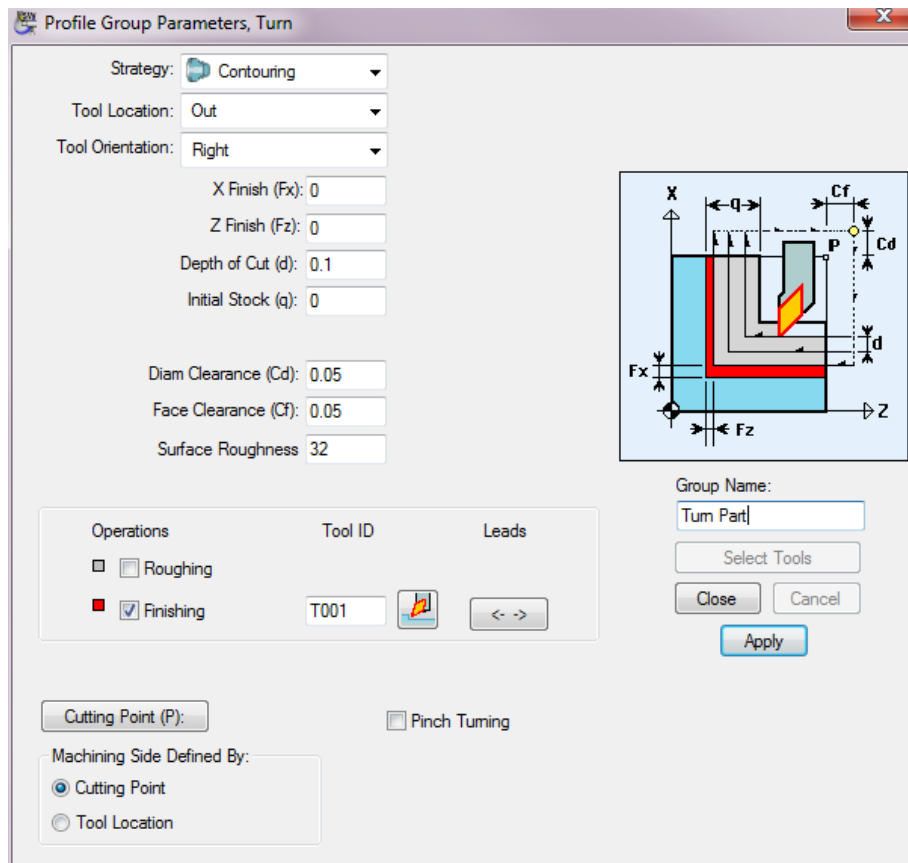


Note Features
Required for
Turning



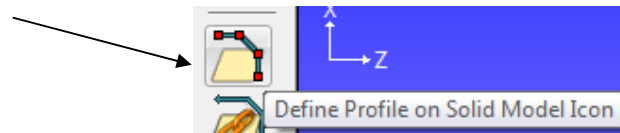
Creating the Profile Group

Create Profile Group as Normal

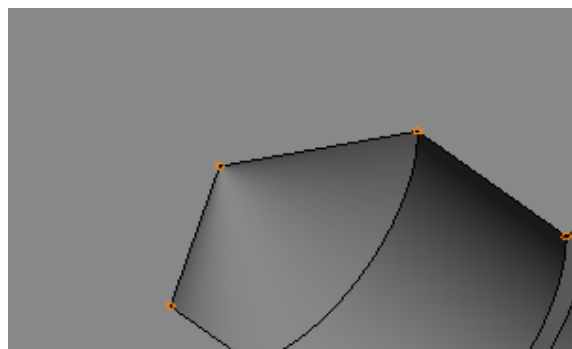


Selecting the Profile

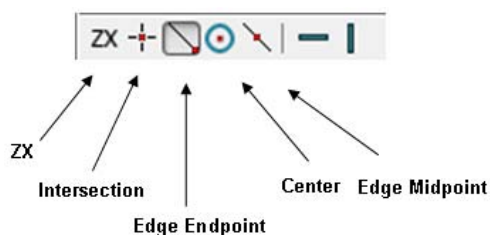
- 1 Select by choosing the Profile Points
- 2 Choose the **Define Profile on Solid Model Icon**



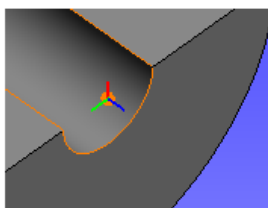
- 3 At this point you will see Red Points show on the Solid Model at the intersection of the Face Plane. Select these points to create your Profile.



- 4 Use the Snap Modes to choose the Profile Points on the Solid Model.

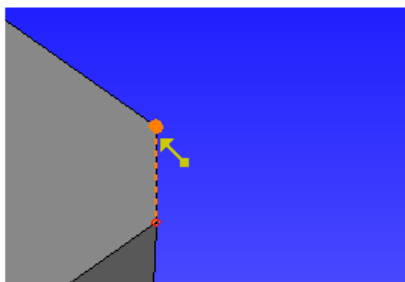


- 5 Click on the Point to Start the Profile.



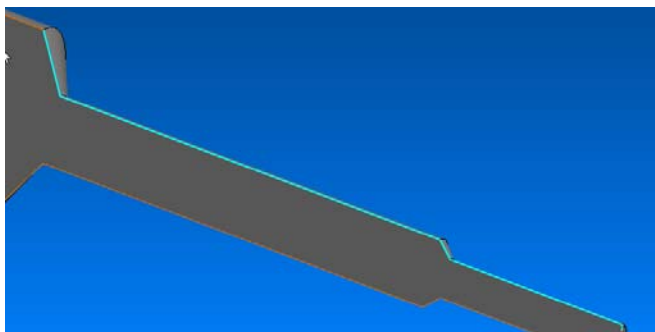
Starting Point at Center Using Edge Midpoint

- 6 Move the Mouse Cursor to the Next Point of the Profile. There will be a Dotted Line indicating Profile and a Dot where next Point will be. Click to connect the profile to this Point.

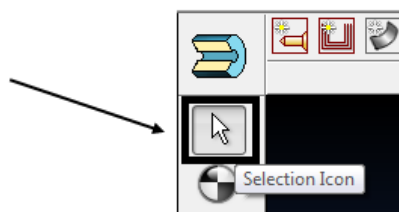


Next Point Select using Edge End Point

- 7 Continue selecting points for the Profile.



- 8 Click the **Selection** icon when done.



Selecting by Chain

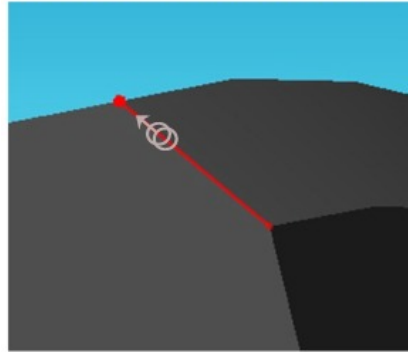
You may chain a profile on a Solid Model by selecting the **Chain Profile on Solid Model** icon.

To use this icon, you need to select the Starting Point and Direction.

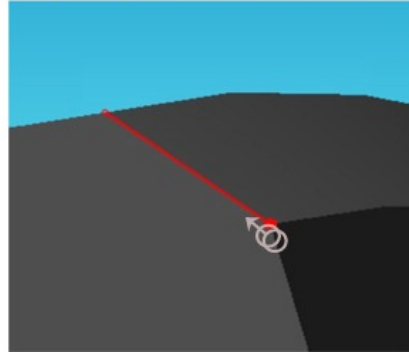


Note: Depending on where you place the mouse cursor, the starting point changes and the Direction Reverses.

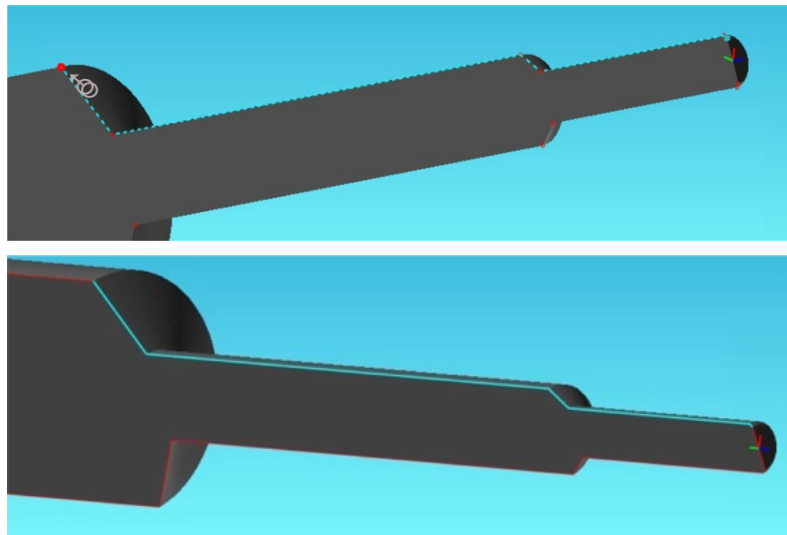
Incorrect Start Point and Direction



Correct Start Point and Direction



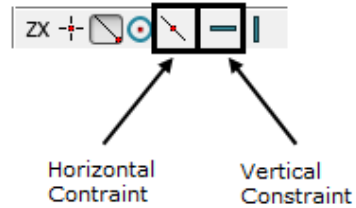
Select Endpoint of Chain. During Selection, a Dotted Line will indicate the Profile. Click the End Point to Finish Profile.



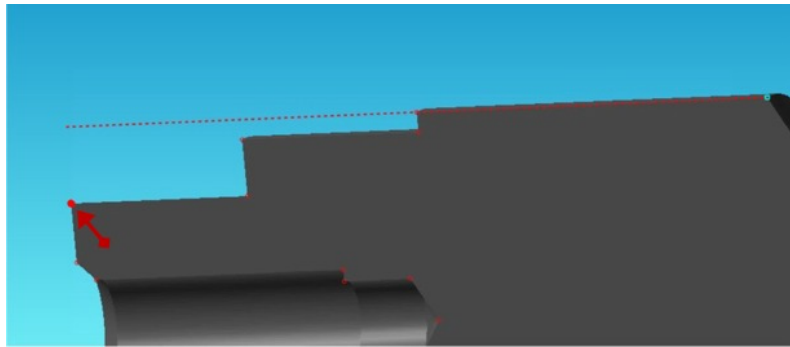
Using Horizontal and Vertical Constraints

You can use the Horizontal and Vertical Constraints to Extend Profiles that may not have a Point to choose from on the Model.

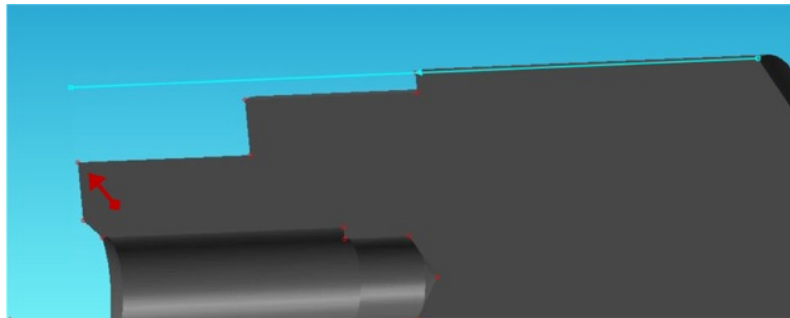
When selecting your Profile using the **Define Profile on Solid Model** icon, select either **Horizontal** or **Vertical Constraint** to lock the Profile in a Horizontal or Vertical move.



- 1 Move the mouse cursor over a Point that is not directly Horizontal or Vertical from the Currently Selected Point. You will see the Profile Extend to this point in a Horizontal or Vertical Line shown by a dashed line.



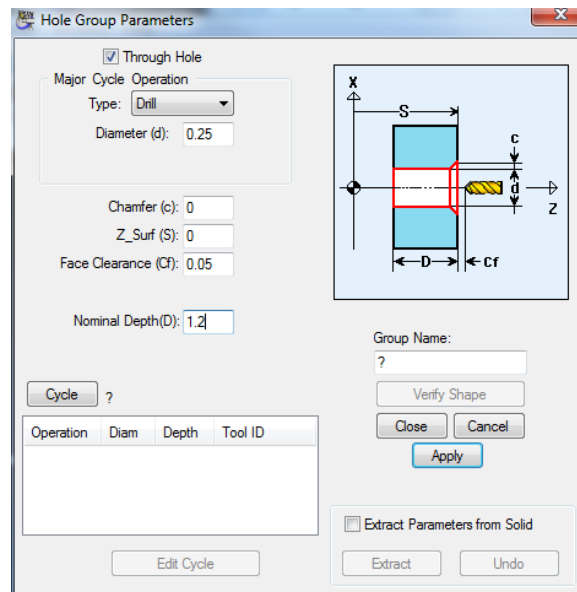
- 2 Click the Point to Extend the Profile to this Point.



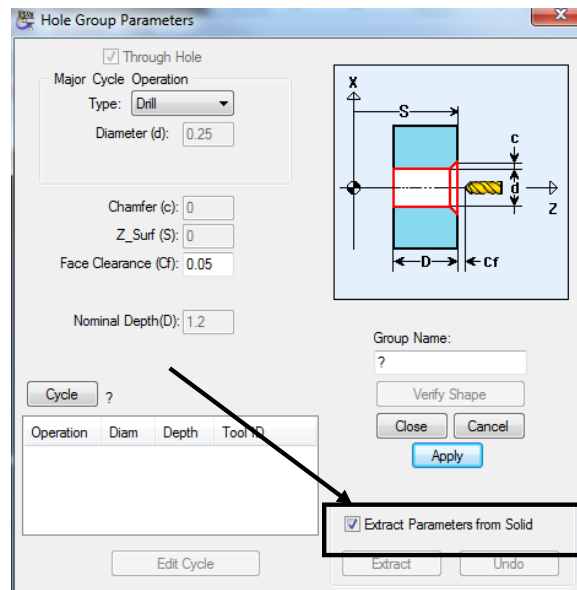
- 3 Depress the Horizontal or Vertical Constant icon to turn off the constraint.

Creating a Hole Group

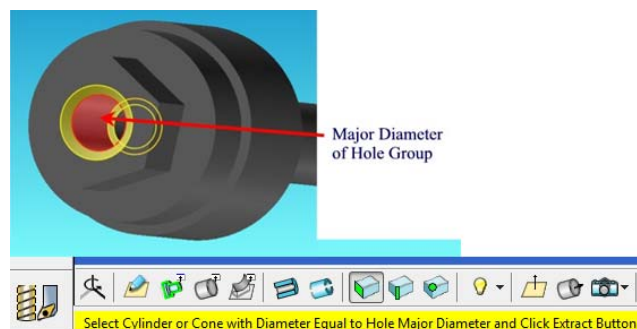
- 1 Create New Hole Group Parameters dialog



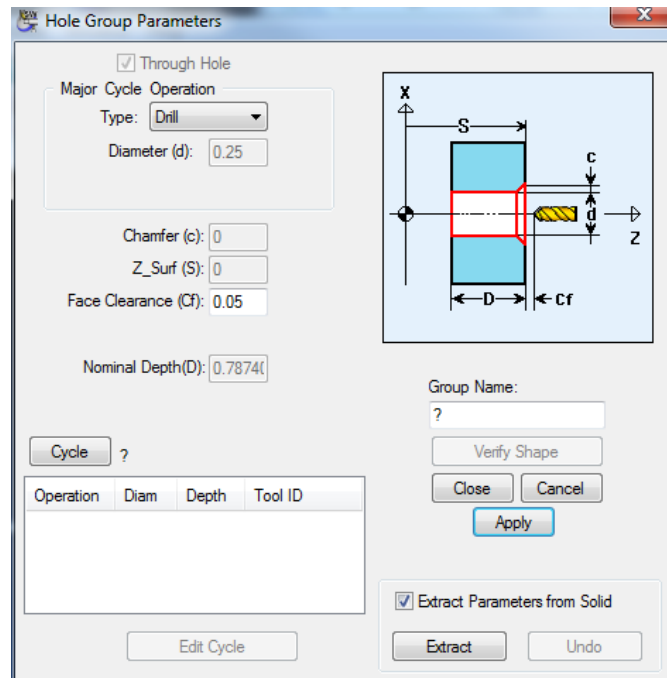
- 2 Check **Extract Parameters from Solid**.



- 3 Select Major Cylinder for Hole on Solid Model. (Note: See Explanation at Bottom of Screen)



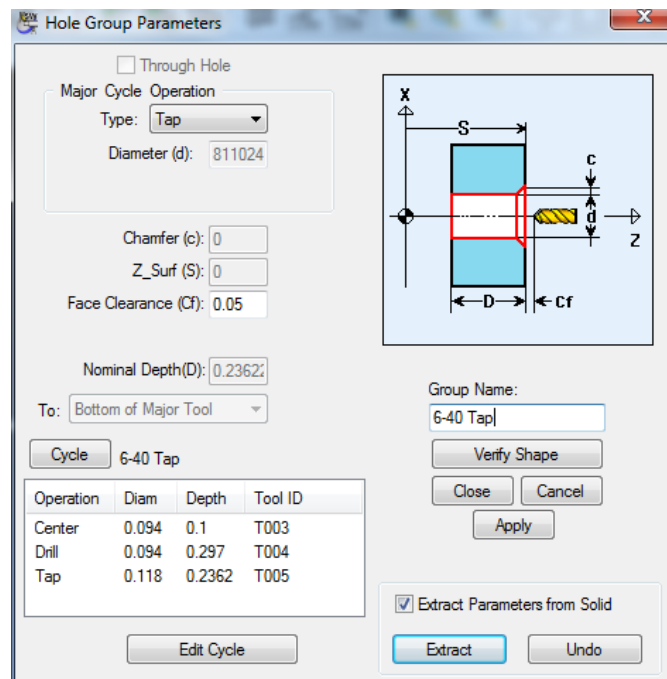
4 Click **<Extract>**



Remove Check from Extract Parameters from Solid if editing of a Hole Group is required.

5 Edit Hole Group as required

6 Click **<Cycle>** and Select the cycle.



7 Click **<Apply>** then **<Close>**

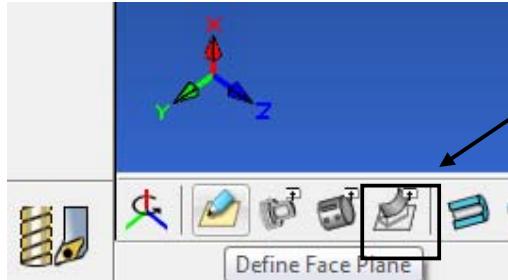
Milling Functions Face Window Programming

Describes Programming on a Solid Model in Milling Functions Face Windows. This applies to PartMaker SwissCAM and Turn-Mill.

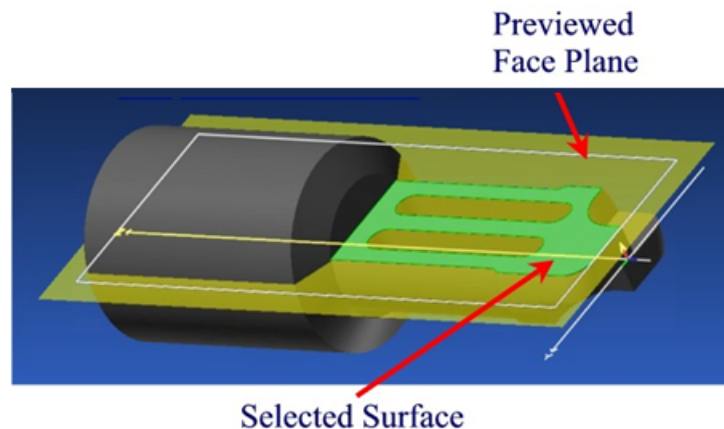
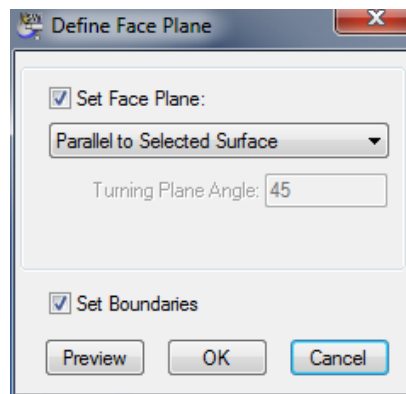
Creating a New Profile Group (Mill ZY window shown)

Setting the Face Plane

- 1 Click the **Define Face Plane** icon to open the dialog.



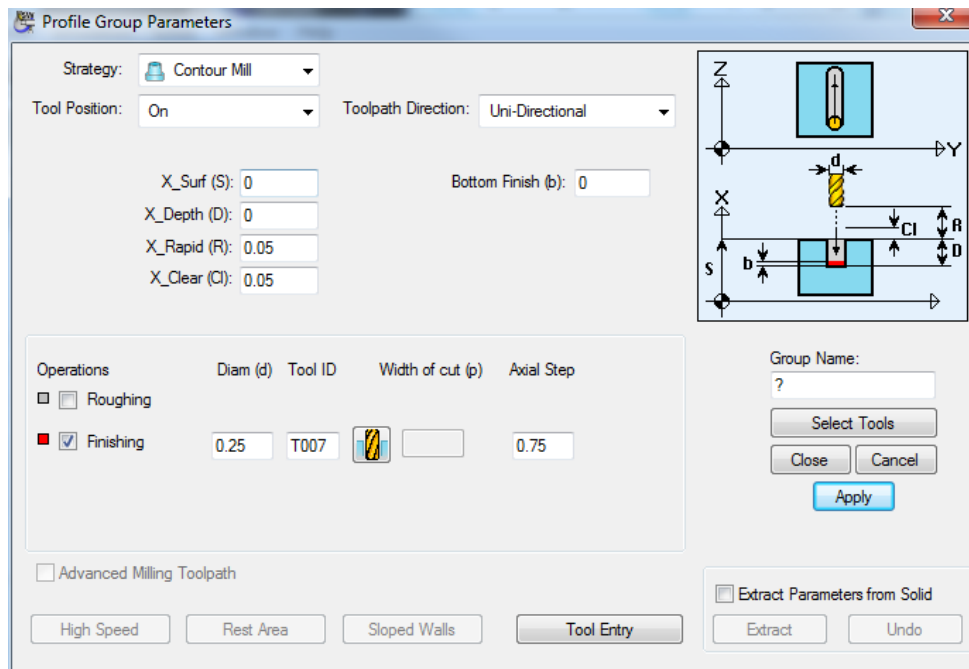
- 2 Choose desired method of Selecting the Face Plane. When choosing **Parallel to Selected Surface**, click on Surface to set as Face Plane. Use the Preview button to preview the changes.



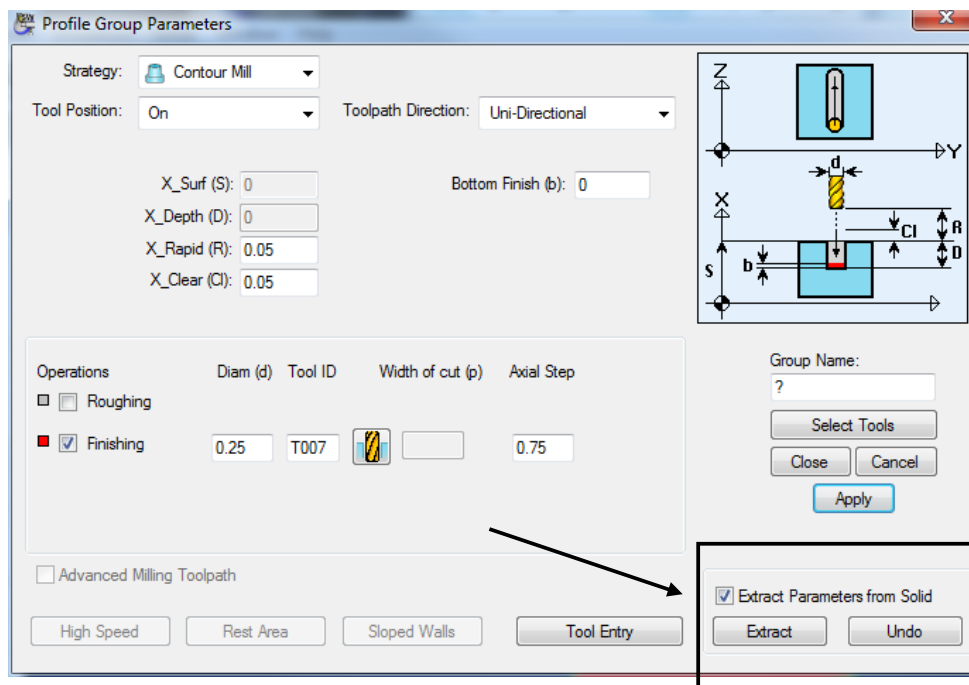
- 3 Click **<OK>** to accept Face Plane Setting. This will fill in the Set-up Window with the correct settings for the Face Plane.

Creating the Feature

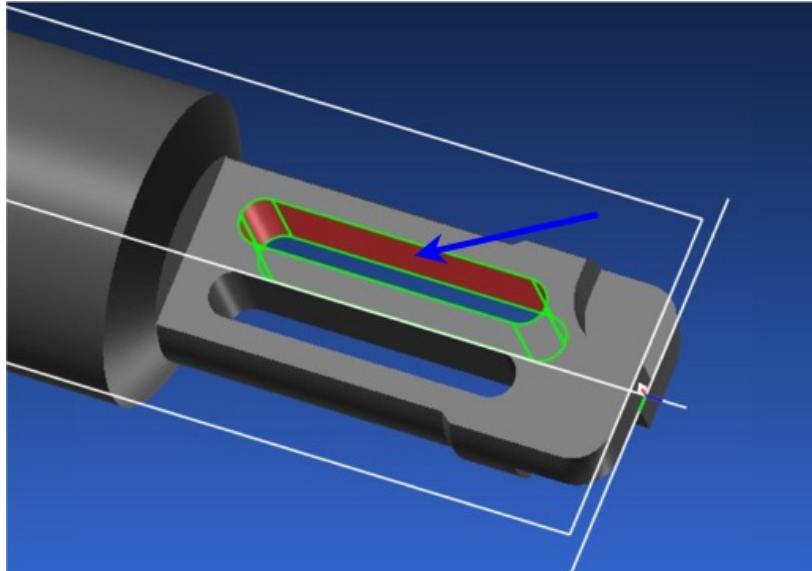
1 Create Profile Group as Normal



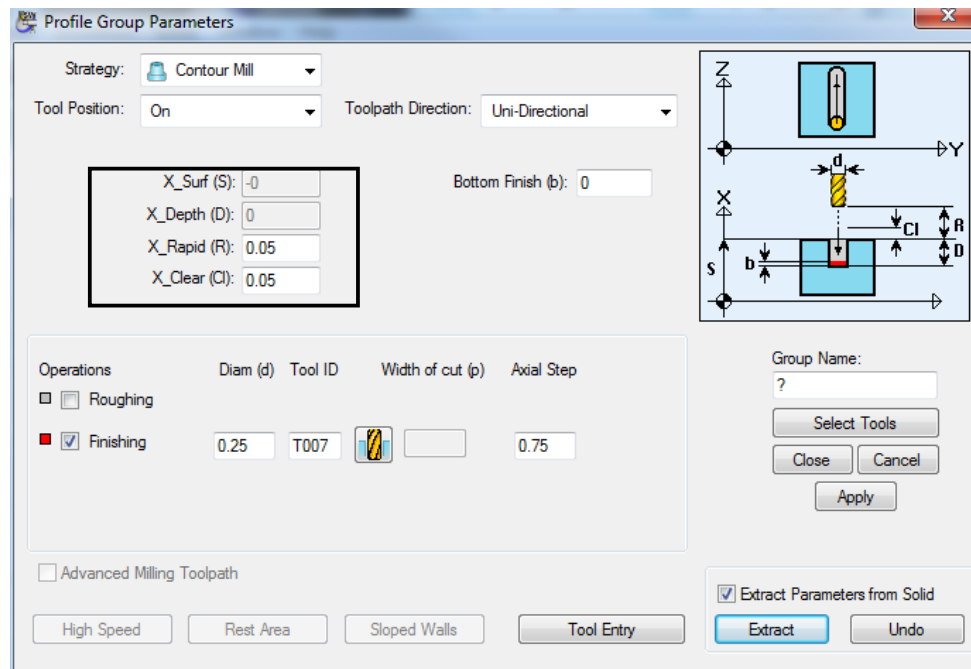
2 Check **Extract Parameters from Solid**



- 3 Select Feature to Extract Properties from.

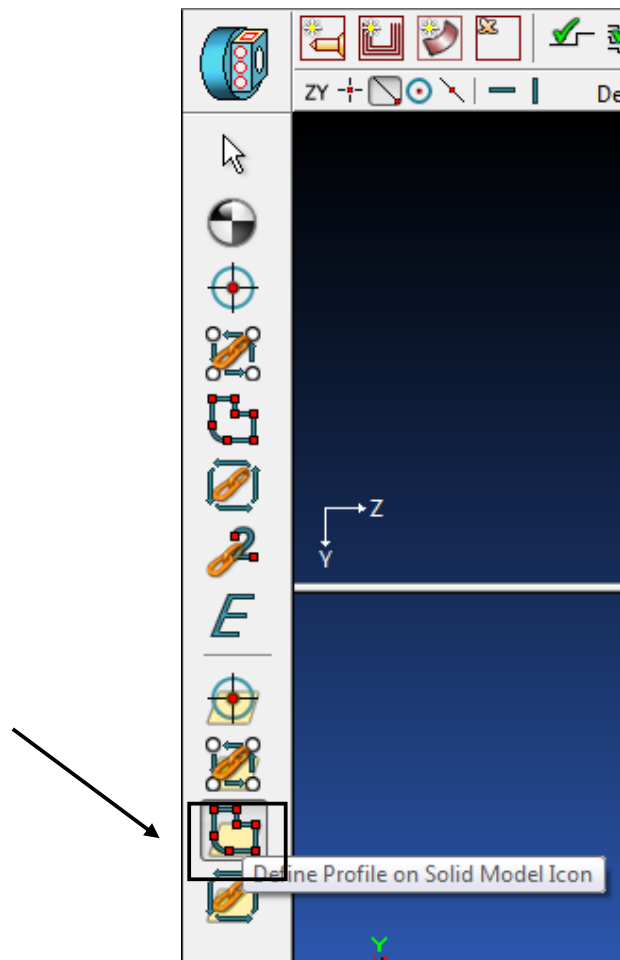


- 4 Click <Extract>. This will extract the information to the Profile Group.

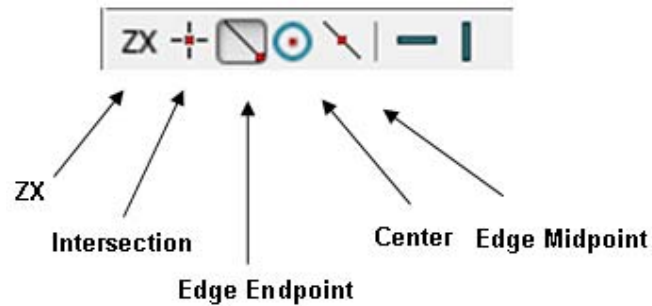


Creating the Profile

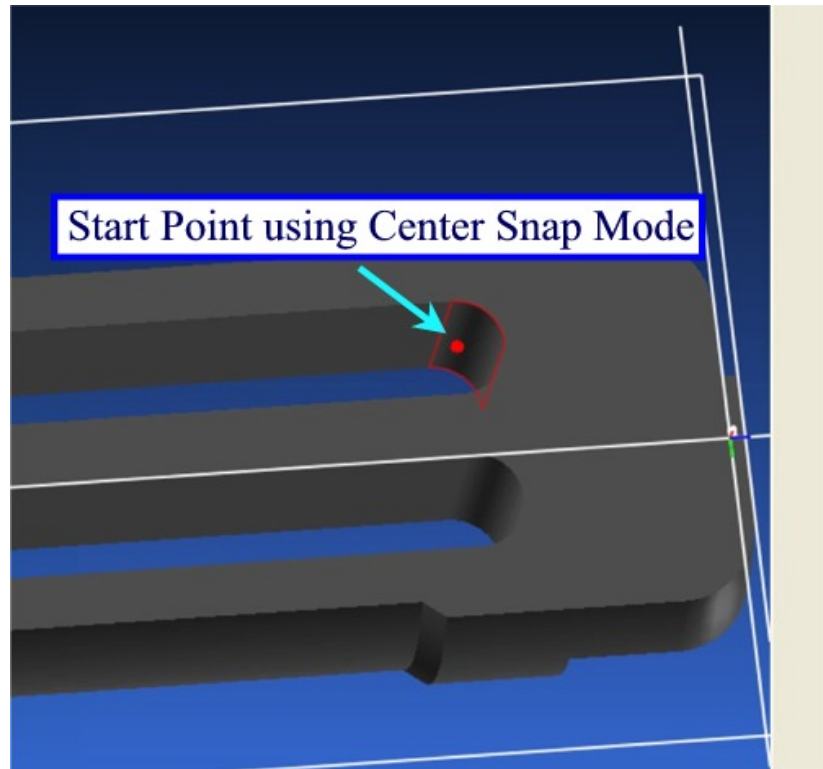
- 1 Choose the **Define Profile on Solid Model Icon**



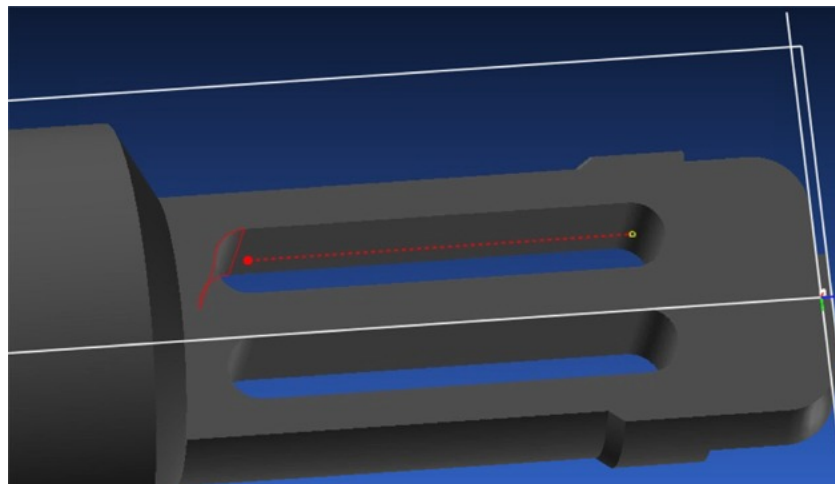
- 2 Use the Snap Modes to choose the Profile Points on the Solid Model.



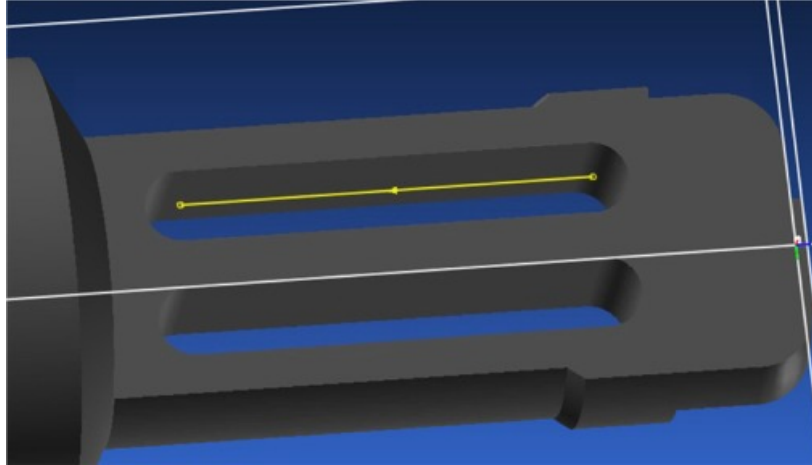
- 3 Click on the Cylinder Surface to select the center point as the Start of the Profile.



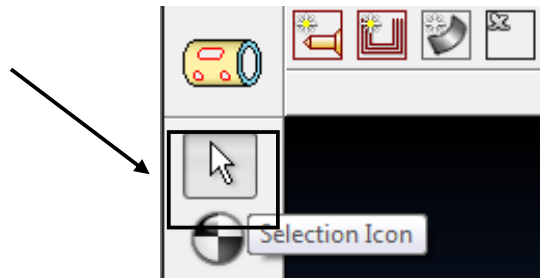
- 4 Move the Mouse Cursor to the Next Point of the Profile. There will be a Dotted Line indicating Profile and a Dot where next Point will be. Click to connect the profile to this Point.



- 5 Continue selecting points for the Profile until finished.



- 6 Click the Selection Icon when done.



Selecting by Chain

You can chain a Profile on a Solid Model by selecting the Chain Profile on Solid Model icon.

- 1 Select Starting Point and Direction



Note: Depending on where you place the Mouse Cursor, the Starting Point changes and the Direction Reverses.

- 2 Select Endpoint of Chain. During Selection, a Dotted Line will indicate the Profile. Click the End Point to Finish Profile
- 3 You may use Horizontal and Vertical Constraints if required.

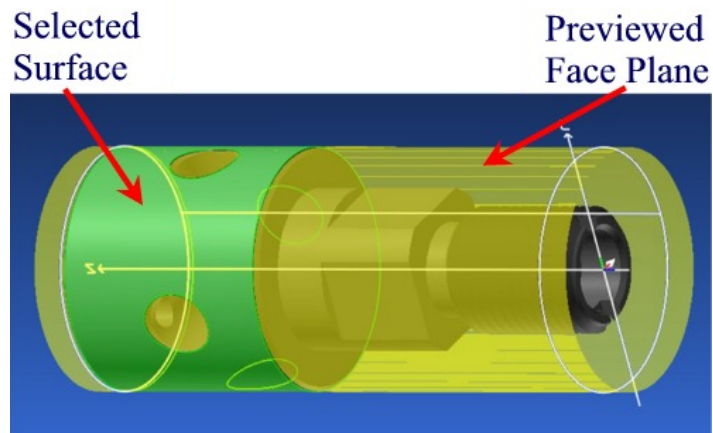
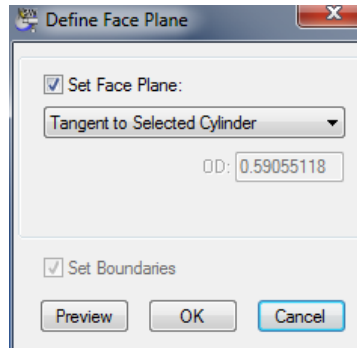
Creating a Hole Group (Mill Diameter Index Face window shown)

Setting the Face Plane

- 1 Open the Define Face Plane Dialog.



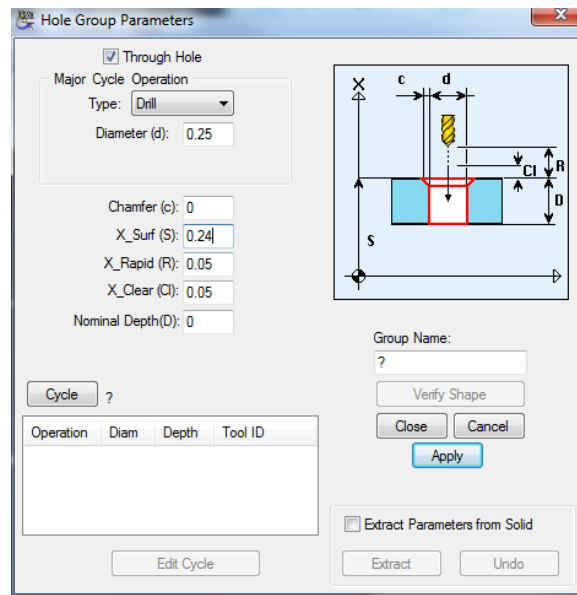
- 2 Choose desired method of selecting the Face Plane. When choosing **Tangent to Selected Cylinder**, click on the Cylinder to set as Face Plane. Use the **<Preview>** button to preview the changes.



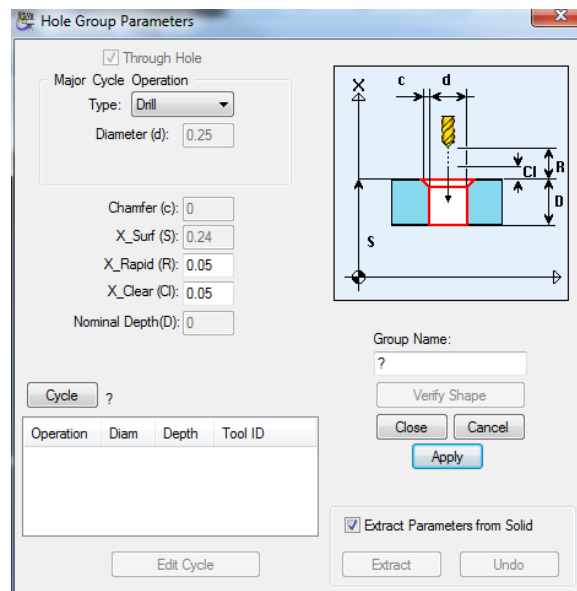
- 3 Click **<OK>** to accept Face Plane Setting. This will fill in the Set-up Window with the correct settings for the Face Plane.

Creating the Feature

- 1 Create New Hole Group Parameters.

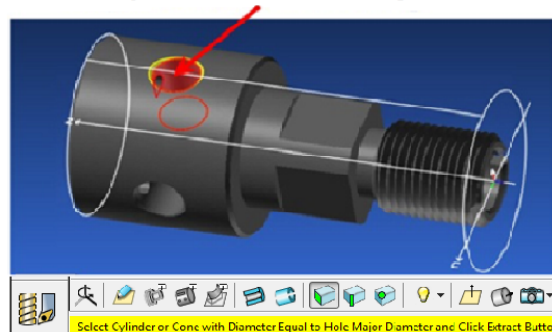


- 2 Check **Extract Parameters from Solid**

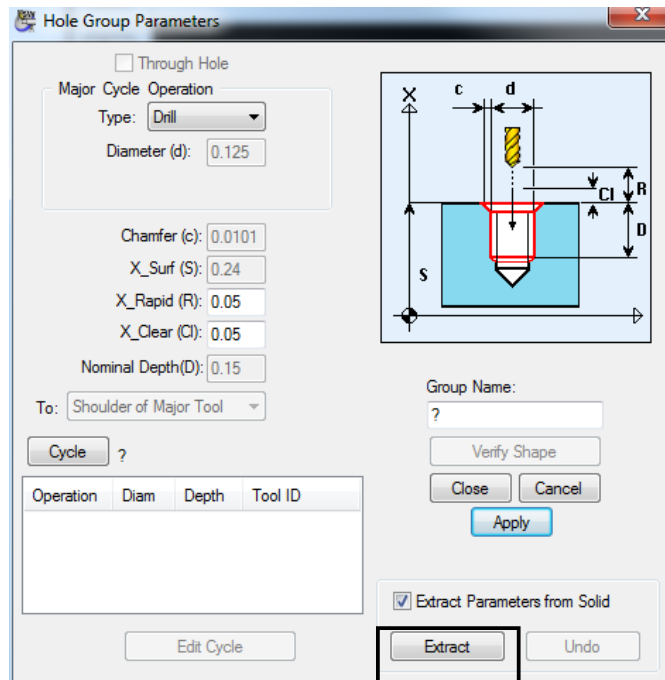


- 3 Select **Major Cylinder for Hole on Solid Model**. (Note: See Explanation at Bottom of Screen)

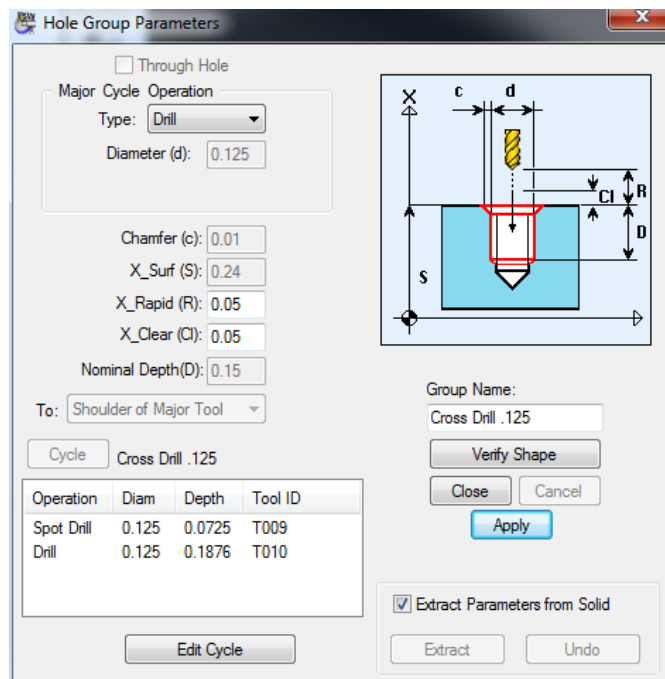
Major Diameter of Hole Group



- 4 Click **<Extract>**



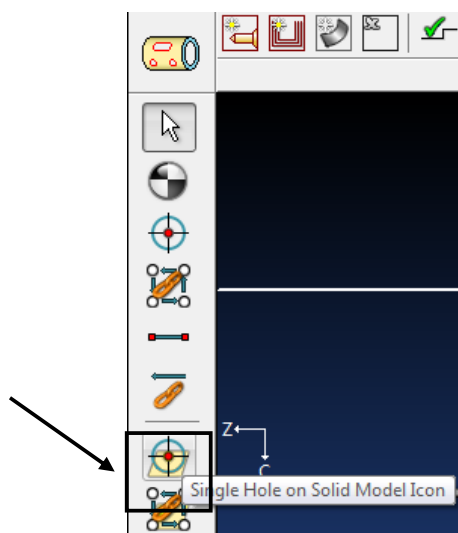
- 5 Remove the check from the **Extract Parameters from Solid** box if editing of a Hole Group is required
- 6 Edit Hole Group as Required
- 7 Click **<Cycle>** and select a cycle



- 8 Click **<Apply>** then **<Close>**

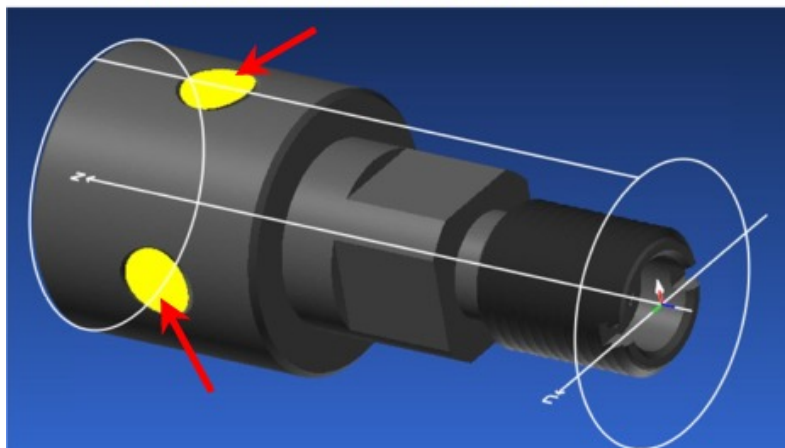
Selecting Single Holes

- 1 Choose the **Single Hole on Solid Model Icon**.



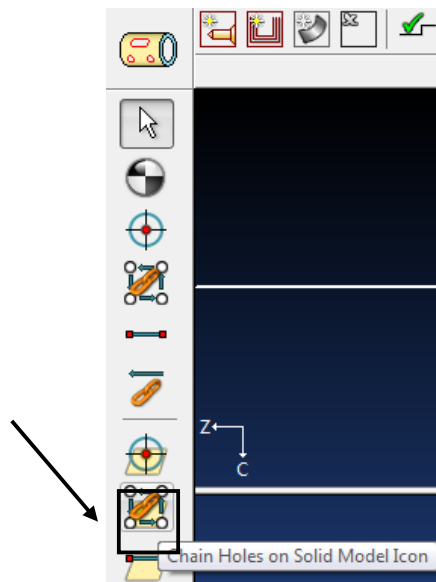
- 2 Select each hole that will be drilled.

Click at each Hole to be Drilled



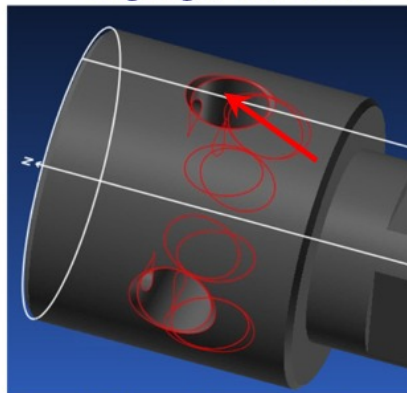
Selecting by Chain

- 1 Choose **Chain Holes on Solid Model** icon

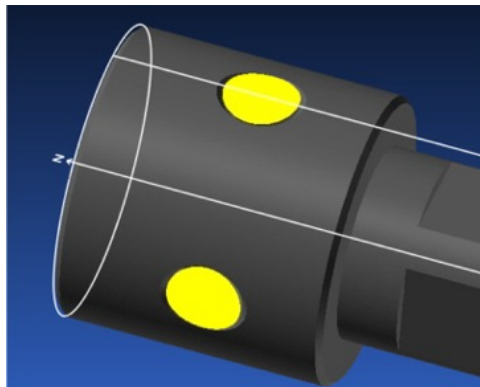


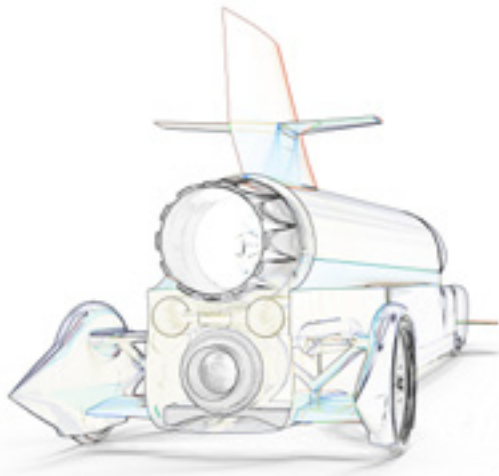
- 2 Select First Hole, PartMaker will then automatically select holes with the same feature in the Face Plane.

Highlight first Hole



Note: Holes with same features automatically become Highlighted

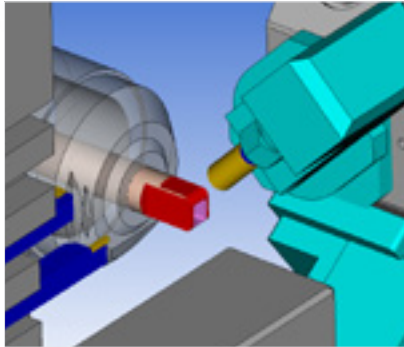




PartMaker 2015

World-leading CAM software for
production machining

www.partmaker.com



PartMaker 2015



www.partmaker.com



www.delcam.tv

Unlock the power
of Delcam software



LEARNING ZONE

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



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



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