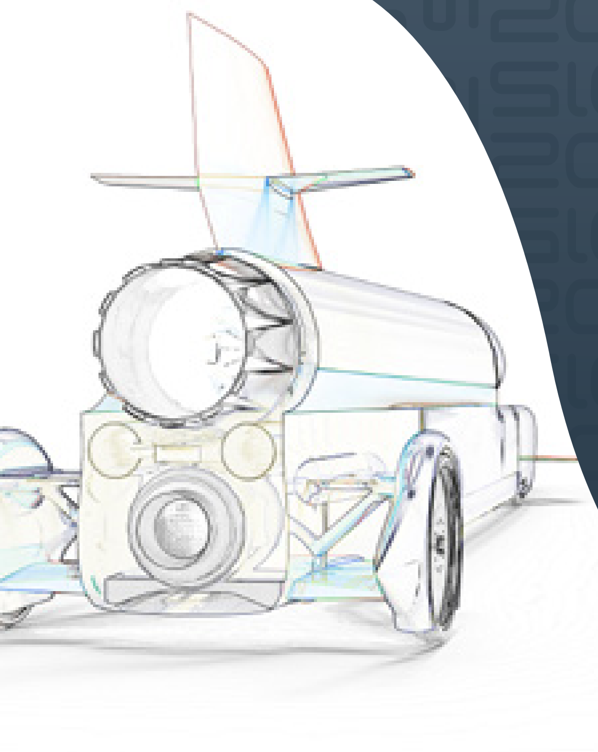


PartMaker 2015

PartMaker Turn-Mill

Version 2015 or Higher

User Guide

Sequence of Programming Steps for PartMaker TurnMill

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**.

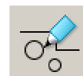
2. Choose the **Tools** command from the **ToolMinder** menu to add new tools.

3. Choose the **Cycles** command from the **ToolMinder** menu to add new cycles.

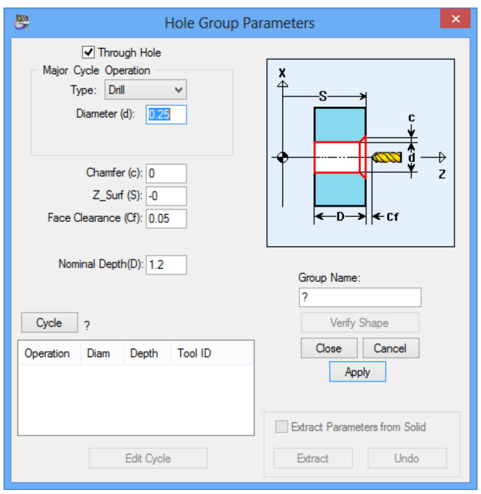
4. Choose the **Open Material File** command from the **File** menu to open a material file from the Material directory.

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

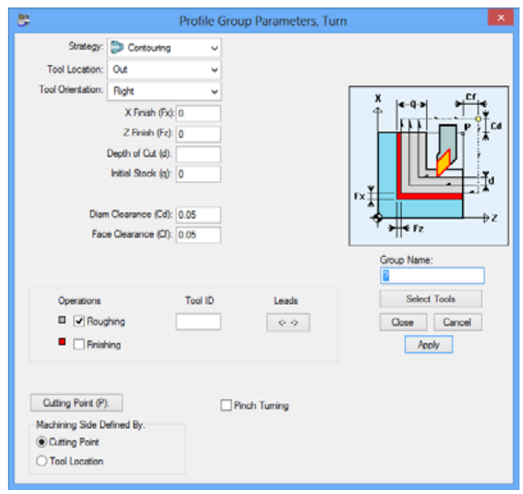
7. Choose the **New Hole Group** command from the **Part Features** menu to create an axial hole. You should see a dialog box as below:



7.1. Enter your desired settings into the dialog.

7.2. Click the **Cycle** button.

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:



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.

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 TurnMill

Important User Notices

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

Page Intentionally Left Blank

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 data base. 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 holemaking 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 Turn-Mill 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 Turn-Mill 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-Mill Tutorial Part – Single Spindle"** shows you how to create a sample part using some of the most popular features of **PartMaker** assuming you are working on a Turn-Mill with one spindle and one turret. We recommend you go through this chapter to familiarize yourself with the **PartMaker** application.

Chapter 4 **"Programming the Advanced Turn-Mill Tutorial Part – Twin Spindle, Twin Turret"** shows you how to create a part on an 8-axis Turn-Mill Center highlighting advanced PartMaker features such as Material Control Processes and Process Synchronization. Recommended to be done after reviewing Chapter 3.

Chapter 5 **"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.

This User Guide is also meant for use in conjunction with a machine specific programming addendum that may have also come with your **PartMaker Turn-Mill** software. The Machine Specific addendum is meant to show you a layout of the machine you are working with as well as describe any machine or post processor specific programming strategies you should be aware of.

Chapter 1: Getting Started With PartMaker

In Your Package

- PartMaker DVD
- 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

A Customer PIN has also come included with your PartMaker software.

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

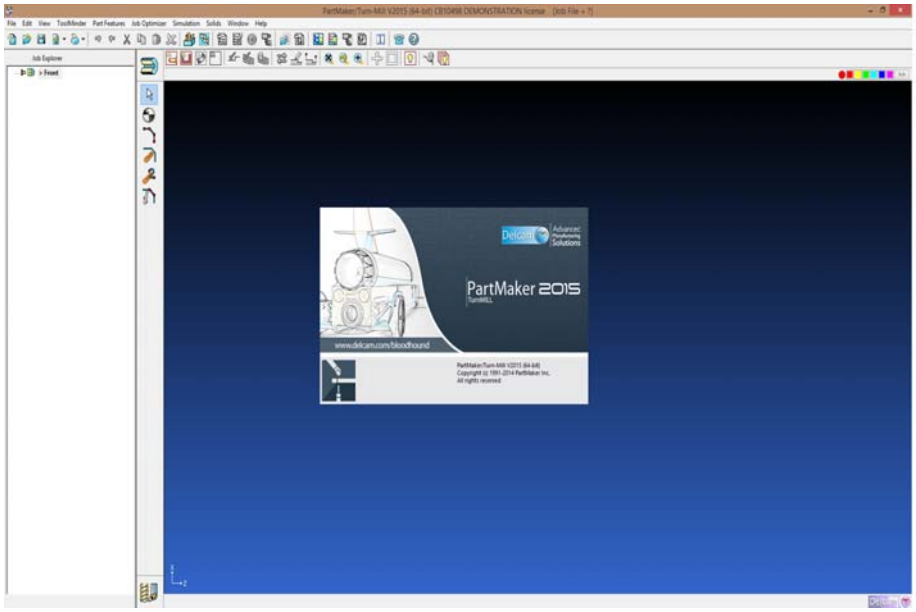
64-bit and 32-bit versions of PartMaker are available and are supported on Windows 7. Only the 64-bit version of PartMaker is compatible with Windows 8 PCs.

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

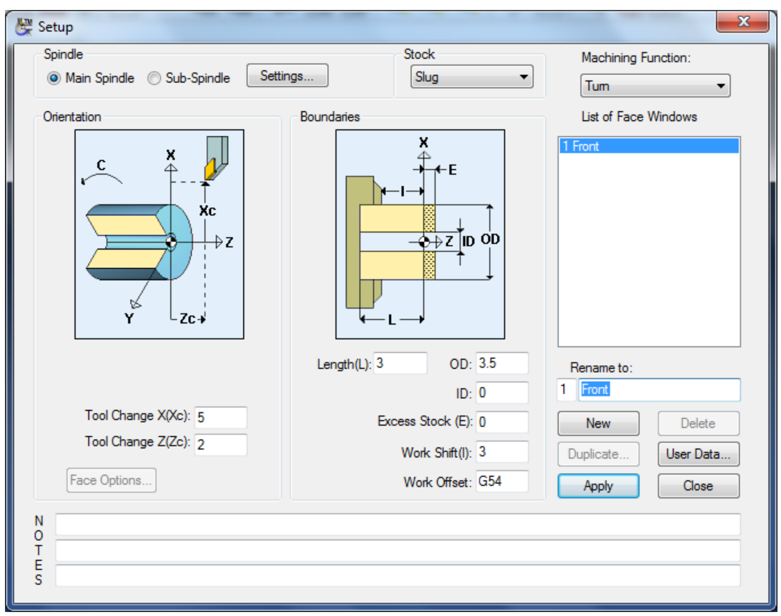
Starting PartMaker

From your desktop, double-click the PartMaker TurnMill 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**.



Installing Multi-Axis TurnMill Post Processors

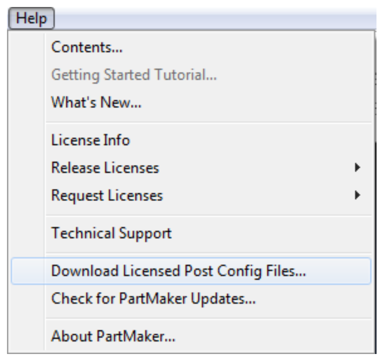
PartMaker TurnMill Version 2010 and higher features a unique capability for downloading your licensed TurnMill post processors and related files directly through the software's user interfaces. This feature is helpful for both new users installing the software for the first time and existing users who wish to update their post processors and related files to the most recent "distribution" Post Processors (see below for more information on "distribution" versus "non-distribution" post processors).

Important! *You must be connected to the internet to automatically download and install your licensed post processors through the PartMaker interface.*

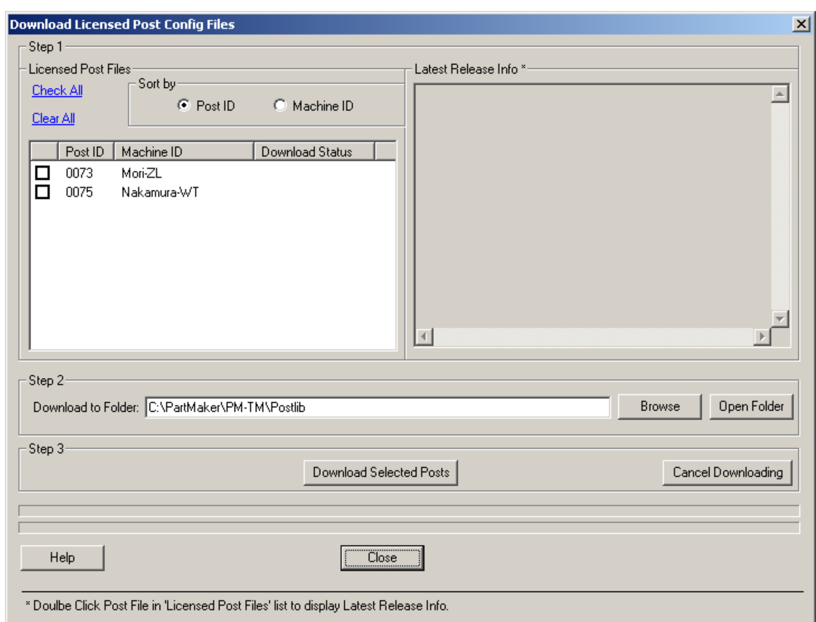
Once you have successfully launched the TurnMill application, you can install the TurnMill post processors that you have licensed directly from the PartMaker user interface.

To do so:

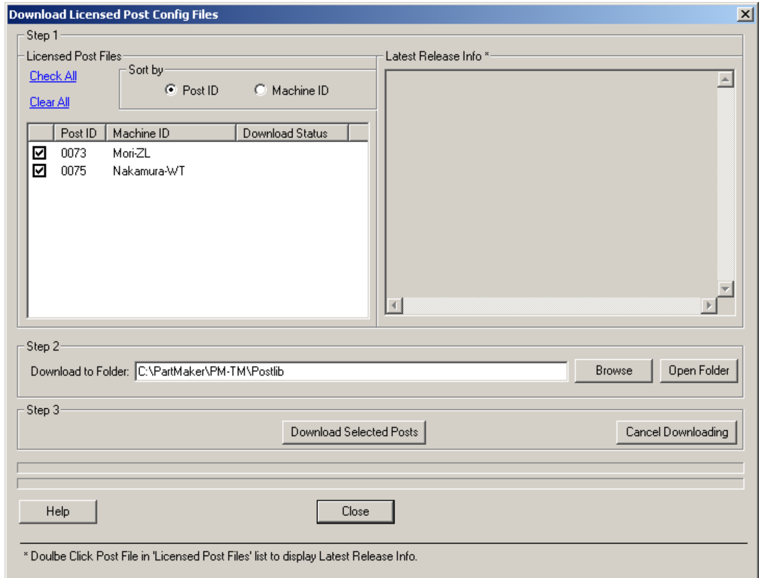
1. From the Help menu, choose Download Licensed Post Config Files...



2. You will be presented with the dialog below (the actual post processors for your system may differ from those shown below):

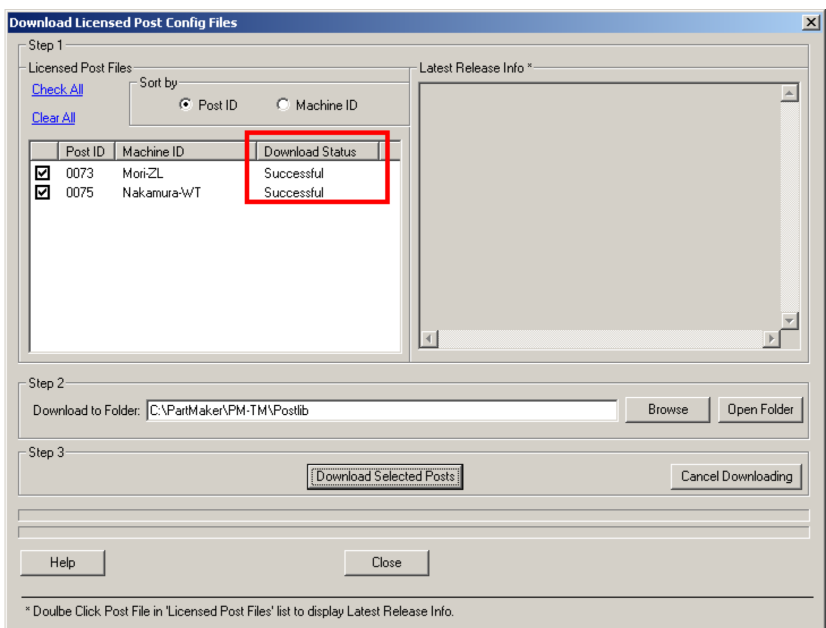


3 Check the boxes to the left of the post processor(s) you wish to download as shown below:



4 Once you have selected the items you wish to download, click the **Download Selected Posts** button to begin downloading those items into the directory specified in **Step 2** of the **Download Licensed Post Config Files** dialog. Upon doing so, you will see a progress bar showing the status of your download.

5 Once you have successfully downloaded the selected items, you will see the **Download Status** column will indicate “Successful” as shown below.



Note: *If you receive any message other than “Successful” after this process, contact PartMaker technical support at support@partmaker.com or your local PartMaker support representative (outside North America).*

6 Once you successfully downloaded the requested files, click the **<Close>** button to continue using PartMaker.

What You Have Just Downloaded

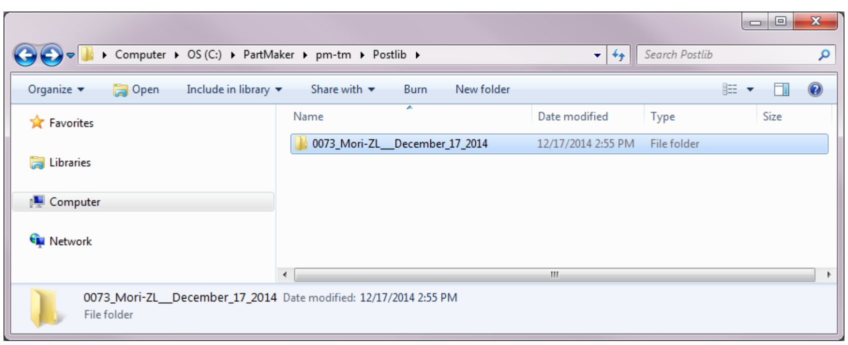
Once you have completed the steps above, you will have downloaded what are known as “distribution post processors.” PartMaker post processors for multi-axis lathes are more than just standard post processors that come with a CAM system. They are highly engineered, actively supported, routinely maintained and well documented machine configuration files to assure you generate accurate NC code for your multi-axis lathes utilizing the full capability of your machine tool.

In fact, you have download much more than just post processors. You have downloaded the following items:

1. **The Post Processors your company has licensed** – These include the *.pst and *.sub (for machines equipped with a sub spindle)
2. **Related Simulation Files** – PartMaker’s Advanced and Full Machine simulation modules use additional files to allow you to simulate the architecture of your machine. You will need to use the *.MCH file corresponding to your post processor to view your machine’s working envelope when using Advanced Simulation. If you have licensed Full Machine Simulation, you will need to use the *.FMS and *.MCH corresponding to your post processor to view your machine’s working envelope when using Full Simulation. Please see the chapter in your user manual for information on using PartMaker Advanced and Full Machine Simulation modules. Included with these files are PartMaker Job (*.job), Tools (*.tdb) and Cycles (*.cdb) files that are configured to run in both Advanced and Full Machine Simulation.
3. **Post Processor Related Documentation** – Documentation related to your licensed post processors and simulation files.
4. **Sample Files** – Sample Job (*.job), Tools (*.tdb) and Cycles (*.cdb) files in both the inch and metric system corresponding to the tutorials in your PartMaker user guide and configured to your machine tool’s specific architecture are also provided.

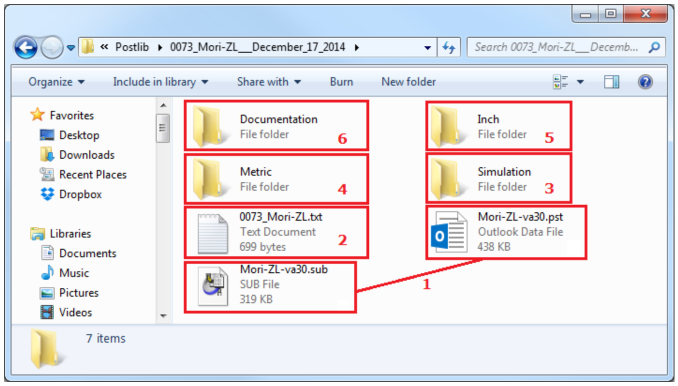
To see an example of this, let’s look at the files downloaded in this example:

1. The picture below shows the result of the download above:



2. Double clicking on the folder above called 0073_Mori-ZL_____Decemeber__17__2014, reveals the following, where:
   1. The *.pst and *.sub, the post processor files themselves
   2. A *.TXT is a readme file explaining changes made in the most recent version of the post processor
   3. A folder containing the simulation, *.MCH and *.FMS referred to above
   4. A folder containing tutorial files in the metric system corresponding to the tutorial in the user guide
   5. A folder containing tutorial files in the inch system corresponding to the tutorial in the user guide

6 A folder containing the machine specific addendum and FMS reference kit referred to above



*__Tip:__ If using Advanced and/or Full Machine Simulation, you may wish to place the *.FMS and *.MCH in the same directory as the *.PST and *.SUB files. This will allow your simulation files to open automatically when you load the post processor. You may also wish to place the tutorial files (in the units of measure in which you work) in the same directory.*

More on Multi-Axis Turning Post Processors

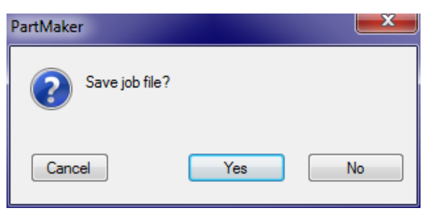
PartMaker Inc's application staff maintains a library of well documented, well supported multi-axis post processors which includes virtually every TurnMill machine ever built. Once you start to customize your own post processor using the ConfigPost application, you create a "non-distribution" post processor.

Please also note, PartMaker is constantly updating and improving its TurnMill post processors to add functionality and correct any possible reported errors. You can use the same procedure above to check for post processor updates periodically if you are experiencing difficulty with your post processor. Even if you have customized your post processor and are using a "non distribution" post processor, you can quickly see any differences between your customized "non distribution" post processor and the most current "distribution" post processor using the Compare function in PartMaker's ConfigPost application. See the ConfigPost user guide that came with your software for more information on this function.

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:

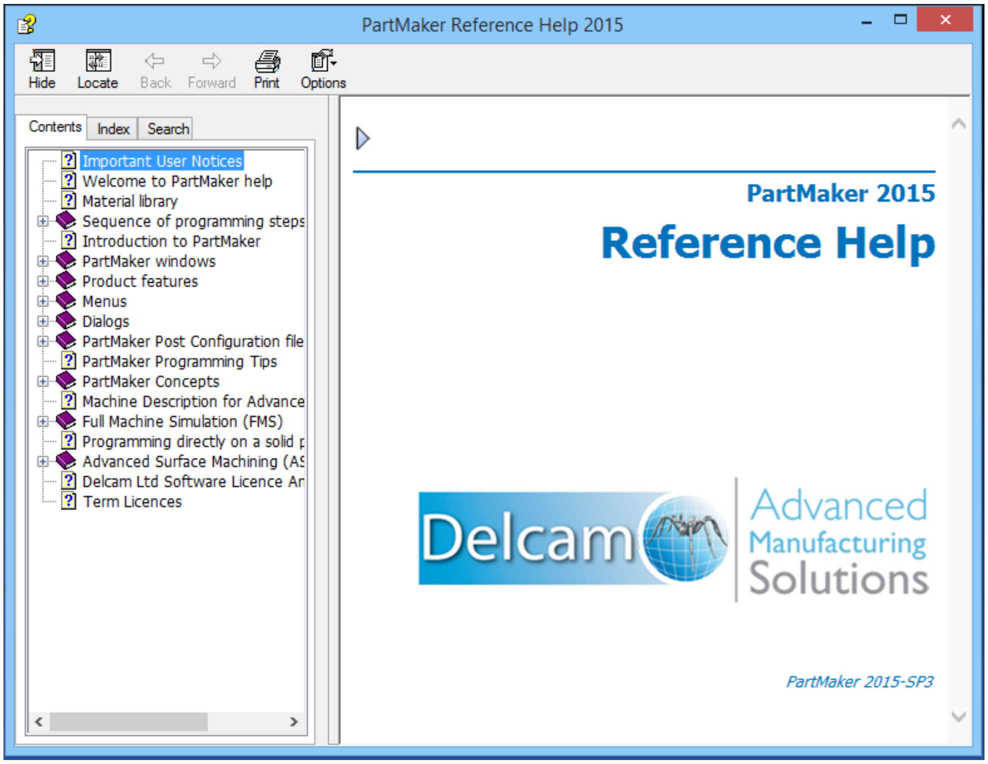
- Click **<Yes>** to save the changes.
- If the file is untitled, select a location and enter a file name in the dialog that appears, then click **<OK>.**
- Click **<No>** to discard the changes and exit.
- Click **<Cancel>** to return to the presentation window without exiting.

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 **Contents** from the **PartMaker Help** menu, click the  icon on the toolbar 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.

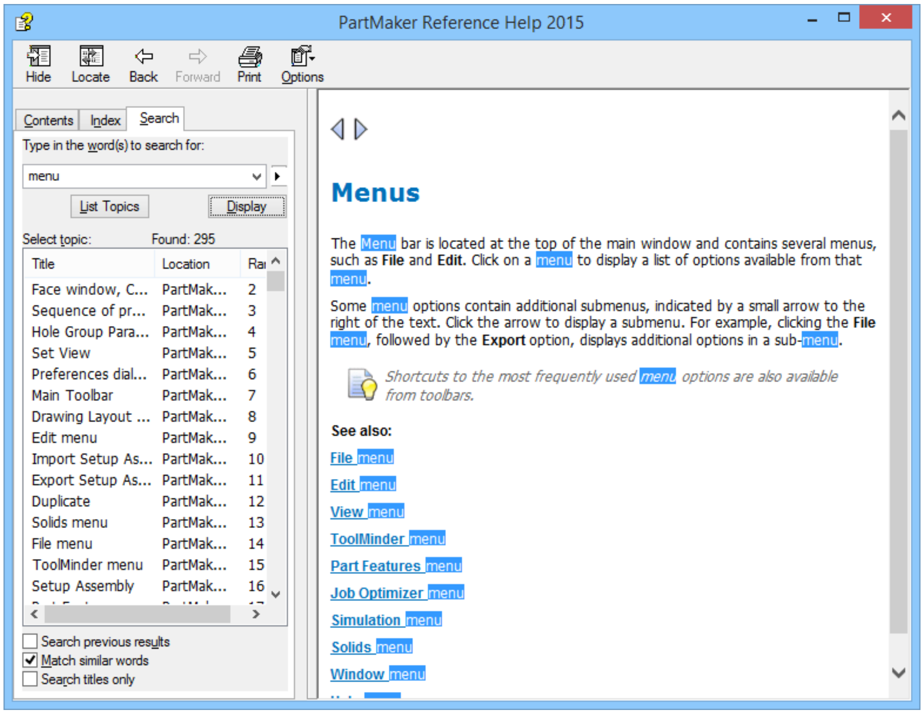
Note: *Some topic lists include sub-lists from which you can select a help topic.*

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.

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



8 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:

1 Display any **PartMaker** dialog.

2 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 **Tool**s 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 TurnMill 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

TurnMill Centers are capable of performing a full set of turning operations: facing, turning, grooving and threading. Many TurnMill Centers are also capable of performing various types of milling as well as drilling holes on different surfaces. Multiple spindles in which such operations can be performed simultaneously further increase the machines' productivity and complexity.

Milling capabilities of the TurnMill Center vary significantly and can be ranked in order of sophistication as follows:

- Index milling/drilling
- Milling on a selected ZX plane
- Polar milling on the face of the part
- XZC continuous milling on cylindrical surfaces
- Milling interpolation in the YZ plane

Programming of these powerful machines is, however, much more difficult than the programming of the individual mill or lathe. Despite significant efforts by many CAM vendors to develop an easy-to-use CAM system for programming multi-axis TurnMill Centers, the majority of programming today is still done manually.

PartMaker CAM Software from PartMaker, Inc. applies a Patented 'divide and conquer' programming strategy to simplify programming of TurnMill 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.

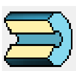
A specific machining function such as turning, polar milling or cylindrical milling is assigned to each face. **PartMaker** lets you create a dedicated window that contains a workspace for the graphic representation of face features.

A dialog box associated with each **Face Window** shows you the type of machining to be performed on a face, the location of the face boundaries and parameters defining the positioning of the face relative to the machine coordinate system.

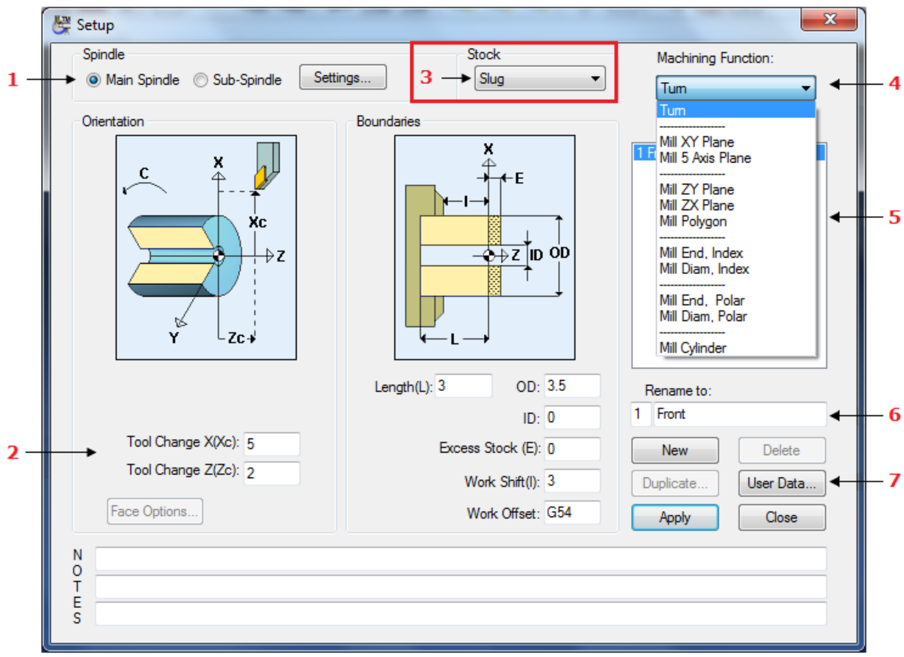
Getting Started: The Setup Dialog

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.

Once divided into much smaller, similar motion components, programming of a multi-axis TurnMill Center becomes a much easier task. To best explain how **PartMaker** handles this task, it is necessary to first explain how each of these individual-machining functions is defined.



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:



1 The Main Spindle and Sub Spindle radio buttons allow you to select whether the programming carried out in a particular **Face Window** is being machined on the Main or Sub-Spindle.

2 The Tool Change X and Z positions for a particular machine should be entered here.

3 The Use Bar Stock indicates to PartMaker whether the part is being made from Bar or Slug. This effects the display of the 3D part simulation.

4 The Machining Function drop down menu allows you to select the type of machining motion being executed in a particular **Face Window**. The machining motions that can be defined in each selection from the Machining Function drop down menu are explained below.

5 The List of **Face Windows** displays a list of **Face Windows** currently existing for a given job.

6 The Rename To: field allows you to name the **Face Window** you are working with. Existing **Face Windows** can be selected from the Window menu based on the names assigned to them in this field of the **Setup** dialog.

7 Clicking the function buttons shown in the **Setup** dialog with the left-hand mouse button does the following:

<New>	Creates new face windows of different machining function types or on different spindles, adding to the List of Face Windows.
<Delete>	Deletes the currently selection from the List of Face Windows.
<Duplicate>	Allows the user to duplicate a series of identical Mill ZX, ZY and Mill XY windows. Only active when one of these Windows is selected.
<User Data>	Allows the user view the User Data for a given post processor. Only active if a post processor is loaded.
<Apply>	Accepts or applies any new parameters or data entered into the various fields in the Setup dialog, and the dialog remains open.
<Close>	Accepts or applies any new parameters or data entered into the various fields in the Setup dialog, and closes the dialog.

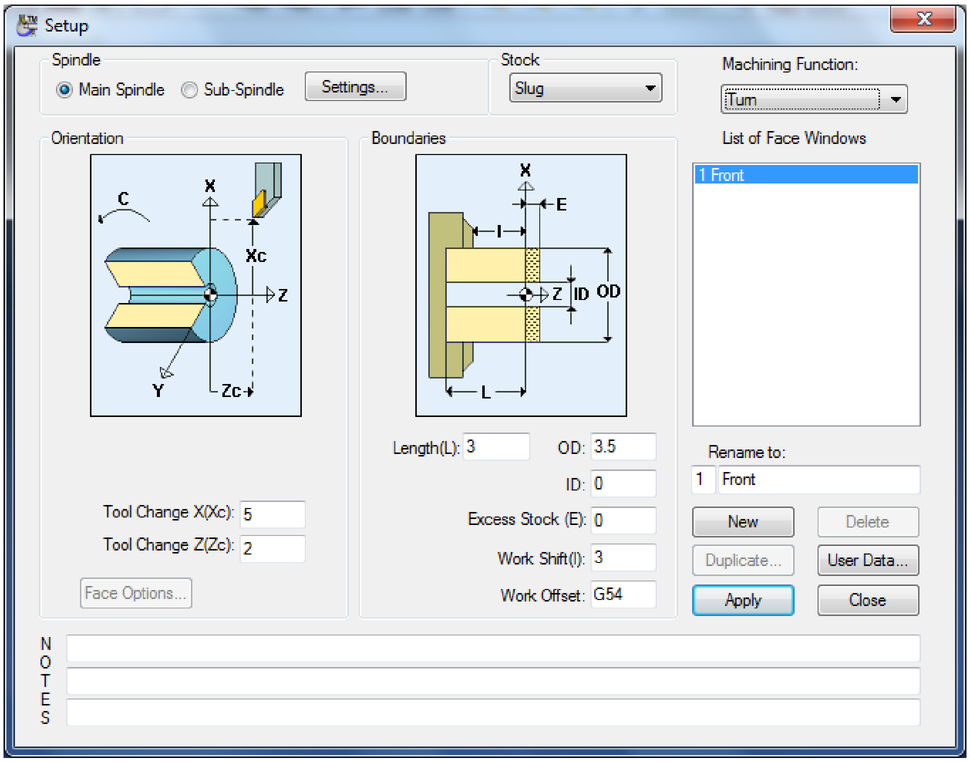
The various selections under the Machining Function drop down menu are explained on the following pages.

The Notes fields allow the user to enter notes about a specific job. These notes can be directly output into the NC program to improve documentation.

Machining Function: Turning

The Turn machining function uses conventional turning tools moving in the X and Z-axes. 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.

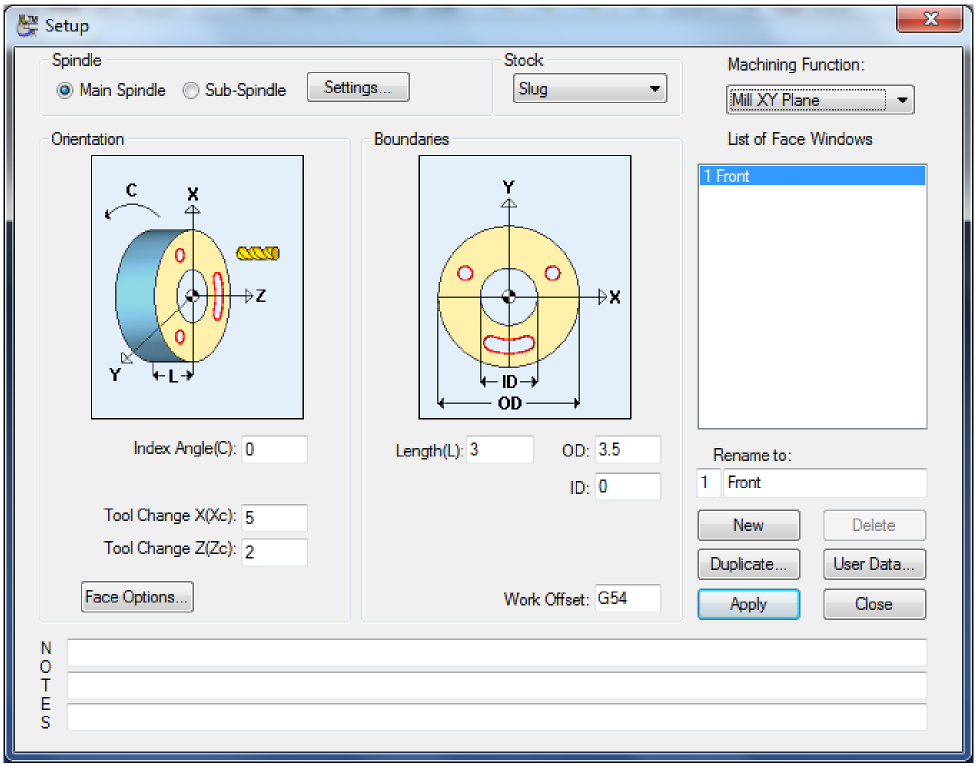


Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Excess Stock (E)	Rough stock left for facing on the front of the part
Work Shift	Specifies amount of the part exposed during machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill XY Plane

The Mill XY Plane machining function supports machining operations on the face of the part using the Y-axis capability of the machine. G-code output will be generated in the form of X and Y coordinates. Features that can be created by using this Face Window include off-center holes on the face of the part, pockets on the face of the part, and contours (such as CAM shapes) on the face of the part.

Programming of this machining function requires that your TurnMill Center have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



Length (L) Length of the finished part

OD Starting outside diameter of bar stock before machining

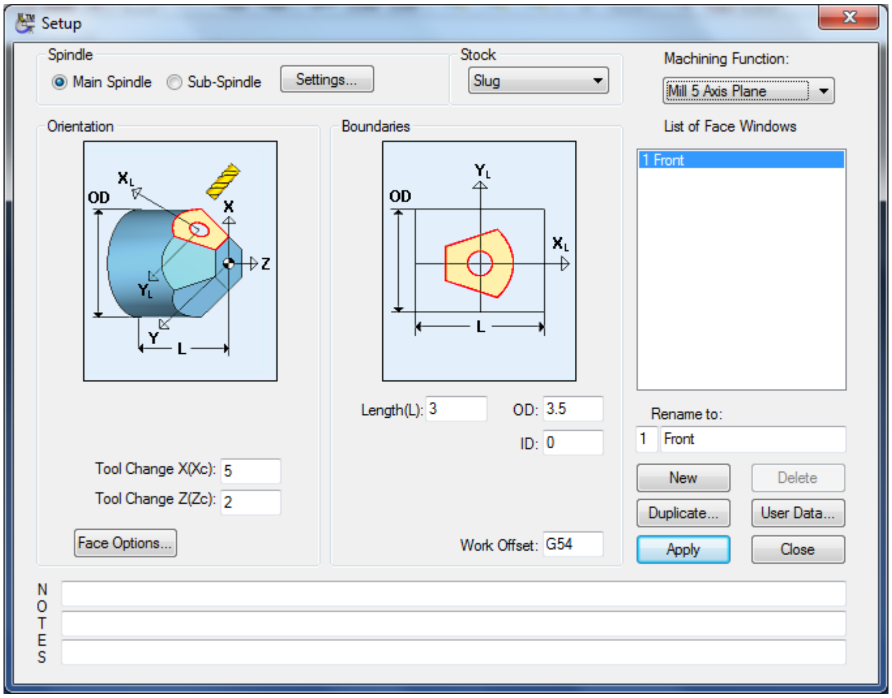
ID Starting inside diameter of bar stock before machining

Work Offset Specifies the work offset for the current face window

Machining Function: Mill 5 Axis Plane

The Mill 5 Axis Plane machining function supports angled machining operations using either a pivoting tool head or a fixed, angled tool. Features that can be created by using this Face Window include any inclined feature including inclined holes, angled flats or any interpolation occurring on an angle.

Use of this machining function requires that your TurnMill Centers has either a programmable tool head or the ability to be fitted with angeled tooling attachments and that you have purchased the **PartMaker** B-axis milling option.

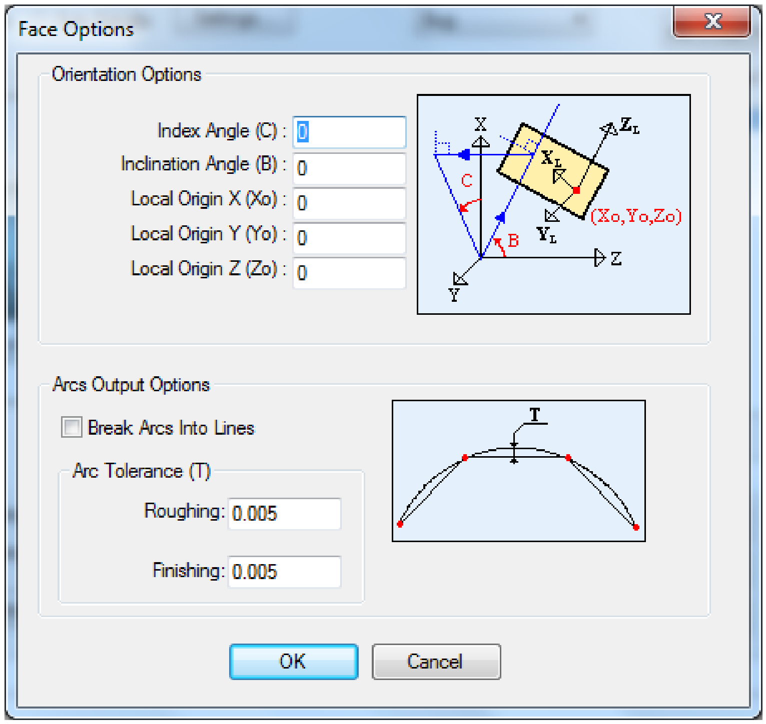


Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining

Machining Function: Mill 5 Axis Plane (cont'd)

Additionally, when using the Mill 5 Axis Plane Face Window, you will also be required to use the **Face Options** dialog which can be accessed by clicking the **<Face Options>** button in the **Setup** dialog.

The **Orientation Options** section of the dialog is used to orient the part and set the local coordinate system being used for the features being machined in a given **Mill 5 Axis Plane** Face Window. When working with a Solid Model, the parameters in the **Orientation Options** section of the dialog will be set automatically.



Orientation Options

Index Angle (C): Specifies the angle to which the c-axis is indexed in this face window.

Inclination Angle (B): Specifies the angle about the Z axis about which the toolhead or inclined too is tilted in this Face Window.

Local Origin (Xo), Local Origin (Yo), Local Origin (Zo): These fields represent the co-ordinates for the local origin created to machine the planar face. When working from a Solid Model, these values are determined by placing the local orgin in the mathematical center of the feature being machined.

Arcs Output Options

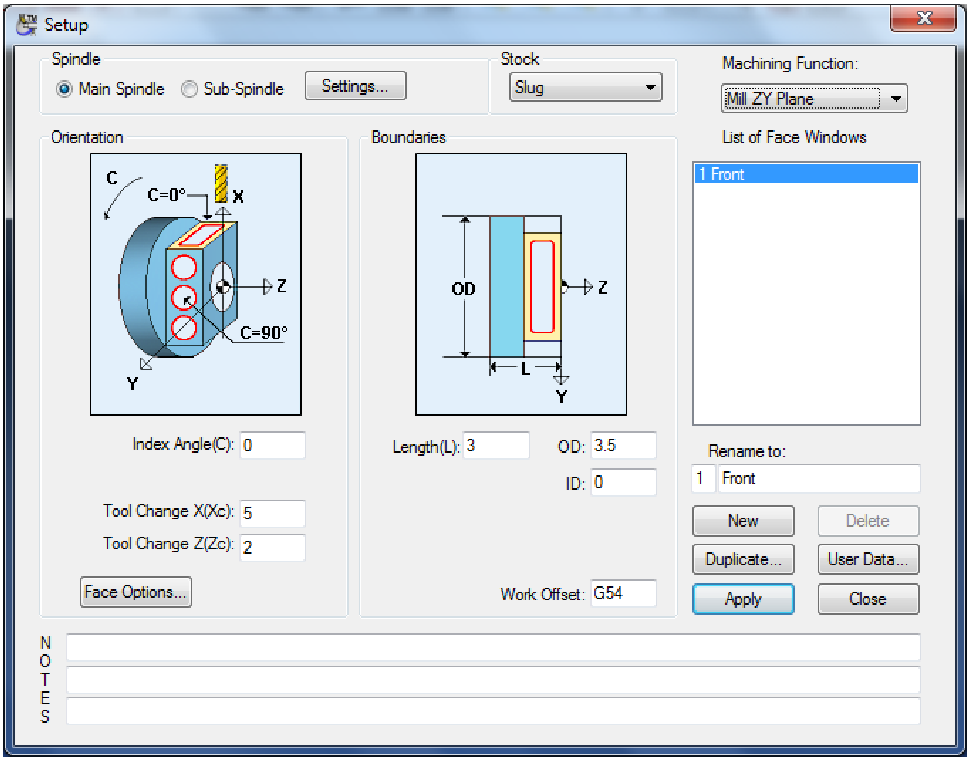
Break Arcs Into Lines: Checking this box indicates to break output for all arcs into small lines. Checking this box is necessary for machines that do not have the ability to support a "local coordinate system" and cannot perform interpolation with an inclined tool.

Arc Tolerance (T): Specifies the sizes of "facets" when breaking arcs into lines.

Machining Function: Mill ZY Plane

The Mill ZY Plane machining function is used to perform milling interpolation in the ZY plane when the spindle is locked at a discreet angular position. All motions are performed with a tool oriented along the machine X-axis.

Programming of this machining function requires that your TurnMill Center have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.

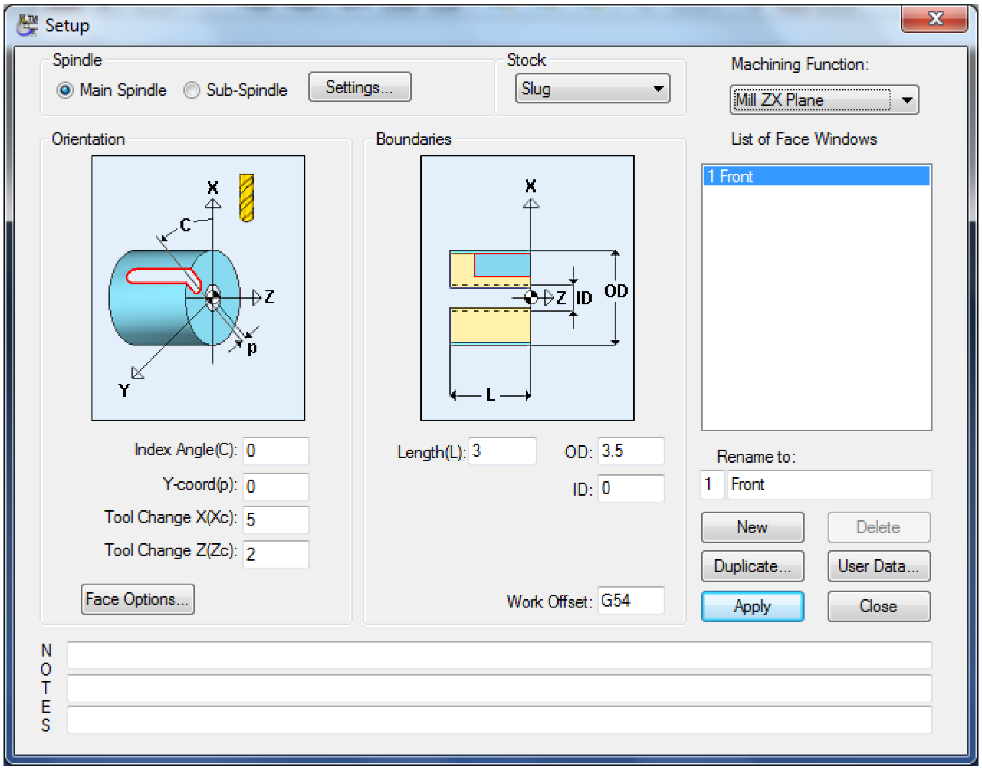


Angle (C)	Orientation angle of the stock to achieve the desired face position
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill ZX Plane

The Mill ZX Plane machining function is used to perform milling in the Z and X planes only. Such operations typically involve following a turning profile with an end mill (often ball nose) with the machine spindle in a locked position. This window can not be used to machine crossholes.

Programming of this machining function requires that your TurnMill Center have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.

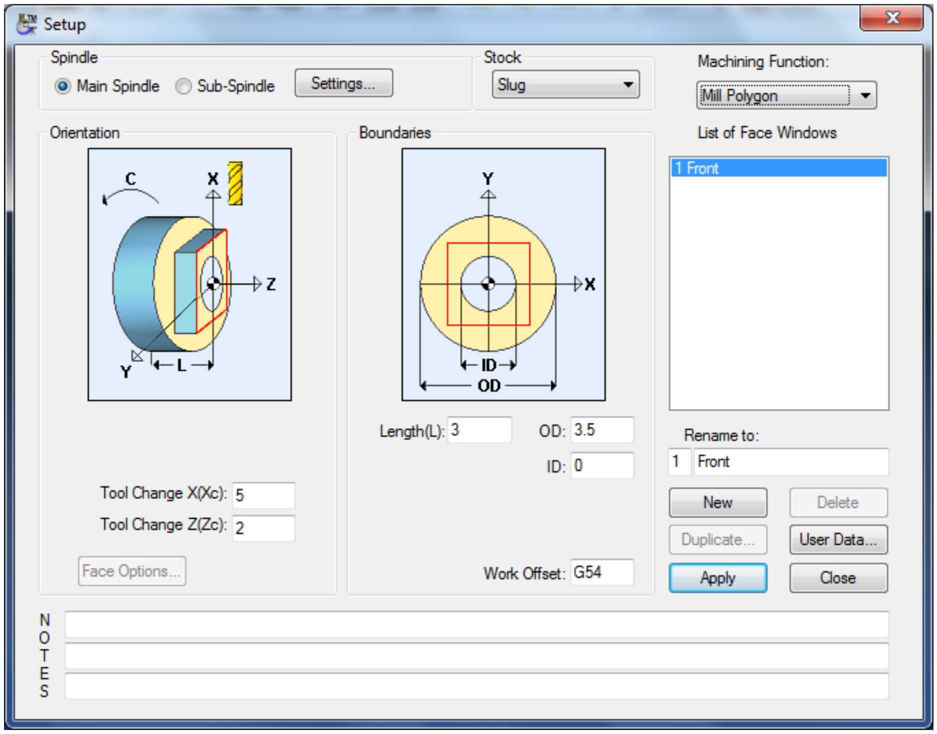


Index Angle (C)	Orientation angle of the stock to achieve the desired face position
Y-coord (p)	Y coordinate value measuring the distance from the centerline of the part the tool is offset during machining. This value should always be positive.
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill Polygon

The Mill Polygon machining function allows you to create polygon shapes on the OD of a part using the Y-axis capability of your machine and a tool oriented along the X-axis of the machine.

Programming of this machining function requires that your TurnMill Center have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



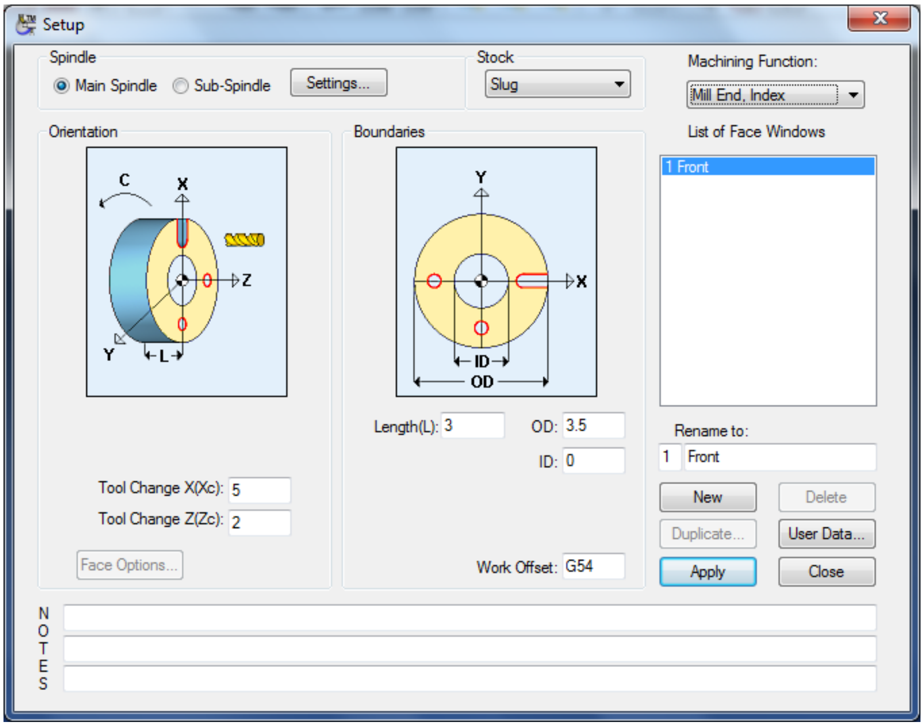
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill End, Index

The Mill End, Index machining functon is used when drilling holes and milling slots on the end face of the part. The holes are located in the XY plane and oriented along the Z-axis.

The shapes that can be created with this machining function are:

- Holes in the end of the work piece. created by Z-axis feed. This motion type supports off-center drilling on the end of the part, using C-axis indexing and X-axis positioning to locate the hole position.
- Radial slots which are milled using C-axis indexing to position and hold the part while X-axis motion is used to cut the part. The tool would be a milling tool oriented along the Z-axis.



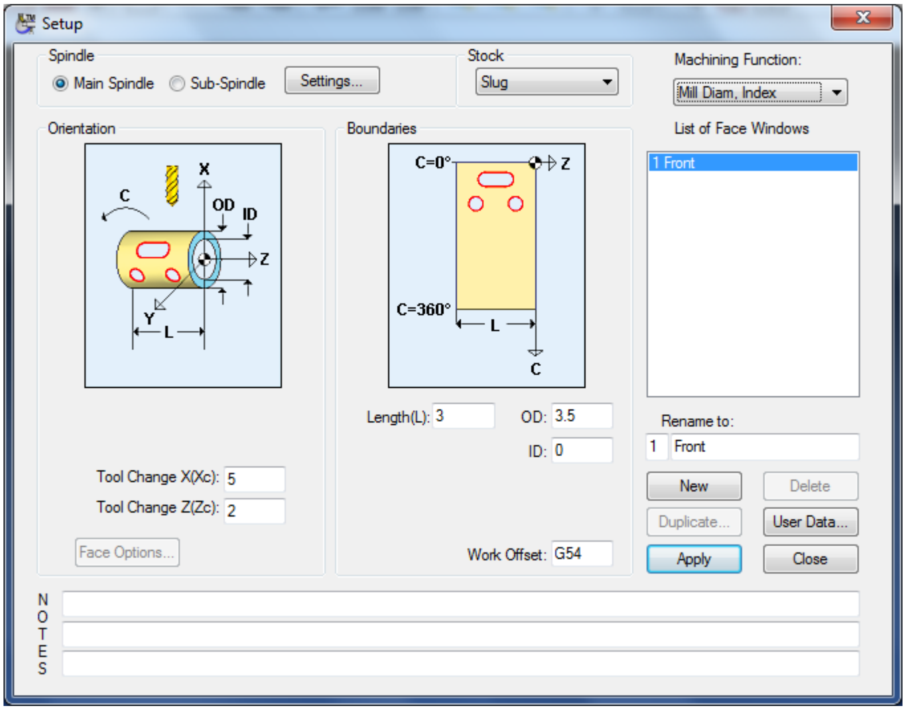
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill Diam, Index

The Mill Diam, Index machining function allows you to create “cross holes” or radial slots on the OD of the part. In order to support this machining function, your TurnMill Center must be able to support C-axis indexing, which may not be performed while the tool is in contact with the work.

This machining function supports the following features cut on the outside diameter of the work using a tool oriented along the machine's X-axis:

- Slots oriented along the length (Z-axis) of the stock.
- Holes drilled anywhere along the cylinder except the end of the stock.

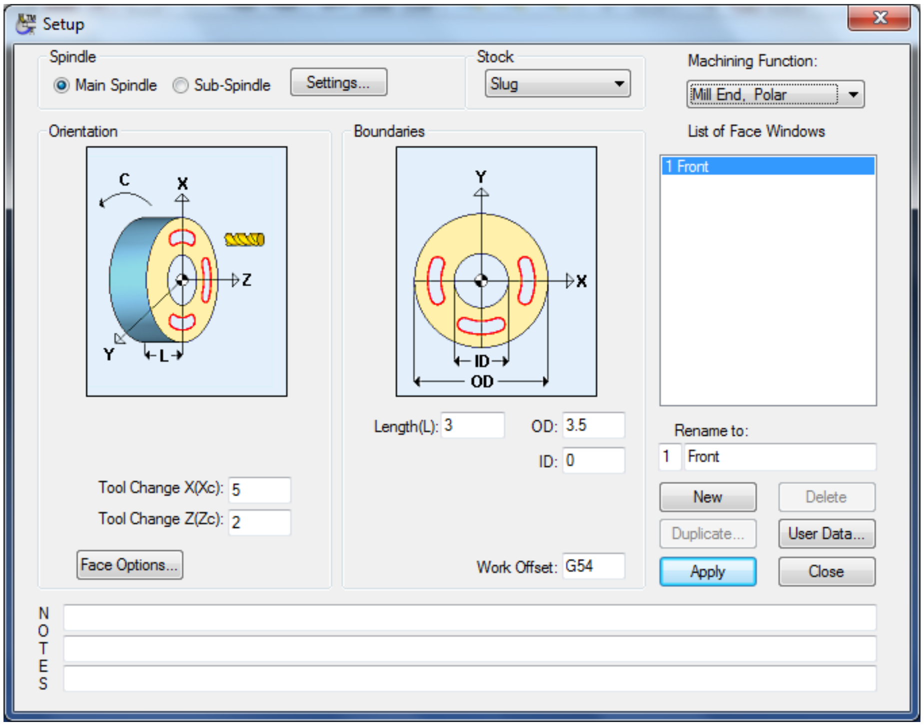


Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill End, Polar

The Mill End, Polar machining function allows you to perform polar interpolation on the face of the part. This machining function assumes that the TurnMill Center has the ability to perform simultaneous feed motions with the X, Z and C-axes.

Programming of this machining function requires that your TurnMill Center has a programmable C-axis and that you have purchased the **PartMaker** C-axis milling option.

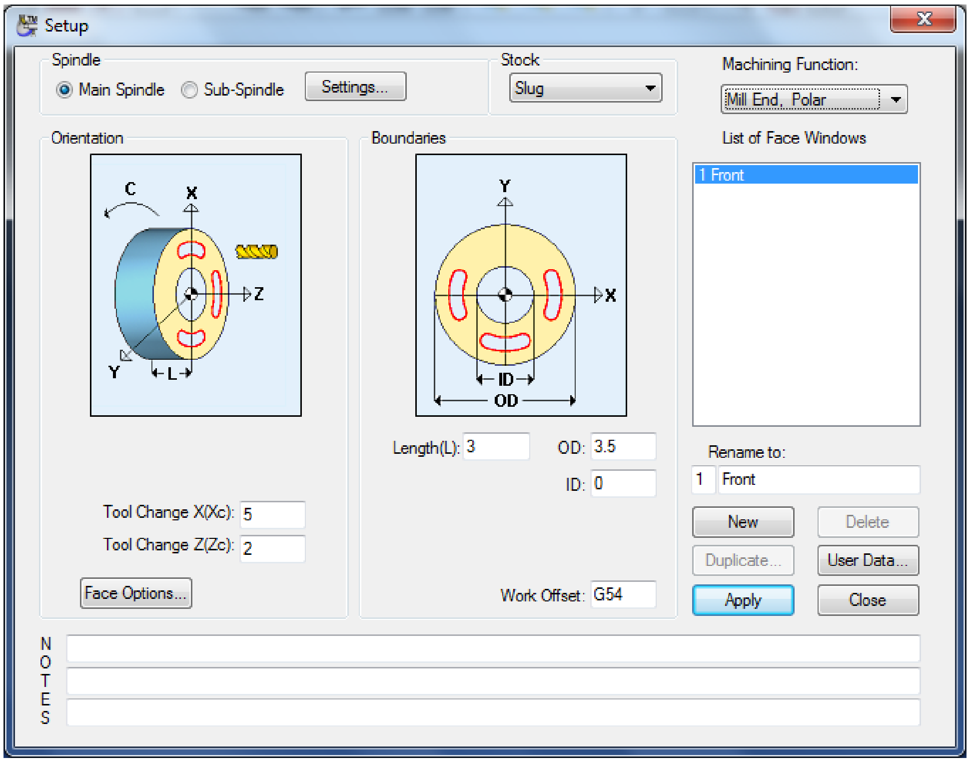


Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Work Offset	Specifies the work offset for the current face window

Machining Function: Mill Diam, Polar

The Mill Diam, Polar machining function supports polar milling anywhere on the stock using an X-axis oriented milling tool. This machining function assumes that your TurnMill Center has the ability to perform simultaneous feed motions with the X, Z and C-axes. The Mill Diam, Polar machining function is most commonly used to machine polygon features on the OD of the part using continuous C-axis feed. Programming of this type in **Face Windows** is carried out in much the same manner as the Mill Polygon machining function.

Programming of this machining function requires that your TurnMill Center has a programmable C-axis and that you have purchased the **PartMaker** C-axis milling option.



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
Work Offset	Specifies the work offset for the current face window

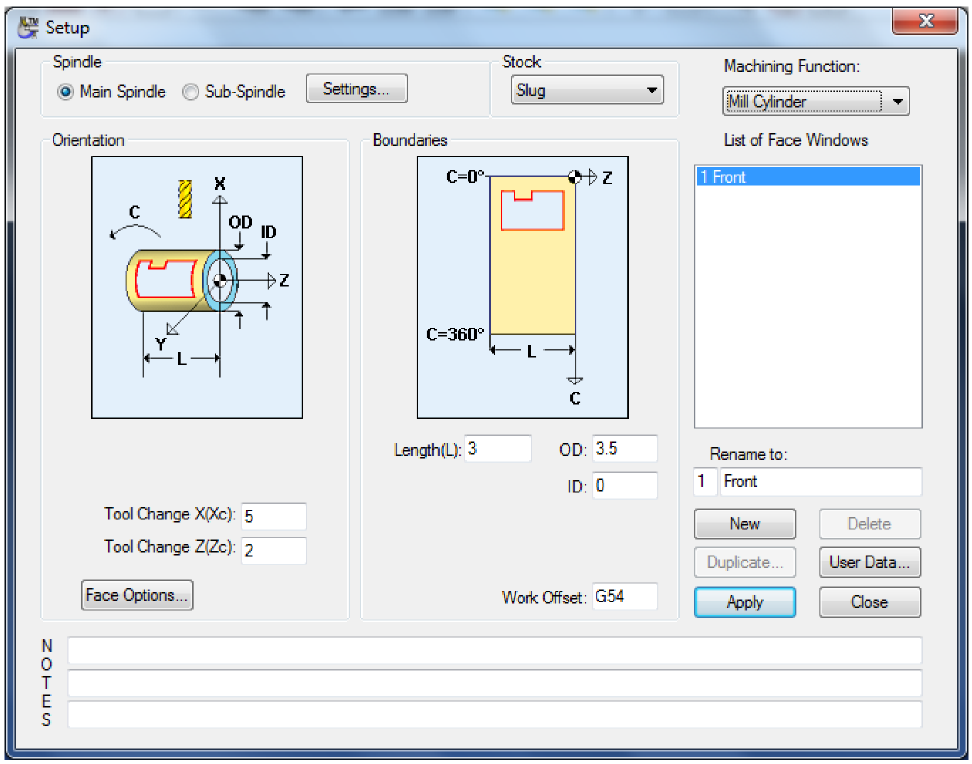
Machining Function: Mill Cylinder

The Mill Cylinder machining function supports continuous milling on the outside diameter of the part. Cylindrical milling assumes that your TurnMill Center has the ability to simultaneously feed in the X, Z and C axes.

Using motions of this type, combined with a tool oriented along the X axis of the machine, profiles can be created on the outside diameter of the stock.

Tool positioning is programmed using X, Z, and C axis coordinates. The C position is programmed in terms of degrees of rotation relative to machine C=0 which is clearly shown in the graphic below.

Programming of this machining function requires that your TurnMill Center has a programmable C-axis and that you have purchased the **PartMaker** C-axis milling option.



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
Work Offset	Specifies the work offset for the correct face window

PartMaker Programming Convention

When using PartMaker TurnMill 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, the same programming convention employed on a conventional 2-axis CNC Lathe. The same programming convention also applies for work being performed on the sub-spindle. 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

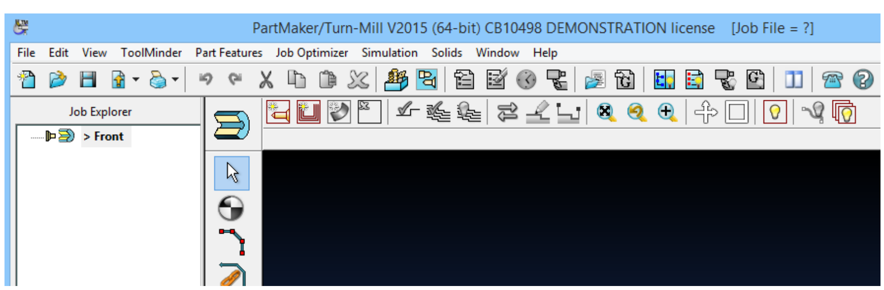
PartMaker uses several kinds of files for storing tools, cycles, and material information. Your jobs and post processor files used to create NC programs for machining can also be stored in their own files.

- 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 **Tool**s 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 material, tools, and cycles databases to store the information needed for each job. These three databases represent the knowledge-based components of **PartMaker** because they allow you to store shop specific information that can be reapplied to future jobs automatically, thus letting **PartMaker** learn from you. The more you use **PartMaker**, the smarter it gets!

Important! *Before you begin programming a job, make sure you properly setup your databases in* ***PartMaker****. This means creating the tools you need for the job, selecting the material you are cutting, and setting-up the holemaking cycles you may need to machine the part.*

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. Each tool database can store up to 3,000 different tools.

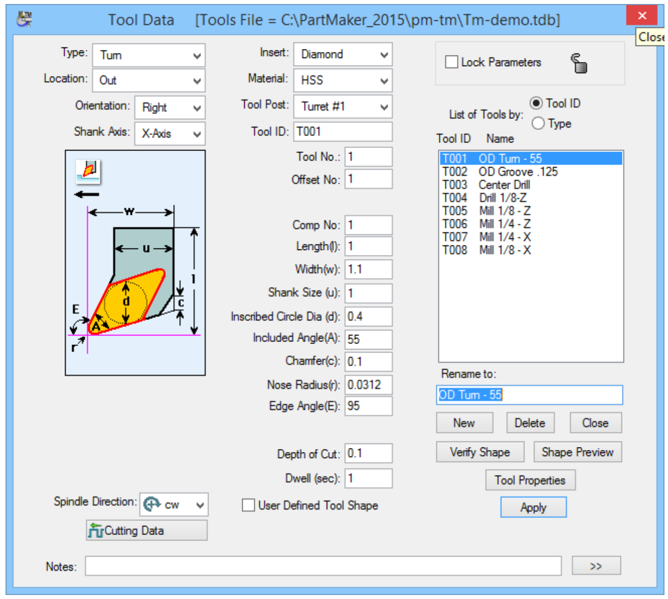
Note: Due to the unique nature of tool placement and off-set information required by different TurnMill Centers, when using **PartMaker** TurnMill, it is recommended that you set up one tools file for each TurnMill machine you are programming. Doing so will avoid a great deal of confusion.

Please refer to the Machine Specific Addendum(s) that came with your TurnMill software for information on properly setting up tools for your machine.

You can access **PartMaker**'s Tools Database by choosing **Tools** from the **ToolMinder** menu. The picture on the following page highlights some of the key elements of the **Tool Data** dialog for both turning and milling tools. Please take the time to review and understand these now.

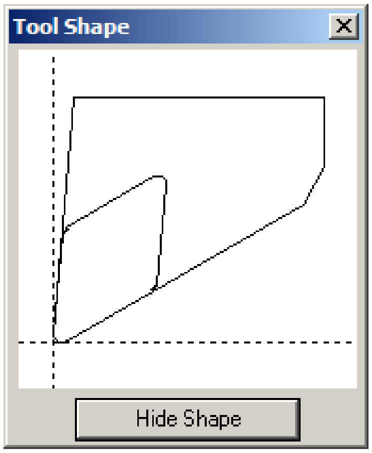
Tools for Turning

The turning tool below can be viewed by choosing **Tools** from the **ToolMinder** menu and choosing OD Turn-55.



- The **Tool Data** dialog box title bar shows the directory path of the tools database.
- The **Type**, **Location** and **Orientation** fields determine general appearance of the current tool. These fields apply to turning tools – turning, grooving and threading tools. It is important to be mindful of the tool orientation with respect to **PartMaker** TurnMill's programming convention. You must remember programming is done assuming the collet or workholding to be to the left.
- For Turning tools only. **Insert** allows you to choose the type of insert being used.
- **Material** allows you to choose the material for the current tool. This selection will directly impact feed and speed calculations.
- The **Tool Post** drop down menu allows you to choose the location or tool post in the TurnMill Center where a tool is mounted. Various TurnMill Center machines have different configurations, but all varieties are accommodated by using the selections from the **Tool Post** drop down menu. Be sure to designate the correct tool group when setting up a tool. Each tool group selection can be explained as follows:
  - **Turret** #1 – For single turret machines, this tool post should always be selected.
  - **Turret #2** – Indicates that the current tool resides in the second turret on the machine. For use with TurnMill Centers that have a two-turret structure.
  - **Turret #3** – Indicates that the current tool resides in the third turret on the machine. For use with TurnMill Centers that have a three-turret structure (least common).
- The **List of Tools by:** radio buttons allow you to list tools either by their **Type** or **Tool ID**, their unique catalogue number in the tool database. Tools will be shown sorted in the list based on their **Tool ID** and assigned name from the **Rename To:** field.
- Clicking the function buttons shown in the **Tool Data** dialog with the left-hand mouse button does the following:

- **<New>** – Allows you to create new tools to be added to the tools database.
- **<Delete>** – Allows you to remove or delete an existing tool from the tools database.
- **<Close>** – Accepts or applies any new parameters or data entered into the various fields in the **Tool Data** dialog, and closes the dialog.
- **<Verify Shape>** – Allows you to verify the shape of the currently selected tool according to the specified parameters. For the first tool shown, when clicking this button, you should see the following:



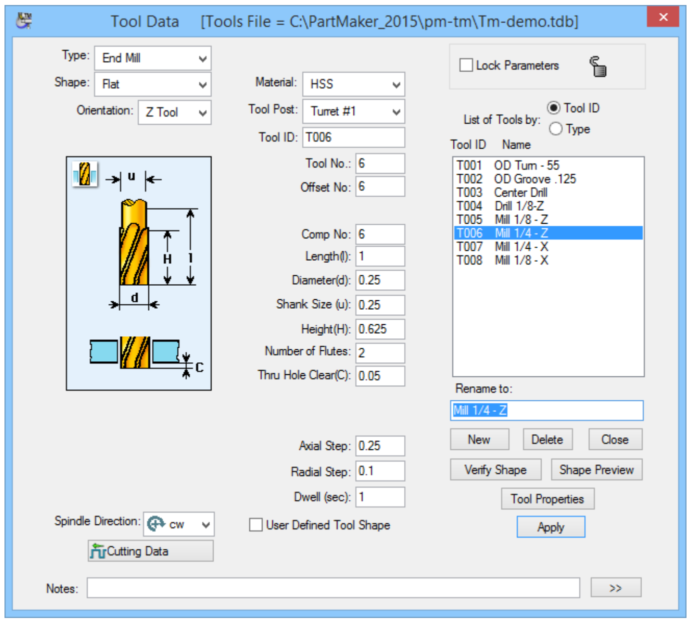
  Click **<Hide Shape>** to return to the **Tool Data** dialog.
- **<Tool Properties> -** Clicking this button displays the **Tool Properties** dialog. The tool properties dialog allows you define tool specific offsets for each tool.
- **<Apply> -** Accepts or applies any new parameters or data entered into the various fields in the **Tool Data** dialog, and the dialog remains open.

Tools for Milling and Holemaking

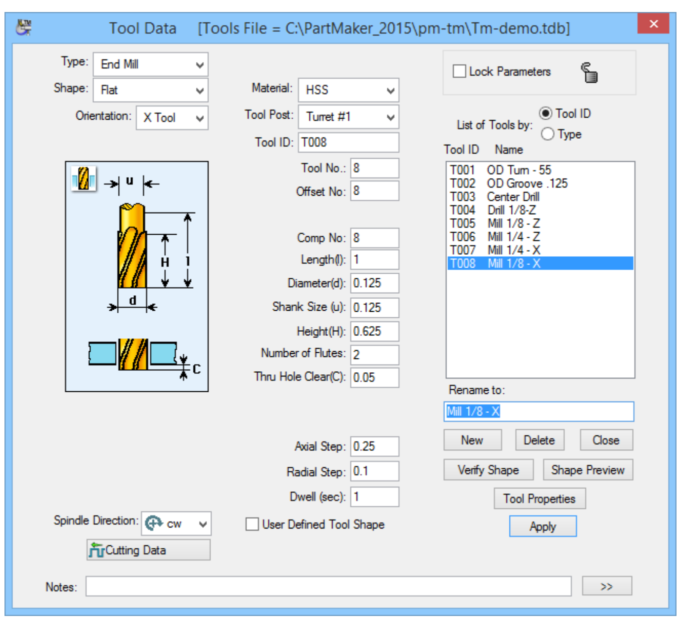
Tools used for Holemaking and Milling are defined in the same way as turning tools. The major difference is the orientation. When setting up a milling or holemaking tool – regardless of whether it will be “live” or “dead” (used on centerline in Machining Function Window of Type Turning) you must specify its orientation with respect to the stock being machined.

Tool **T006** in the list of tools below is an End Mill. This end mill has been designated as a “Z Tool” meaning it is oriented *horizontally* with respect to the stock being machined. The type of **Face Window** in which a tool with a “Z Tool” orientation is applied will determine if it is live or dead.

See **Tools for Turning** in the previous section of this chapter for more information on the functionality of this dialog.



Next, look at tool **T008** in the list of tools to select the End Mill. This end mill has been designated as an “X Tool” meaning it is oriented *vertically* with respect to the stock being machined. Tools with “X Tool” orientations are always “live” tools.



Material Database

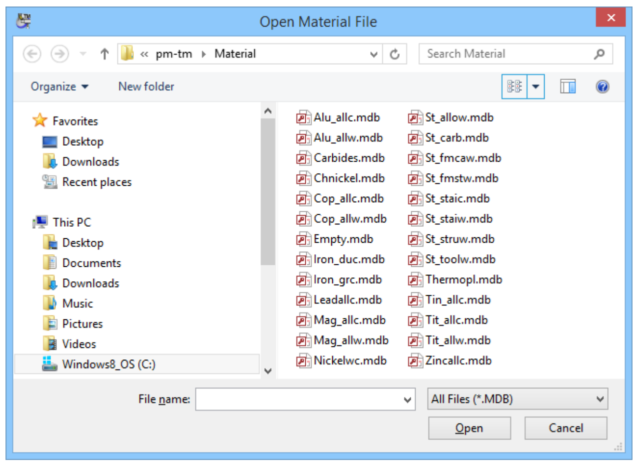
The **Material Database** allows you to store your shop's accumulated materials data. **PartMaker** uses materials data in the automatic calculation of feedrates and spindle speeds. The Materials Database allows you to choose from a variety of different materials and even adjust them and store your changes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

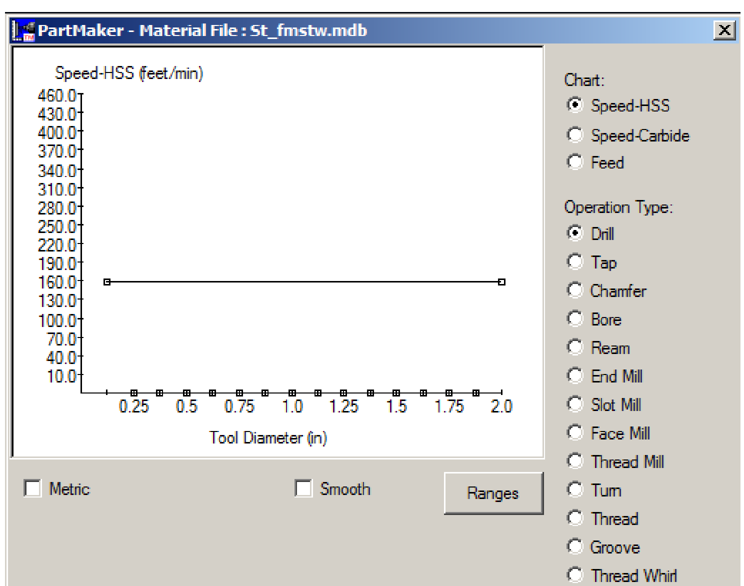
When creating a part, it is important to select a material. To do so:

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**.

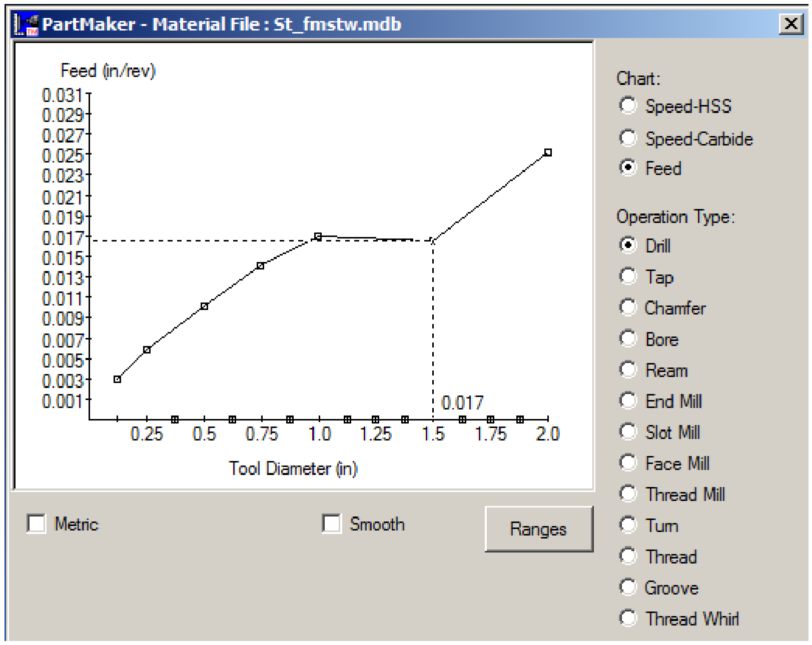
3. To view the cutting characteristics of the selected material, choose **Material** from the **ToolMinder** menu and you will the see the dialog shown below:



Customizing a Material File

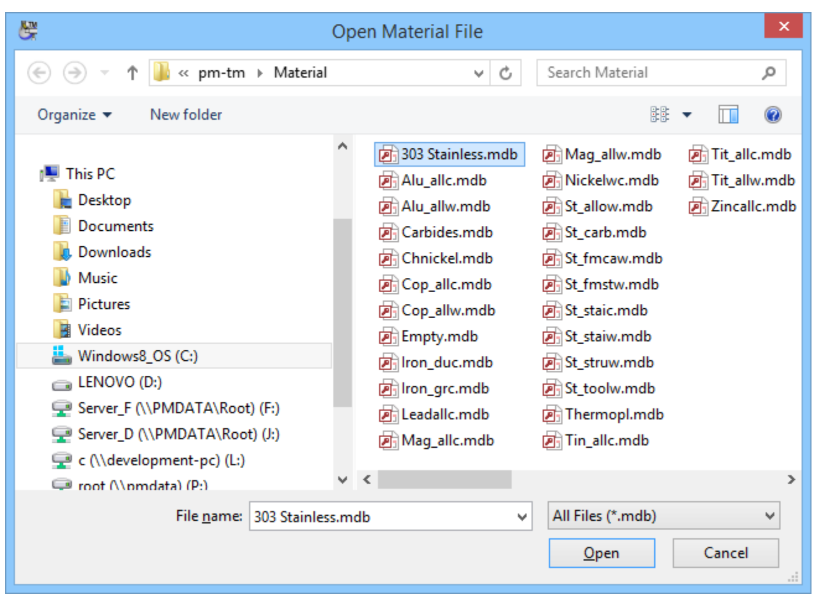
You can customize materials files in **PartMaker** by adjusting the feed and speed charts displayed in the **Material** dialog. To do so:

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

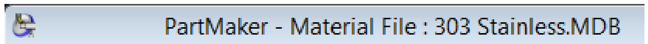


2. Click the third radio button on the right hand side to display the feed chart. By clicking on and dragging the node points lying on the chart or the axes, you can customize feed rates for different sized drill. The same technique can be employed for spindle speeds as well for all the tools listed.

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 "303 Stainless" and click **<Save>** to save the material.

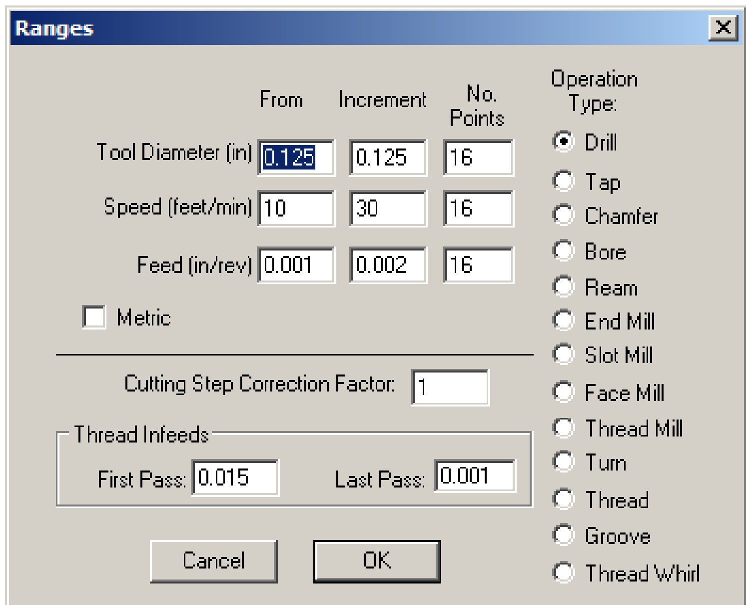


4 You will now notice the new name of the material file at the top of the **Material File** dialog as shown below:



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.*

5 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.



6 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

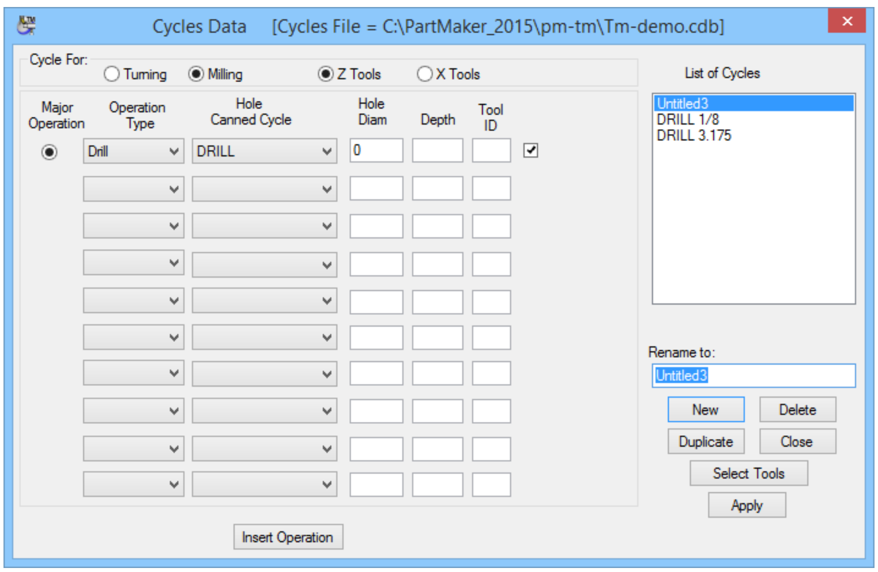
PartMaker lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a Cycle. All cycles are saved in a **Cycles Database**. The Cycles Database can store up to 1,800 cycles. For each cycle operation, you can designate a tool from 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 off-center holes on the face of a part using existing tools in the tools database. This cycle will contain a center drill and a drill.

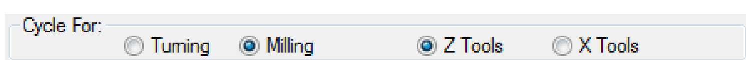
You can access **PartMaker**'s Cycles Database by choosing **Cycles** from the **ToolMinder** menu. Once you are in the Cycle Data dialog, to create a new cycle:

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



2 The next step in creating a Cycle is determining how it will be used. You must determine if the cycle will use on-centerline tools or if it will use “live” tools and if these live tools will be used for face work (Z Tools) or for cross work (X Tools).

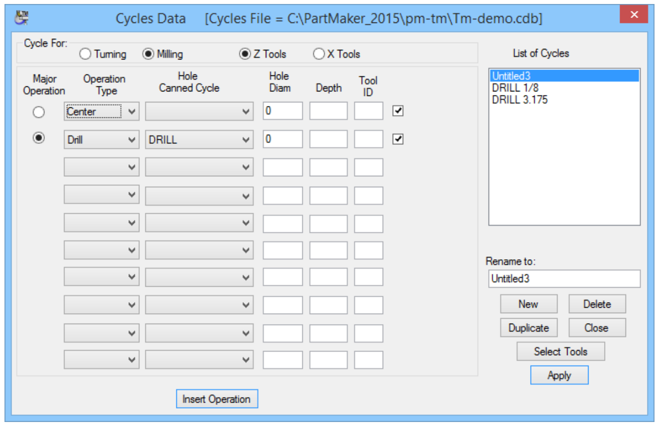
The radio buttons at the top of the cycles dialog allow you to make this determination. By default the Mill and Z Tools radio button are checked.



3 This configuration is correct for the off-center face hole.

4 Next, choose the operations that will comprise the cycle. To do so, choose Center and Drill, respectively, from the Operation Type drop-down menu.

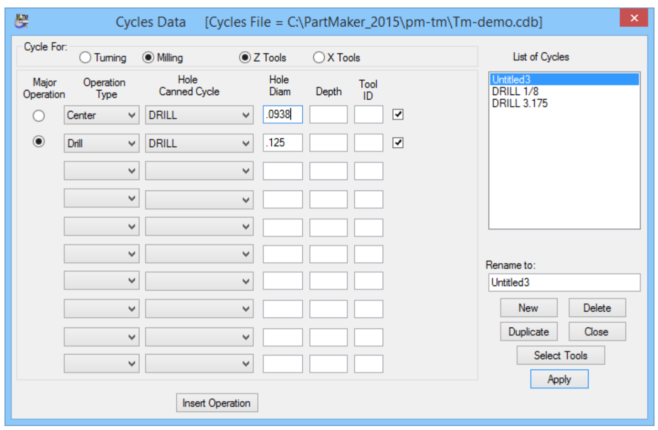
After doing so, your dialog should appear as shown below:



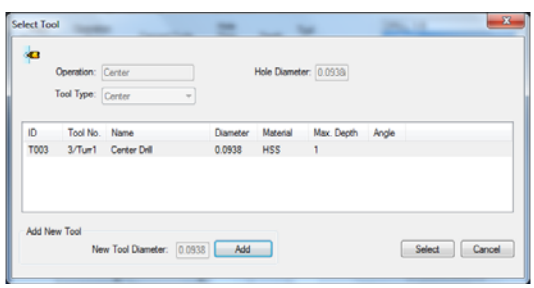
5 Next, choose the corresponding Canned Cycles from the Canned Cycles drop menu for each of these operations, which will be DRILL and DRILL, respectively.

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

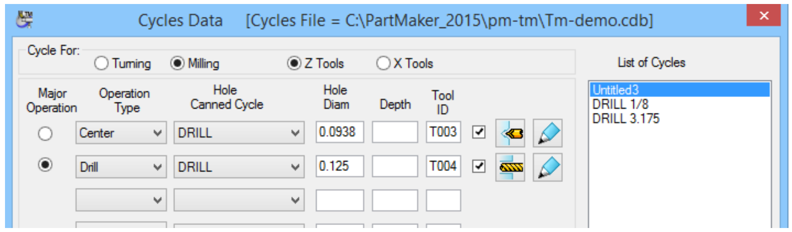
After entering the parameters as specified above, your dialog should appear as shown below:



6 Click the **<Select Tools>** button and **PartMaker** will prompt you for the tools being selected in the tools database:

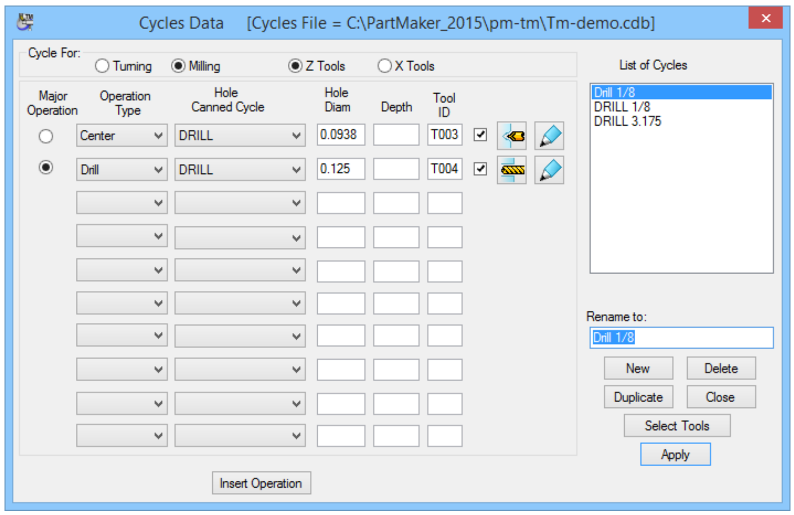


7 Click the **<Select>** button to select each tool from the library. The **Cycles Data** dialog should now appear as shown below:



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

9 The completed Cycle Data dialog should appear as shown below:



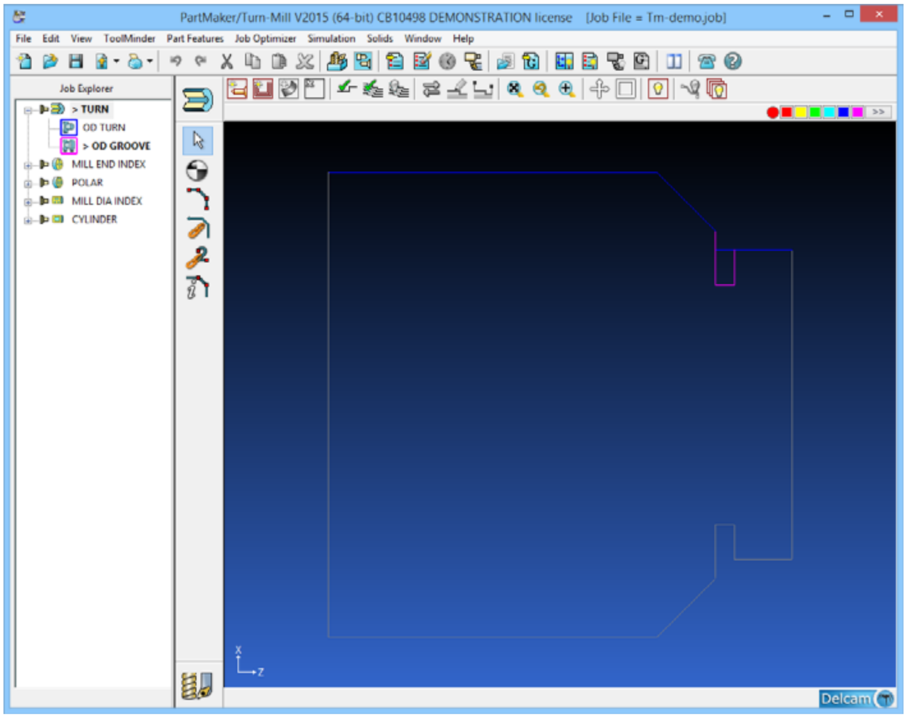
Notice that the column for depth has not been filled out. It has been intentionally left blank in order to allow **PartMaker** to calculate the depths of cut for each tool automatically when applied to the part. This automatic calculation is based on the nominal hole depth and specified geometric characteristics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

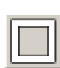
One cycle may be applied to many different jobs, and allowing **PartMaker** to calculate the depth of cut makes each cycle more flexible. It may at times be desirable to enter a value into the depth field. When doing so, be advised that this value will be “hard coded” and that whenever using the cycle,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

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

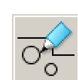
Working in a Face Window

A **Face Window** is the work area in which you define groups machining features or tool paths. The main area of the window displays the part boundaries; the left frame displays the program groups in a tree format, which have the cycle name labels needed to machine the current part. Just to the left of the main screen area displays the Graphics Icons as well as the Short Cut to the Setup dialog icon.



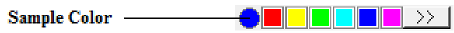
	**Show Axis**	You can display the axes in a **Face Window** by choosing Show Axes from the **View** menu.
	Show Grid	You can display the grid in a **Face Window** by choosing Show Grid from the **View** menu.
	**Show Boundaries**	You can display the part boundaries in a **Face Window** by choosing Show Boundaries from the **View** menu.

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* ***Face Window****, 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 6 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**.

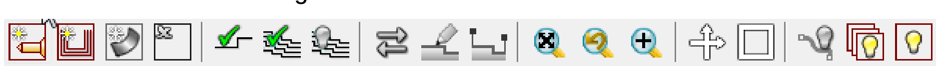
Main Toolbar

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. Each command is explained in greater detail in Chapter 4 of this manual.*

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.

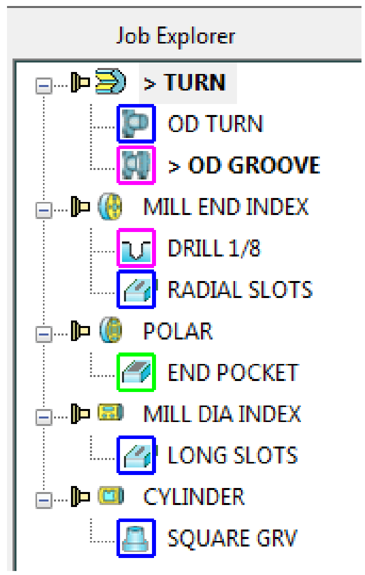
Creating Groups of Part Features

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. Group Symbols are displayed on the left side of the screen on the **Job Explorer Tree** in the CAM mode

To create a new Group Symbol:

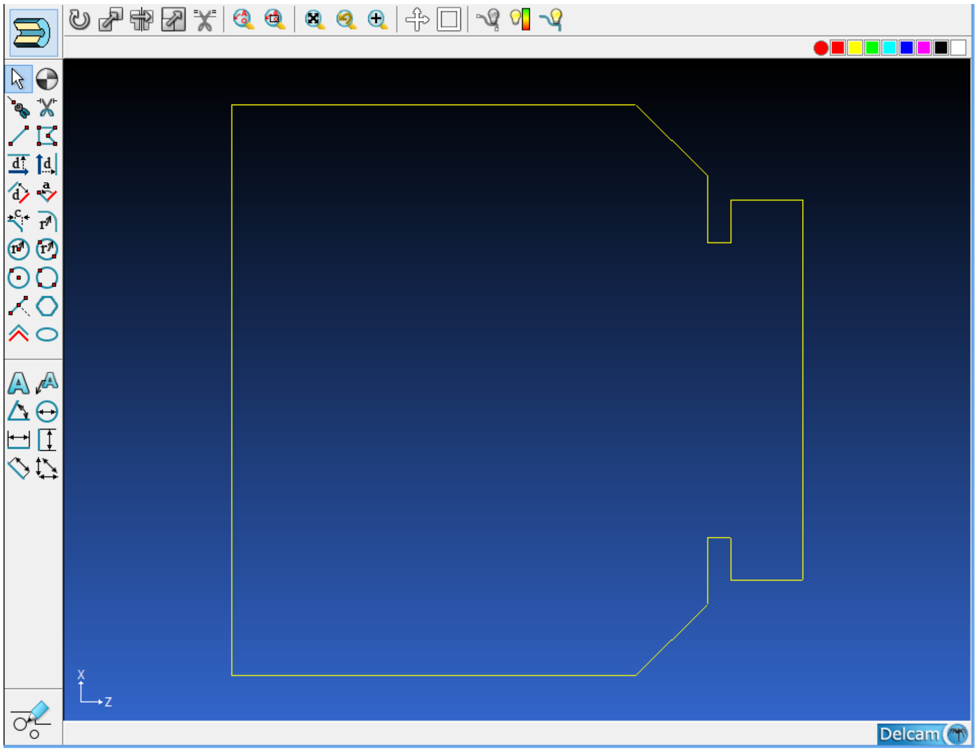
1. Click the desired color on the color bar.
2. Choose **New Profile** or **New Hole Group** (depending on the type of Part Feature being created) from the Part Features menu to display the Group Parameters dialog.

   When you exit the dialog, the new group is created using the color you specified.



Using the Graphics Icons in PartMaker CAD mode

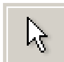
You can create your part drawings in **PartMaker** in the CAD mode. Alternatively, you also import drawings from other CAD systems into **PartMaker** by using the **Import** command from the **File** menu. Imported geometry can be altered in **PartMaker**'s CAD mode.



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 geometry drawing icons are defined below.

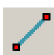


Selection Icon indicates that the system is in the selection mode. In this mode you can select lines, arcs and circles in the Face Window either by clicking on them or dragging a selection rectangle around them.

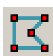
Zero Icon allows you to define a new location for the zero point using Snap Modes.

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.

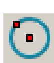
Line on an Angle Icon allows you to create a line on an angle to a cursor-selected line passing through a point specified using Snap Modes.

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

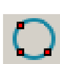
Chamfer Icon allows you to insert a chamfer between two cursor-selected lines or arcs.

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.

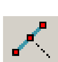


Circle with a Known Center and a Point Icon allows you to define a circle with a center and a point on the circumference specified using Snap Modes.



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.

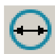
Dimensioning Icons

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

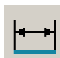
Note Icon Allows you to create a note on a dimensioned drawing.

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.

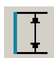
Angular Dimension This icon allows you to insert an Angular Dimension between two lines..



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



Horizontal Dimension Icon This 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 This 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 This 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 This 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 snap mode 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

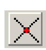
Polar Coordinates: Used for entering Polar coordinates

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

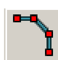
The two icons below will have the same appearance and usage regardless of which machining function **Face Window** you are using.

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.

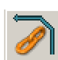
Selected profiles in a Face Window are highlighted; unselected profiles are not highlighted. Selected holes in a Face Window are highlighted; unselected holes are hollow. You can delete selected elements by pressing the <Backspace> key on the keyboard or by choosing the Delete command from the Edit menu.

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

The following icons will have the same function in any given Face Window. However, the list changes depending on which machining function **Face Window** you are using.

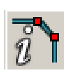


Profile Icon enables you to specify that the currently active group symbol is to be applied to the profile described by either coordinate entry or by clicking on geometric elements.

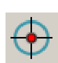


Chain Icon enables you to specify that the currently active group symbol is to be applied to a profile that is created when PartMaker automatically connects all the elements that form an unbroken chain when an initial line or arc is clicked.

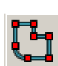
2-Point Chain Icon The 2-Point Chain icon allows you to define a profile by the start point in the chain, and the end point of the chain.



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.



Single Hole Icon enables you to use Snap Modes to define the location on the geometry where a currently active Hole Group Symbol is to be executed.



Define 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 Geometry 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 with an initial line or arc clicked.

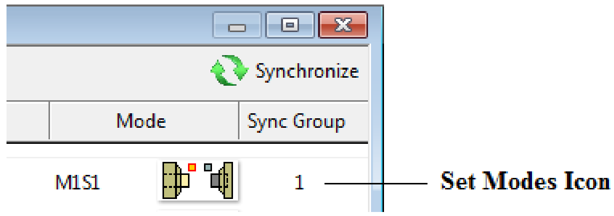
Profile Icon enables you to specify that the currently active group symbol is to be applied to the profile described by either coordinate entry or by clicking on geometric elements. When performing indexing operations, the Profile Icon may only join two points during its usage.

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 with an initial line or arc clicked. When performing indexing operations, the Chain Icon may only join two points during its usage.

Engrave Icon enables you to use the Engrave Data dialog box to create a profile containing numbers and letters.

Process Synchronization in PartMaker TurnMill

After a part has been programmed in **PartMaker** TurnMill, the user has the opportunity to synchronize the processes defined in each of the **Face Windows**. Process synchronization is done visually by **PartMaker** via the **Set Modes** dialog accessed from the **Process Table** by clicking on the **Set Modes** icon to the right of every process.



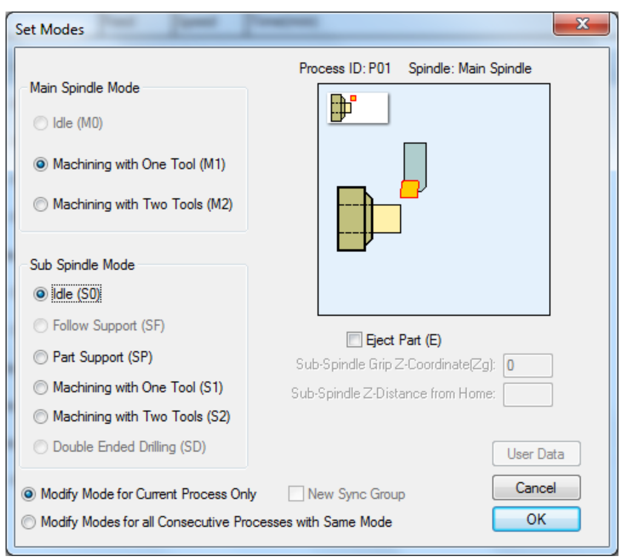
Introduction to Synchronization

Generally speaking, today's TurnMill Centers can allow five (5) types of synchronous machining:

Type 1: The first type of synchronous machining, which can be performed on a TurnMill Center, is actually NOT synchronized at all. This involves machining on one spindle at a time with a single tool while the other spindle remains idle. This type of synchronous operation is represented by the following **PartMaker** modes:

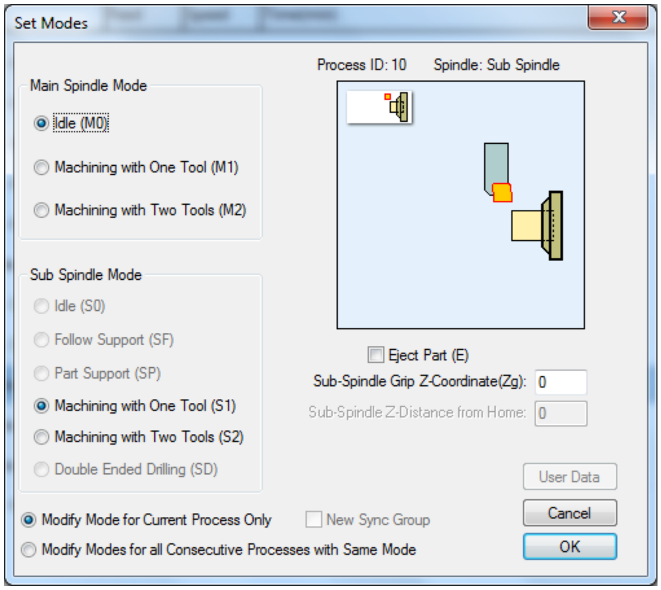
M1S0: Main Spindle Cutting with One Tool, Sub Spindle idle. This mode of operation involves cutting with only one tool on the main spindle, while no machining occurs on the sub-spindle. This mode may be used if a part only requires main spindle work, if all simultaneously occurring operations have already been synchronized, or if the machine does not allow synchronous operation under certain circumstances (i.e. when end working is being performed or when live tools are engaged).

This mode of operation may only be assigned to processes defined in a **Face Window** created for work on the Main Spindle.



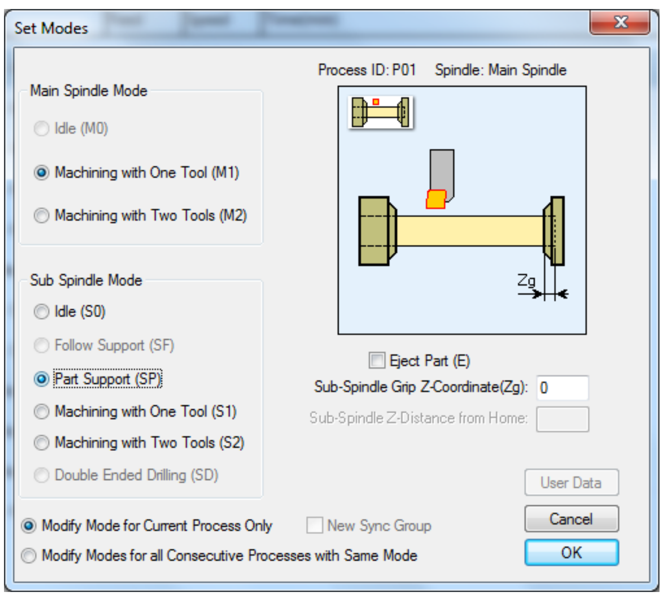
M0S1: Main Spindle Idle, Sub Spindle Cutting with One Tool. This mode of operation involves cutting with only one tool on the sub spindle, while no machining occurs on the main spindle.

This mode of operation may only be assigned to processes defined in a **Face Window** created for work on the Sub Spindle.



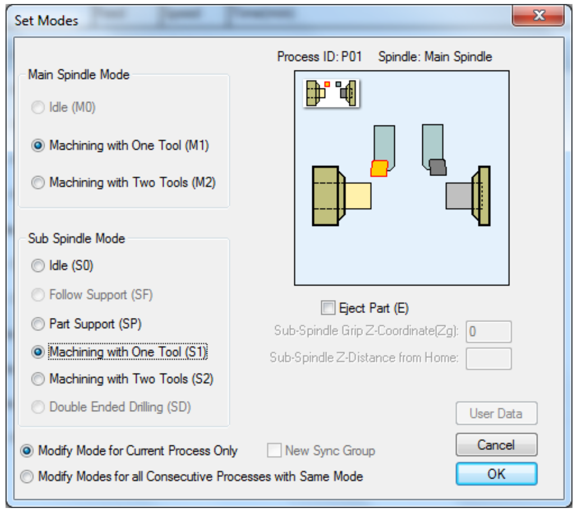
Type 2: One tool cutting while the part is being supported with the sub-spindle. This type of synchronization is most often used during part cut-off and transfer from main to sub spindle. It is also used provide additional support when machining longer parts.

M1SP: Main Spindle Cutting with One Tool, Part Support. This synchronous mode of operation involves the use of one tool cutting while the part is being supported with the sub spindle.



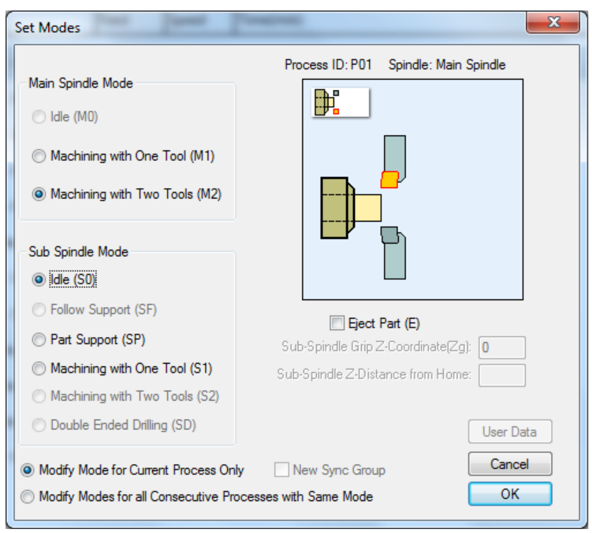
Type 3: Two Tools working on different spindles simultaneously. This type of synchronous operation is represented by the following **PartMaker** modes:

> **M1S1:** Main Spindle Cutting with One Tool, Sub Spindle Cutting with One Tool. This synchronous mode of operation involves the use of two tools on separate turrets (as created in the **PartMaker** Tool Database), cutting on separate spindles.

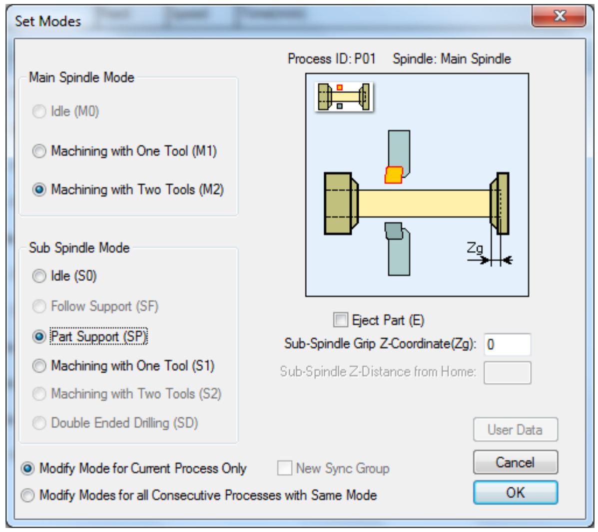


Type 4: Two Tools working simultaneously on the main spindle. This type of synchronization is often referred to as “Pinch Turning” or “Balance Turning” when it involves two outside diameter turning operations. It may also be used when performing turning on the outside diameter and inside diameter (ID boring or drilling on the end of a part) simultaneously. This type of synchronous operation is represented by the following **PartMaker** modes:

> **M2S0:** Main Spindle Cutting with Two Tools, Sub Idle. This synchronous mode of operation involves the use of two tools on separate turrets (as created in the **PartMaker** Tool Database) on the same spindle. These tools can either both be turning tools or one can be a turning tool while the other is a “dead” holemaking tool performing on centerline work on the end of the part.

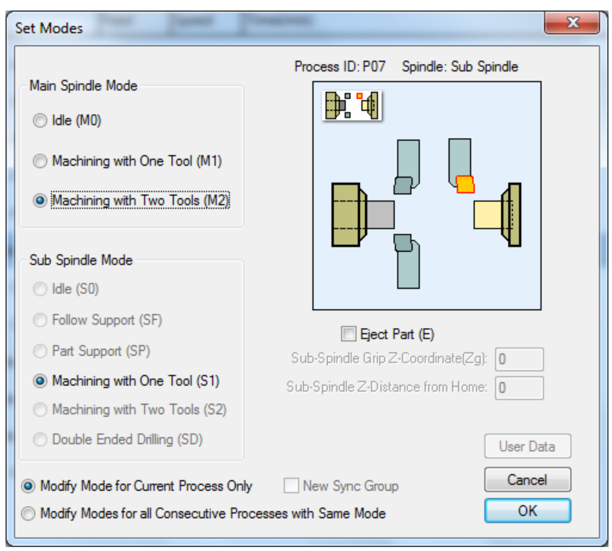


M2SP: Main Spindle Cutting with Two Tools support with Sub-Spindle. This synchronous mode of operation involves the use of two tools on separate tool posts (as created in the **PartMaker** Tool Database) working on the same spindle while the sub-spindle is supporting the stock. In this case, the tools must both be working on the outside diameter.



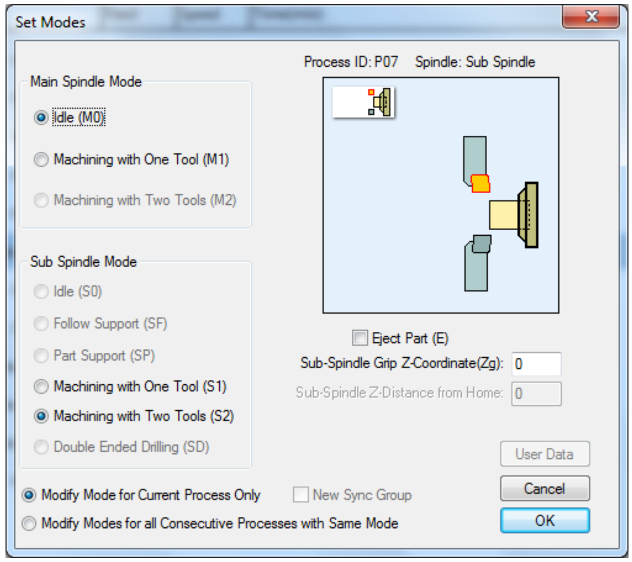
Type 5: Three tools working simultaneously, where two tools are working on the main spindle and one tool is working on the sub spindle. In such a case, two tools can be working simultaneously on the main spindle while a third is working on the sub-spindle. Only the most advanced TurnMill Centers on the market are capable of such operation. This type of synchronous operation is represented by the following **PartMaker** mode:

M2S1: Main Spindle Cutting with Two Tools, Sub Spindle Cutting with One Tool. This synchronous mode operation involves the use of two tools on separate tool posts (as created in the **PartMaker** Tool Database) on the main spindle, while using a third tool mounted on a third tool post on the sub spindle.

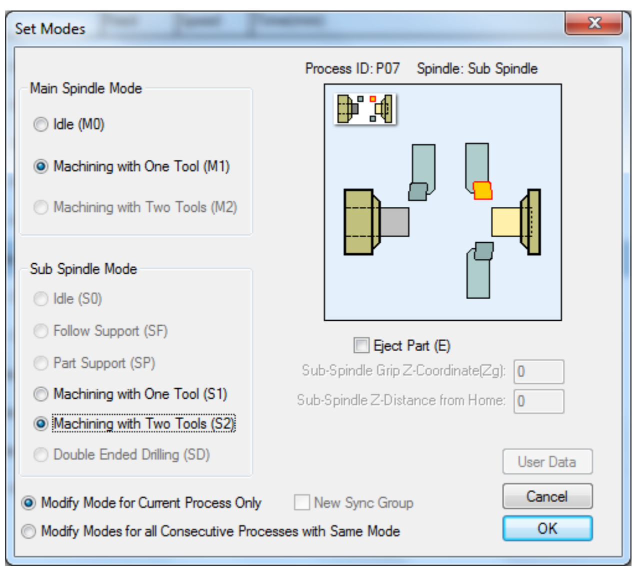


Type 6: Two Tools working simultaneously on the sub spindle. This type of synchronization is often referred to as “Pinch Turning” or “Balance Turning” when it involves two outside diameter turning operations. It may also be used when performing turning on the outside diameter and inside diameter (ID boring or drilling on the end of a part) simultaneously. This type of synchronous operation is represented by the following **PartMaker** modes:

M0S2: Sub Spindle Machining with Two Tools, Main Idle. This synchronous mode of operation involves the use of two tools on separate tool posts (as created in the **PartMaker** Tool Database) on the same spindle. These tools can either both be turning tools or one can be a turning tool while the other is a “dead” hole making tool performing on centerline work on the end of the part.

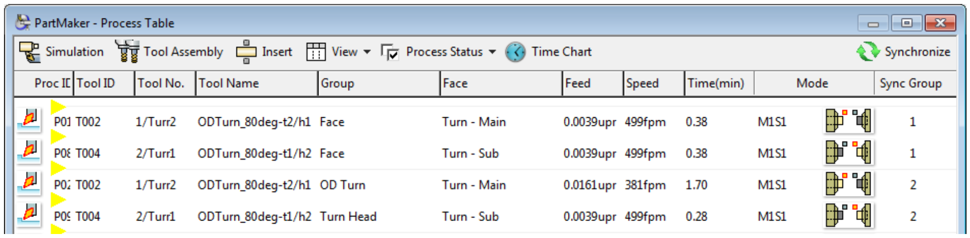


M1S2: Main Spindle Machining with One Tool, Sub Spindle Machining with Two Tools. This synchronous mode operation involves the use of two tools on separate tool posts (as created in the **PartMaker** Tool Database) on the sub spindle, while using a third tool mounted on a third tool post on the main spindle.



Working with Sync Groups

Once the user has chosen the manner in which processes will be synchronized on the Process Table, he has to determine the order in which these processes must be grouped. A group of two or more consecutive processes being executed with the same **PartMaker** Synchronization Mode, as described in Synchronization Types 2 – 4 above, is called a **Sync Group**. Sync Groups are created automatically when the **Synchronize** button is clicked or when the **Generate NC Program** command is chosen. Each Sync Group will have a number assigned to it, denoting separate groups of synchronous operations, such as those shown below:


PartMaker - Process Table

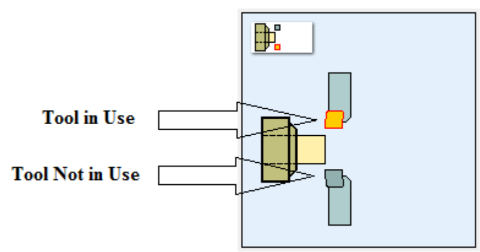
Simulation | Tool Assembly | Insert | View | Process Status | Time Chart | Synchronize

Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h1	Face	Turn - Main	0.0039upr	499fpm	0.38	M1S1	1
P08	T004	2/Turr1	ODTurn_80deg-t1/h2	Face	Turn - Sub	0.0039upr	499fpm	0.38	M1S1	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h1	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1S1	2
P09	T004	2/Turr1	ODTurn_80deg-t1/h2	Turn Head	Turn - Sub	0.0039upr	499fpm	0.28	M1S1	2

PartMaker will perform error checking when the Sync Groups are created. Error checking assures that groups of processes being executed simultaneously have been ordered correctly on the Process Table. This error checking also includes determining whether proper tool post assignments have been made.

Determining the Active Tool in a Sync Group

The user can differentiate between Active and Inactive Tools for any process in a Sync Group by noting the color of the tool in the **Set Modes** dialog assigned to that process. The tool in use during the current process will always be colored in yellow, while the tool that is not cutting will be shaded in gray as shown in the diagram to the right, taken from the **M2S0** Set Modes dialog.



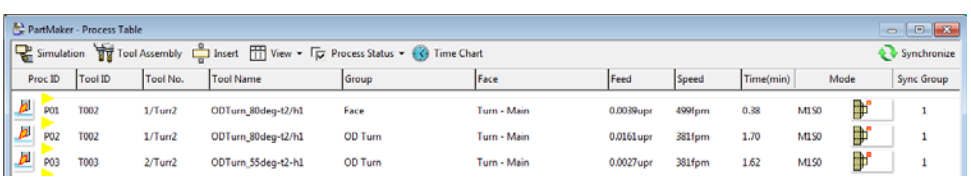
Rules for Grouping Synchronized Operations

Rules for grouping operations assigned various **PartMaker** Modes are explained below:

M1S0: Main Spindle Cutting with One Tool, Sub Spindle idle.

Processes assigned M1S0 modes must involve one or more sequential processes defined in **Face Windows** designated for work on the Main Spindle. Processes may use tools from any tool post capable of working on the main spindle.

Process Table Example:


PartMaker - Process Table

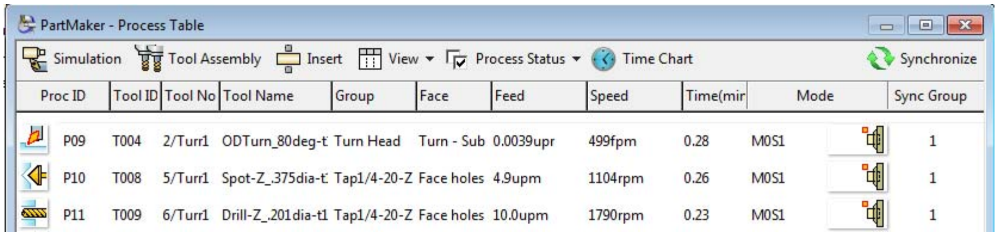
Simulation | Tool Assembly | Insert | View | Process Status | Time Chart | Synchronize

Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h1	Face	Turn - Main	0.0039upr	499fpm	0.38	M1S0	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h1	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1S0	1
P03	T003	2/Turr2	ODTurn_55deg-t2-h1	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M1S0	1

M0S1: Main Spindle Idle, Sub Spindle Cutting with One Tool.

Processes assigned M0S1 modes must involve one or more sequential processes defined in **Face Windows** designated for work on the Sub Spindle. Processes may use tools from any tool post capable of working on the sub spindle.

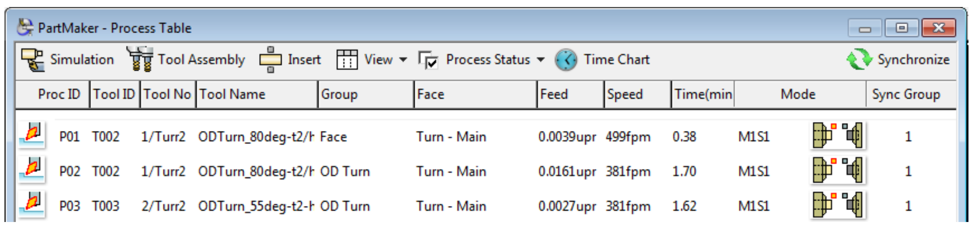
Process Table Example:



Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(mir	Mode	Sync Group
P09	T004	2/Turr1	ODTurn_80deg-t	Turn Head	Turn - Sub	0.0039upr	499fpm	0.28	M0S1	1
P10	T008	5/Turr1	Spot-Z_.375dia-t	Tap1/4-20-Z	Face holes	4.9upm	1104rpm	0.26	M0S1	1
P11	T009	6/Turr1	Drill-Z_.201dia-t1	Tap1/4-20-Z	Face holes	10.0upm	1790rpm	0.23	M0S1	1

M1S1: Main Spindle Cutting with One Tool, Sub Spindle Cutting with One Tool. Processes assigned M1S1 modes must involve one or more sequential processes defined in **Face Windows** designated for work on the Main Spindle followed by one or more sequential processes defined in **Face Windows** designated for work on the Sub Spindle. Within a group of processes labeled M1S1, **PartMaker** will generate an error if the same tool post is used on both a main spindle and sub spindle process. In addition, the stock should either be stationary or moving for all Main Spindle processes in an M1S1 sync group.

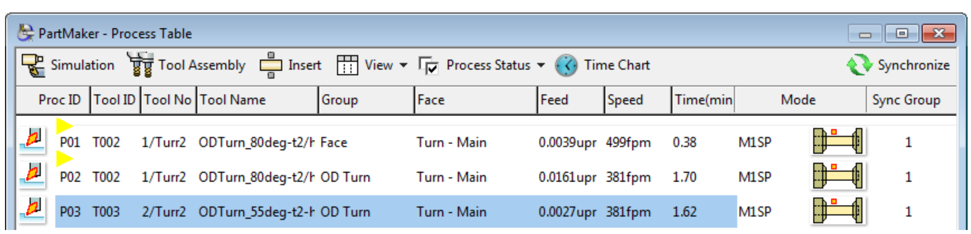
Process Table Example:



Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(min	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/I	Face	Turn - Main	0.0039upr	499fpm	0.38	M1S1	1
P02	T002	1/Turr2	ODTurn_80deg-t2/I	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1S1	1
P03	T003	2/Turr2	ODTurn_55deg-t2-I	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M1S1	1

M1SP: Main Spindle Cutting with One Tool, Sub Spindle providing Part Support (Closed Sub-Spindle Collet) or Non-Following Support (Open Sub-Spindle Collet). Processes assigned M1SP modes must involve one or more sequential processes defined in **Face Windows** designated for work on the Main Spindle. Processes assigned these modes should use tools whose movements are independent of the sub-spindle. In addition, **PartMaker** will display errors if **Sub-Spindle Support Z-Coordinate (Sz)** parameters are not provided, or if negative values are entered in this field.

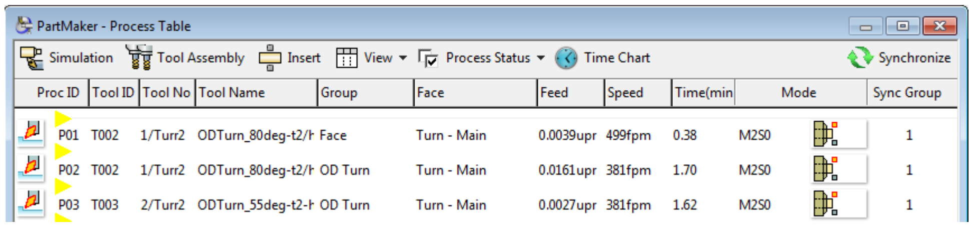
Process Table Example:



Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(min	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/I	Face	Turn - Main	0.0039upr	499fpm	0.38	M1SP	1
P02	T002	1/Turr2	ODTurn_80deg-t2/I	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1SP	1
P03	T003	2/Turr2	ODTurn_55deg-t2-I	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M1SP	1

M2S0: Main Spindle Cutting with Two Tools, Sub Idle. This synchronous mode of operation involves the use of two tools on separate tool posts (as created in the **PartMaker** Tools Database) on the same spindle. Pinch Turning is handled in this mode and the M2S0 mode will be designated automatically on the process table if the **Pinch Turning** box for two groups is checked. Simultaneous OD and ID work is also programmed using this mode.

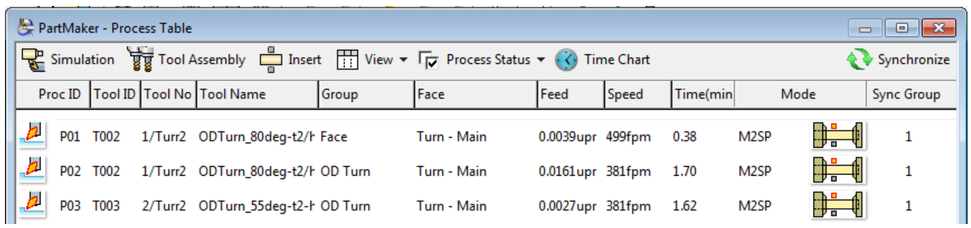
Process Table Example:



Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(min	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h	Face	Turn - Main	0.0039upr	499fpm	0.38	M2S0	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M2S0	1
P03	T003	2/Turr2	ODTurn_55deg-t2-h	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M2S0	1

M2SP: These two synchronous modes of operation observe the same rules as the M2S0 Mode, except that they involve sub-spindle support during machining.

Process Table Example:

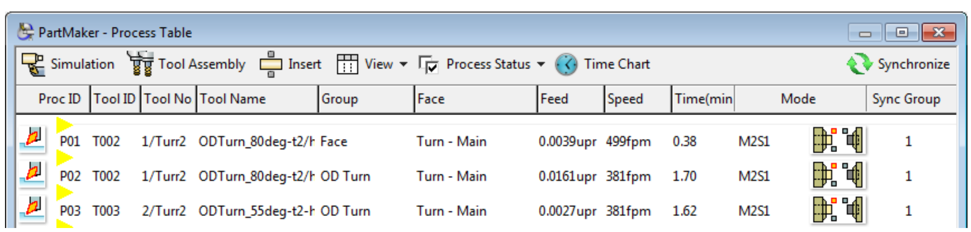


Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(min	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h	Face	Turn - Main	0.0039upr	499fpm	0.38	M2SP	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M2SP	1
P03	T003	2/Turr2	ODTurn_55deg-t2-h	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M2SP	1

M2S1: Main Spindle Cutting with Two Tools, Sub Spindle Cutting with One Tool. A Sync Group with processes assigned the M2S1 mode must involve one processes defined in a **Face Window** designated for work on the Main Spindle involving a **Primary Tool**. This should then be followed by one processes designated for work on the Main Spindle involving a Secondary Tool. Finally, the main spindle operations should be followed by one or more sequential processes defined in **Face Windows** designated for work on the Sub Spindle.

Three different turrets should be used within a Sync Group involving the M2S1 mode.

Process Table Example:



Proc ID	Tool ID	Tool No	Tool Name	Group	Face	Feed	Speed	Time(min	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h	Face	Turn - Main	0.0039upr	499fpm	0.38	M2S1	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M2S1	1
P03	T003	2/Turr2	ODTurn_55deg-t2-h	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M2S1	1

More Information on Sync Groups

You can learn more about using and applying Sync Groups for your machines by referring to the machine specific addendum, which came with your **PartMaker** TurnMill software. This addendum contains information about machine-specific synchronization techniques as well as which synchronous modes of operation your post processors support.

Material Control Processes

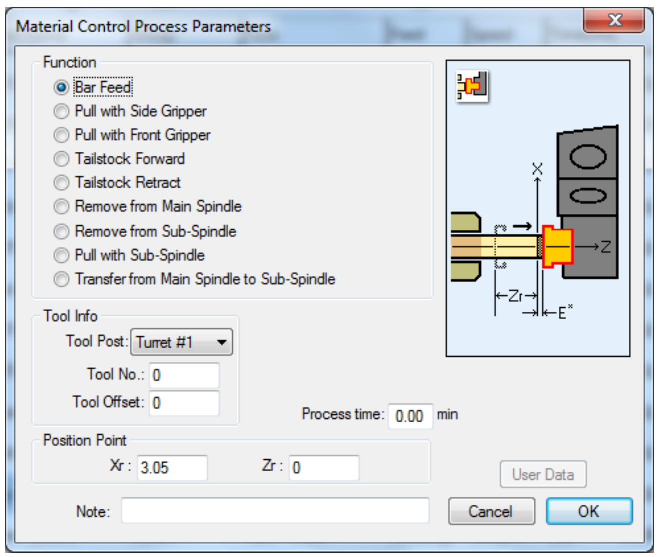
Material Control Processes (MCPs) are turning processes which, do not involve material removal but do require NC programming. MCPs can be inserted into the Process Table via the **<Insert Material Control Process>** command under the **Job Optimizer** menu. Upon insertion into the Process Table, MCPs 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 TurnMill 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 Material Control Process 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 Post: Choose the turret at which the Bar Stop is mounted.

Turret #1 – Defined as the upper turret for slant bed lathes or the front turret for flat bed lathes.

Turret #2 - Defined as the lower turret for slant bed lathes or the rear turret for flat bed lathes.

Turret #3 – Defined as the third turret for three turret lathes – Please contact **PartMaker** Technical Support for more information on three-turret capability.

Note: *If using a lathe with only one turret, select* ***Turret #1.***

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.

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.

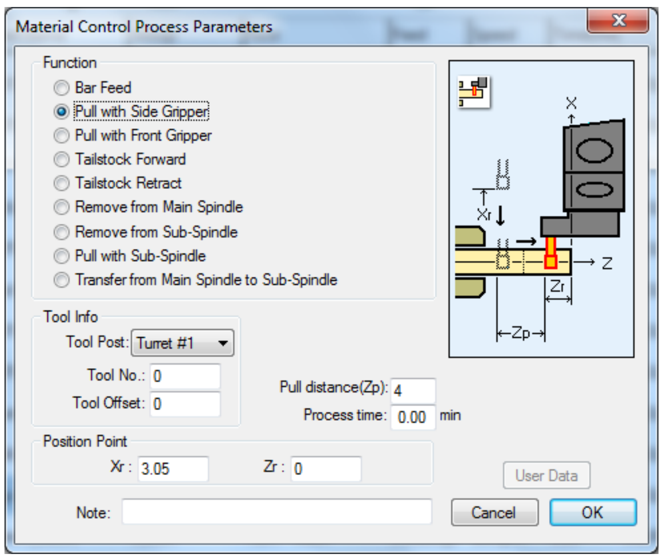
***Note:** If using the cutoff tool as the Bar Stop, enter a negative x coordinate in order to move the cutoff tool insert below the stock diameter.*

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.

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.

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.



Tool Post: Choose the turret at which the Side Gripper is mounted.

Turret #1 – Defined as the upper turret for slant bed lathes or the front turret for flat bed lathes.

Turret #2 - Defined as the lower turret for slant bed lathes or the rear turret for flat bed lathes.

Turret #3 – Defined as the third turret for a three turret lathe – Please contact **PartMaker** Technical Support for more information on three turret capability.

Note:** If using a lathe with only one turret, select **Turret #1.

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.

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 that the bar stock diameter.

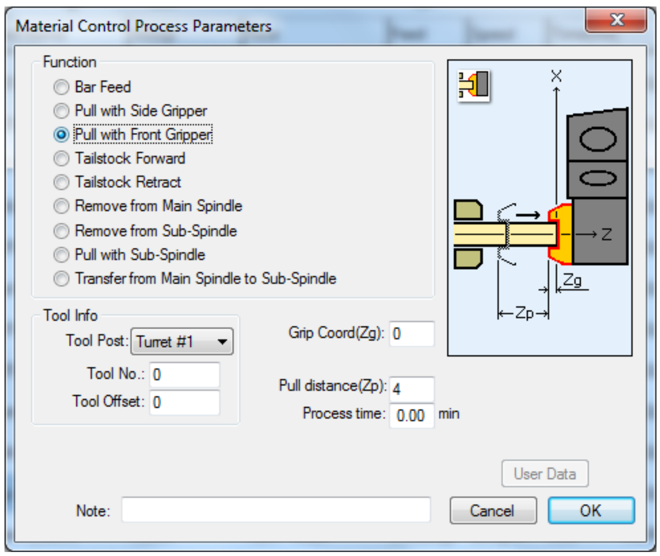
Zr: Enter an incremental z distance (measured from the final cutoff z coordinate) to position the Side Gripper above the bar stock before it moves in to clamp the bar.

Pull 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.

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.



Tool Post: Choose the turret at which the Front Gripper is mounted.

Turret #1 – Defined as the upper turret for slant bed lathes or the front turret for flat bed lathes.

Turret #2 - Defined as the lower turret for slant bed lathes or the rear turret for flat bed lathes.

Turret #3 – Defined as the third turret for a three turret lathe – Please contact **PartMaker** Technical Support for more information on three turret capability.

Note: *If using a lathe with only one turret, select* ***Turret #1.***

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.

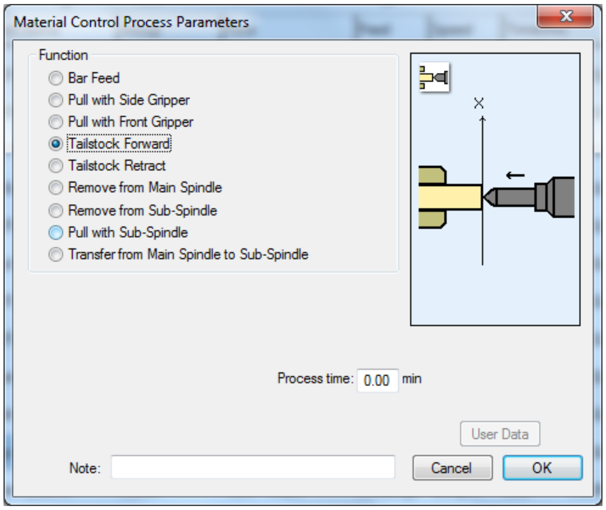
Grip Coord (Zg): Enter an incremental z distance (measured from the face of the bar stock) to where the Front Gripper is to clamp on to the bar.

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

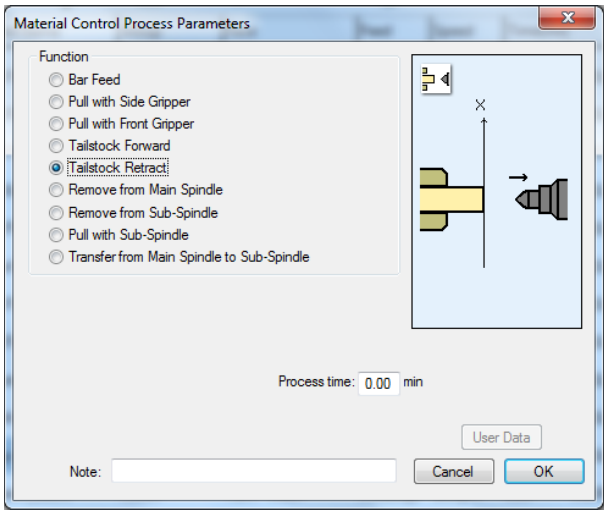
The Tail Stock Forward Material Control Process allows the user to insert code for bringing a tailstock forward for support as a steady rest. This MCP is used for machines equipped with a hydraulic Tailstock. The tailstock will be in the advanced position until the Tailstock Retract MCP is activated.



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

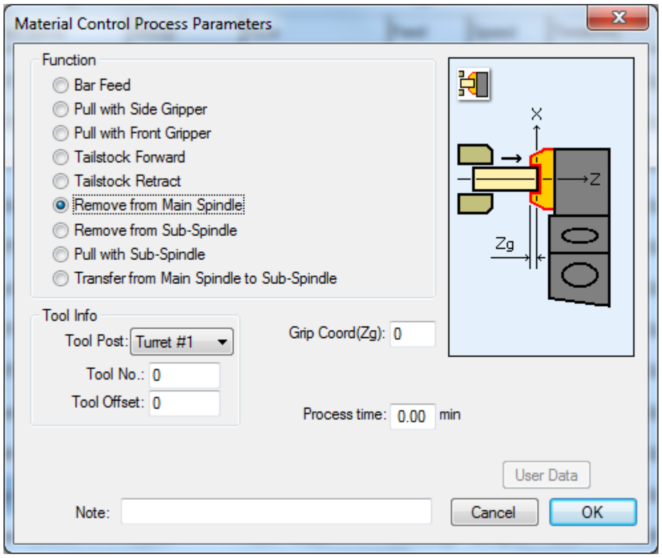
This Tailstock Retract Material Control process is used for machines equipped with a hydraulic tailstock. If selected, the tailstock retracts away from the face of the part. The tailstock will be in the retracted position until **Tailstock Advance** is activated.



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 Post: Choose the turret at which the Part Ejector is mounted.

Turret #1 – Defined as the upper turret for slant bed lathes or the front turret for flat bed lathes.

Turret #2 - Defined as the lower turret for slant bed lathes or the rear turret for flat bed lathes.

Turret #3 – Defined as the third turret for a three turret lathe – Please contact **PartMaker** Technical Support for more information on three turret capability.

Note: *If using a lathe with only one turret, select **Turret #1.***

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

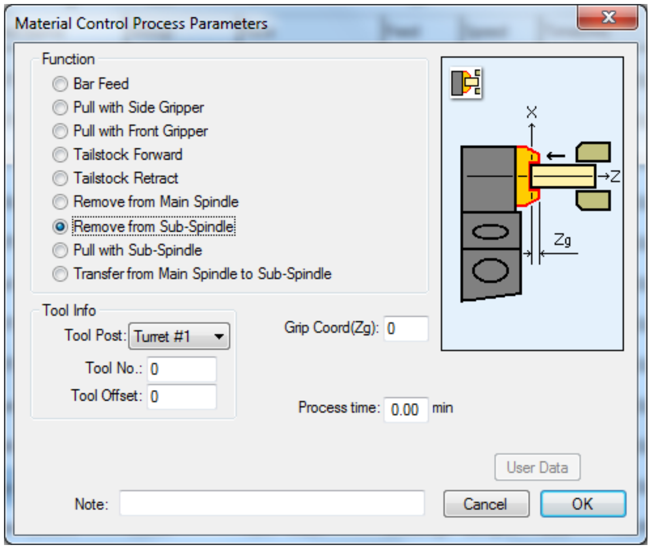
Tool Offset: Enter the offset number for the Part Ejector. The number must be a one or two digit integer.

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.

Remove from Sub Spindle

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



Tool Post: Choose the turret at which the Part Ejector is mounted.

Turret #1 – Defined as the upper turret for slant bed lathes or the front turret for flat bed lathes.

Turret #2 - Defined as the lower turret for slant bed lathes or the rear turret for flat bed lathes.

Turret #3 – Defined as the third turret for a three turret lathe – Please contact **PartMaker** Technical Support for more information on three turret capability.

Note: *If using a lathe with only one turret, select* ***Turret #1.***

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

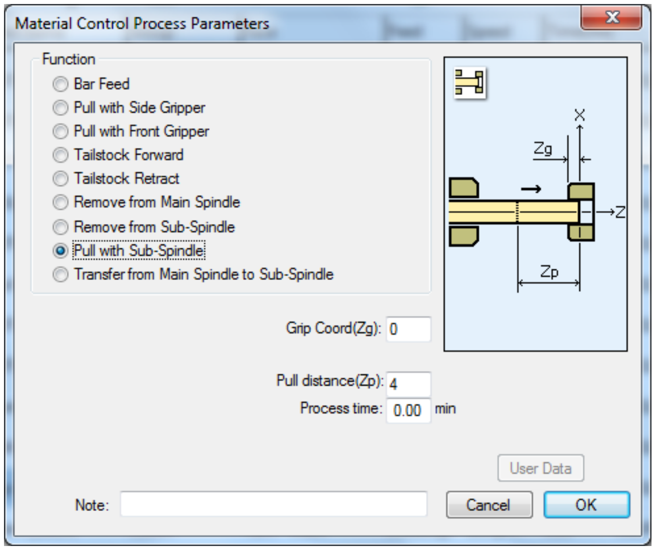
Tool Offset: Enter the offset number for the Part Ejector. The number must be a one or two digit integer.

Grip Coord (Zg): Enter an incremental z distance (measured from the face of the work piece on the Sub 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.

Pull with Sub Spindle

The Pull with Sub Spindle Material Control Process allows the user to insert code for drawing bar stock out of the main spindle by clamping onto the part with the sub-spindle.



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

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

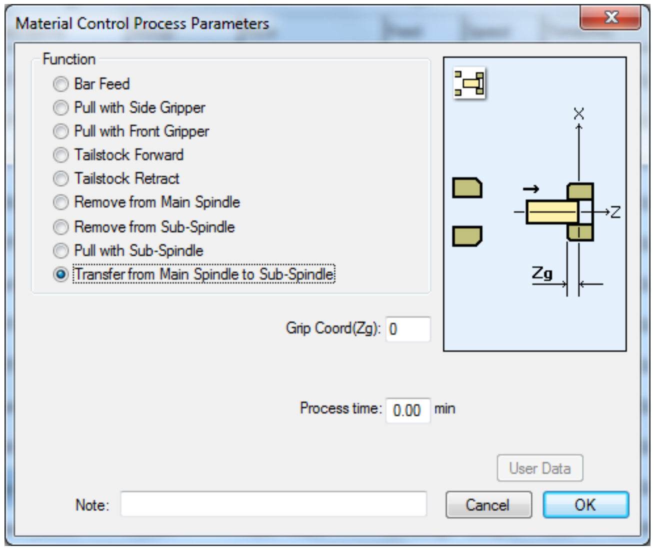
Note: *If pulling right before a cutoff/transfer process, the value entered should be the [part length + cutoff insert width + excess stock for facing on Sub Spindle (if necessary)]*

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

Transfer from Main Spindle to Sub Spindle

The Transfer from Main Spindle to Sub Spindle Material Control Process allows the user to insert code for transferring a work piece from the main to the sub spindle. This MCP should only be used when working with slug parts.

Note: Use the M1SP Mode for performing part transfers when working with Bar Stock.



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

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

Chapter 3: TurnMill Tutorial Part: Single Spindle

Introduction

This tutorial will help you learn the steps to go through in using PartMaker to program TurnMill parts. The emphasis in this tutorial will be on choosing the correct Machining Function when programming parts on a TurnMill center. Once you have completed this chapter, you may choose to go on to Chapter 4, which contains another example focusing more on process synchronization and work on main and sub spindles.

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.

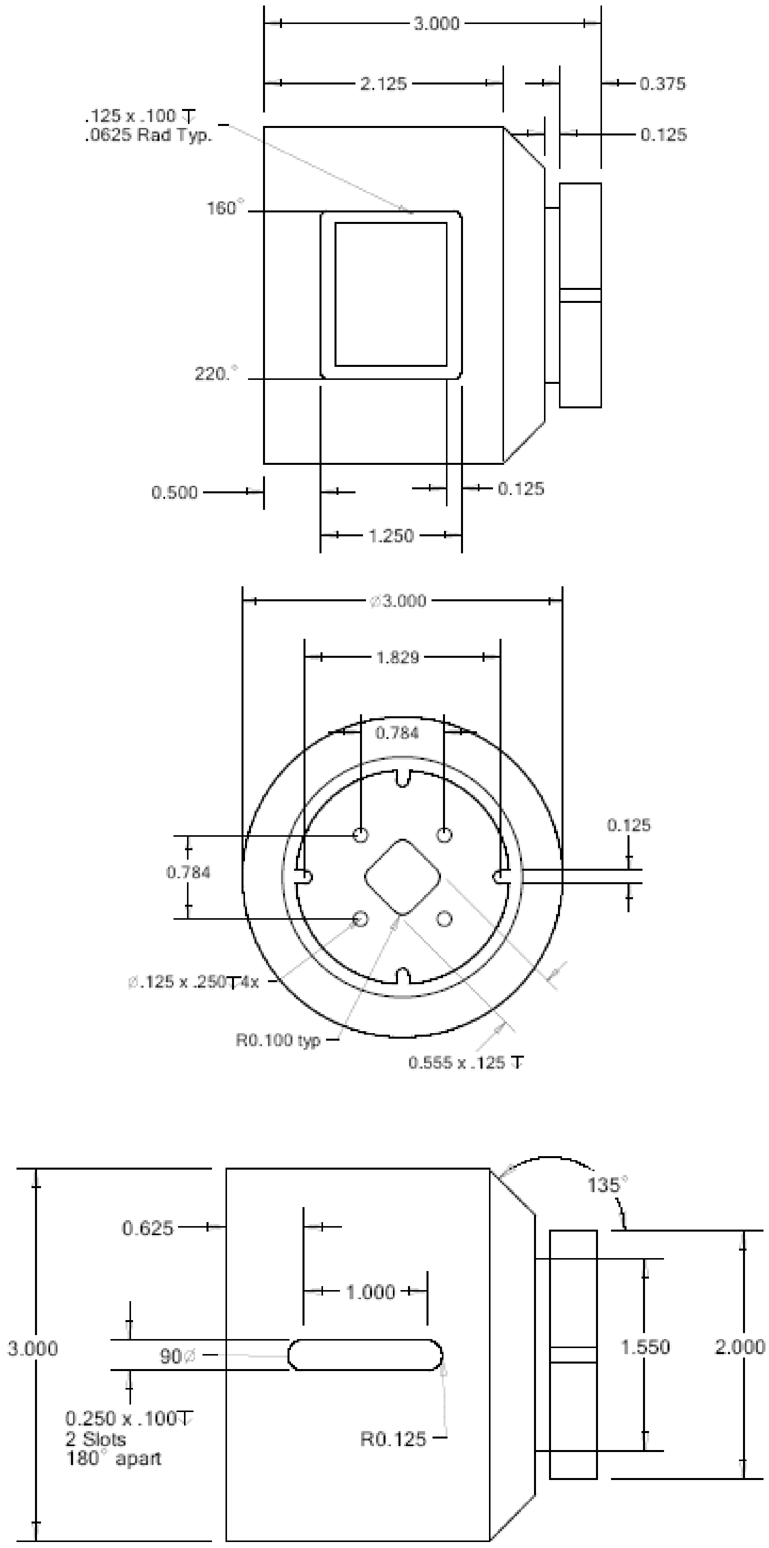
How You Will Create the TurnMill Tutorial Part

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

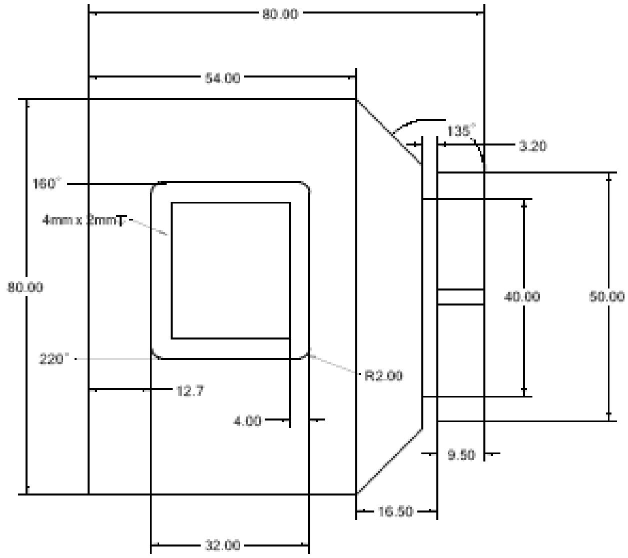
- √ Start **PartMaker**
- √ View tool information for the part
- √ View cycle information for the part
- √ Select a material for the part
- √ Setup defaults for TurnMill
- √ Define the stock boundaries for the face window
- √ Open a new face window
- √ Create Geometry
- √ Perform Turning operations
- √ Perform Milling operations on the end of the part using C-axis indexing
- √ Perform polar milling operations
- √ Mill slots on the cylinder using C-axis indexing
- √ Perform continuous C-axis milling
- √ Generate the Process Table
- √ Simulate the cutting of the part
- √ Generate an NC Program

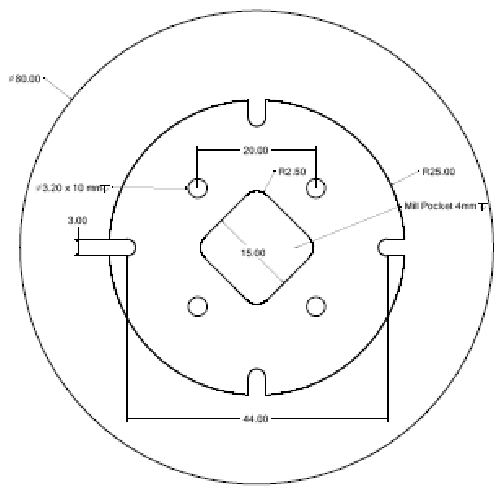
The print below shows the part to be programmed in this tutorial.

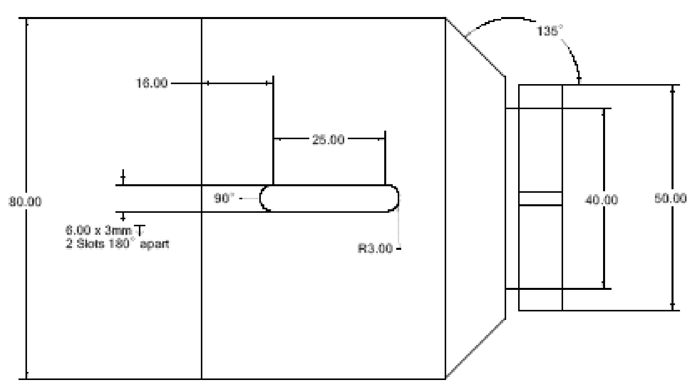
Inch Units



Metric Units







Getting Started

The first part of any programming exercise with PartMaker/TurnMill is to decide which machining functions will be used. The machining functions used in this tutorial are as follows:

Turning

Use conventional turning tools and programming techniques to create the basic OD contour of the part.

Mill End - Index

Use a powered drill and the machine's C-axis indexing capabilities to drill the four holes arranged in the square pattern shown.

Use a powered end mill with C-axis indexing to create the four radial slots.

Mill End - Polar

Create the rectangular pocket shown in the end of the stock. Programming of this pocket will use the simultaneous X and C-axis feed capabilities of the turn mill center.

Mill Diameter - Index

Use an X oriented, powered end mill, in conjunction with C-axis indexing to create the two longitudinal slots oriented at 90 and 270 degrees on the OD of the part.

Mill Cylinder

Use an X-axis oriented, powered end mill combined with X and C axis simultaneous feed to cut the rectangular groove on the OD of the cylinder.

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 TurnMill part can be found in your PartMaker TurnMill directory under the following file names:

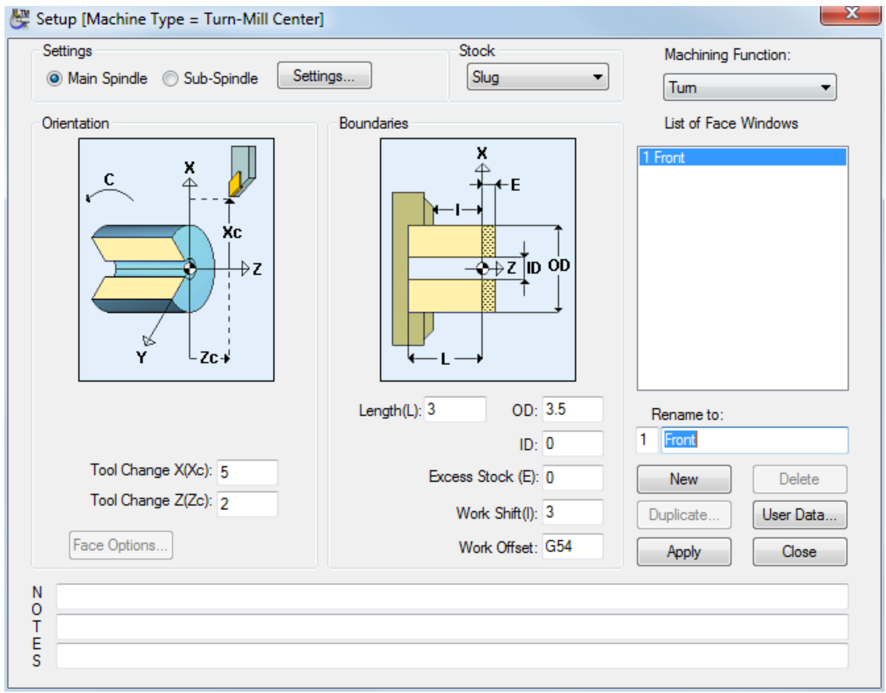
Completed Job File:	C:\PartMaker\PM-TM\Tm-demo.job
Completed Tools Data Base:	C:\PartMaker\PM-TM\Tm-demo.tdb
Completed Cycles Data Base:	C:\PartMaker\PM-TM\Tm-demo.cdb

In the TurnMill tutorial, the default **Tools Data Base** and **Cycles Data Base** will be the same as Tm-demo.tdb and Tm-demo.cdb found in the **PM-TM** root directory. Although it is shown, you will not need to open a tools or cycles database at this point.

Starting PartMaker TurnMill

All programming for TurnMill parts done with **PartMaker** should be done in the TurnMill application. To start the **PartMaker** TurnMill application:

1. Double click on the TurnMill icon on your desktop or choose **TurnMill** from the **PartMaker** Group accessed through Programs under the Windows Start button.
2. An icon containing the **PartMaker** logo will appear for a few moments. You will then see the **Setup** definition dialog box shown below. Click the **<Close>** button.

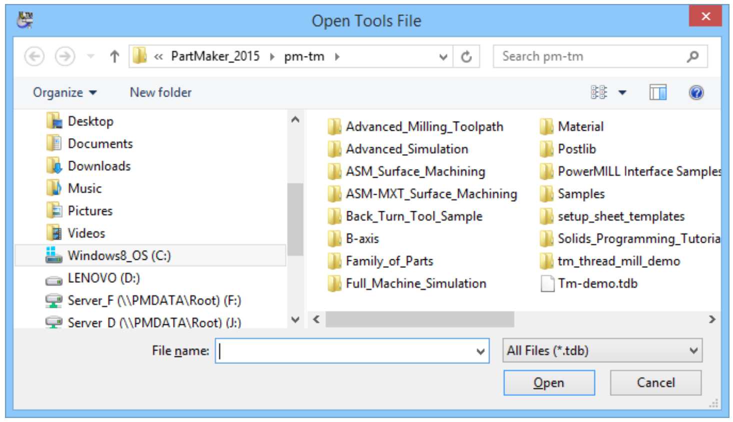


Opening the Tools file

The tool library for the TurnMill tutorial has been created.

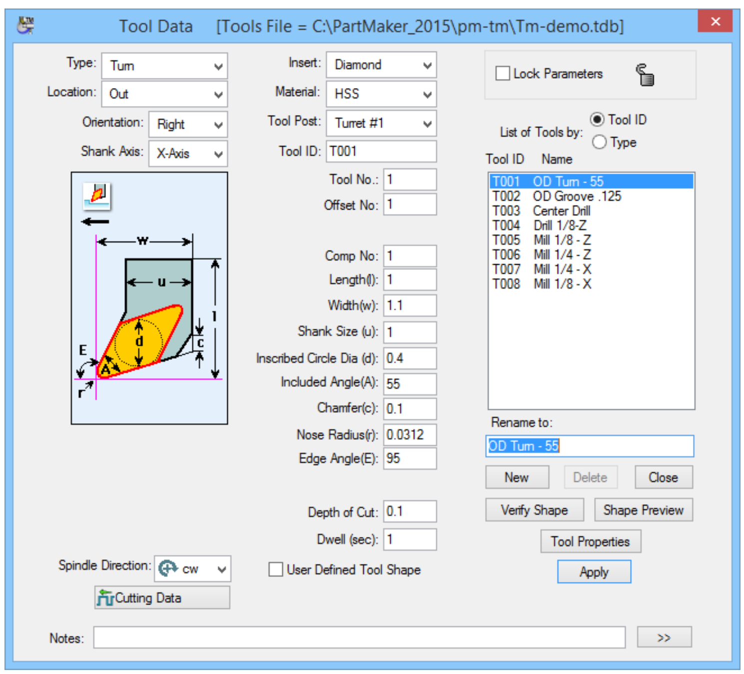
1 Choose **Open Tools File** from the **File** menu.

2 Open the **Tm-demo.tdb** from the **C:\PartMaker\PM-TM** directory as shown below:

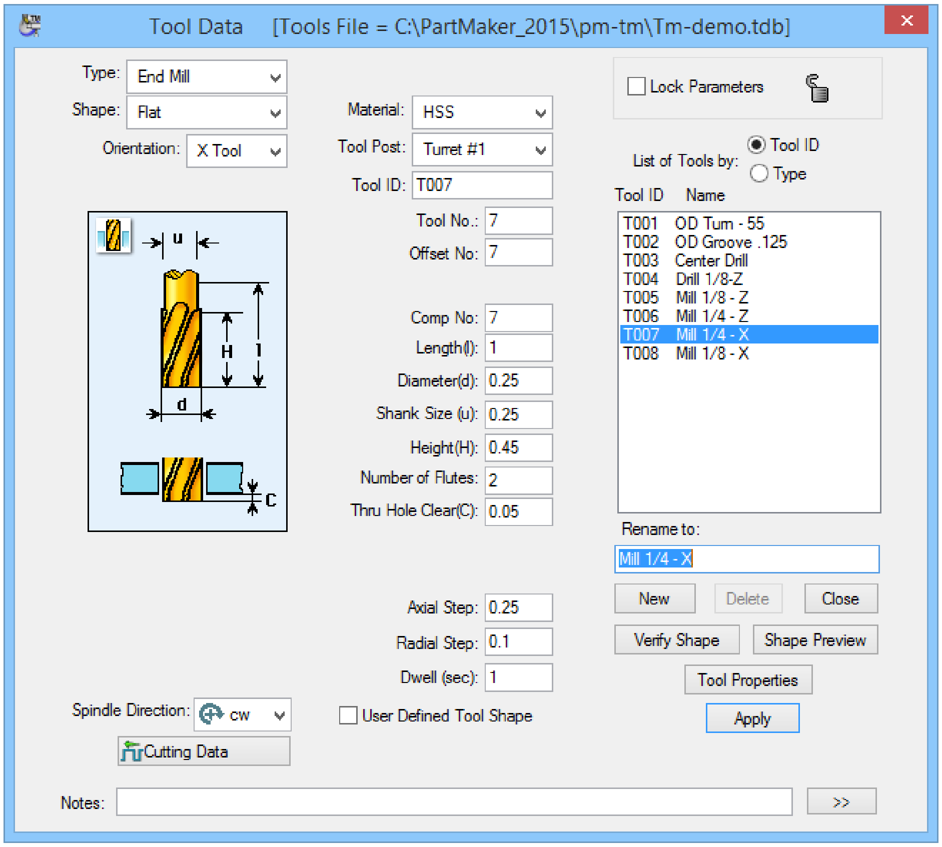


3 To view the tools, choose **Tools** from the **ToolMinder** menu.

Look at the first two tools in the library. Notice **T001** below is an OD turning tool. It is setup in the same manner of OD Turning as those used in conventional lathe operations.



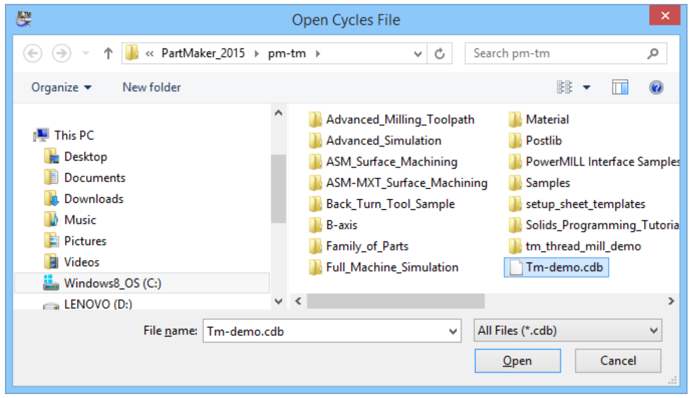
Next, select **T007**. Notice the **Orientation** field. In PartMaker TurnMill, when defining milling tools, you must be careful to designate their orientation, either along the X-axis or Z-axis.

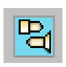


Opening the Cycles file

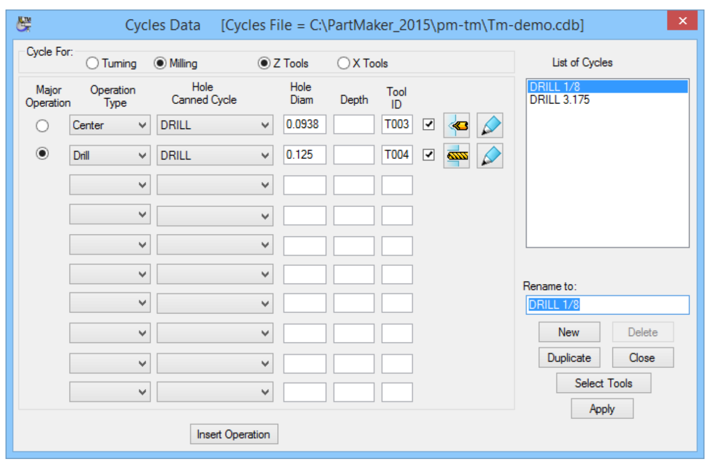
The cycles library for the TurnMill tutorial has also been created.

1. Choose **Open Cycles File** from the **File** menu.
2. Open the **Tm-demo.cdb** from the **C:\PartMaker\PM-TM** directory as shown below:





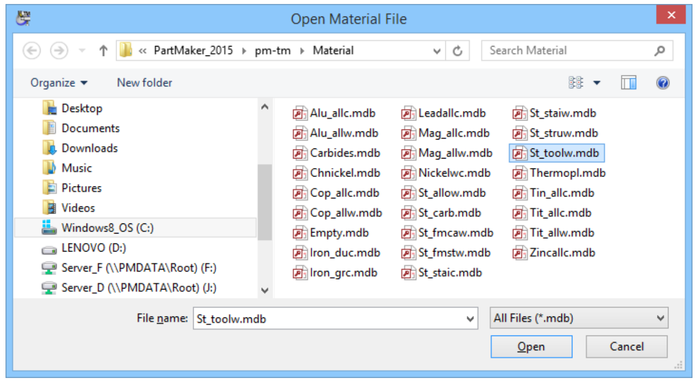
3. To view the cycles, choose **Cycles** from the **ToolMinder** menu. As with tool definition, in PartMaker TurnMill, when defining holemaking cycles, you must be careful to designate their orientation, either along the X-axis or Z-axis. The cycle below will be used for machining holes on the face of the part. You must also designate whether a cycle will be used in a **Face Window** of machining function Turn or one involving milling.



Selecting a Material for the Part

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

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



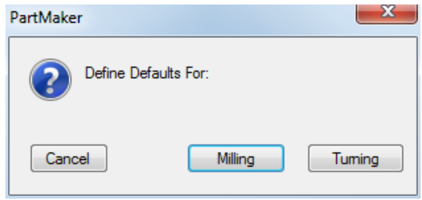
You have now loaded the tools, cycle, and material files that were previously developed for the TurnMill tutorial. PartMaker has also provided a completed job file. At this point you will not open the job file. You will be guided through a step-by-step process to develop a job file for the tutorial.

Setting Defaults

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

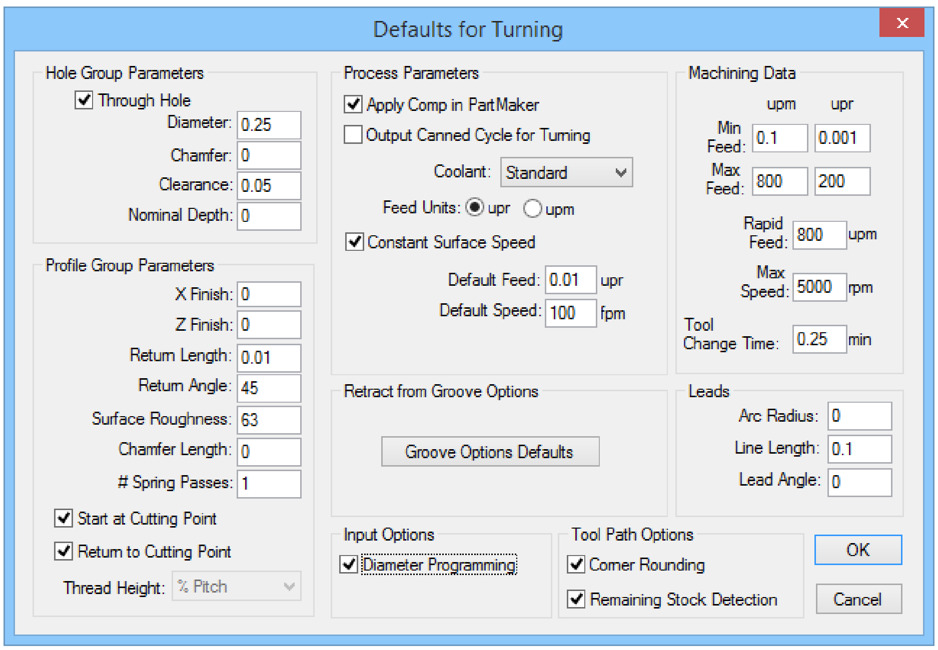
1 To access the **Defaults** dialog, select **Defaults** from the **Job Optimizer** menu.

2 After doing so you will see the dialog below:



Click the <**Turning**> button to access the **Turning** defaults dialog.

3 After doing so, you will see the dialog below:



4 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.

5 Set a default **Line Length** for your programmed **Leads**. For the TurnMill tutorial this value should be 0.100 (2.5).

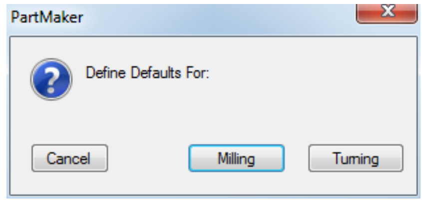
Note: *Setting a generally applicable lead as a default will save you a great deal of time when programming because you will not have to reenter the same data from feature to feature.*

6 Set the **Thread Height:** There are two options available:

% Pitch. The user will enter the height as a percent and resulting height will be Pitch * Thread Height / 100.

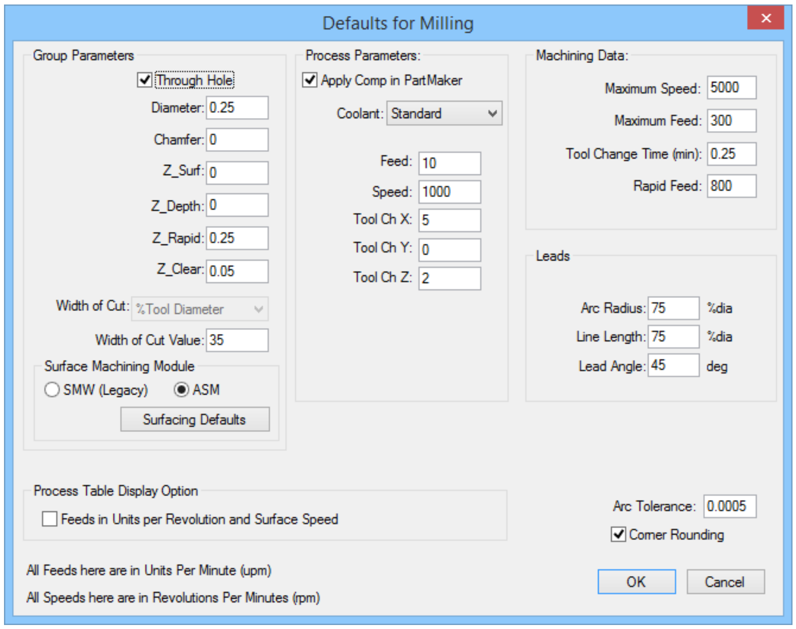
Part Units. The user will enter the actual measured height of the thread in the current part units.

7 Access the **Defaults** dialog, select **Defaults** from the **Job Optimizer** menu.



Click the <**Milling**> button to access the **Milling** defaults dialog.

Set the Milling default as shown.



Width of Cut. This parameter specifies whether the "**Width of Cut**" field in the **Mill Profile Group Parameters Dialog** will be defined in terms of percentage of tool diameter or an absolute value.

Once you have looked at the tools and cycles databases, selected a material for the job, and setup the defaults for your job as shown above, you can move on to begin programming the TurnMill tutorial part.

Turning

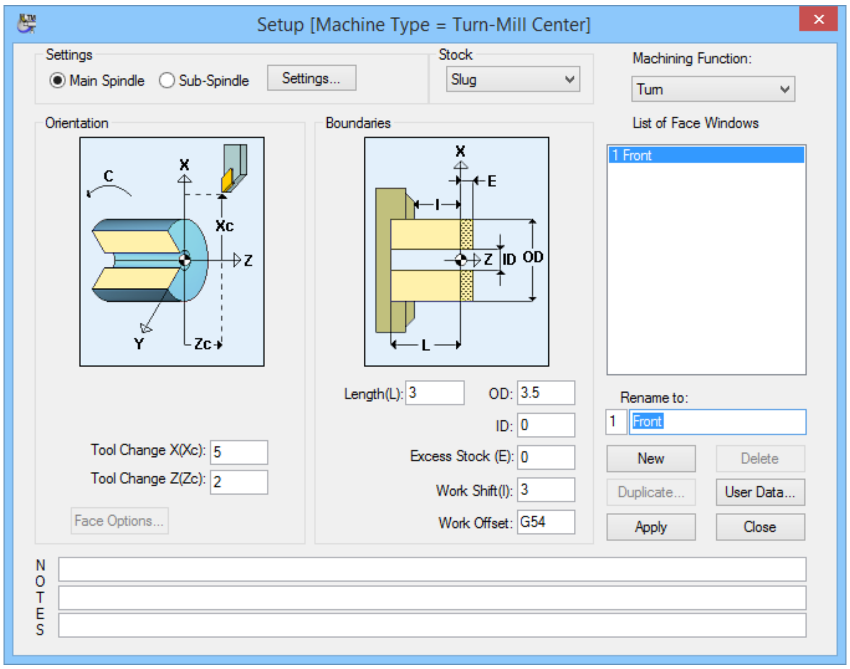
This section of the tutorial will instruct you in the steps necessary to perform the turning operations.

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:

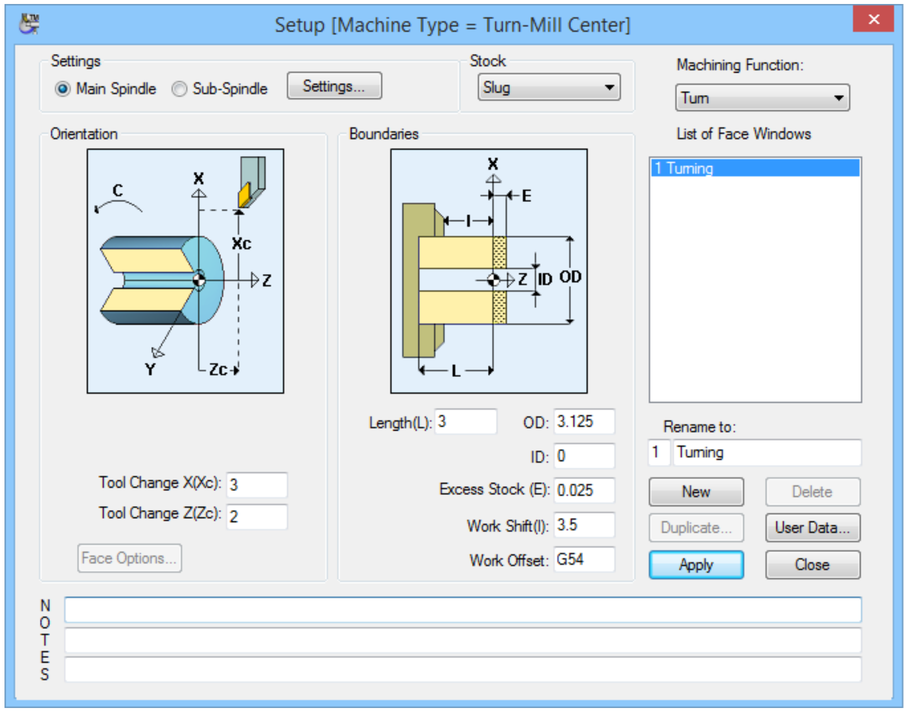


2 Rename the current face window by clicking on **Front** in the **List of Face Windows:** field and type **Turning** in the **Rename To:** field and click the **<Apply>** button.

3 Now enter the parameters listed below in the appropriate fields:

Length (L)	3.0	(80.0)
OD	3.125	(85.0)
ID	0	(0)
Excess Stock (E)	0.025	(0.5)
Work Shift(I):	3.5	(100)
Work Offset:	G54	
Tool Change X (Xc)	3	(100)
Tool Change Z (Zc)	2.0	(100)

When you are finished, your **Setup** dialog should appear as shown 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 in the next section of the tutorial.

Creating Geometry in CAD Mode

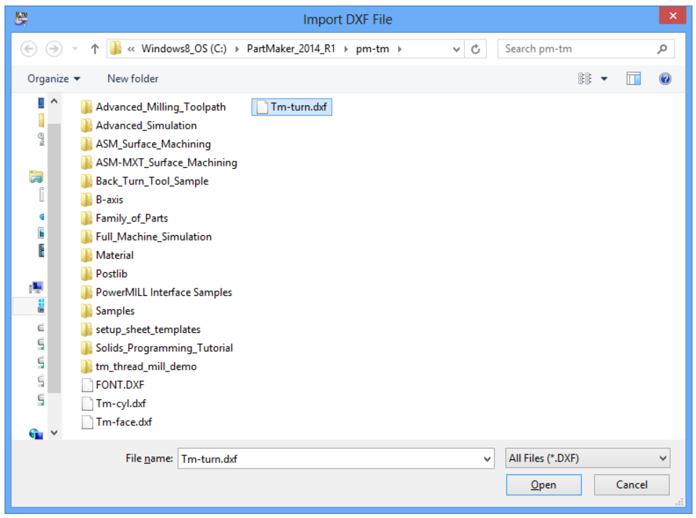
In this section, you will create the geometry for the turning operations.

Since you should already be familiar with creating turning geometry from programming lathes with PartMaker, import the geometry for this operation from a .dxf file. The .dxf file is called **TM-TURN.DXF** and is located in the **\PM-TM** directory.

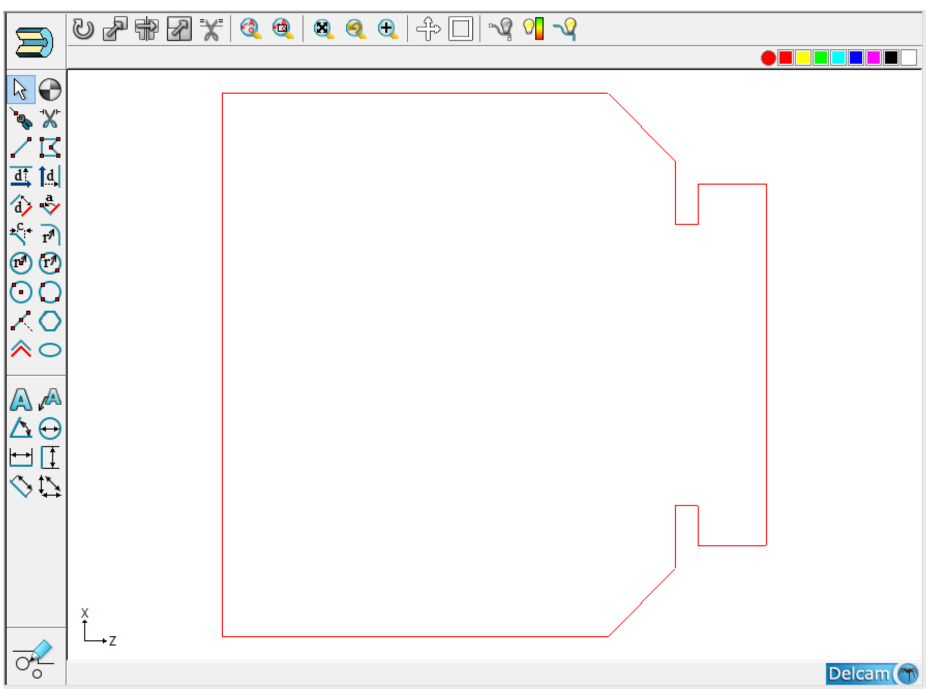
To import the geometry as follows:

1 Choose **Import DXF file** from the **File** menu.

2 Select **Tm-turn.dxf** from the **C:\PartMaker\PM-TM** directory as shown here:

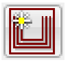


Your screen should now appear as shown below:

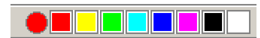


Defining the Profile for OD Turning

The first profile to create will be the OD shape previously defined by the imported DXF geometry.

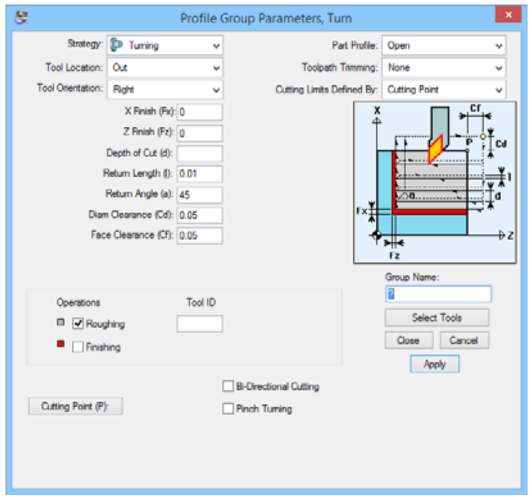


1 Click the **CAD/CAM** switch to go to CAM mode.



2 Choose the red square in the **Color Bar**

3 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Parameters** dialog. You should see the dialog below:



4 From the **Strategy** menu, select **Turning**.

5 From the **Tool Location** menu, select **Out**.

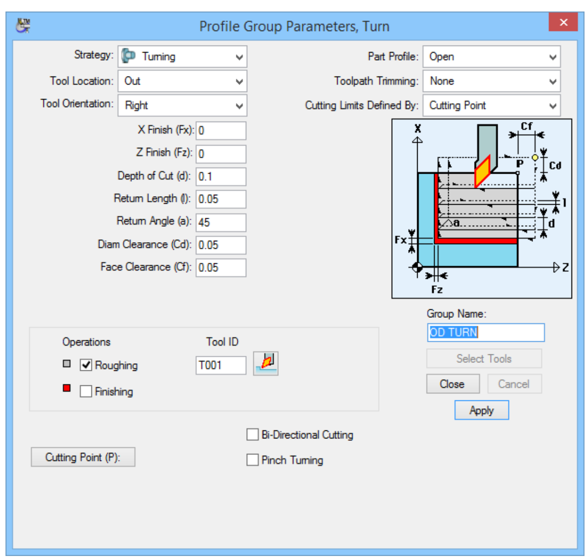
6 From the **Tool Orientation** menu, select **Right**.

7 Enter a **Depth of Cut** of 0.1 (2.5).

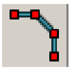
8 In the **Group Name:** field enter **OD TURN**.

9 Click the **Select Tools** button and allow PartMaker to select the appropriate tool.

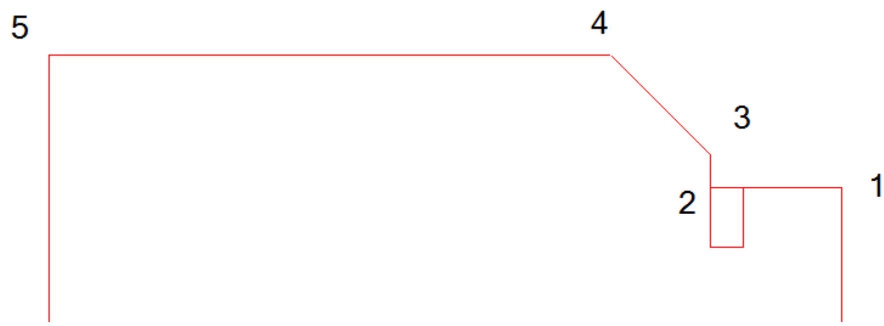
The completed **Profile Group Parameters** dialog should appear as shown below:



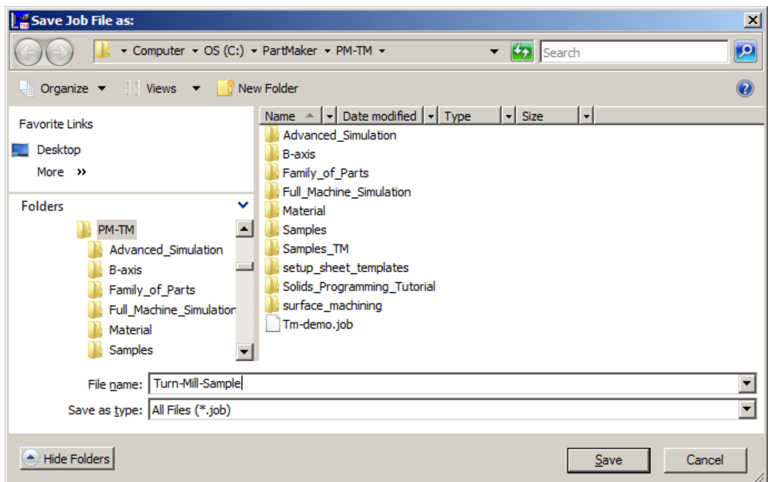
When your dialog box appears as the one above, click the **<Close>** button.



10 Choose the **Define Profile** icon and the **End of an Element Snap Mode** to select the profile to be cut as shown here, starting from the right and working toward the left, staring at point 1 and going to 5, click your cursor at the endpoint of each line, as shown below:



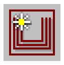
*Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**. Give your job file a name you will remember, such as TurnMill-Sample.*



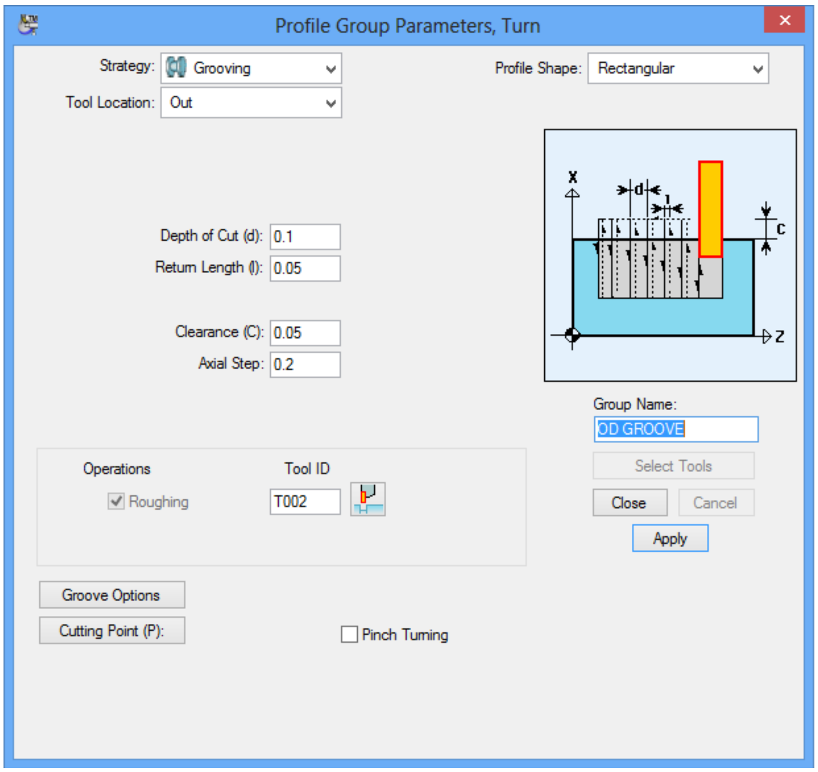
Next, you will create the profile for the groove.

Defining the Profile for Grooving

The next operation to be performed will create an OD groove.

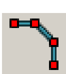


1. Click the blue square from the **Color Bar**.
2. Choose New Profile Group from the Part Features menu.
3. From the **Strategy** menu, select **Grooving**.
4. From the **Tool Location** menu, select **Out**.
5. Enter a **Depth of Cut** of 0.1 (2.5).
6. In the **Group Name:** field enter **OD GROOVE**.
7. Click the **Select Tools** button and allow PartMaker to select the appropriate tool.



When you are satisfied that your dialog box appears as the one above, click the **<Close>** button.

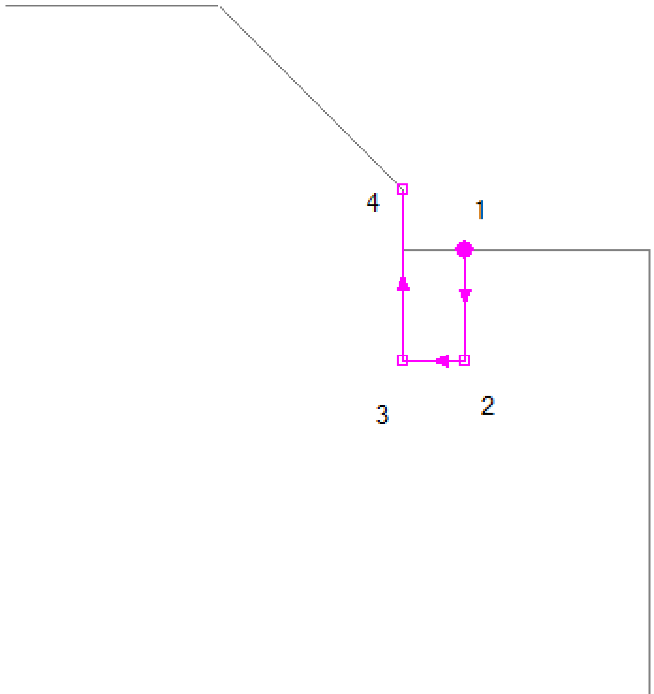
You can now assign the **OD GROOVE** operation to the geometry:




1. With the **OD GROOVE** group symbol selected, select the **Define Profile** icon and the **Closest Intersection Snap Mode.**

__Note:__ To better focus on the grooving profile, choose __Show Work Group Only__ from the __View__ menu.

2 Click on the groove to be cut, starting at point one and finishing at point 4, then toward the left, as shown here:

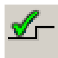


Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>** or click on the save button.

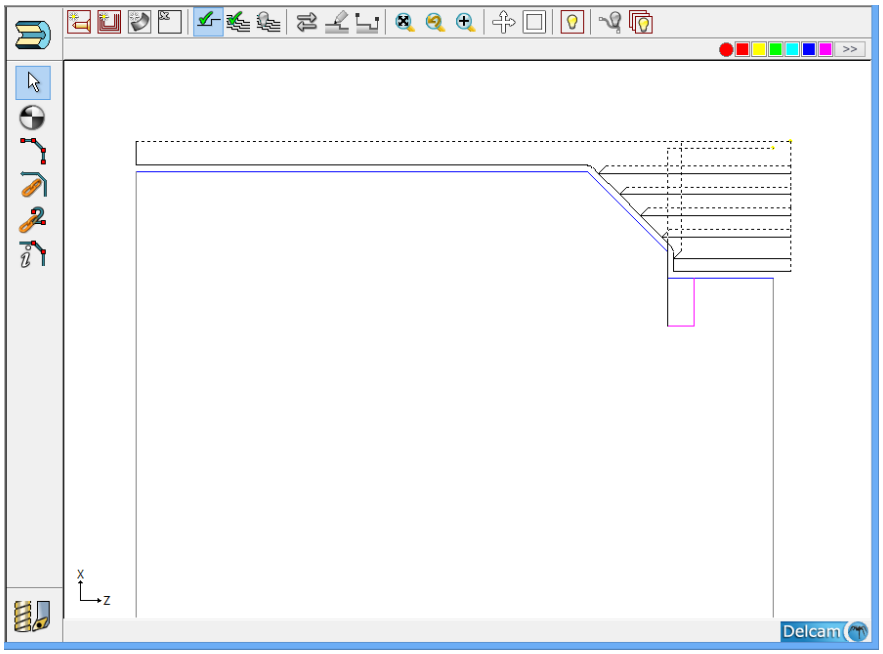
Verifying the Tool Path

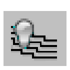
You can now verify the tool paths of the operations you have just programmed. To do so:

1. Choose **Show All** from the **View** menu to show both part features on screen.



2. Select **Verify Work Group Tool Path** from the **Part Features** menu to check your work.
3. In the Verification Options dialog choose Do Not Show Tool.
4. PartMaker should show you the following:





5. When you are satisfied that your screen matches that of the picture above, return to the **Part Features** menu and select **Hide Every Tool Path.**

You have finished the turning portion of this tutorial. The next step is to program the end work using C-axis indexing programming.

Mill End, Index

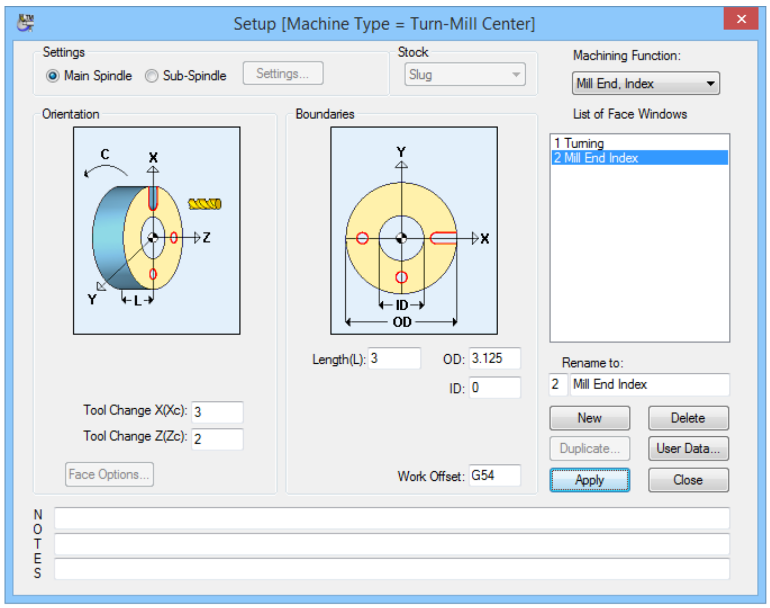
You will now program the four holes and four radial slots on the face of the part.

Setting Up the Face Window

In this section, you will set up the **Face Window** for to drill the holes and mill the slots on the end of the part. To do so:

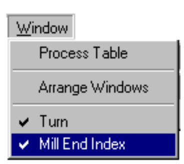
1. Choose **Setup** from the **View** menu or use the **Shortcut to the Setup** icon located above the profile icons on the left-hand side of the **Face Window**.
2. Click the <**New>** button to create a new face window.
3. In the **Rename to:** dialog, enter the name **Mill End Index**.
4. From the Machining Function drop down menu, select **Mill End, Index**.

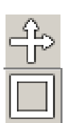
   The size and origin location of the stock should remain the same as previously defined in the Turning Setup Window. When you are finished, the Setup dialog should appear as shown below:



5. 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.

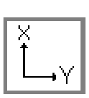
6. Make sure you are in the correct face window by selecting the **Mill End Index** window under the **Window** menu as shown below:





7. Choose **Show Axes** from the **View** menu.
8. Choose **Show Boundaries** from the **View** menu.

__Note:__ If the image in the face window appears distorted or oblong, choose __Full View__ from the __View__ menu to refresh the screen for it to appears as shown above.



Notice in the lower left-hand corner of the window where the X-Y axis icon appears. This icon is used to remind you which axes are currently being used.

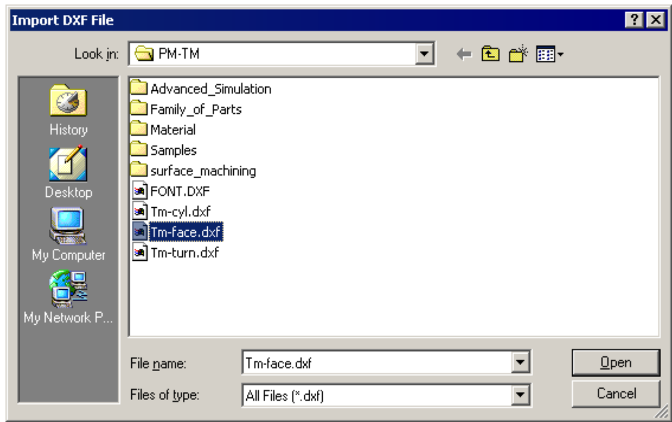
Creating Geometry in CAD Mode

A DXF file of the necessary geometry has again been provided for this portion of the tutorial.

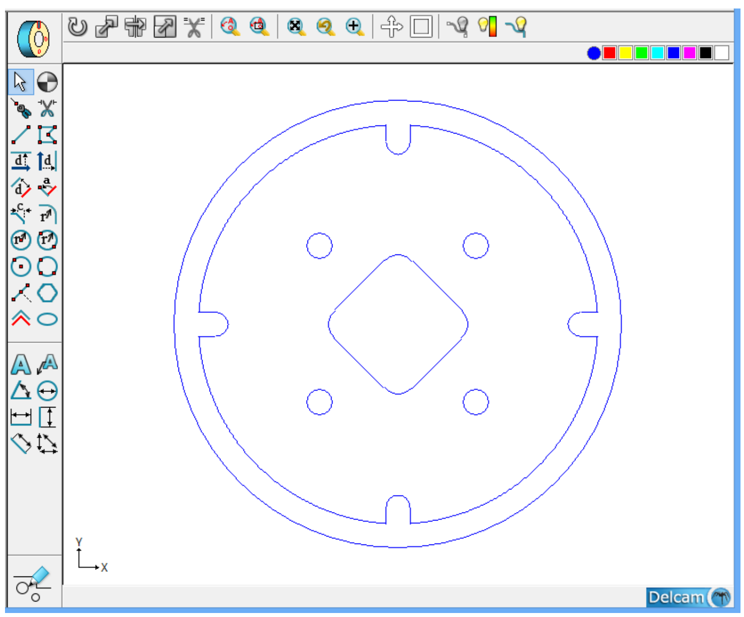
The file to import is called **Tm-face.dxf** and is found in the **C:\PartMaker\PM-TM** directory.

To import the geometry as follows:

1 Choose **Import DXF file** from the **File** menu.
2 Select **Tm-face.dxf** from the **\PM-TM** directory as shown here:



Your display should look like this (the picture in the foreground below is shown in CAD mode:



Before proceeding, click on the CAD/CAM switch and make it appear as a drill to make sure you are in CAM mode.

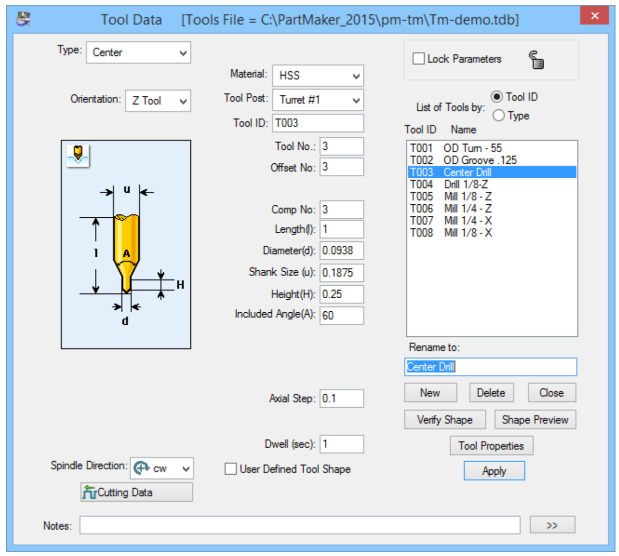
Programming Off-Center Holes

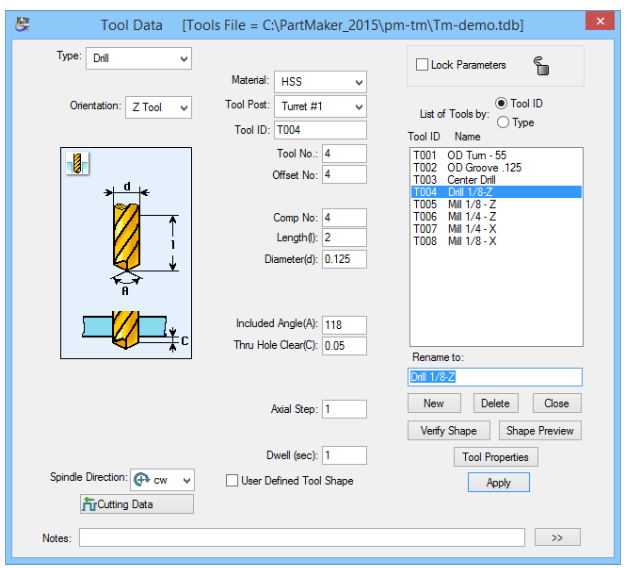
The first step is to program the four drilled 1/8" (3.2mm) holes. PartMaker / TurnMill uses drilling cycles just like PartMaker / Mill and PartMaker / Turn.

Using Milling Tools in PartMaker TurnMill

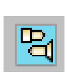
The two tools that will be used to create the off-center holes are a center drill and a 1/8" (3.2) drill. These tools have already been created and can be viewed by selecting **Tools** from the **ToolMinder** menu or clicking on the tool box icon. Note the orientaton of these tools.

Observe the two new tools as shown here:



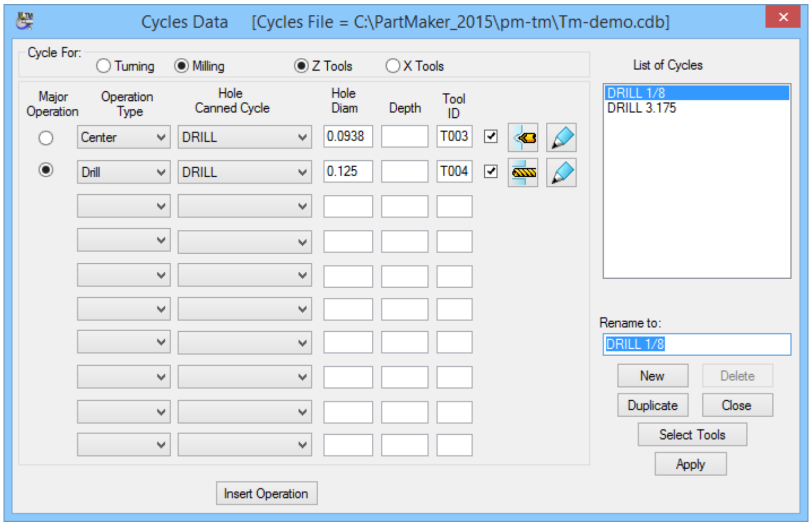


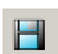
Once you have observed the tools as shown on the previous page, close the dialog box and from the **ToolMinder** menu, select **Cycles**.



PartMaker holemaking cycles are also described in the same manner. To see this, choose **Cycles** from the **ToolMinder** menu**.**

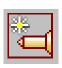
The cycle that will be used to drill the four holes has been defined like any other mill or lathe drilling cycle you may be used to. Note however, that this cycle has been specified as a **Mill** cycle with component tools oriented along the Z-axis. Had the cycle been designated for use in turning Face Window (with the **Turn** radio button clicked) it would not have been necessary to designate orientation. For **Turn** type holemaking cycles, tools may only move on center-line.

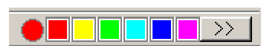




Once again, this would be a good time to save your work. Press **<Ctrl+S>** to save your work up to this point.

Once you feel comfortable with setting tools and cycles for TurnMill centers, proceed to program the four holes. To do so:

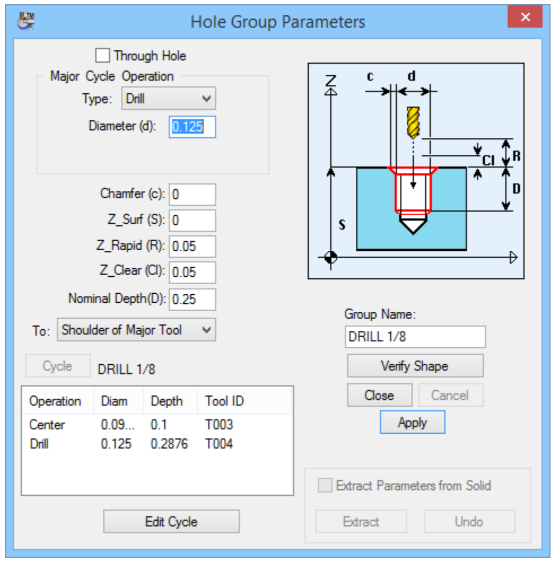


1. Click the red square from the **Color Bar**. 
2. Choose New Hole Group from the Part Features menu.
3. Uncheck the **Through Hole** box
4. In **Major Cycle Operation** select **Drill** (it may already be selected).
5. Enter a diamter of 0.125 (3.2)
6. Other parameters should be set as follows:

Chamfer (c):	0	
Z-Surf (S):	0	
Z-Rapid (R):	0.05	(1.0)
Z_Clear (Cl):	0.05	(1.0)
Nominal Depth:	0.25	(10.00)

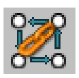
PartMaker will calculate the necssary programmed depth from the desired nominal depth taking into account the diamter and point angle of the drill.

7 Press the **Cycle** button and PartMaker will select the appropriate cycle to create the desired hole. The selected cycle should be **Drill 1/8 (3.20)**.



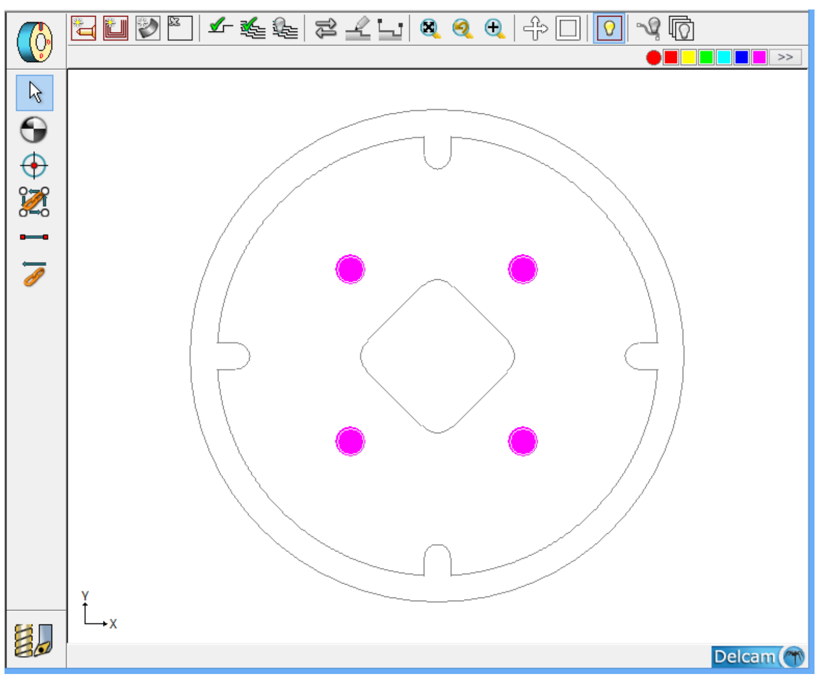
When you are satisfied your dialog appears as the one above, click the **<Close>** button.

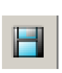
You are now ready program the holes. To do so:



1 From the left hand tool bar, select the **Chain Circle** icon.

2 Click on any one of the four circles and PartMaker will select the other three. After doing so, your screen should appear as shown below:



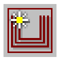


Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Programming Radial Slots

The next step of the programming process is to define the radial slots.

1 Click the blue square from the **Color Bar**.



2 Choose New Profile Group from the Part Features menu.

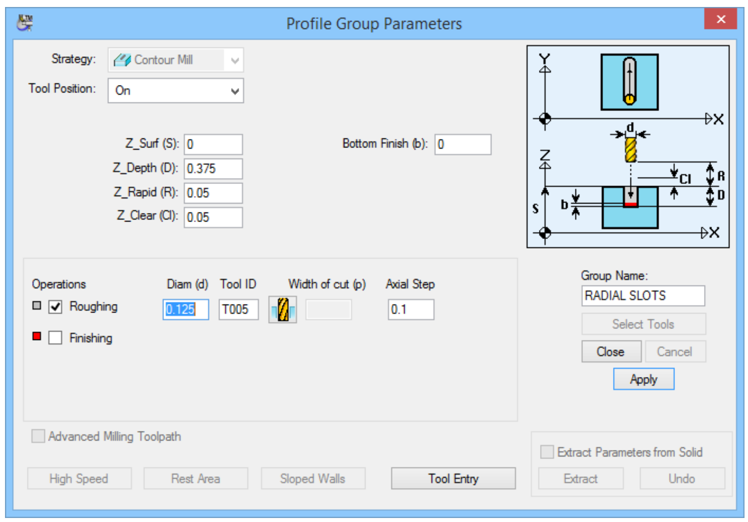
3 In the Group Name box enter RADIAL SLOTS.

4 From the **Strategy** drop down menu choose **Contour Mill**. Choose a **Tool Position** of **On**.

5 Other parameters should be set as follows:

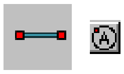
Z_Surf (S):	0
Z_Depth (D):	0.375 (6.5)
Z_Rapid (R):	0.05 (1.00)
Z_Clear (C):	0.05 (1.00)
Bottom Finish:	0
Roughing:	0.125 (3.20)

6 Click the **Select Tools** button and allow PartMaker to select the appropriate tool.



When you are satisfied your dialog appears as the one above, click the **<Close>** button.

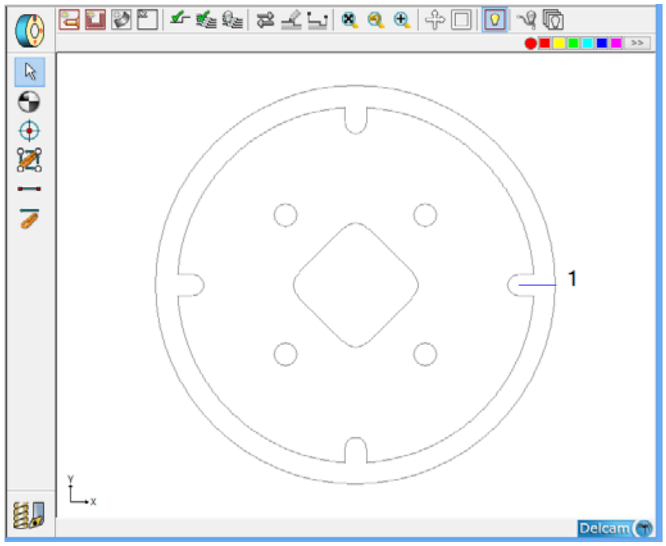
The next step is to actually define the path that the tool will be following. To do so:



1 Choose the **Linear Profile** icon followed by the **Point on an Circle** snap mode.

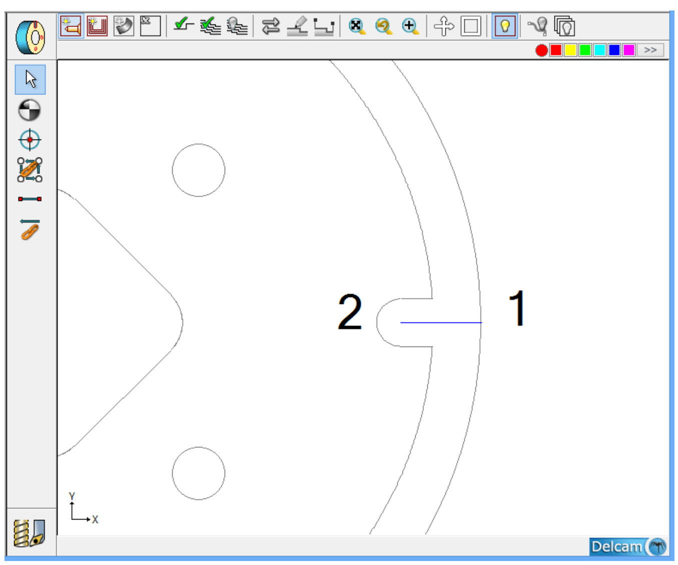
2 At the **A <0>** field located in the lower left hand corner of the screen, enter the number 0 at the prompt for an angle and press **<Enter>**.

3 You will now be prompted to **Select an Arc or Circle**. To do so, click on point 1 as shown in the picture below:



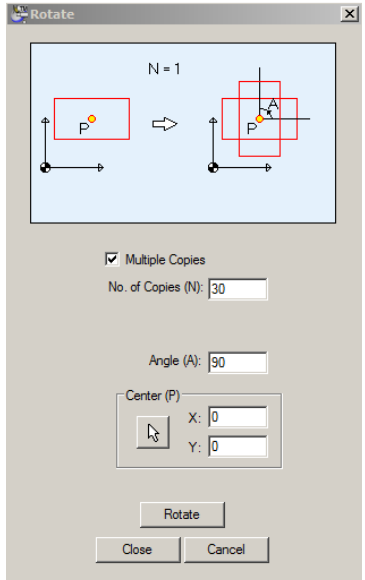
***Note:** In the .dxf file imported above, a circle with a radius 0.2 inches larger than that of the radius to be milled has been included. (The inner circle with a radius of 1” represents the turned down radius from the previous face.) In this example, in order to avoid having the tool begin cutting on the stock, you will use the outer circle to begin the tool’s feed away from the actual part.*

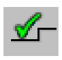
4 Next, select the **Circle Center** snap mode and click on the arc at the bottom of the slot at point 2 as shown below:



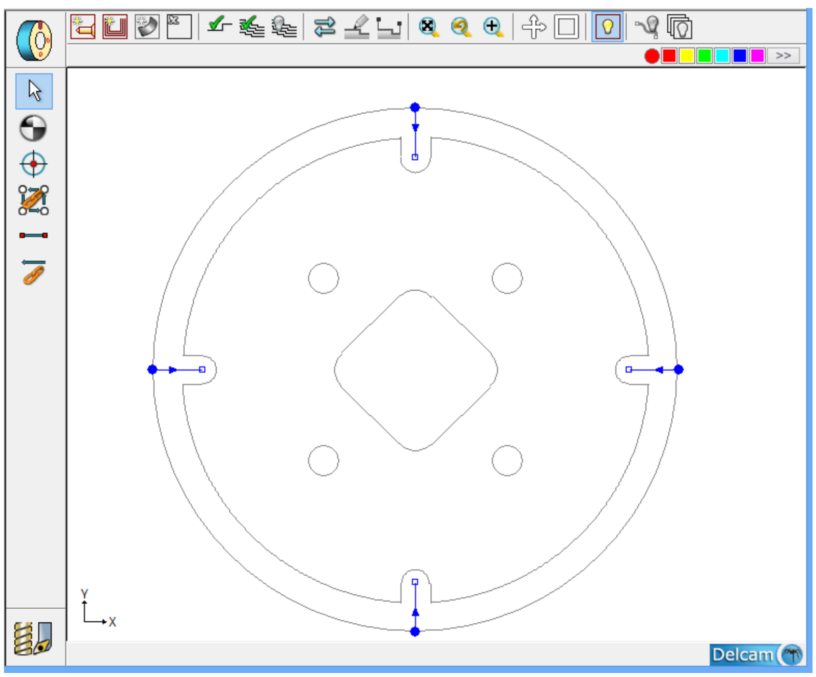
Once you have done so choose the **Selection Icon**. Select the slot you just defined by clicking on it with your cursor.

5 To mill the three remaining slots, with the first slot selected, choose **Rotate** from the **Edit** menu. Complete the **Rotate** dialog as shown below:

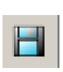




6 Having completed defining tool paths for the radial slots, select **Verfiy Work Group Tool Path** from the **Part Features** menu. You should see the following:



When you are satisified your screen appears as shown above select **Hide Every Tool Path**, from the **Part Features** menu.



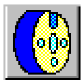
Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Polar Milling on the End

The next feature to be programmed will be the rectangular pocket in the end of the part. The machining function used will be **Mill End, Polar**.

Setting Up the Face Window

In this section, you will set up the **Face Window** to drill the holes and mill the slots on the end of the part. To do so:



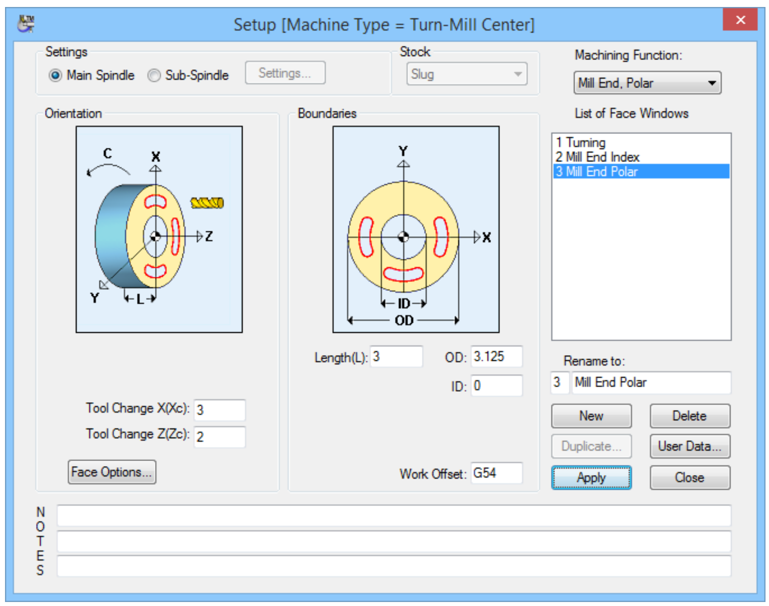
1 Choose **Setup** from the **View** menu or use the **Shortcut to the Setup** icon located above the profile icons on the left-hand side of the **Face Window**.

2 Click the <**New**> button to create a new face window.

3 In the **Rename to:** dialog, enter the name **Mill End Polar**.

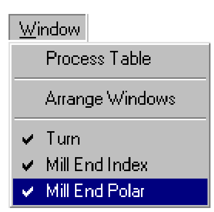
4 From the Machining Function drop down menu, select **Mill End, Polar**.

The size and origin location of the stock should remain the same as previously defined in the Turning Setup Window. When you are finished, the Setup dialog should appear as shown below:



5 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.

6 From the **Window** pull down menu, click on the menu item for the new window just created, as shown on the right:

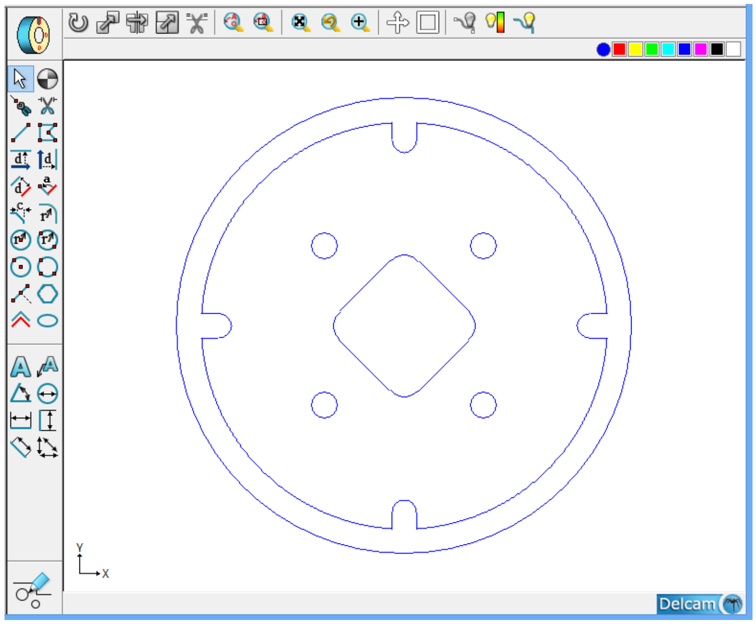


Creating Geometry in CAD Mode

Next, geometry to program is required. The geometry previously imported into the **Mill End, Index** face contains what is needed, so again, import that geometry into PartMaker as you did in the previous section.

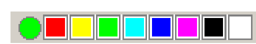
The name of the file is \PM-TM\Tm-face.dxf

Since you have already programmed the holes and radial slots, the only geometry to be concerned with is that of the diamond shaped pocket in the center of the face, as shown here.

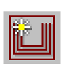


Programming the Pocket

You will now program the pocket on the end of the part.



1 Click the Green square from the **Color Bar**.



2 Choose New Profile Group from the Part Features menu.

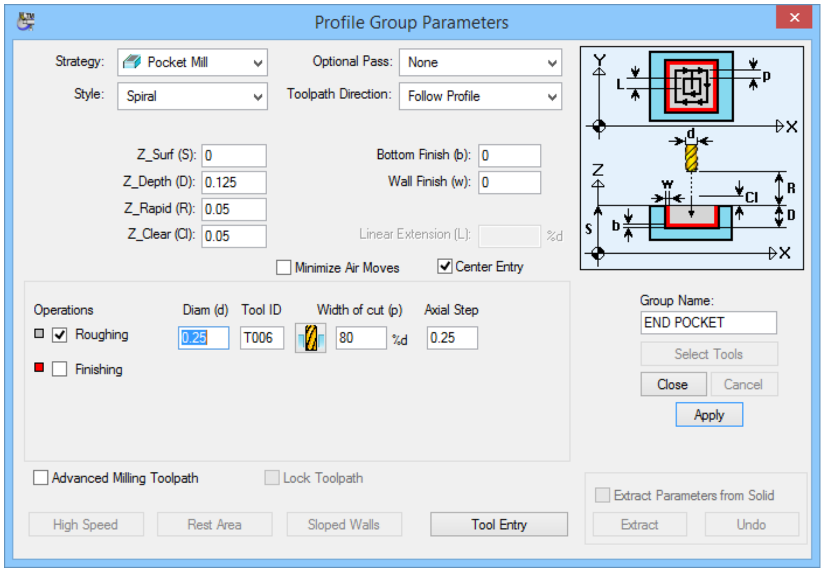
3 In the **Group Name** box enter **END POCKET**.

4 From the **Strategy** drop down menu choose **Pocket Mill**

5 Other parameters should be set as follows:

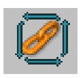
Z_Surf (S):	0
Z_Depth (D):	0.125 (3.00)
Z_Rapid (R):	0.05 (1.00)
Z_Clear (C):	0.05 (1.00)
Bottom Finish:	0
Wall Finish:	0
Diam (d):	0.25 (6.00)

6 Click the **Select Tools** button and allow PartMaker to select the appropriate tool.



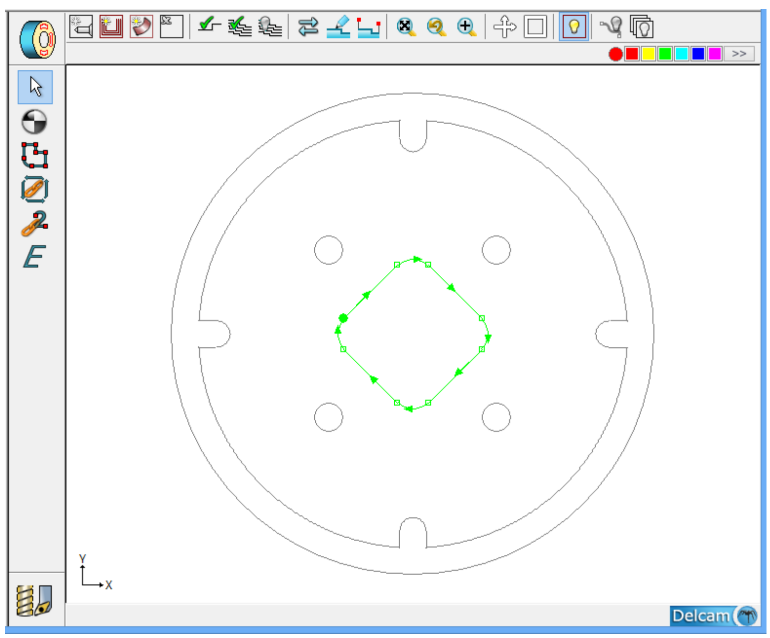
7 When you are satisfied that your dialog appears as the one above, click the **<Close>** button.

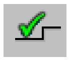
The next step is to develop the tool path for the pocket:



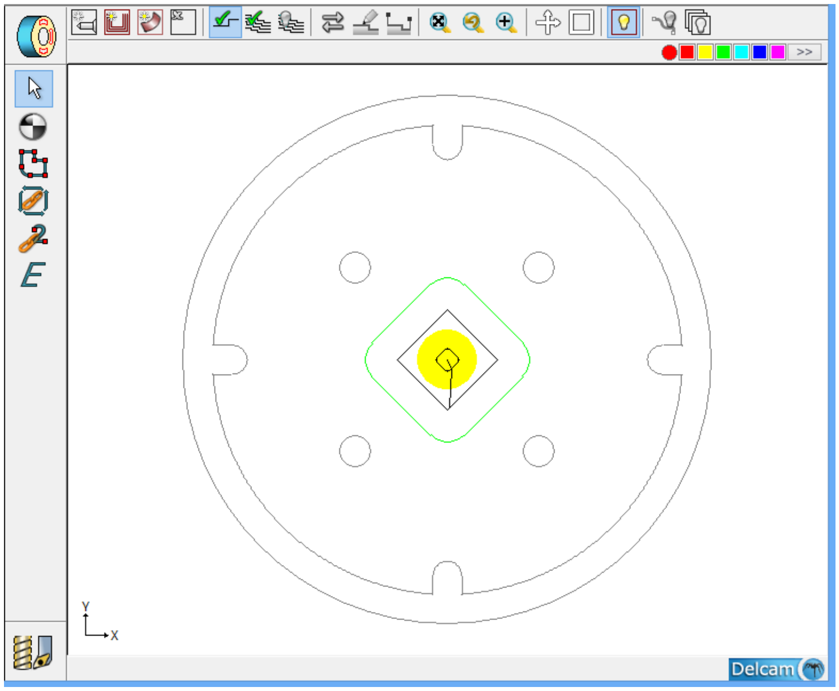
1 Select the Chain Geometry Icon.

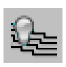
2 Select any portion of the diamond pocket as you would have with conventional milling, as shown here:



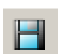


3 To check your work, select **Verify Work Group Toolpath** from the **Part Features menu**. Your completed pocket should appear as shown below:





4 From **Part Features** choose **Hide Every Tool Path** when you are satisfied.



Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Mill Diameter, Index

This section has been placed at the end of the tutorial because it incorporates drawing and programming concepts that are not encountered in basic milling or turning operations - the concept of 'unwrapping' a cylinder and programming it as a flat surface.

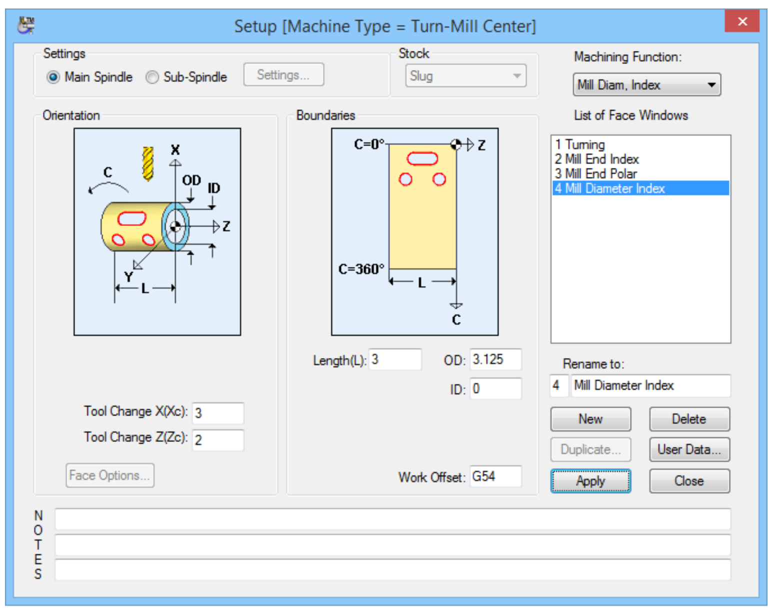
To better understand the concept of unwrapping, imagine the outside diameter (OD) of the cylinder as a paper wrapper on a tin can. Slit that wrapper and peel it off the can and lay it flat on the table. This is how PartMaker handles cylindrical (OD) machining. Instead of using X and Y coordinates, programming is done with Z and C, where Z is the 'length' of the cylinder along the machine Z-axis and C represents the degrees of rotation, starting from 0 and progressing to 360 degrees.

Setting Up the Face Window

In this section, you will set up the **Face Window** to mill the slots on the OD of the part. To do so:

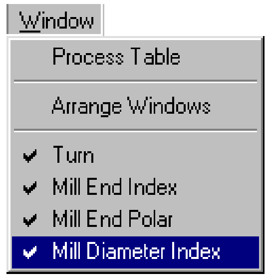
1. Choose **Setup** from the **View** menu or use the **Shortcut to the Setup** icon located above the profile icons on the left-hand side of the **Face Window**.
2. Click the <**New**> button to create a new face window.
3. From the Machining Function drop down menu, select **Mill Diam, Index**.
4. In the **Rename to** dialog, enter the name **Mill Diameter Index.**

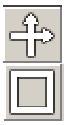
   The size and origin location of the stock should remain the same as previously defined in the Turning Setup Window. When you are finished, the **Setup** dialog should appear as shown below:



5. 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.

6 Make sure you are in the correct face window by selecting the **Mill Diameter Index** window under the **Window** menu as shown below:





7 Choose **Show Axes** from the **View** menu.

8 Choose **Show Boundaries** from the **View** menu.

Note how Z and C are used to define the position of the features (slots and holes) that can be programmed using this machining function.

Creating Geometry in CAD Mode

Since the concept of drawing and programming and an unwrapped cylinder is an entirely new concept, you will work through both the CAD creation and CAM programming of the desired features.

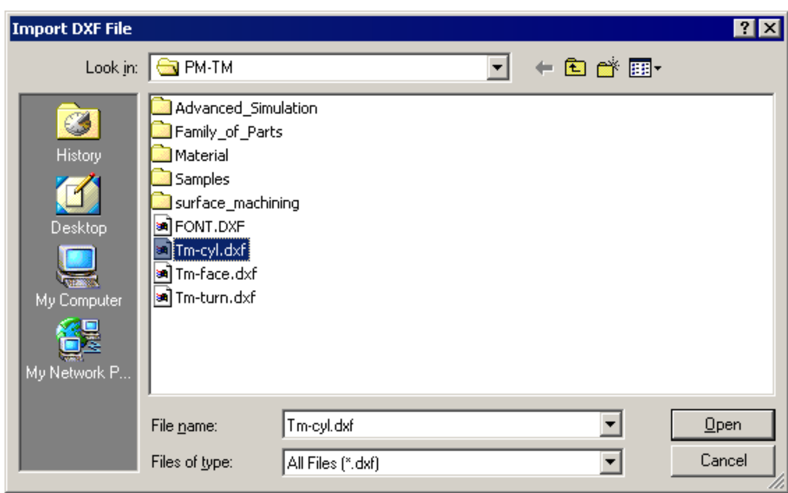
At this point you can choose to either create the geometry or import a .DXF file.

Method 1: Import a .DXF file

A DXF file of the necessary geometry has again been provided for this portion of the tutorial.

The file to import is called **Tm-cyl.dxf** and is found in the \ PM-TM directory. If you choose to use this method:

1 Import this file now by clicking **<Import DXF File>** from the **File** menu and choose **Tm-cyl.dxf**.



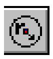
2 Please skip ahead to the section titled **Programming Slots on the Cylinder**.

*Please note that the geometry you import will look slightly different from the one you will see in the **Programming Slots on the Cylinder** section because the imported geometry includes a square that will be used in the next section in which you will perform cylindrical milling. You should ignore this square for the **Programming Slots on the Cylinder** section.*

Method 2: Defining Geometry in PartMaker

If you would like to create the geometry yourself, make sure you are in CAD mode. by clicking on the CAD/CAM switch in the lower left-hand corner of the screen, making sure it appears as a pencil.

Look at the supplied drawing for the tutorial part on the second page of this chapter. Note that there are two longitudinal slots, oriented at 90 and 270 degrees. To draw the slots:

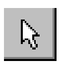


1 Select the **Circle with Known Radius and Center** icon. At the prompt in the lower left-hand corner of the screen, enter a value of **0.125 (3.00)** for the radius and press **<Enter>**.

2 Select the **ZC Snap Mode** icon from the snap mode icon bar. Enter a value of **-2.375 (-64.00)** for Z. Press the **<Tab>** key or use your cursor to advance to the <C> field. Enter a value of **90** for C and press **<Enter>**.

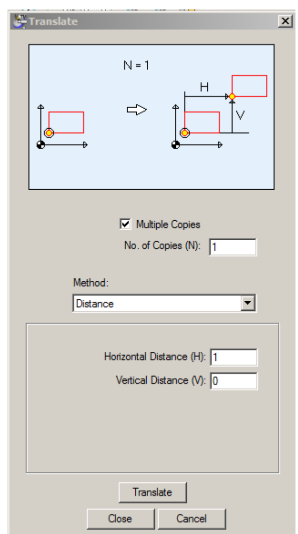
3 Create the second circle using the technique shown above, i.e. using the **Circle with Known Radius and Center** icon and **ZC Snap Mode**. This time you need only change the C value from 90 to 270.

You will now translate these two circles along the Z-axis. To do so:

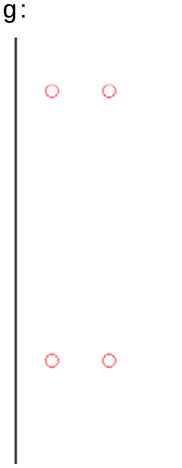


1 Choose the **Selection** icon. Holding down the **<Shift>** key, and click your cursor on each circle such that they are highlighted.

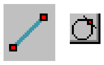
2 With both circles selected (highlighted), choose **Translate** from the **Edit > Transform** menu. Complete the dialog as shown below and click <**OK**>: **Z 1.0 (25)**



PartMaker will show the following:

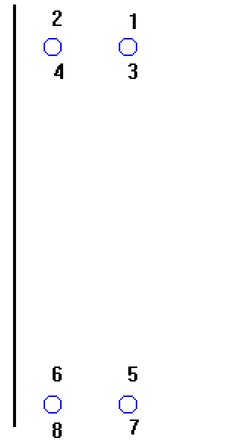


You will now complete the slots. To do so:



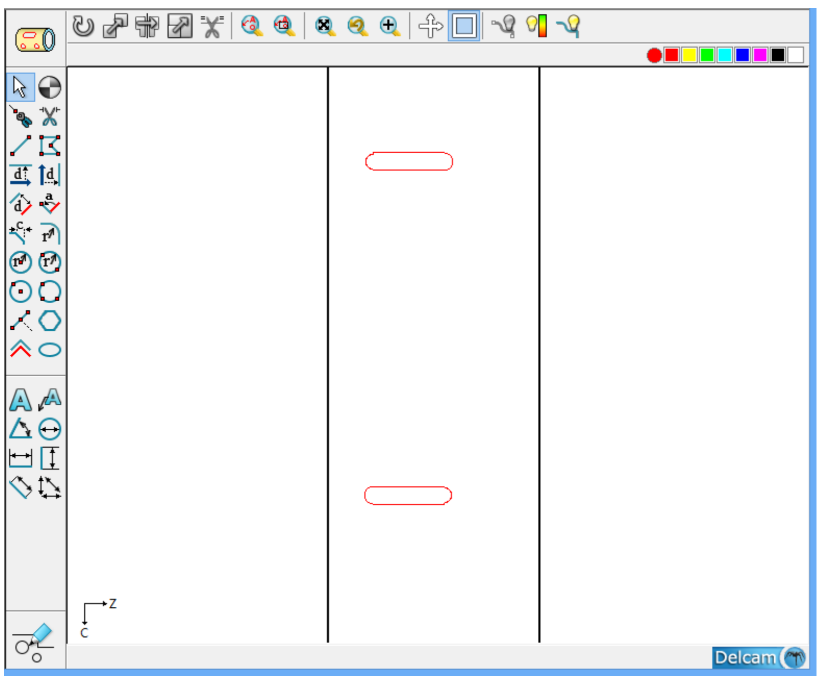
3 Select the **Line Thru Two Points** icon along with the **Tangency** snap mode.

4 To close the slots click your mouse from point 1 to point 2, repeating this procedure from point 3 to 4, 5 to 6 and 7 to 8 as you move down the drawing.



5 Once you have closed the slots, select the **Remove** icon to remove unnecessary geometry. To do so, click you cursor on the on the interior of the slots.

When you are finished, your slots should appear as shown here.



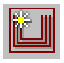
When you are satisfied that the geometry has been correctly created and are confident in your understanding of the techniques used, switch to the CAM mode by clicking on the pencil icon in the lower left hand corner of the screen.

If you are at all unsure that your geometry is correct, please go back to Method 1 before proceeding to the next section.

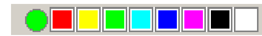
Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Programming Slots on the Cylinder

The next operation is to program the tool motion to machine slots oriented along the Z-axis.



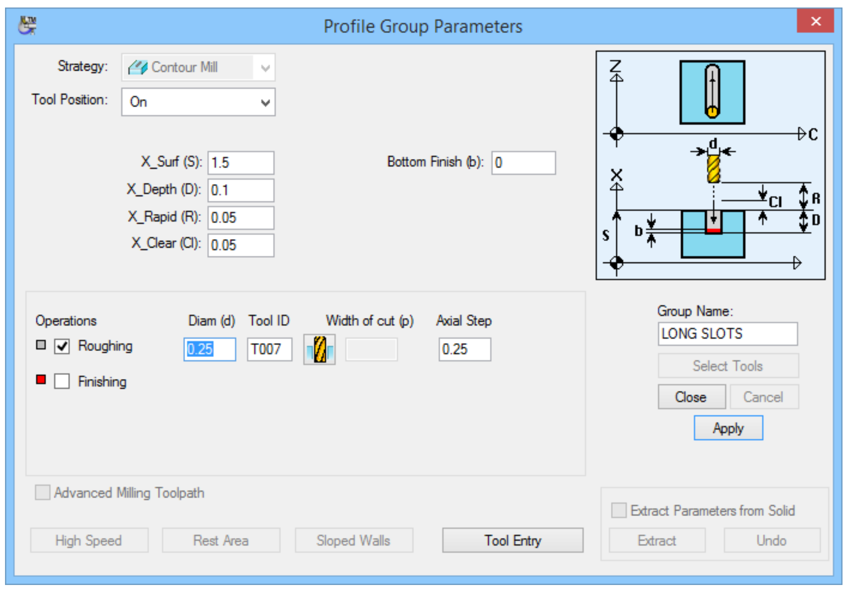
1. Click the green square from the **Color Bar**.



2. Choose New Profile Group from the Part Features menu.
3. From the **Strategy** menu, select **Contour Mill**.
4. In the Group Name box enter **LONG SLOTS**.
5. Other parameters should be set as follows:

X_Surf (S):	1.5	(40.00)
X_Depth (D):	0.1	(3.00)
X_Rapid (R):	0.05	(1.00)
X_Clear (C):	0.05	(1.00)
Bottom Finish:	0	
Diam (d):	0.25	(6.00)

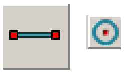
6. Click the **Select Tools** button and allow PartMaker to select the appropriate tool.



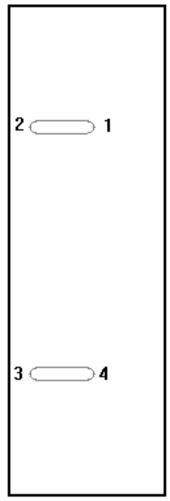
Note: *Notice that the (S) value is now shown as X_Surf and the (D) value is shown as X_Depth. Note that the X_Surf value is shown as the radius of the part, not the diameter, and that the depth is programmed as a radius depth.*

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

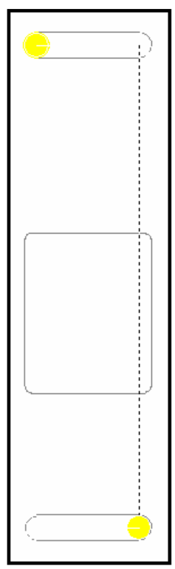
The next step is to develop the tool path for the slots. To do so:



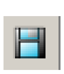
1 Choose the **Linear Profile** icon and the **Circle Center** snap mode.
2 Click you cursor on the inside of the arc starting at point 1 (as shown below) and then doing the same while moving from to point 2.
3 With the **Linear Profile** icon and the **Circle Center** snap mode still selected do the same from point 3 to point 4. Be sure to select both ends of one slot before moving to the next slot.



4 When you are finished, and have selected **Verify Workgroup Toolpath** from the **Job Optimizer Menu**, your screen should appear as shown below:



5 Select Hide Every Tool Path from the Part Features menu.



Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Cylindrical Milling

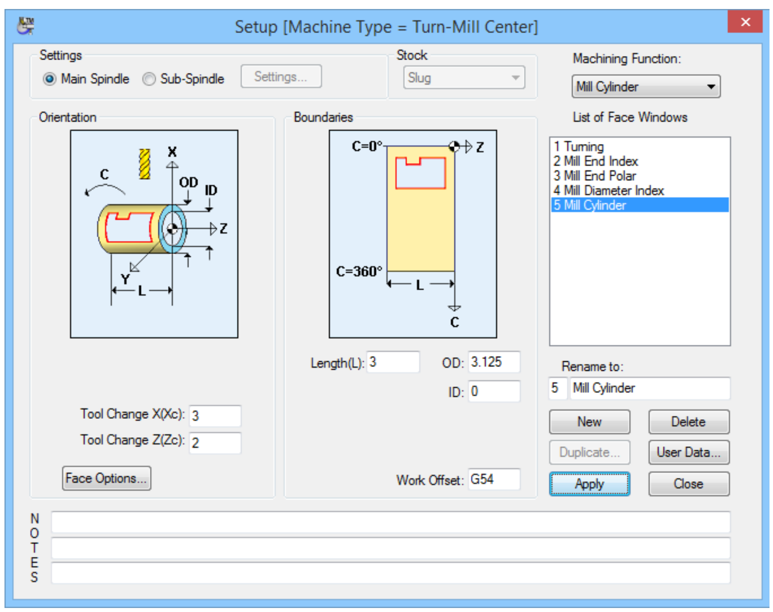
A TurnMill center capable of cutting cylindrical milling geometry has the ability to simultaneously feed in X, Z and C-axes, often referred to as continuous C-Axis capability. Looking at the tutorial drawing, you can see that a rectangular groove is to be machined on the OD.

As you may have noticed in the previous example, if you are programming along the center line of the tool, it is not actually necessary to draw the groove - only the center line for the tool to follow. We will use this technique to simplify the geometry creation portion of this example.

Setting Up the Face Window

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

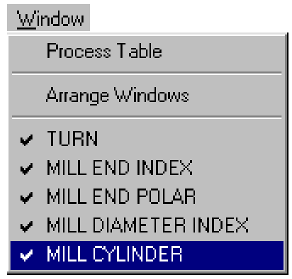
1. Choose **Setup** from the **View** menu to view the **Setup** dialog box or use the **Shortcut to the Setup** icon located above the profile icons on the left-hand side of the Face Window.
2. Click the **<New>** button to create a new face window.
3. In the Rename To: field, enter Mill Cylinder.
4. From the **Machining Function** drop down menu select **Mill Cylinder**.
5. When you are finished, your **Setup** dialog should appear as shown below:



Once again, notice how PartMaker will be ‘unwrapping’ the cylindrical geometry and creating a flat layout, with coordinates of Z in linear units and C in degrees of rotation.

6. 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.

7 To switch to the **Mill Cylinder** Face Window, choose **Mill Cylinder** from the Window menu.



8 Choose **Show Axes** from the **View** menu. This will display the horizontal and vertical axes lines which will help you create the geometry in the next section of the tutorial.

9 Choose **Show Boundaries** from the **View** menu.

Creating Geometry in CAD Mode

Cylindrical milling geometry is created exactly like **Mill Diameter, Index** geometry. The difference is in the capabilities of the TMC, not PartMaker.

At this point you can choose to either create the geometry or import a .DXF file.

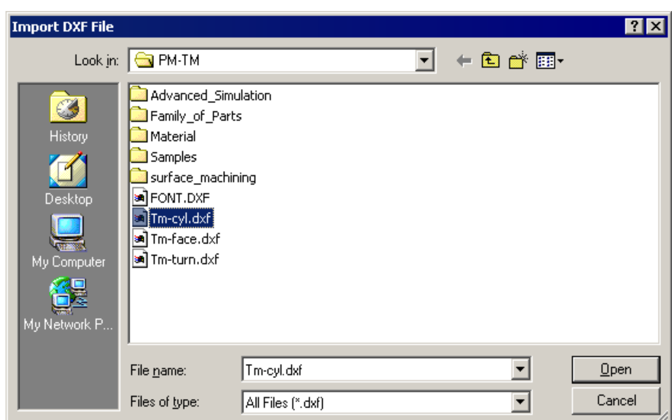
Method 1: Import a .DXF file

A DXF file of the necessary geometry has again been provided for this portion of the tutorial. To import this .DXF file:

1 Choose **Import DXF File** from the **File** menu.

2 Select **tm-cyl.dxf** from from the pm-tm\tutorial subdirectory.

If you choose to use this method, import this file now and please skip ahead to the section titled **Programming Cylindrical Milling**.

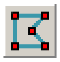


***Note:** The geometry you import will look slightly different from the one you will see in the **Programming Cylindrical Milling** section. This is because the imported geometry includes the slots that were used in the previous section in which you performed only index milling on the cylinder. You should ignore these slots for the **Programming Mill Cylinder** section.*

Method 2: Defining Geometry in PartMaker

Here you will create the CAD geometry for the rectangle:

1. Start by selecting the **CAD/CAM switch** in the lower left hand corner of the screen and making sure it is a pencil.



2. Click on the **Connected Lines** icon. After doing so, select the **ZC** snap mode from the Snap Mode icon bar at the top of the Face Window.

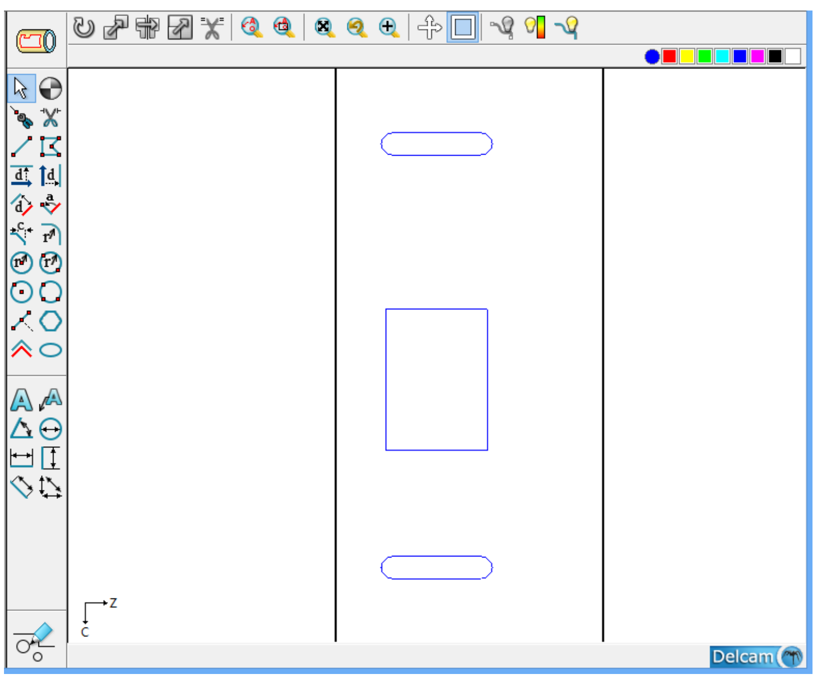
3. After selecting the **ZC Snap Mode** two data entry fields will appear in the lower left hand corner of the Face Window.

   Note: *Remember, when plotting coordinates in PartMaker CAD, 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. You can move between data entry fields by pressing the* ***<Tab>*** *button or using your mouse.*

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

Z Coordinate	C Angle
-2.4375 (-65.30)	160
Default	220
-1.3125 (-37.30)	default
Default	160
-2.4375 (-65.30)	default

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

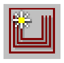


If you are not comfortable that your geometry resembles the one above, please revert to Method 1 of this section and import the appropriate DXF file before proceeding with programming this geometry.

Programming Cylindrical Milling

The next operation is to program the tool motion to machine the rectangle oriented along the Z-axis.

1 Click the CAD/CAM switch such that it appears as a drill bit. This will put you back into the CAM mode.



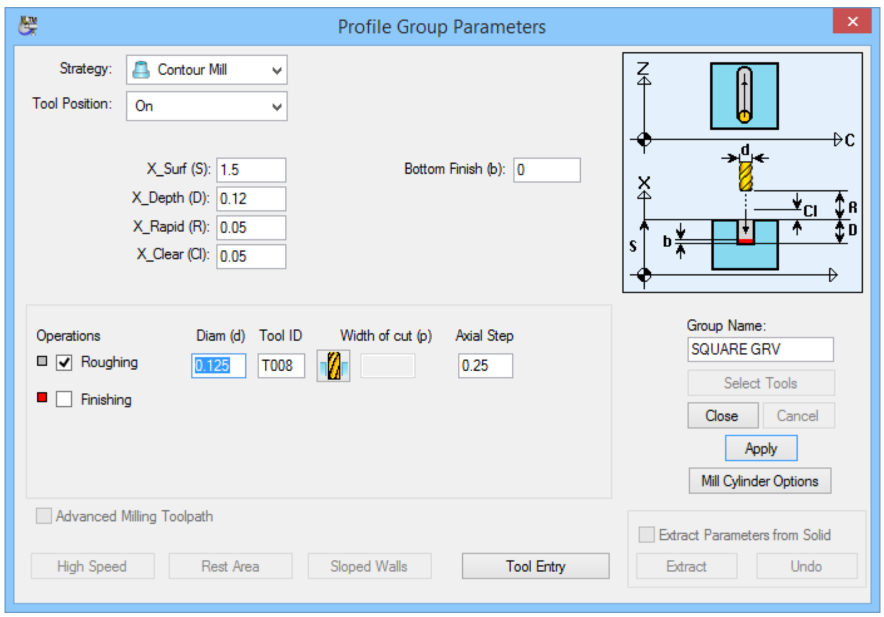
2 Select New Profile Group from the Part Features menu.

3 In the Group Name box type **SQUARE GRV**.

4 Other parameters should be set as follows:

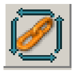
X_Surf (S):	1.5	(40.00)
X_Depth (D):	0.12	(3.0)
X_Rapid (R):	0.05	(1.0)
X_Clear (C):	0.05	(1.0)
Bottom Finish:	0	
Diam (d):	0.125	(4.0)

5 Click on the **<Select Tools>** button. An end mill icon will appear to the right of the **Tool ID** box. The completed **Profile Group Parameters** dialog should appear as shown below:



6 When you are satisfied that your dialog appears as the one above, click the **<Close>** button.

The next step is to develop the tool path for the cylindrical milling operation:

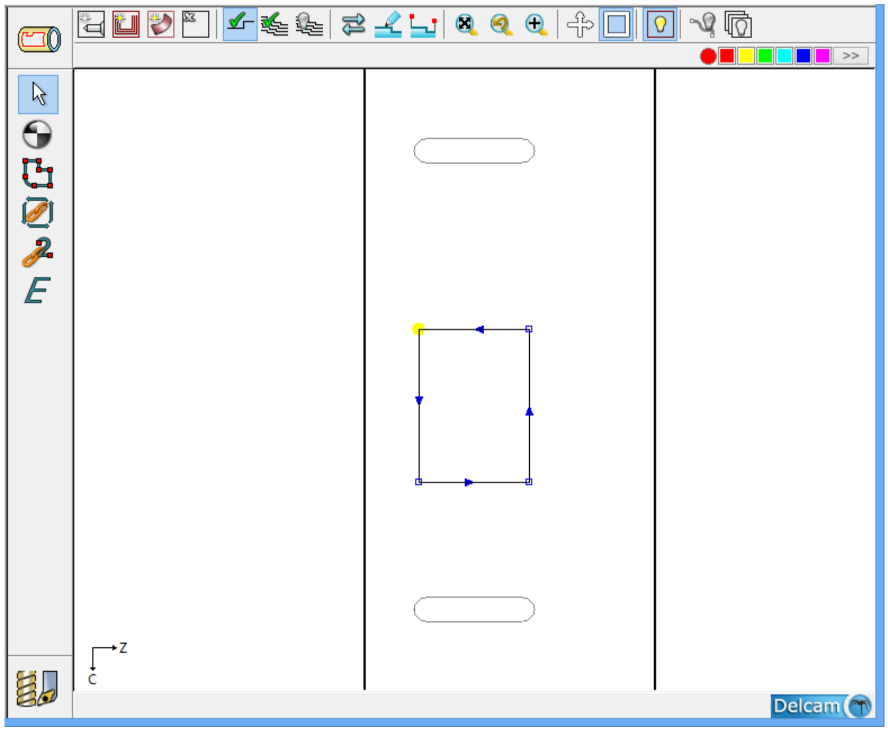


1 Use the **Chain Geometry** icon and click on the groove to be machined.

2 Click the rectangular geometry with your cursor to assign the part feature to your geometry:

3 Check your selection using **Verify Workgroup Toolpath** found in the **Part Features** pull down menu.

Your PartMaker screen should appear as shown below:



4 Select Hide Every Tool Path from the Part Features menu.

Before moving on, remember to save your work. Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

Creating an NC Program

The programming portion of the tutorial is now complete. At this point, all tool and machine motions necessary to create this part have been created.

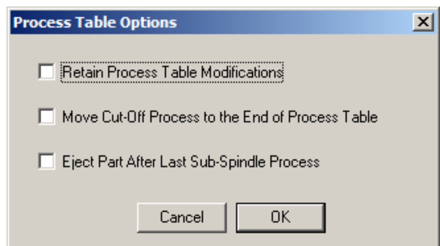
The remaining steps are:

- Choose the control
- Create the Process Table
- Generate the NC code

Generating a Process Table

The first step in generating an NC program is creating a Process Table. A **Process Table** shows you all of the machining processes for a part. When PartMaker generates a process table, all cutting conditions such as feed rate and spindle speed (RPM) are calculated automatically based on the tools and material information previously entered.

1. Choose **Generate Process Table** from the **Job Optimizer** menu. You will be promtped with **Process Table Options** dialog.

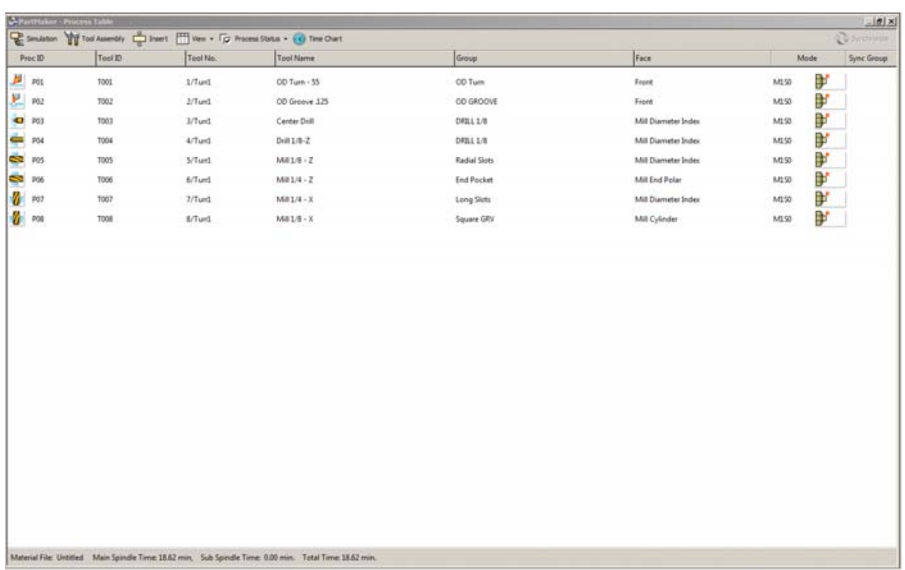


Retain Process Parameters Modifications. Checking this box will retain previous modifications to the process table. Since this is your first time Generating the Process Table it will have no effect.

Move Cut-Off Process to End of Process Table. Checking this box will automatically place your cut-off process at the end of the Process Table.

Eject Part After Last Sub-Spindle Process. Checking this box will automatically place your eject after the last Sub-Spindle process.

2. Click on **<OK>** to display the **Process Table** shown below:

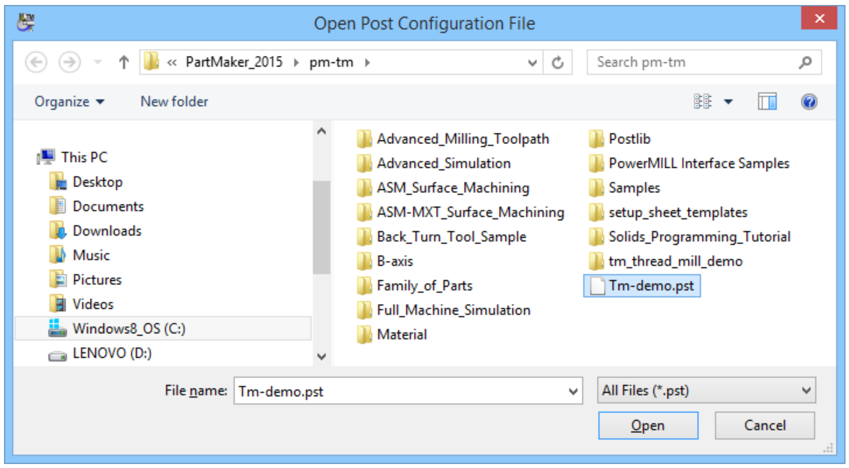


Selecting a Post Processor

Having generated a Process Table you must now select a Post Configuration File before you can output an NC progrm. This post configuration file is stored in the Pm-tm\Samples\8_axis-demo directory. If you do not have any other post configuration files loaded in this directory the **Tm-demo.pst** will be loaded automatically. If you do have other post configuration files in this directory:

1 Choose **Post Config File = ?** from the Job Optimizer menu.

2 Select the **TM_demo.pst** post as shown below. PartMaker for TurnMill comes standard with a post configuration for a Fanuc 18t control which supports, live tooling, c-axis and subspindle operations.

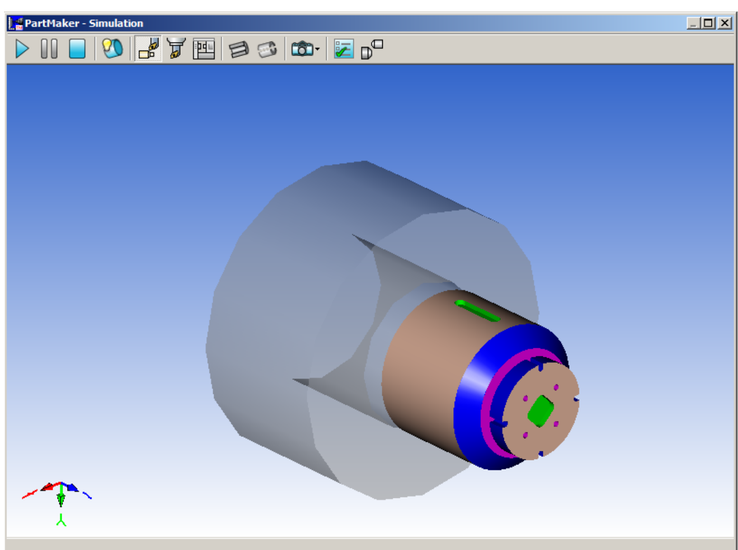


3 Click **<OK>** to continue.

Simulating the Part

Once you are satisified 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 lower left hand corner of the Process Table.

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

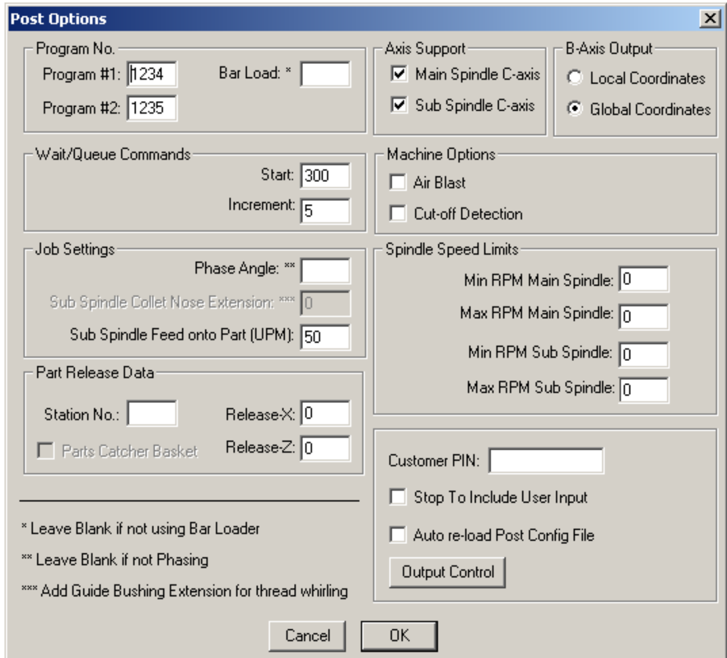


You can learn more about using PartMaker's simulation module in Chapter 5.

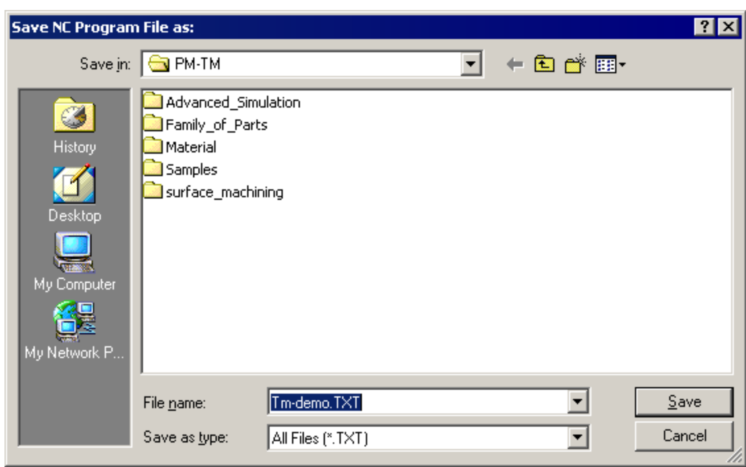
Generating an NC Program

Having selected a Post Configuration file, you are now ready to generate an **NC Program.**

1 Choose **Generate NC Program** from the **Job Optimizer** menu. Set the **Post Options** dialog as shown below. Make sure to uncheck the **Stop To Include User Input** box. More information about this dialog is contained in the machine specific addendum supplied with your PartMaker software.

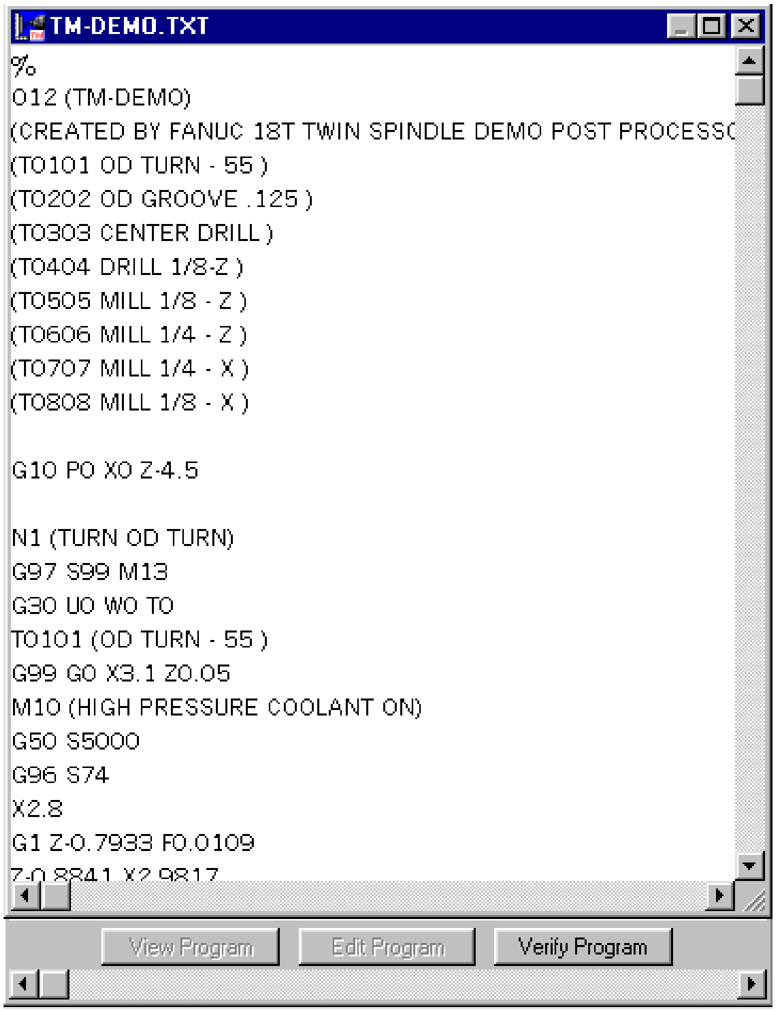


2 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog. In this dialog, enter a name for your program.



3 Click **<Save>** to generate the NC program.

After you save to generate the NC Program this information will appear as shown below:



Chapter 4: TurnMill Tutorial Part: 8-Axis Twin Spindle Twin Turret

Introduction

This tutorial is designed to help you learn the steps to go through in using **PartMaker** TurnMill to program TurnMill lathe parts. The illustration on the following page represents a typical part that would be cut on a CNC TurnMill.

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.

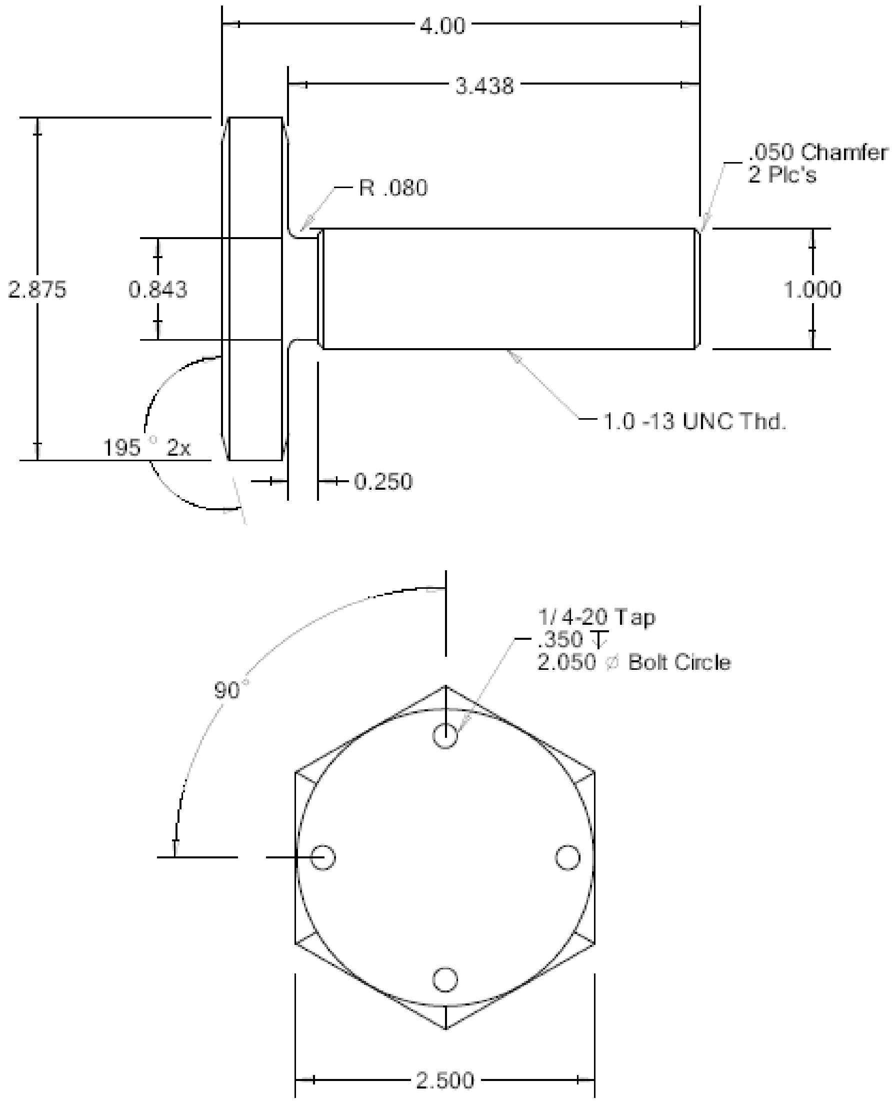
How you will create the advanced TurnMill tutorial part

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

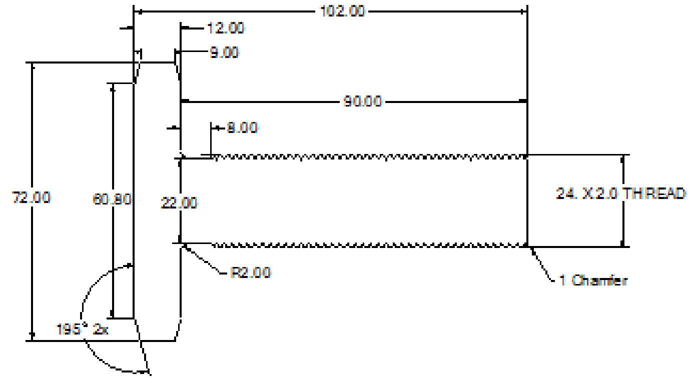
- √ Start PartMaker
- √ View tool information for the part
- √ View cycle information for the part
- √ Select a material for the part
- √ Define the stock boundaries for the Face Window
- √ Open a new Face Window
- √ Create geometry
- √ Perform turning operations on the main spindle
- √ Perform turning operation on the sub-spindle
- √ Mill a hex on the sub-spindle
- √ Drill and tap on the sub-spindle
- √ Generate a Process Table
- √ Synchronize machining operations on the main and sub-spindles
- √ Simulate the cutting of the part
- √ Generate an NC Program

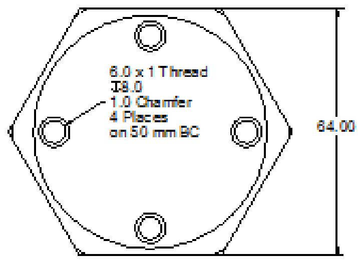
The print below shows the part to be programmed in this tutorial.

Inch Units

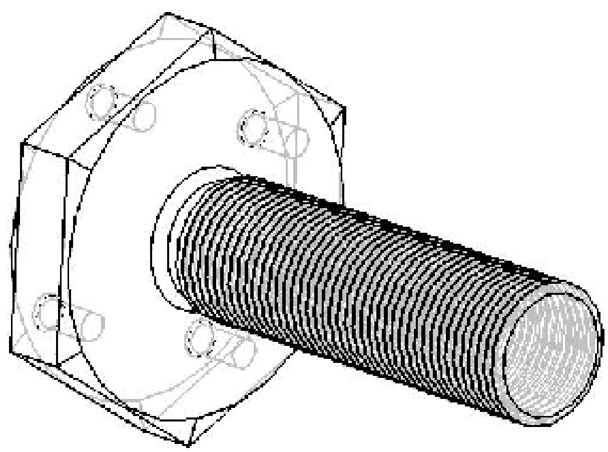


Metric Units





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



Getting Started

The first part of any multi-axis turning programming exercise with **PartMaker** TurnMill is to decide which machining functions will be used for each part face to be programmed.

The machining functions used in this tutorial are as follows:

Turn – Main Spindle

Use turning tools to perform a variety of turning operations on the main spindle including facing, turning, grooving, and threading.

Turn – Sub-Spindle

Use turning tools to perform a variety of turning operations on the sub-spindle including facing and turning.

Mill Polygon – Sub-Spindle

Use an end mill to create a set of hexagonal flats

Mill End Index – Sub-Spindle

Use an end mill to create a set of hexagonal flats.

Programming TurnMill lathe operations with **PartMaker** is performed in a unique manner. No matter what the orientation of the part in the main spindle or sub spindle, **PartMaker** programs all turning operations as if being machined in a conventional lathe, i.e. all programming is carried out from right to left.

Note: This tutorial covers both general **PartMaker** programming techniques as well as those unique to TurnMill. For more information on using **PartMaker** to program CNC Milling, 2-axis turning and Swiss-type applications, please see the manuals which correspond to those applications.

Now you will look in detail at the steps necessary to program each of the components of the finished part. The components of the completed TurnMill tutorial part can be found in the following directories:

Completed Job File:

c:\partmaker\pm-tm\samples\8-axis_demo\Inch-8-axis_demo\8-axis_demo.job

Completed Tools File:

c:\partmaker\pm-tm\samples\8-axis_demo\Inch-8-axis_demo\8-axis_demo.tdb

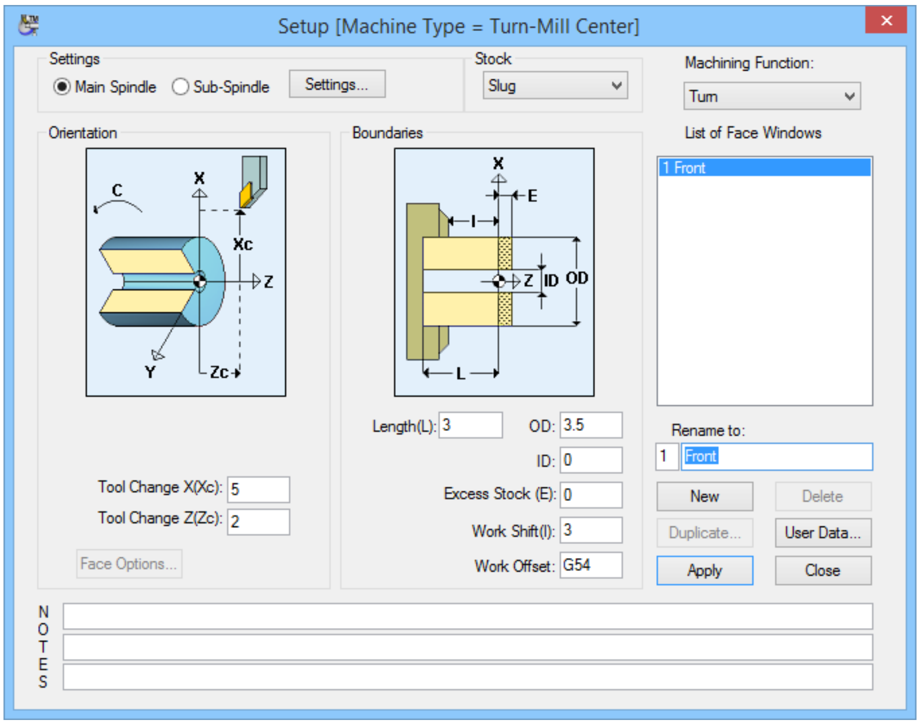
Completed Cycles File:

c:\partmaker\pm-tm\samples\8-axis_demo\Inch-8-axis_demo\8-axis_demo.cdb

Starting PartMaker TurnMill

All programming for TurnMill parts done with **PartMaker** should be done in the TurnMill Application. To start the **PartMaker** TurnMill application:

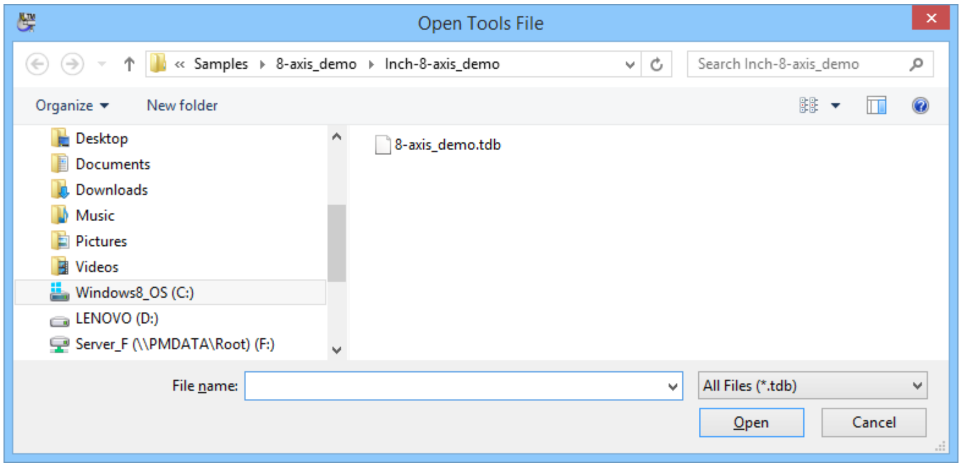
1. Double click on the TurnMill icon on your desktop or choose **TurnMill** from the **PartMaker** Group accessed through Programs under the Windows Start button.
2. An icon containing the **PartMaker** logo will appear for a few moments. You will then see the **Setup** definition dialog box shown below. Click the **<Close>** button.



Opening Files

The tool library for the TurnMill tutorial has already been created.

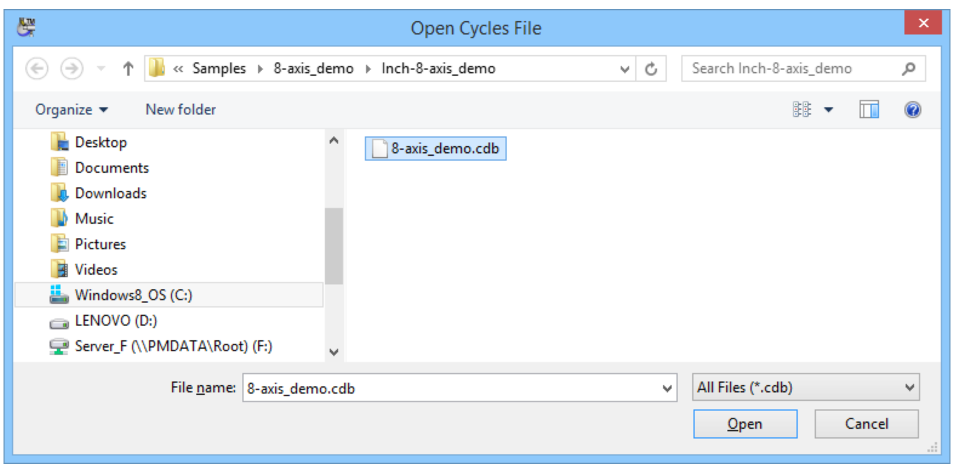
1 Open the Tools file by choosing **Open Tools File** from the **File** menu and choosing the 8-axis_demo.tdb from the **c:\partmaker\pm-tm\samples\8-axis_demo\ Inch-8-axis_demo** directory.



2 Click the **8-axis_demo.tdb** file as shown above and press **<Enter>**.

The cycles library for the TurnMill tutorial has already been created.

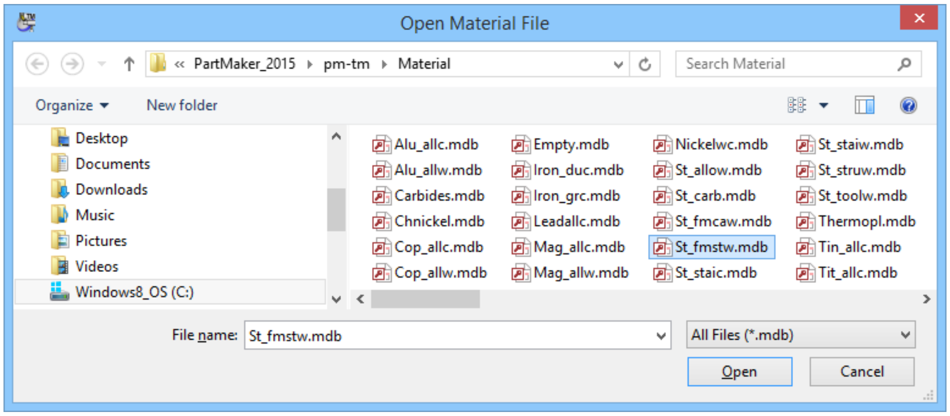
3 Open the Cycles file by choosing **Open Cycles File** from the **File** menu and choosing the 8-axis_demo.cdb from the **c:\partmaker\pm-tm\samples\8-axis_demo\ Inch-8-axis_demo** directory.



4 Click the **8-axis_demo.cdb** file as shown above and press **<Enter>**.

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

5 Open the Material file by choosing **Open Material File** from the **File** menu.



6 Choose **st_fmstw.mdb** as shown below and press **<Enter>**.

You have now loaded the tools, cycle, and material files that were previously developed for the TurnMill tutorial. **PartMaker** has also provided a completed job file. At this point, do not open the job file. You will be guided through a step-by-step process to develop a job file for the tutorial.

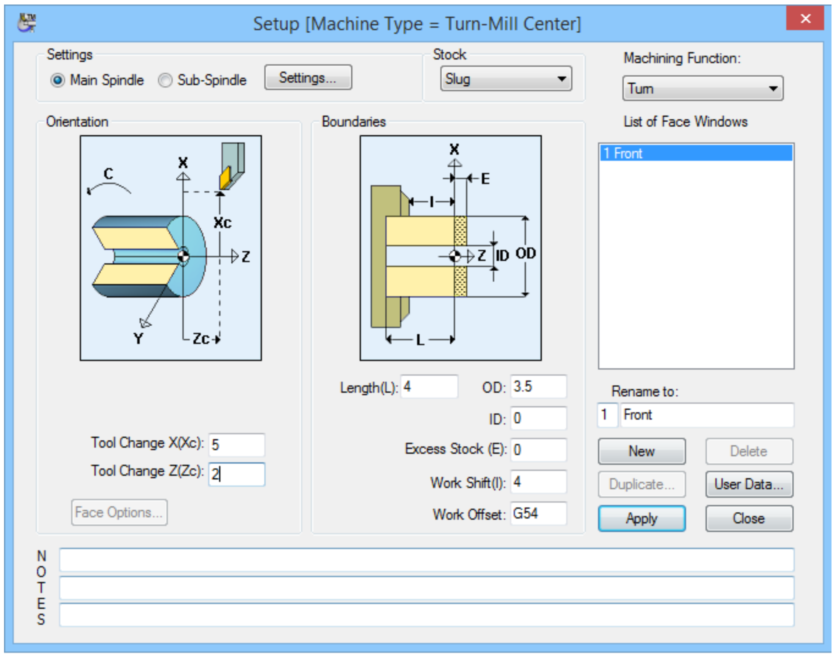
Turning on the Main Spindle

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

Setting Up the Face Window

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

1 Choose **Setup** from the **View** menu to view the **Setup** dialog box (or click the **Shortcut to the Setup** icon in the upper right hand corner of the screen). The following default dialog will display:



In this dialog, you will define the machining methods, part boundaries, and location of the stock with respect to the programming origin.

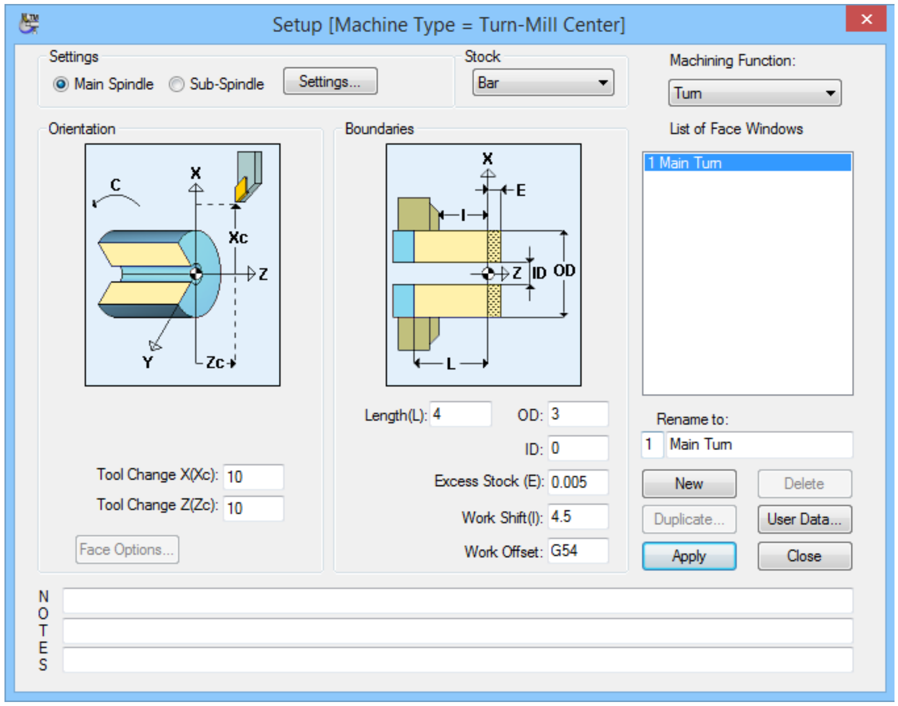
2 Make sure **Stock** is to **Bar**

3 Rename the current **Face Window** by clicking on **Front** in the **List of Face Windows:** field and type **Turn– Main** in the **Rename To:** field and click the **<Apply>** button.

4 Now enter the parameters listed below in the appropriate fields:

Length (L)	4.0	(102.0)
OD	3.0	(75.0)
ID	0	(0)
Excess Stock (E)	0.005	(0.5)
Work Shift(I):	4.5	(125)
Work Offset:	G54	
Tool Change X (Xc)	10.0	(100)
Tool Change Z (Zc)	10.0	(100)

Your completed **Setup** dialog should appear as shown below:



Note: *In the dialog above, the **Main Spindle** radio button is **<u>checked</u>**. This designates that this operation will be programmed on the main spindle of the machine.*

5 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.

6 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.

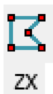
Tip: *If at any time the menu commands become "grayed out" and unusable, this indicates that the **Face Window** you are working in has become inactive. To reactivate the window, click anywhere within your **Face Window**.*

Creating Geometry in CAD Mode

In this section, you will create the geometry for the turning operations to be performed on the main spindle. You will either create the geometry for the turning operations to be performed on the main spindle or import a DXF file. Here, you will create the part geometry in PartMaker CAD:

To create the geometry in **PartMaker** CAD:

1 Click on the **CAD/CAM** switch in the lower left-hand corner of the window. The icon will change its appearance from a tool bit to a pencil.



2 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.

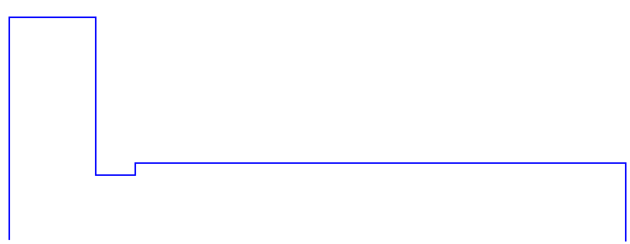
After selecting the **ZX Snap Mode** two data entry fields will appear in the lower left hand corner of the **Face Window**.

Note: *When plotting coordinates in **PartMaker** CAD, 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. You can move between data entry fields by pressing the **<Tab>** button or using your mouse.*

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

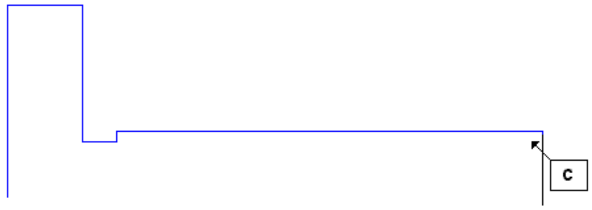
0,	0
<default>,	1 (24)
-3.187 (-82),	<default>
<default>,	0.843 (22)
-3.438 (-90),	<default>
<default>,	2.875 (72)
-4 (-102),	<default>
<default>,	0
1.1 (28.0),	<default>
<default>,	0.146 (3.75)
1.2 (30.5),	<default>

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



3 You will now place a chamfer on the front of the part. To do so, select the **Chamfer** icon. A field will then open up at the bottom of the window prompting you to enter the size of the chamfer. Enter a value of 0.04 (1.0) and press **<Enter>**.

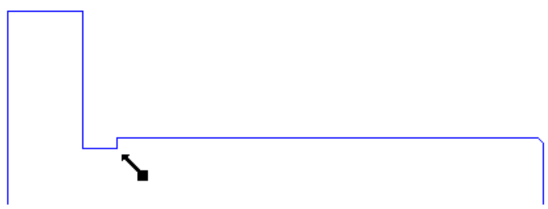
To place the chamfer, position your cursor on the inside of the intersection of the lines at the front of the part as shown below:



With your cursor in position, click the left mouse button and a chamfer will inserted as shown below:

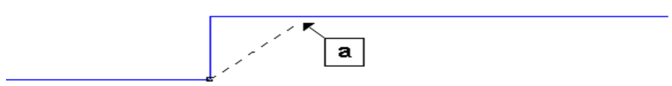
4 You will now place a second chamfer on the part, at a distance 3.187 (82) from the front of the part. This time, however, no chamfer dimension is given on the print, but you do want to create 45 degree chamfer from the 0.843 (22) diameter to the major thread diameter.

Click on **Line on an Angle Icon** and select **End of an Element Snap Mode** icon. Drag your cursor into place as shown below and click the left mouse button.



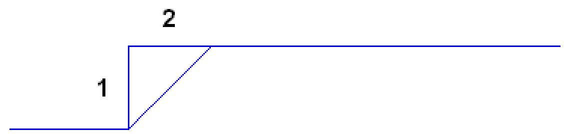
You will now be prompted to enter an angle for this line. At the prompt enter a value of 45 and press **<Enter>**.

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.

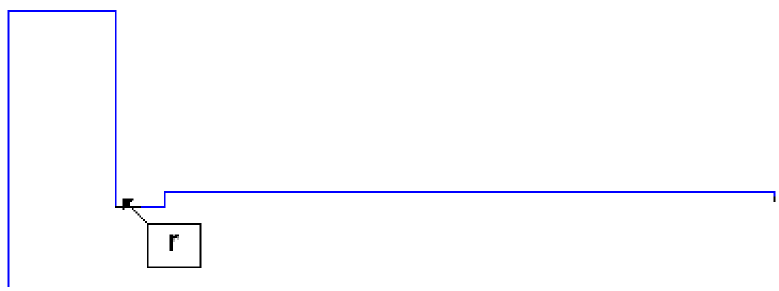


After having done so, your geometry will appear as shown below:

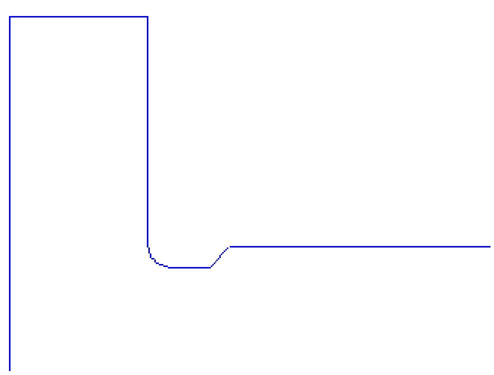
5 Next you will create the .08 (2.0) radius at the back of the relief groove. To do so, choose the **Fillet Icon**.

You will now be prompted to enter the arc radius. At the prompt enter a value of .08(2.0) press **<Enter>**.

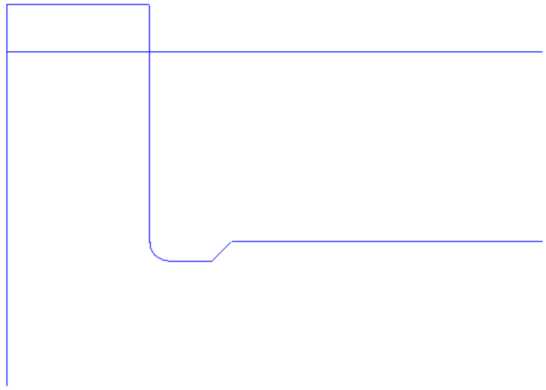
Move your cursor so the arrow is at the inside of the intersection of the two lines at the back of the relief groove.



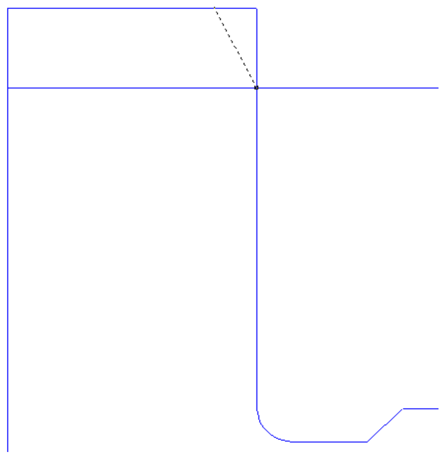
Having done so, you will now have a drawing which appears as shown below.



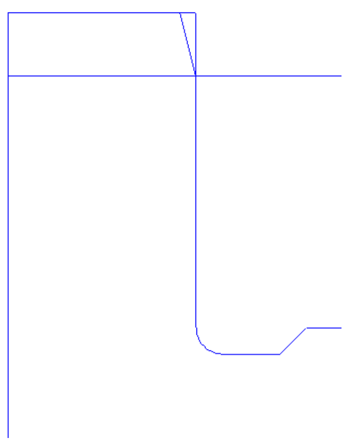
6 Next, choose the **Line Parallel horizontal axis icon**. Enter a value of 2.5 (1.5) and press **<Enter>**. A line as shown below will be created.



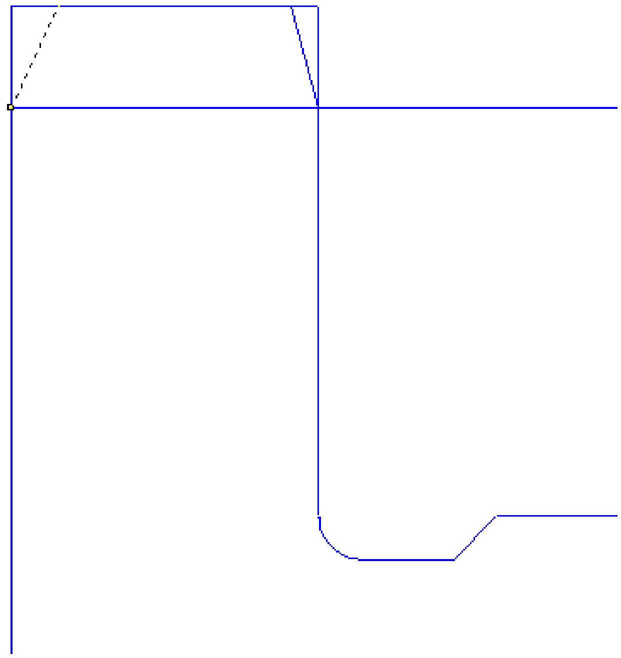
7 Choose **Line on an Angle Icon** . Choose the **Closest Intersection Snap Mode** icon and enter a value of 105 in the angle field. Select the point as shown in figure below and then click on the horizontal line above it as shown.



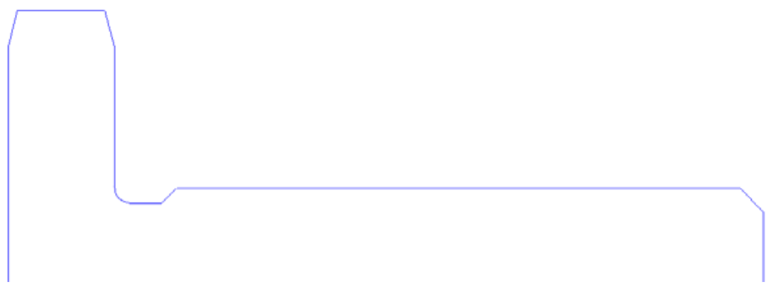
The geometry should appear as shown below:



8 Choose **Line on an Angle Icon**. Choose the **Closest Intersection Snap Mode** icon and enter a value of 75 in the angle field. Select the point as shown in figure below and then click on the horizontal line above it as shown.



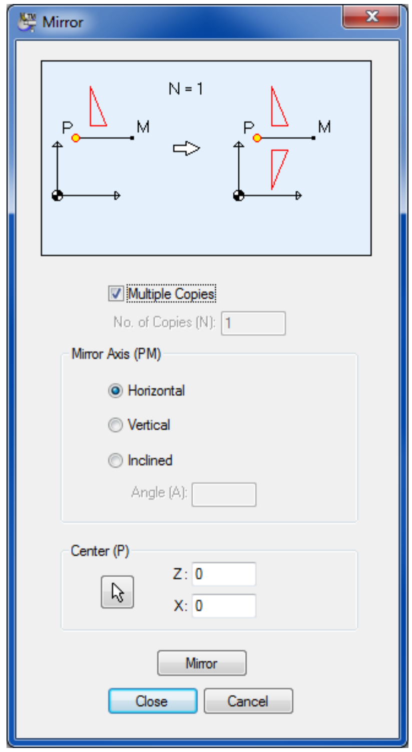
9 Now trim the irrelevant geometry using the scissors icon and the final geometry should look as below



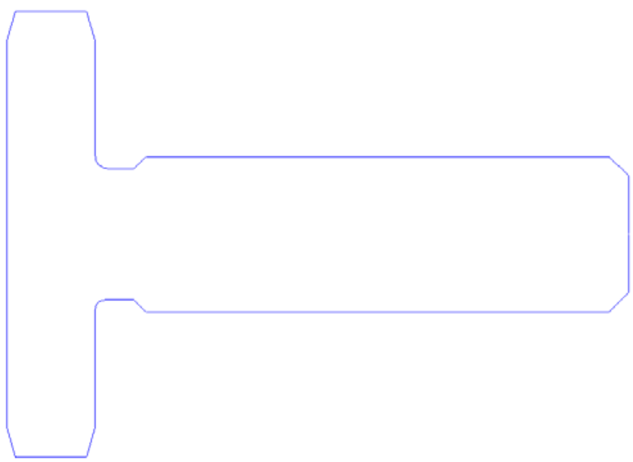
10 Once you have created the two chamfers, choose the **Selection Icon** and select the geometry you have created by choosing **Select All** from the **Edit** menu.

Note: *In* ***PartMaker*** *selection always preceeds action. Before you can manipulate an entity, be it a piece of geometry or a part feature, you must select it. Selection must always be carried out in selection mode. To be in selection mode, you must have the* ***Selection*** *icon chosen.*

11 You will now mirror the selected geometry by choosing **Mirror** from the **Edit** menu. Complete the **Mirror** dialog as shown below:



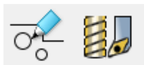
When you have completed the dialog as shown above, click the **<Mirror>** button and close the dialog. Click anywhere to deselect the geometry. The completed drawing should appear as below:



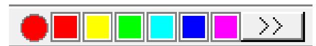
Defining the Profile for Face Turning

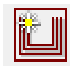
The first turning operation to be performed on the Main Spindle is to face the stock.

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



2 Click anywhere in the **Main-Turn** Face Window to make it active.

3 Choose the red square in the **Color Bar**. 



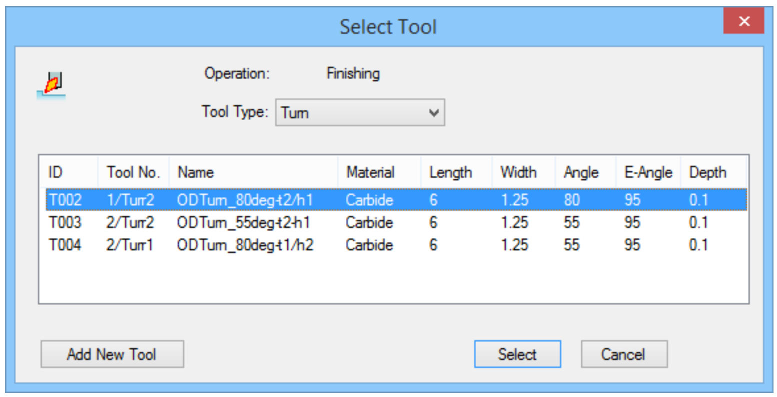
4 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Group Parameters** dialog, or select the button from the tool bar.

5 Choose **Contouring** from the **Strategy** drop down menu.

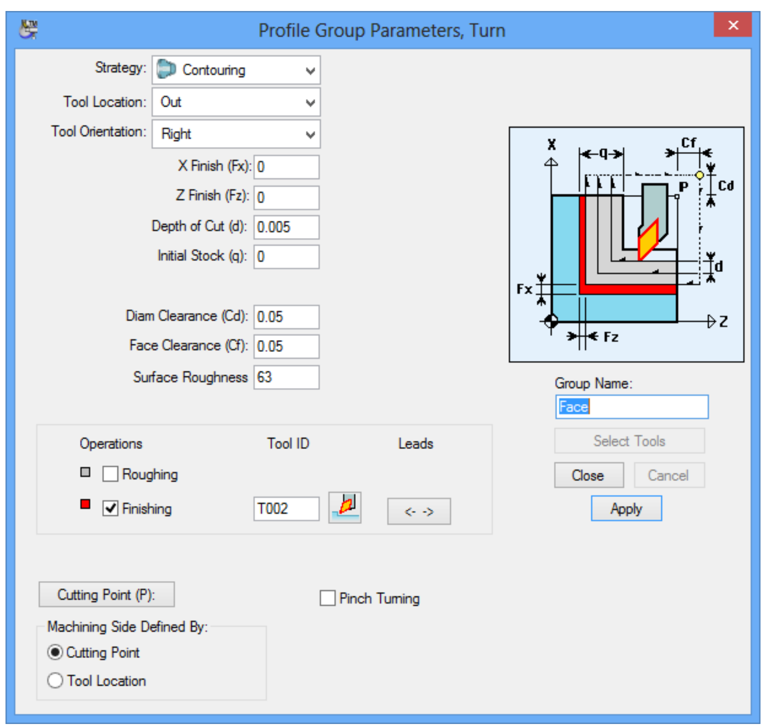
6 Choose **Face** from **Tool Location**.

7 Deselect Roughing which will select Finishing automatically

8 Click the **Select Tools** Button. You will be prompted with a choice of three available turning tools. Choose **T002** and click **<OK>**.



9 In **Group Name** type **Face**. Your completed **Profile Group Parameters** dialog should appear as shown below:



At this point, **do not** close the dialog box shown above. Proceed for instructions on how to complete setting up this profile group.

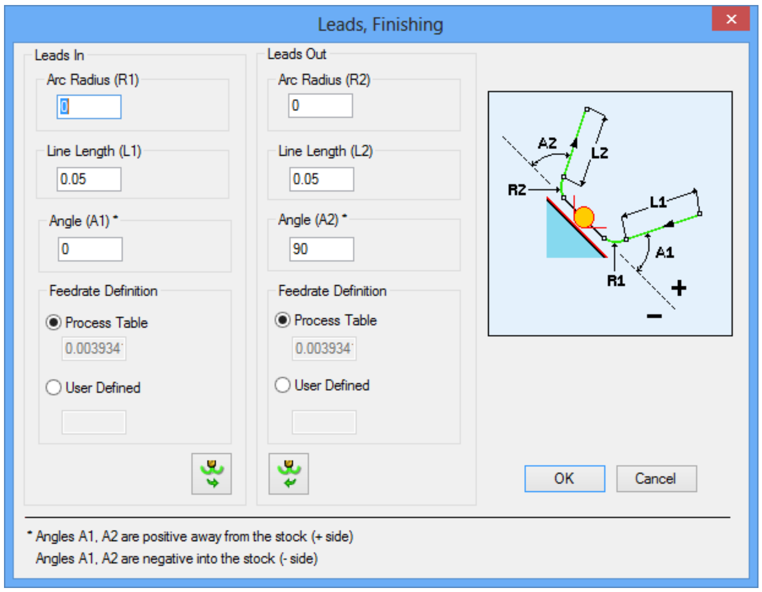
When programming parts in **PartMaker**, you have the ability to control how far and at what angle a tool feeds on and off of a part. These values are controlled via **Leads**. The **Lead In** is the distance from the part the tool will begin feeding before cutting. Likewise, the **Lead Out** is the distance that the tool will feed away from the material before tool change or rapidding to its next position.

Note: *If you are using the same **Leads In** and **Leads Out** in most of your operations, you can set them as default values in the **Defaults for Turning** dialog accessed under the **Job Optimizer** menu. Setting your own default Leads In and Out values will save time when programming in **PartMaker**.*

1 The **Leads In** and **Out** dialog box combine the settings for controlling the movement of the tool as it approaches (Leads In) and leaves (Leads Out) the stock.

2 Set the **Leads In** values for the operation. Now set the **Leads Out** values for the operation.

3 You can also set the values from one **Leads** to the other by selecting the Copy Leads button.

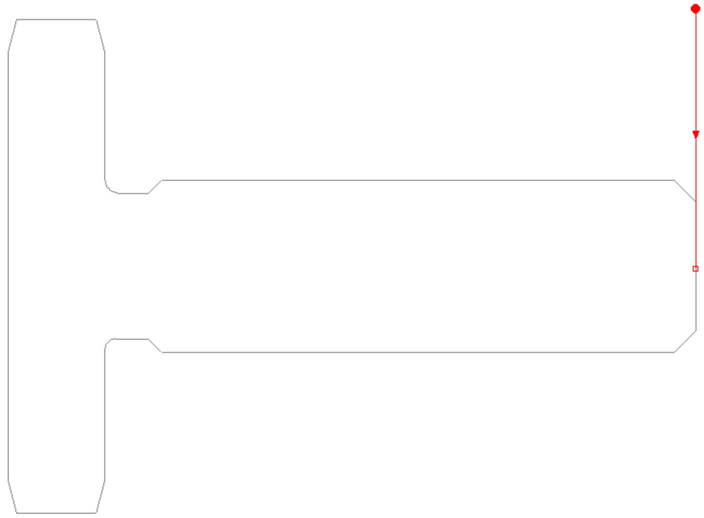


Press **<OK>** to return to the **Profile Group Parameters** dialog.

Note: *The **Lead** angle is always set normal to the path of the tool by **PartMaker**. Above, setting the **Leads Out** angle will allow the tool to retract in a manner perpendicular to the part. For example, if turning a 45 degree chamfer, a 0 degree lead angle would allow the tool to exit the stock on a 45 degree angle.*

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

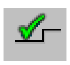
Click the <**Close**> button to close the **Profile Group Parameters** dialog box. When you close the dialog box it will automatically create a tool path on the face of the part.



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.

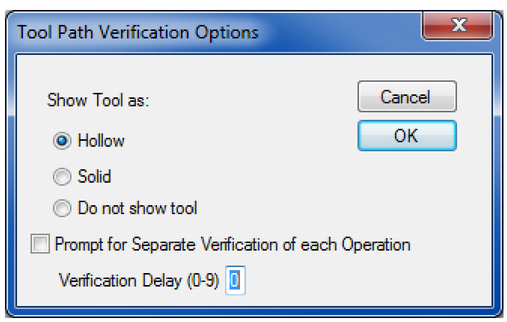
Tool Path Verification

Once a tool path is defined, you can verify its shape to gauge the accuracy of the cut you have defined. To do so:



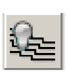
1 Select Verify Work Group Tool Path from the Part Features menu or the button:

You will be prompted with the **Tool Verifications Options** dialog as shown below:



2 Check the **Solid** or **Hollow** radio button and enter a delay between 0 and 9, zero being the fastest and nine the slowest, relative to the processor speed of your computer.

3 Click **<OK>**. You should see your tool path verified.

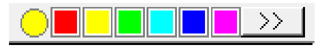
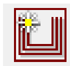


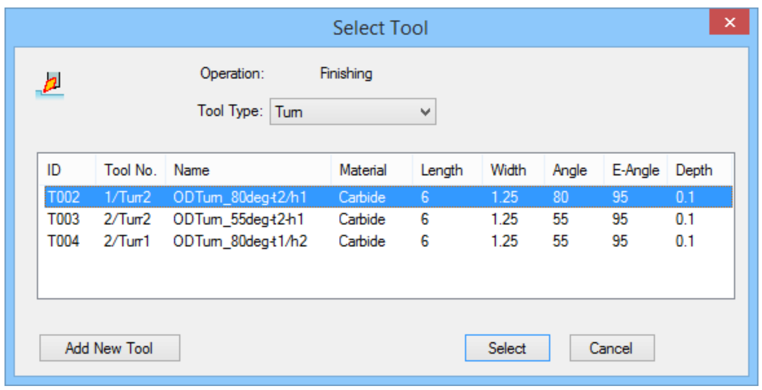
4 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu, or select the button.

Note: *Whenever you verify a tool path (or every tool path) you MUST choose* ***Hide Work Group Tool Path*** *before moving on and continuing to work in* ***PartMaker****. Not doing so could hamper performance of your system. In order to stop verification before it is complete, hit the* ***<Esc>*** *key.*

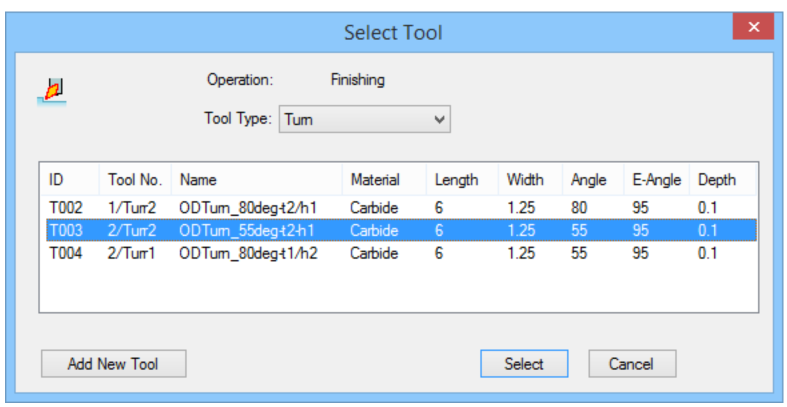
Defining the Profile for OD Turning

The next operation to be performed will be OD turning, starting from the bottom of the chamfer on the face of the part.

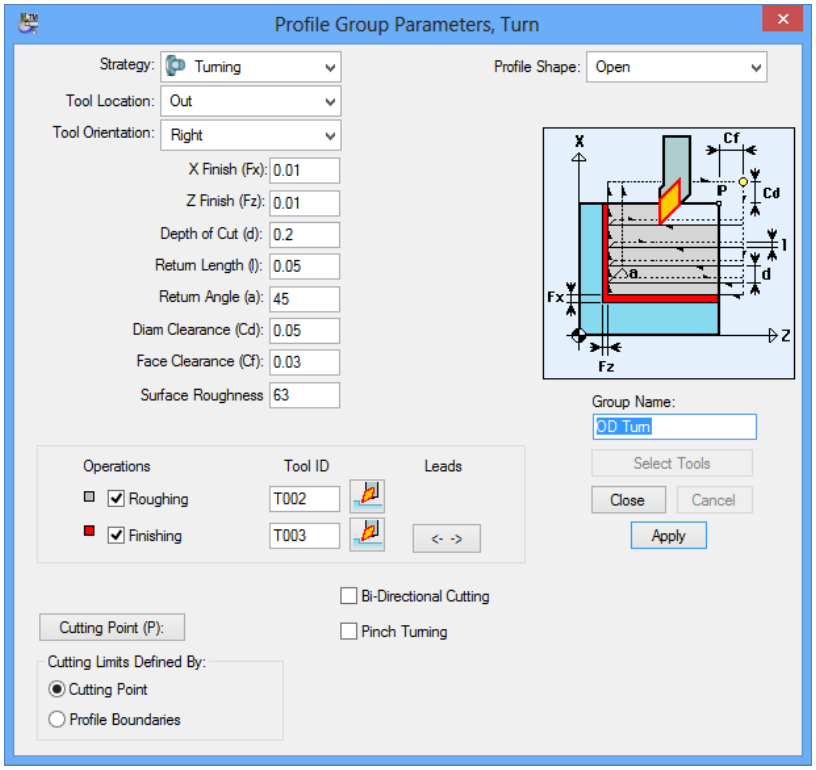
1. Choose the yellow square in the **Color Bar.** 
2.  Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog, or select the icon.
3. From the **Strategy** menu, select **Turning**.
4. From the **Tool Location** menu, select **Out**.
5. From the **Tool Orientation** menu, select **Right**.
6. Enter a **Depth of Cut** of 0.2 (2.4).
7. In the **Group Name:** field enter **OD Turn**.
8. Check the box labeled **Finishing Tool ID** to indicate that you want to perform a finishing opertation.
9. Click the **Select Tools** button. Click the **<OK>** button to choose **T002** as the roughing tool.



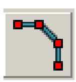
10. Upon doing so, you will be immediately be prompted with the same dialog, prompting you to select a finishing tool. Click on **T003** from the list and click **<OK>**.



The completed **Profile Group** Parameters dialog should appear as shown below.

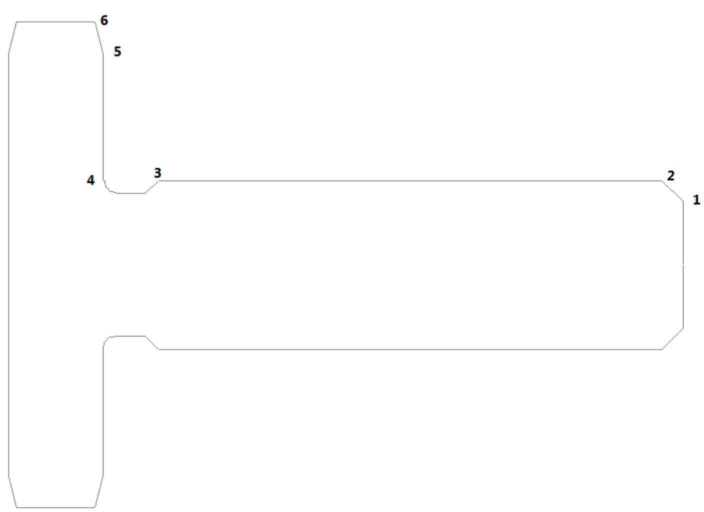


Now that you have defined the cycle to be used, you must tell **PartMaker** the location of the tool path. Click <**CLOSE**>

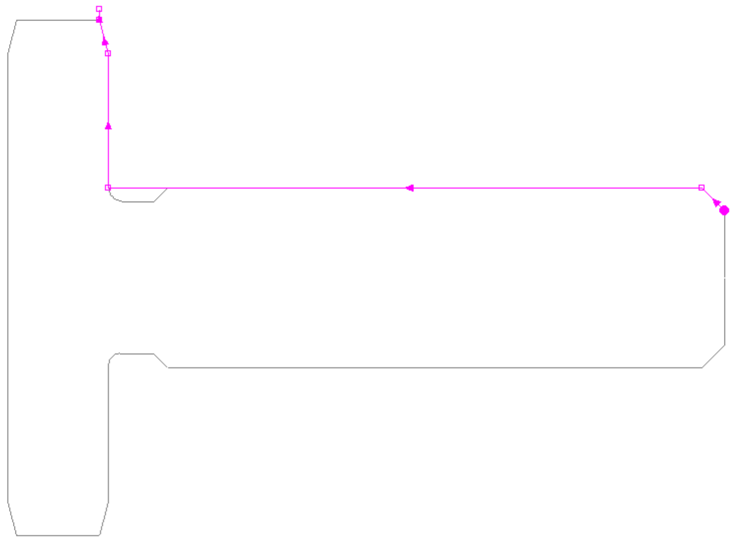


1 From the left-hand side of the screen select the **Define Profile** icon.

2 From the top of the screen, select the **End of an Element snap mode** icon. To define the OD Turn path, click your left mouse on positions to match the diagram below.

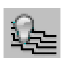


3 To finish the tool path, use **ZX Coordinates Snap Mode** and enter a value of 3.0 (76) in X field. The completed tool path should appear as shown below:



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

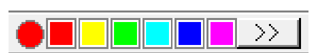
5 Select **Verify Work Group Tool Path** from the **Part Features** menu to check your work., or select the tool-bar icon.

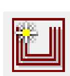


6 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu, or the tool-bar icon.

Defining the Profile for OD Grooving

The next operation to be performed will be OD grooving.

1 Choose the red square in the **Color Bar.** 



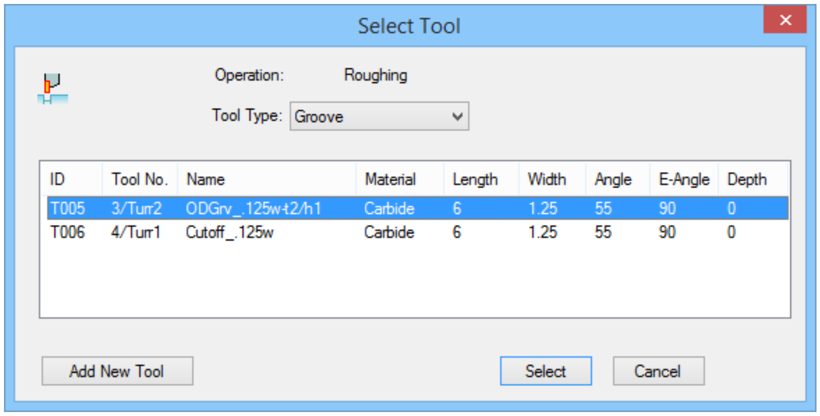
2 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog or select the icon.

3 Change your **Strategy** to **Grooving**.

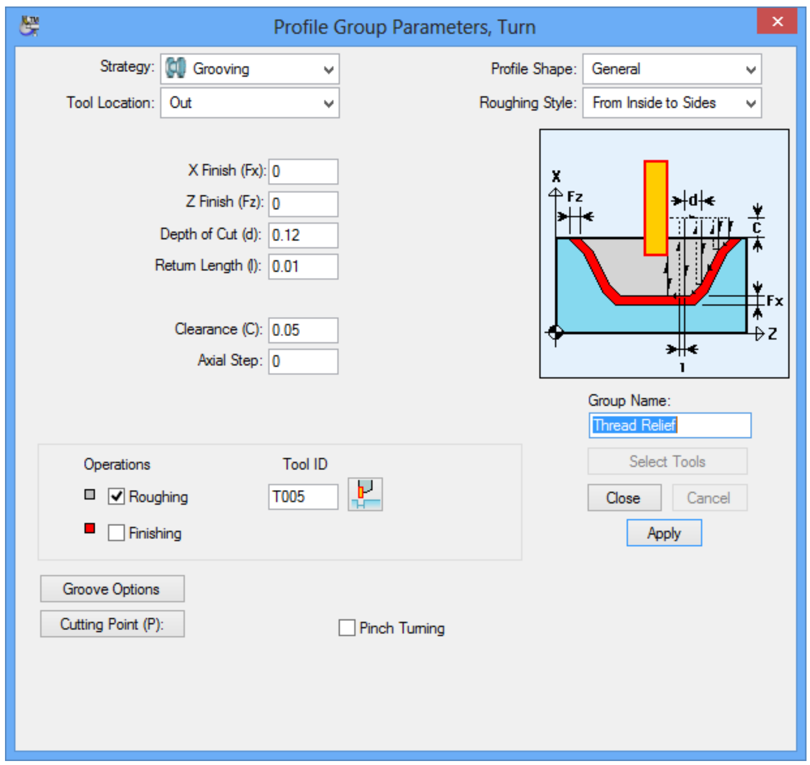
4 Choose a **Tool Location** of **Out**.

5 Look at the graphic. Notice that there are two options for **Profile Shape**: **Rectangular** and **General**. **Rectangular** would be used where the groove is a rectangular shape. **General** would be used if there were a contour your grooving tool would have to follow. It would also be used if you wanted to take a rough and finish pass. Take a minute and look closely at both graphics and the parameters for each. When you are finished, leave this setting at **General**.

Click the **Select Tools** button. You will be prompted with a choice of two tools. Choose **T005** and click **<OK>**.

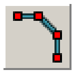


6 Rename the group to **Thread Relief**. Your completed profile group parameters dialog should now appear as shown below:



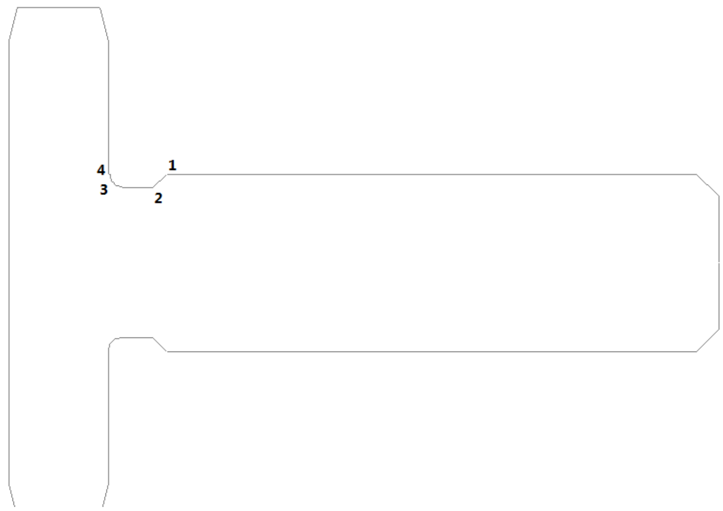
7 Click <**CLOSE**>

Now that you have defined the cycle to be used, you must tell **PartMaker** the location of the tool path.

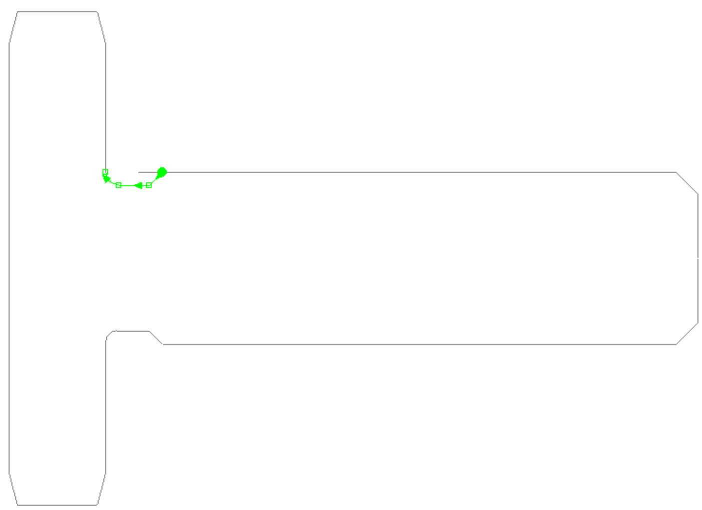


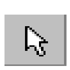
1 From the left-hand side of the screen select the **Define Profile** icon.

2 From the top of the screen, select the **End of an Element snap mode** icon. To define the Groove path, click your left mouse on positions to match the diagram below.

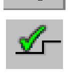


The completed profile should appear as shown below:

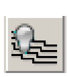




3 Click the **Selection** icon to bring yourself to a home or neutral position



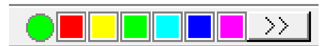
4 To check your work, select **Verify Work Group Tool Path** from the **Part Features** menu or select the icon.

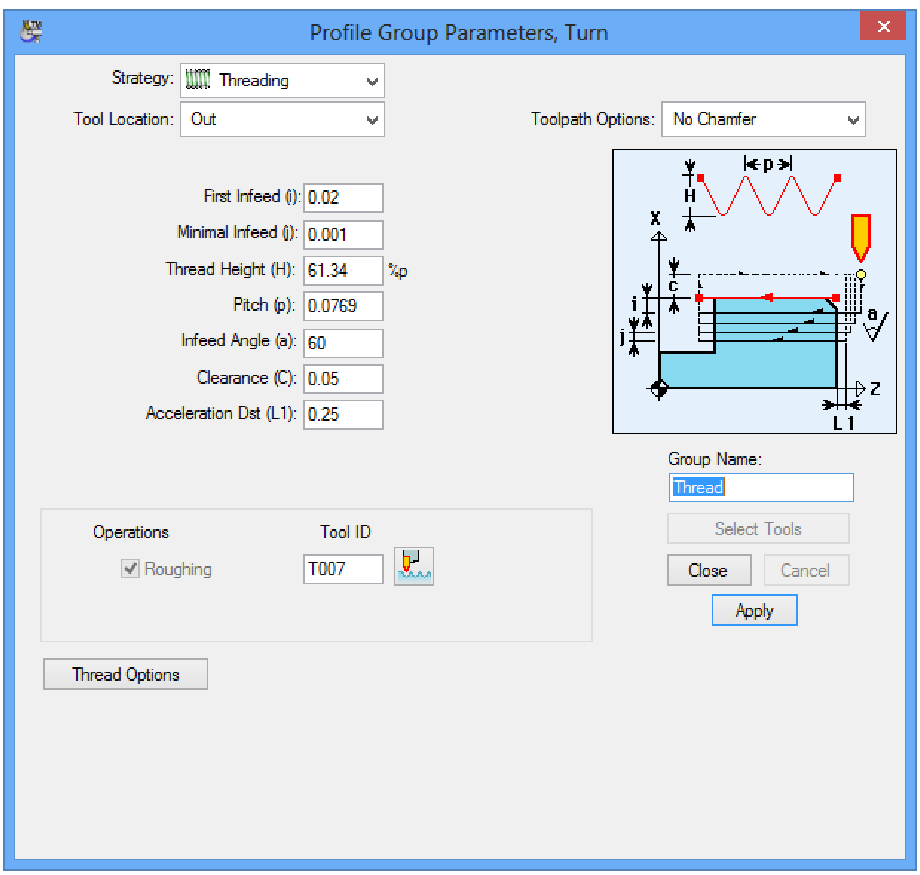


5 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu.

Defining the Profile for OD Threading

In this operation you will create the OD thread on the main spindle. In **PartMaker**, threading operations are always programmed based on the crest diameter of the thread (i.e. major diameter on the OD, minor diameter on the ID).

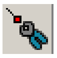
1. Choose green from the **color bar**. 
2. Select New Profile Group from the Part Features menu.
3. From the **Strategy** menu, select **Threading**.
4. Look at the graphic. Notice that there are two **Toolpath Options**: **No Chamfer** and **Chamfer**. **No Chamfer** would be used where there is a relief groove at the end of the thread. **Chamfer** would be used if there were no relief groove at the end of the thread. Take a minute and look closely at both graphics and the parameters for each. When you are finished, leave this setting at **No Chamfer**.
5. Enter a **Pitch** of 1/13 (1.5). You can enter the fractional value for the pitch in the field
6. Click the **Select Tools** button and allow PartMaker to select the appropriate tool.
7. In the **Group Name:** field enter **Thread**.



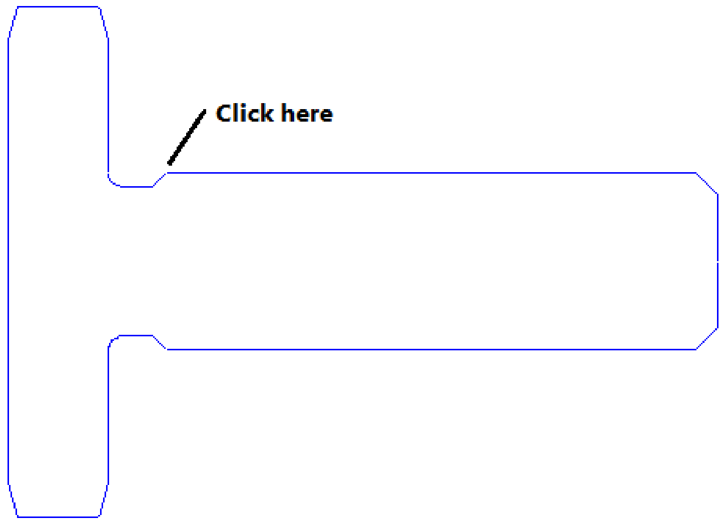
8. Close the **Profile Group Parameters** dialog for Threading by clicking the <**Close**> button.

Before you assign the **Threading** cycle to the geometry, it is necessary to extend the line along which you will be threading into the middle of the relief groove. This is a common technique when defining threading operations in PartMaker. To do so:

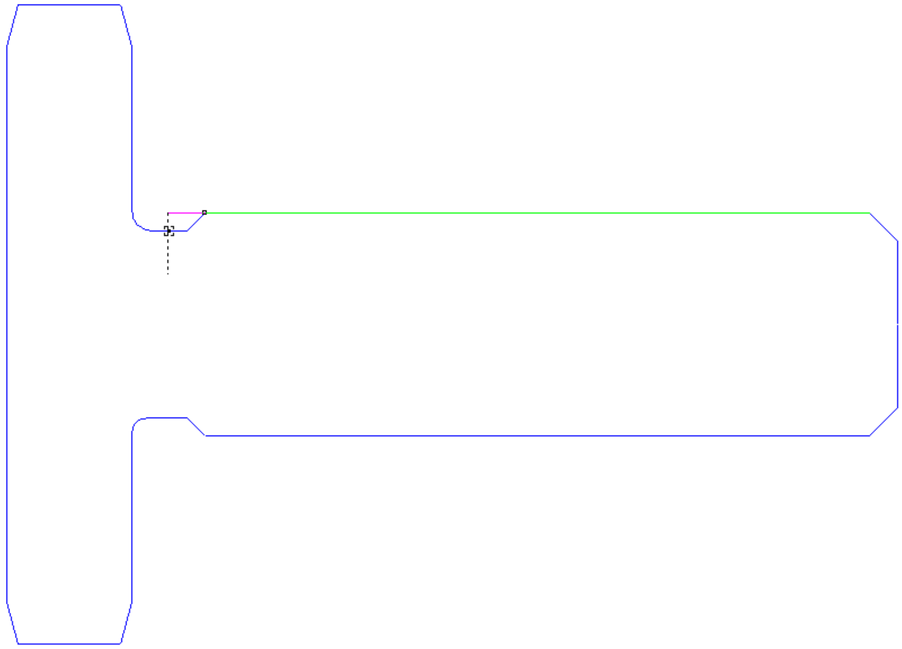
1 Click on the CAD/CAM switch to enter the CAD mode.



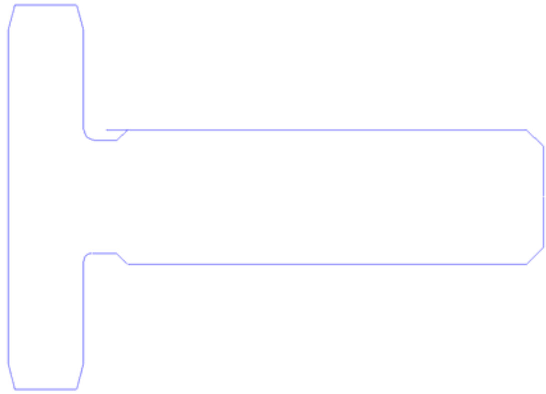
2 Choose the **Trim and Extend** icon on the left hand side of the screen. Click toward the left side of the 1 inch diameter along which you will be threading by positioning your cursor and clicking as shown below:



3 Choose the **Middle of an Element snap mode** icon on the left hand side of the screen. Click toward the left side of the 1 inch diameter along which you will be threading and drag your cursor to the left and clicking near the middle of the bottom of the groove as shown below:



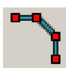
The completed geometry should appear as shown below:

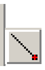


You can now assign the **Threading** cycle to the geometry you created.

4 Return to CAM mode.

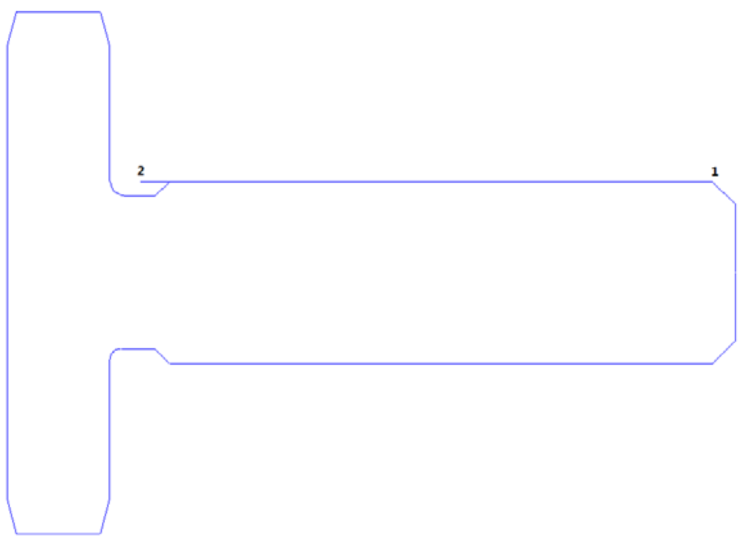
5 From the **View** menu, choose **Show Work Group Only** to turn off all the other tool paths other than the one you are about to create.



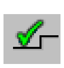


6 From the left-hand side of the screen select the **Define Profile** icon.

7 From the top of the screen, select the **End of an Element snap mode** icon. Click on points 1 and 2 a shown below.



8 Click the **Selection** icon to bring yourself to a home or neutral position



9 Select **Verify Work Group Tool Path** from the **Part Features** menu or the icon to check your work.

10 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu or use the icon.

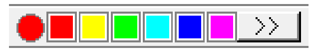
Note: *To better focus on the **Work Group** you are currently working with, you can select **Show Work Group Only** from the **View** menu or the icon.*

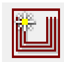
This has been done above to isolate the area along which you will thread. You can see all the work groups again by selecting **Show All** or the icon:

This programming methodology will be employed throughout this tutorial.

Defining the Profile for Cut-off

The next operation to be performed will be the cut-off.

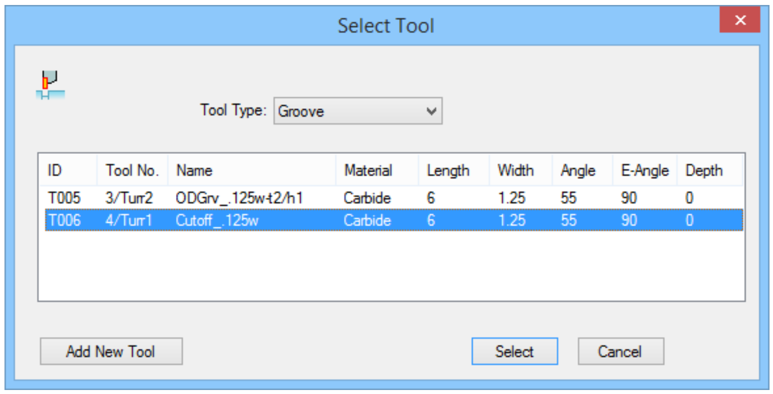
1 Choose the cyan square in the **Color Bar.** 



2 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog, or select the icon.

3 Change your **Strategy** to **Cut-off**.

4 Click the **Select Tools** button. You will be prompted with a choice of two tools. Choose the second tool on the list as shown below and click **<OK>**

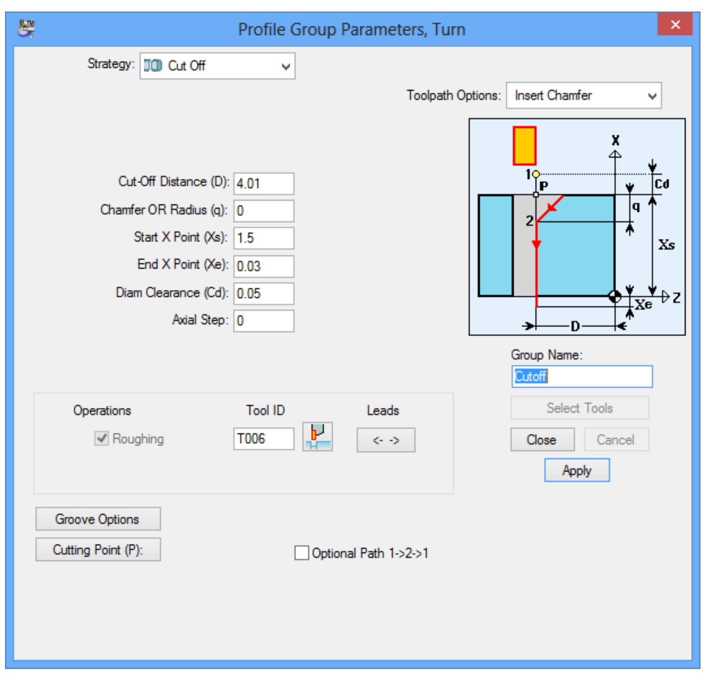


5 Look at the graphic. Notice that there are two options: **Insert Chamfer** and **Insert Fillet**. **Insert Chamfer** would be used when you would like to create a chamfer on the end of the part. **Insert Fillet** would be used when you would like to create a radius on the end of the part. Take a minute and look closely at both graphics and the parameters for each. When you are finished, leave this setting at **Chamfer**.

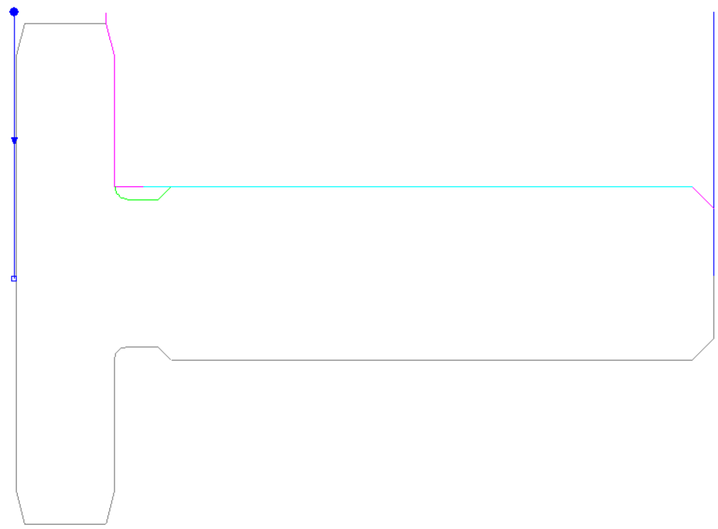
6 In the **Group Name:** field enter **Cutoff**.

7 To the **Cut-Off Distance (D):** field add 4.01 (.25)

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



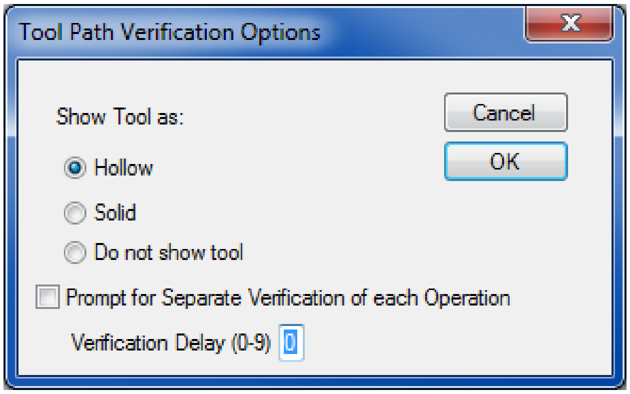
Click the <**Close**> button to close the **Profile Group Parameters** dialog box. When you close the dialog box it will automatically create a tool path on the end of the part. Your tool path should look like the one in the diagram below.



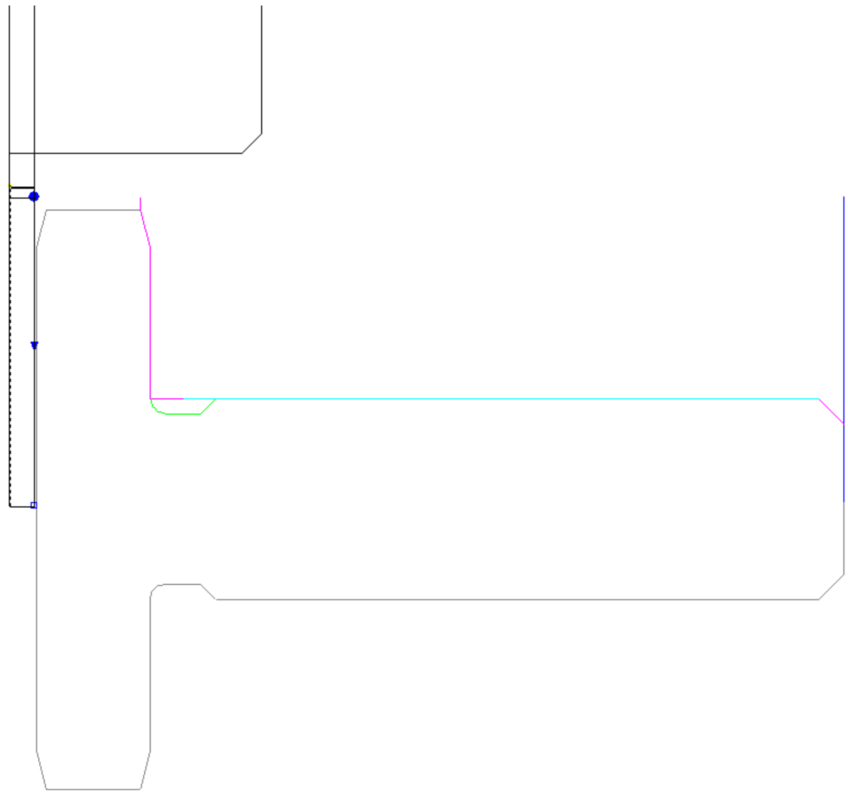
You have now completed the turning work on the main spindle. Now that all the programming in the **Main Spindle - Turn Face Window** is complete, take a moment to see exactly what you have programmed.

1 Choose **Verify Every Tool Path** from the **Part Features** menu or the button

In the **Verification Options** dialog, choose **Hollow** or **Solid** as shown in the box below and click **<OK>**:



PartMaker will show you exactly what you have programmed in a graphical manner as shown below:



2 When you are satisfied your screen appears as the one above, choose **Hide Every Tool Path** from the **Part Features** menu, or use the button:

You have now completed the turning operations on the main spindle for the TurnMill tutorial part. Before moving on, remember to save your work! Do so by selecting **Save** from the **File** menu or by using the shortcut key **<Ctrl + S>**.

Turning on the Sub-Spindle

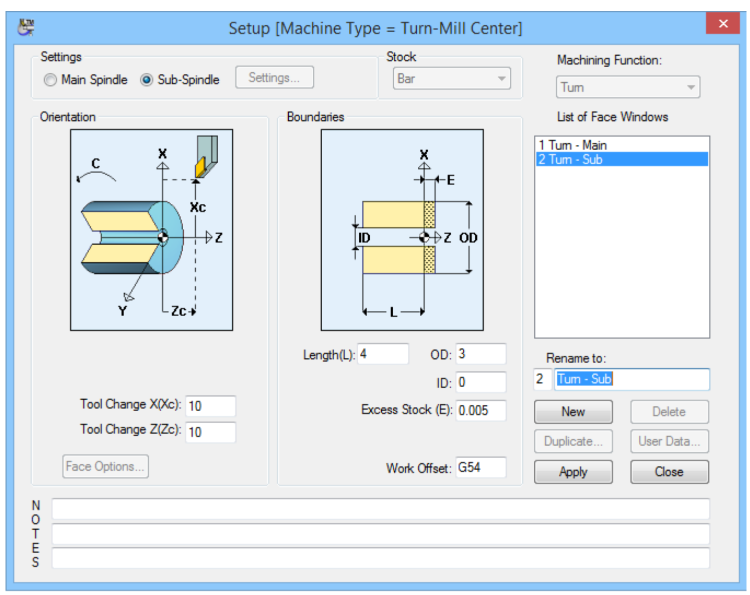
This section of the tutorial will instruct you in the steps necessary to perform a variety of turning operations on the sub-spindle.

Setting Up the Face Window

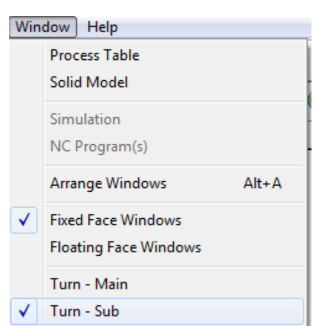
In this section, you will set up the **Face Window** needed to perform turning operations on the sub-spindle. To do so:

1. Choose **Setup** from the **View** menu to view the **Setup** dialog box (or click the **Shortcut to the Setup** icon in the upper right hand corner of the screen).
2. Click the **<New>** button to create a new face Window. Because you will be doing turning in this window, the **Machining Function** will remain of type **Turn**.
3. In the **Rename to:** dialog, enter the name **Turn-Sub**
4. Check the **Sub Spindle** radio button to indicate that machining in this window will be carried out on the sub-spindle
5. When you are satisfied that your **Setup** dialog appears as the one below click the **<Apply>** button. This will apply the parameters to the current **Face Window**. Click the **<Close>** button to close the **Setup** dialog.

   The size and origin location of the stock should remain the same as previously defined in the Turning Setup Window. When you are finished, the Setup dialog should appear as shown below:



To switch to the **Sub Spindle - Turn Window**, choose **Turn-Sub** from the **Window** menu.

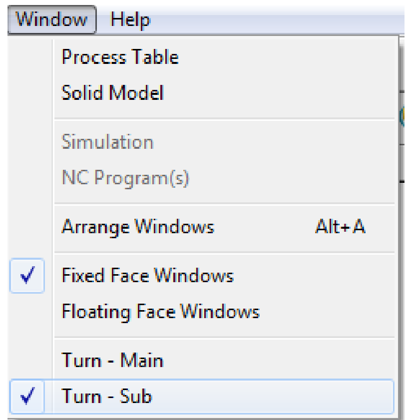


Creating Geometry in the CAD Mode

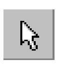
To quickly define geometry for the turning operations to be performed on the sub spindle, you can use the geometry already created in the **Main Turn** window. Remember, in **PartMaker**, all programming is done as if being performed on a conventional lathe, i.e. all programming is carried out from right to left.

As a result, you can copy the geometry previously created in the **Main Turn** window and mirror it to create the desired geometry in the **Turn-Sub** window.

1 Choose **Main Spindle - Turn** from the **Window** menu to return to the first window you created as shown below:



In the **Main-Turn Window**, to copy geometry, click on the **CAD/CAM switch** to return to CAD mode. The icon will change its appearance from a tool bit to a pencil.



2 With the **Selection** icon highlighted, select the geometry you have created by choosing **Select All** from the **Edit** menu.

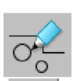
3 With the geometry selected (it should be highlighted), choose **Copy** from the **Edit** menu.

4 Return to the **Turn-Sub Window** by choosing **Turn-Sub** from the **Window** menu.

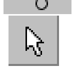
5 In the **Turn-Sub Window**, choose **Paste** from the **Edit** menu.

Remembering the **PartMaker** convention that all programming is carried out from right to left, as if on a conventional lathe, you will now mirror the geometry you just pasted.

To do so:

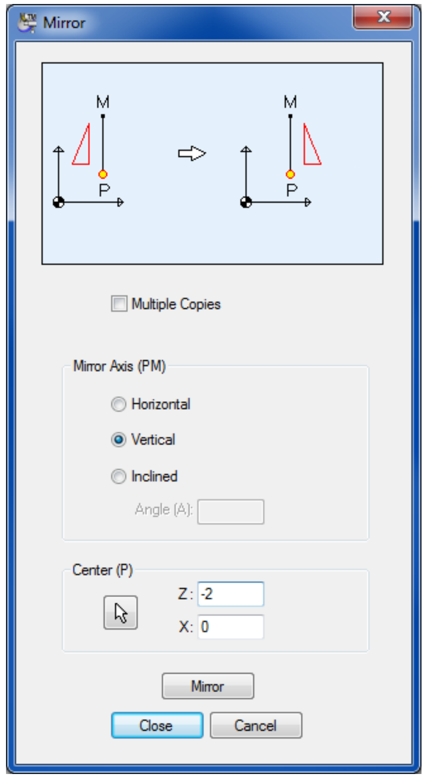


1 Make sure you are in CAD mode in the **Turn-Sub Window**.

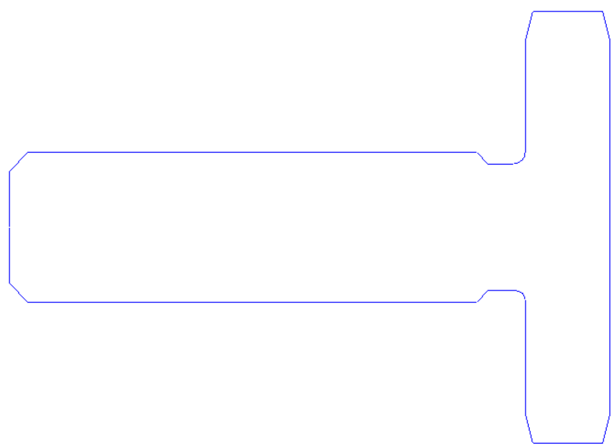


2 With the **Selection** icon chosen, again, choose **Select All** from the **Edit** menu to select the geometry if it is not currently selected.

3 With the geometry selected, choose **Mirror** from the **Edit > Transform** menu. To flip the part in order to perform operations on the back, you will need to mirror with respect to the middle of the part. To do so, complete the **Mirror** dialog as shown below:



The completed (mirrored) part should appear as shown below:

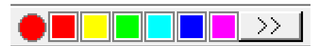


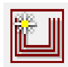
Defining the Profile for Face Turning on the Sub-Spindle

The first turning operation to be performed on the Main Spindle is to face the stock.

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

2 Choose the red square in the **Color Bar.**





3 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Group Parameters** dialog, or use the button:

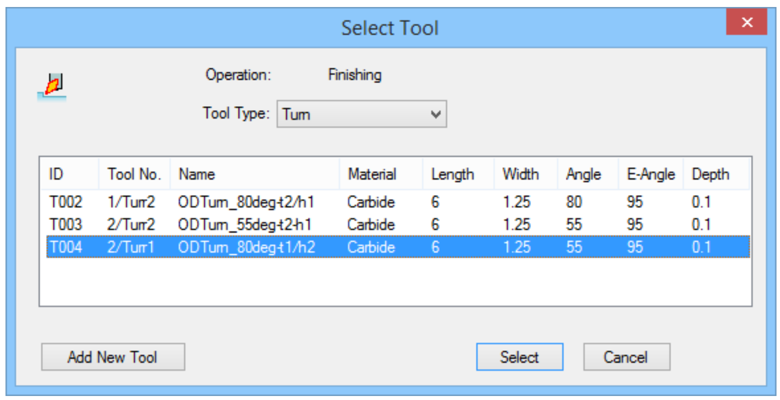
4 Fill out the apropriate information.

5 Choose **Contouring** from the **Strategy** drop down menu.

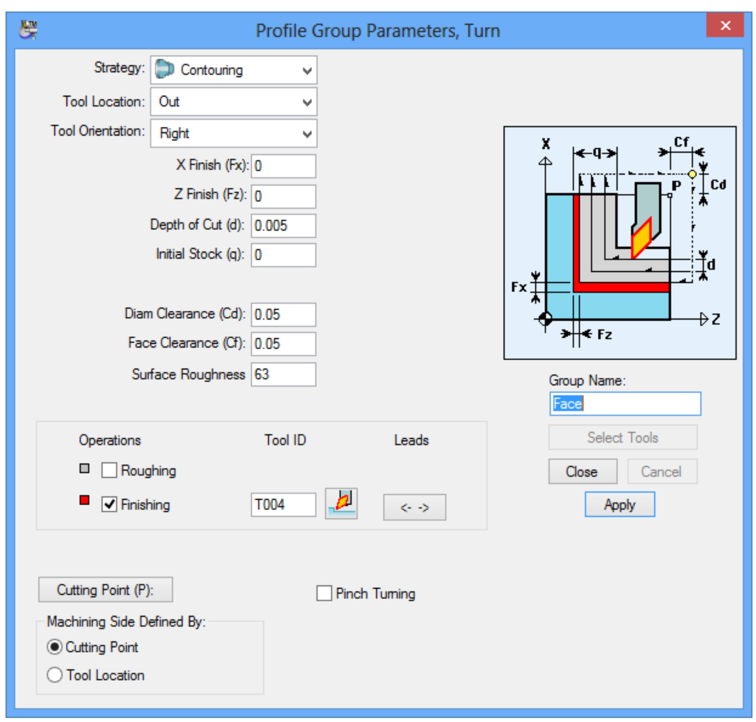
6 Choose **Out** from **Tool Location**.

7 Choose **Right** from **Tool Orientation**

8 Click the **Select Tools** Button. You will be prompted with a choice of three available turning tools. Choose T004 and click **<OK>**.

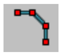


9 In **Group Name** type **Face**. Your completed **Profile Group Parameters** dialog should appear as shown below:



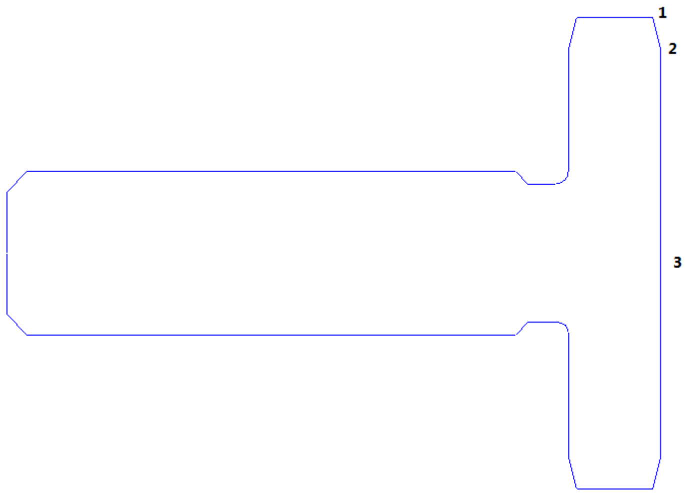
Click the <**Close>** button to close the **Profile Group Parameters** dialog box.

You can now assign the **Facing** cycle to the geometry you created.

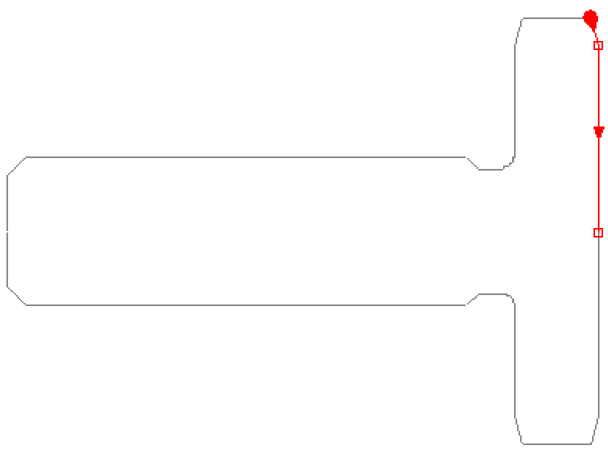


1 From the left-hand side of the screen select the **Define Profile** icon.

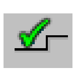
2 From the top of the screen, select the **End of an Element snap mode** icon. To define the turning tool path, click your left mouse on positions to match the diagram below.



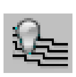
Your part should now appear as shown below:



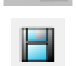
3 Click the **Selection** icon to bring yourself to a home or neutral position



4 To check your work, select **Verify Work Group Tool Path** from the **Part Features** menu, or use the button.



5 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu.

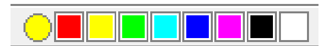


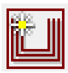
Before moving on, save your work! Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**, or select the icon.

Defining the Profile for Turning the Head on the Sub-Spindle

The first turning operation to be performed on the Main Spindle is to face the stock.

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

2 Choose the yellow square in the **Color Bar.** 



3 Choose **New Profile Group** from the **Part Features** menu or select the icon to display the **Profile Group Parameters** dialog. Fill out the apropriate information.

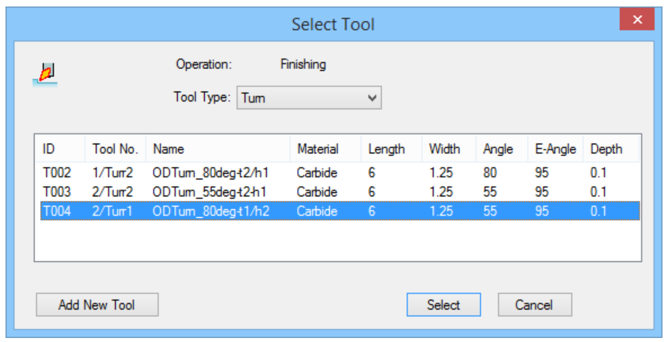
4 Choose **Contouring** from the **Strategy** drop down menu.

5 Choose **Out** from **Tool Location**.

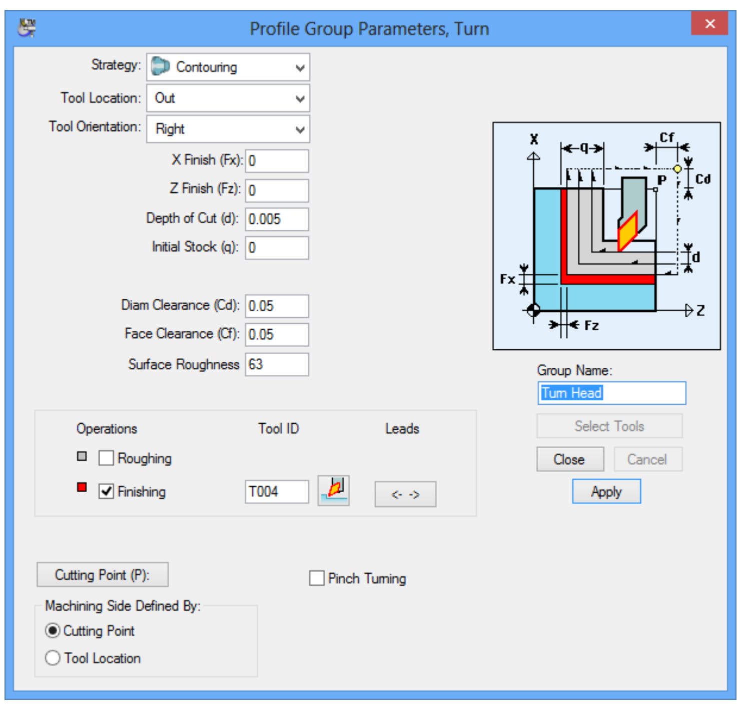
6 Choose **Right** from **Tool Orientation**

7 Click the **Select Tools** Button. You will be prompted with a choice of three available turning tools. Choose T004 and click **<OK>**.

8 Uncheck the Roughing Tool ID and check the Finishing Tool ID.

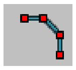


9 In **Group Name** type **Turn Head**. Your completed **Profile Group Parameters** dialog should appear as shown below:



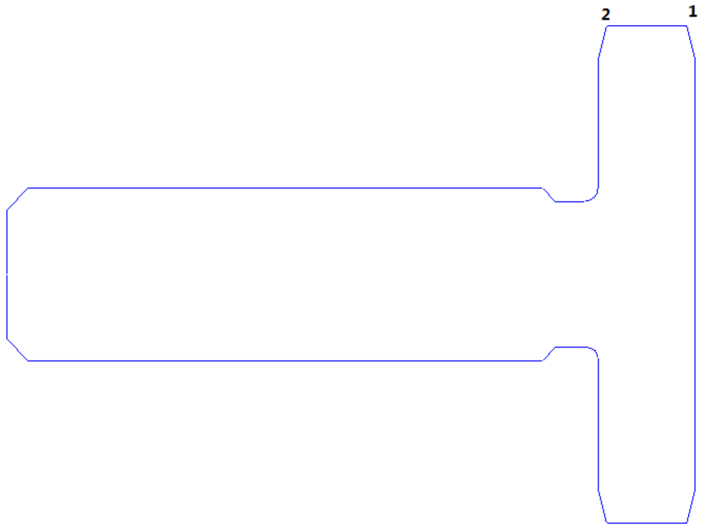
Click the <**Close**> button to close the **Profile Group Parameters** dialog box.

You can now assign the **Turning** cycle to the geometry you created.

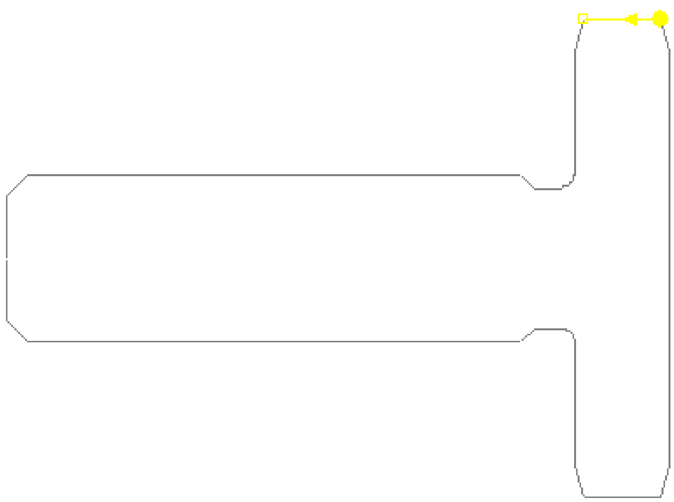


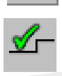
1 From the left-hand side of the screen select the **Define Profile** icon.

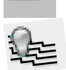
2 From the top of the screen, select the **End of an Element snap mode** icon. To define the turning tool path, click your left mouse on positions to match the diagram below.



Your part should now appear as shown below:



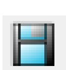




3 Click the **Selection** icon to bring yourself to a home or neutral position

4 To check your work, select **Verify Work Group Tool Path** from the **Part Features,** or select the button.

5 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu, or select the icon.



Before moving on, save your work! Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>.**

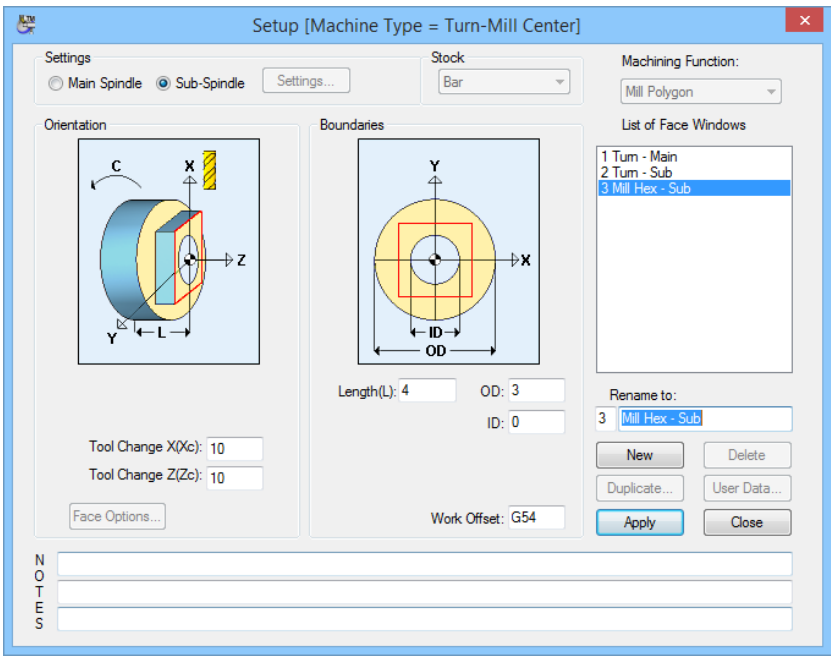
Milling the Hex on the Sub Spindle

This section of the tutorial will instruct you in the steps necessary to perform a hex milling operation on the sub-spindle. The methodology of programming shown here assumes your TurnMill Center is equipped with a Y-axis.

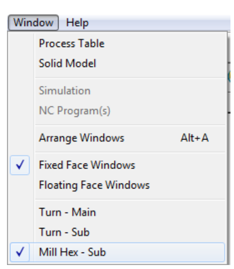
Setting Up the Face Window

1. Choose **Setup** from the **View** menu to view the **Setup** dialog box (or click the **Shortcut to the Setup** icon in the upper right hand corner of the screen).
2. Click the <**New>** button to create a new face window.
3. In the **Rename to:** dialog, enter the name **Mill Hex - Sub**
4. From the Machining Function drop down menu, select **Mill Polygon**
5. Check the **Sub Spindle** radio button to indicate that machining in this window will be carried out on the sub-spindle

   The size and origin location of the stock should remain the same as previously defined in the Turning Setup Window. When you are finished, the Setup dialog should appear as shown below:



6. 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.
7. To switch to the **Mill Hex – Sub** window, choose **Mill Hex – Sub** from the **Window** menu.



Creating Geometry in CAD Mode

In this section, you will create the geometry for the hex milling operations to be performed on the sub spindle.

To create the geometry in **PartMaker** CAD:

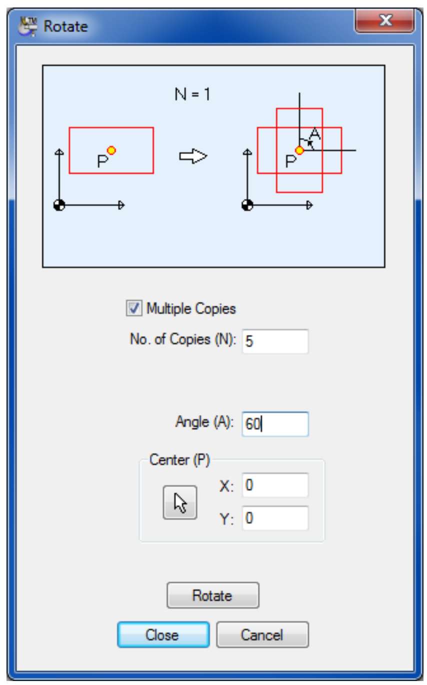
1 Click on the **CAD/CAM** switch in the lower left-hand corner of the Mill-Hex window. The icon will change its appearance from a tool bit to a pencil.

2 From the left side of the screen, choose the **Line Parallel to the Horizontal axis** icon.

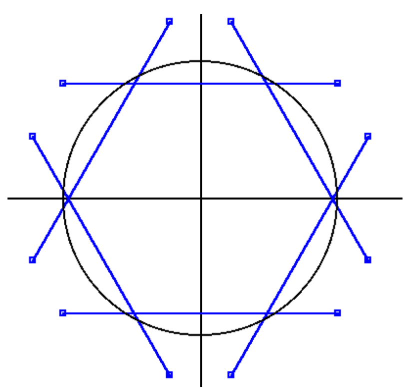
3 Enter a value of 2.5/2 (72/2) at the prompt at the bottom of the screen and press **<Enter>**

4 Choose the selection icon and select the horizontal line drawn on the screen

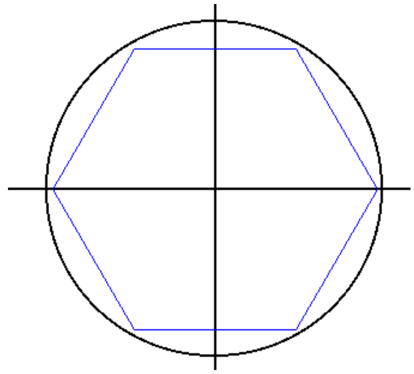
5 From the **Edit > Transform** menu, choose **Rotate**. Complete the rotate dialog as shown below and click **<Rotate>**:



Your window should now appear as shown below:

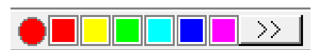


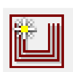
6 Select the **Remove** icon and click on each extra line you wish to remove. Your completed Hex geometry should appear as shown below:



Defining the Profile for Mill Polygon on the Sub-Spindle

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

2 Choose the red square in the Color Bar. 



3 Choose **New Profile Group** from the **Part Features** menu, or select the button icon to display the **Profile Group Parameters** dialog.

4 From **Distance from Part Face** choose **Tool Center (P2)**, positioning the tool in Z relative to the centerline of the flat. Because you are now working in the sub-spindle the front of the hex is now at the Z=0 position.

5 Enter a 0.3125 (6.0) for **P2**.

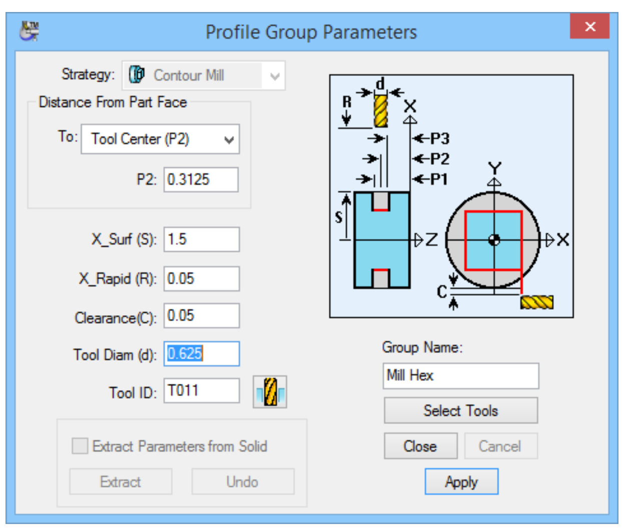
6 Enter 0.625 (10) for **Tool Diameter**.

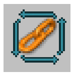
7 Click on the **<Select Tools>** button.

8 Cycle Name Type enter **Mill Hex**

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

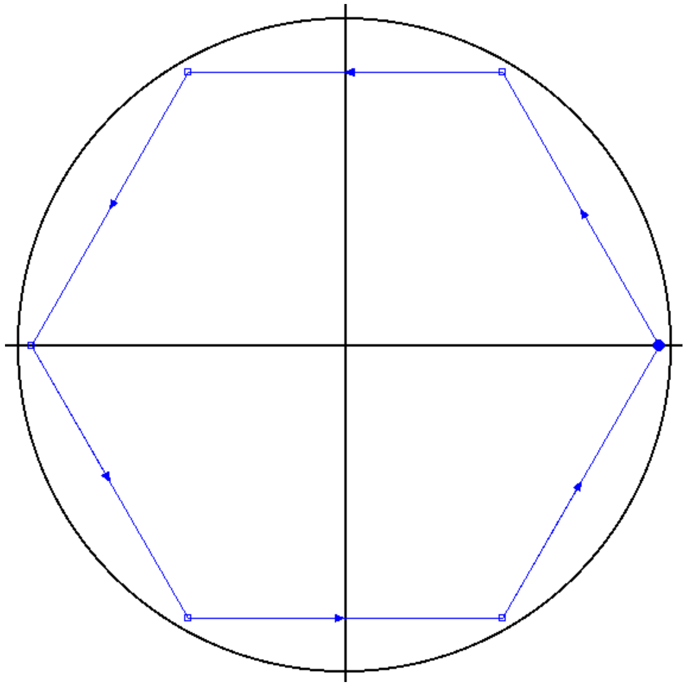
Your profile group parameters should now appear as shown below:

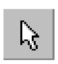




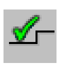
You can now assign the **Milling** cycle to the geometry you created.

1 From the left-hand side of the screen select the **Chain Geometry** icon.
2 To define the milling tool path, click your left mouse on the upper right hand corner of your geometry. Your tool path should match the diagram below.



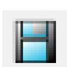


3 Click the **Selection** icon to bring yourself to a home or neutral position.



4 To check your work, select **Verify Work Group Tool Path** from the **Part Features** menu or select the button

5 Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu, or icon.



Before moving on, save your work! Do so by selecting **Save** from the **File** menu or using the shortcut key **<Ctrl + S>**.

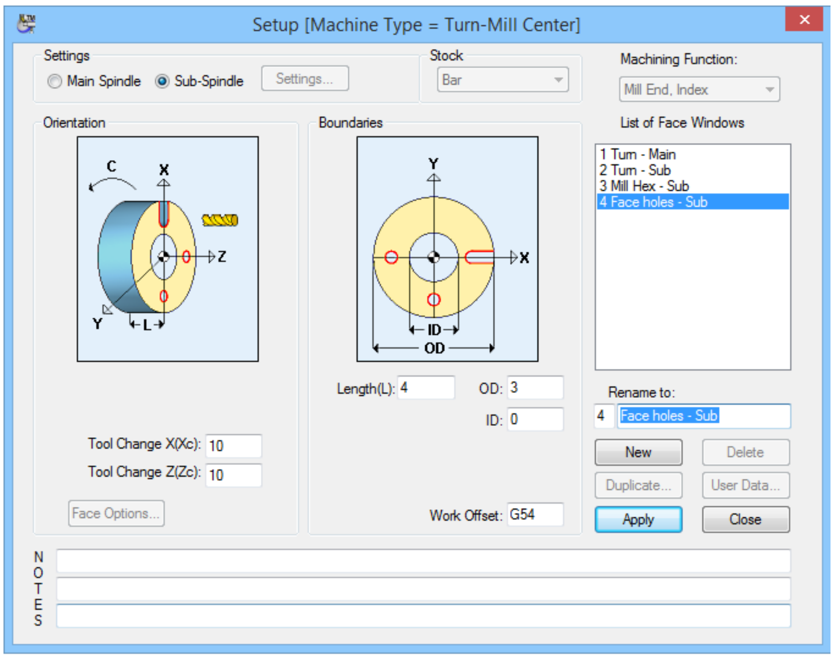
Drilling and Tapping on the Sub-spindle

This section of the tutorial will instruct you in the steps necessary to perform drilling operations on the sub-spindle.

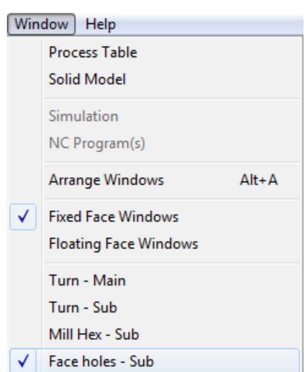
Setting Up the Face Window

In this section, you will set up the **Face Window** needed to perform milling operations on the sub-spindle. To do so:

1. Choose **Setup** from the **View** menu to view the **Setup** dialog box (or click the **Shortcut to the Setup** icon in the upper right hand corner of the screen).
2. Click the <**New>** button to create a new face window.
3. In the **Rename to:** dialog, enter the name **Face Holes - Sub**
4. From the Machining Function drop down menu, select **Mill End, Index**
5. Check the **Sub Spindle** radio button to indicate that machining in this window will be carried out on the sub-spindle



6. 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.
7. To switch to the **Face Holes - Sub**, choose **Face Holes-Sub** from the **Window** menu.



Creating Geometry in CAD Mode

In this section, you will create the geometry for the off center holes to be tapped on the sub spindle. To create the geometry in **PartMaker** CAD:

1 Click on the **CAD/CAM** switch in the lower left-hand corner of the Face Hole-sub window. The icon will change its appearance from a tool bit to a pencil.

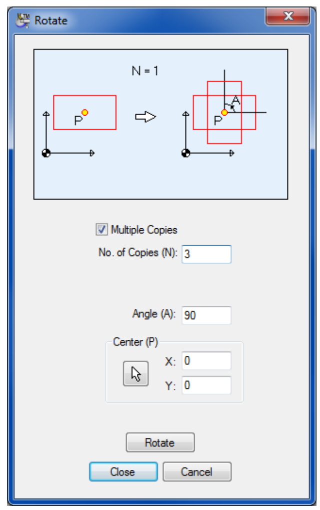
2 From the left side of the screen, choose the **Circle with a Known Radius and Center** icon.

3 Enter a value of .25/2 (3) at the prompt at the bottom of the screen and press **<Enter>**

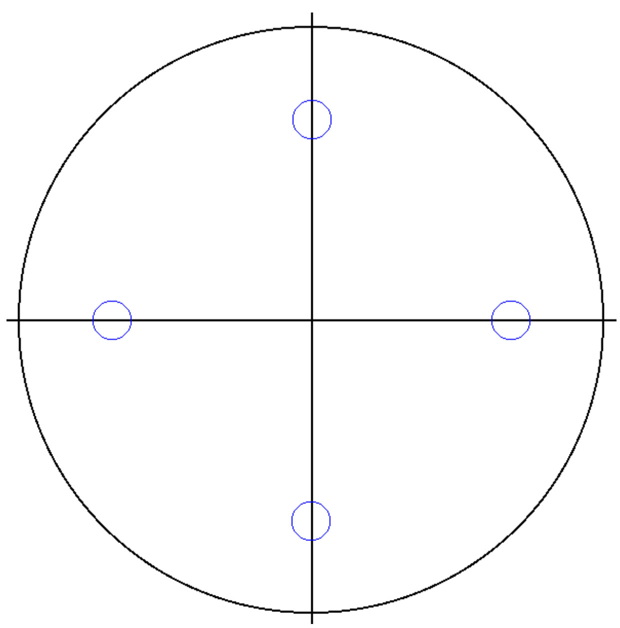
4 Choose the **XY Coordinates** Snap Mode and enter a X value of 2.05/2 (25) and Y value of 0 and press **<Enter>**

5 Choose the selection icon and select the circle drawn on the screen

6 From the **Edit > Transform** menu, choose **Rotate**. Complete the rotate dialog as shown below and click **<Rotate>**:



Your window should now appear as shown below with **Show Axes** and **Show Boundries**:



Defining the Tapped Hole on the Sub-Spindle

Here you will define the cycle and size of the tapped hole to be machined on the subspindle. To do so:

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

2 Choose the red square in the Color Bar.

3 Choose **New Hole Group** from the **Part Features** menu to display the **Hole Group Parameters** dialog, or select the icon.

4 Uncheck the Through Hole box.

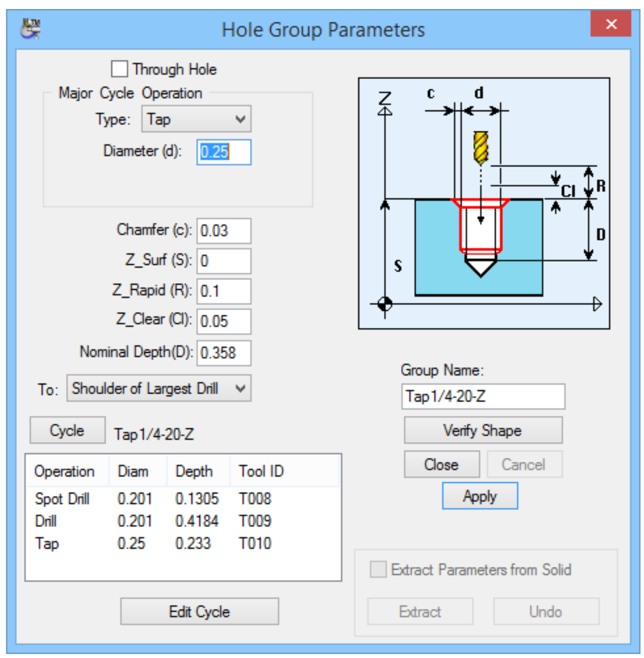
5 In the Major Cycle Operation drop down menu select **Tap**.

6 Enter a **Diameter (d)** of .25 (6)

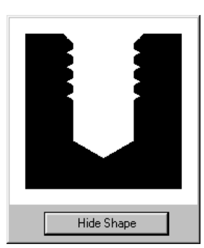
7 Other parameters should be set as follows:

Chamfer (c):	0.03	(1.0)
Z-Surf (S):	0.0	
Z-Rapid (R):	0.1	(1)
Z- Clear (C):	0.05	(1)
Nominal Depth (D):	0.358	(8.0)

Click the **Cycle** button and **PartMaker** will select the appropriate cycle to create the desired hole. The selected cycle should be **Tap-1/4-20-Z (M6 x 1 Tap)**.

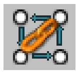


You can verify the shape of the tapped hole by pressing the **Verify Shape** button. If you entered the parameters correctly you should see the following:



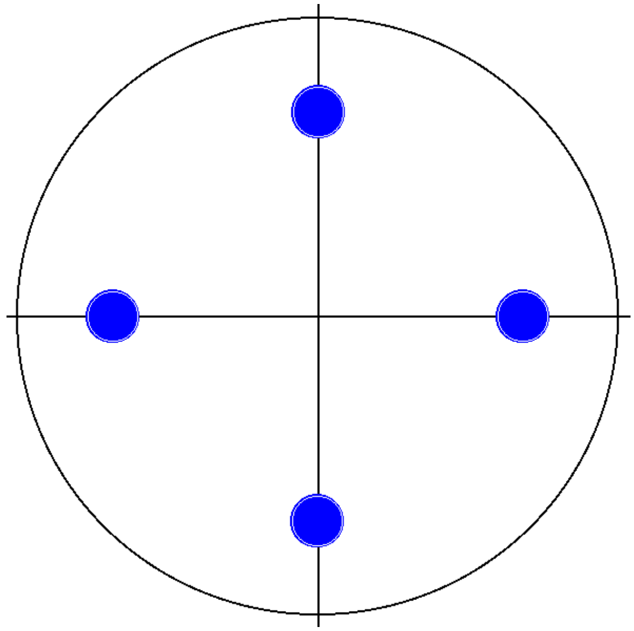
Press the <**Hide Shape**> button and the <**Close**> button to exit the **Hole Group Parameters** dialog.

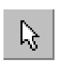
You can now assign the **Drilling** cycle to the geometry you created.



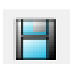
1 From the left-hand side of the screen select the **Chain Circles Icon**.

2 To define the drilling tool path, click your left mouse on any hole on the screen. **PartMaker** will automatically find all holes of the same diameter. Your tool path should match the diagram below.





3 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>** or select the icon.

Creating an NC Program

At this point, you have completed creating the faces and laying out the tool paths for the 8-axis tutorial part. You are now prepared to take the final steps in developing your part program.

The steps required for process plan development and program creation are described using the “generic” 8-axis TurnMill post processor that comes standard with your **PartMaker** TurnMill software. Synchronization and other processing related issues may vary slightly for different machine makers, but the steps below are instructive to work through in developing an understanding of process development in **PartMaker** TurnMill.

Tip: *Refer to the Machine Specific Addendum that came with your* ***PartMaker*** *software for more information on process planning and post processing issues for the TurnMills in your shop.*

Generating a Process Table

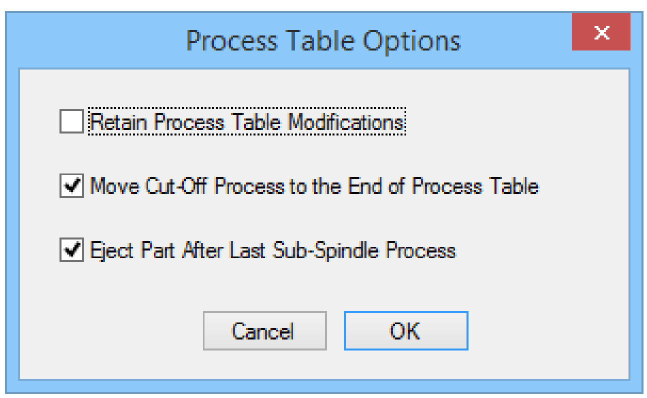
The first step in generating an NC program is creating a Process Table. A **Process Table** shows you all of the machining processes for a part.

When **PartMaker** generates a Process Table, all cutting conditions such as feed rate and spindle speed (RPM) are calculated automatically based on the tools and material information previously entered. A time for each operation is shown, along with a total machining time for both the main and sub spindles. These time calculations give the user a sense of how closely “balanced” machining is for a part, i.e. how much cycle time is expended on both main and sub spindles in addition to the total time to cut the part. Time calculations in **PartMaker** include both “in-cut” time as well as tool change time.

To create the Process Table:

4 Choose **Generate Process Table** from the **Job Optimizer** menu, or select the icon.

Upon doing so, you will be prompted with the dialog below:

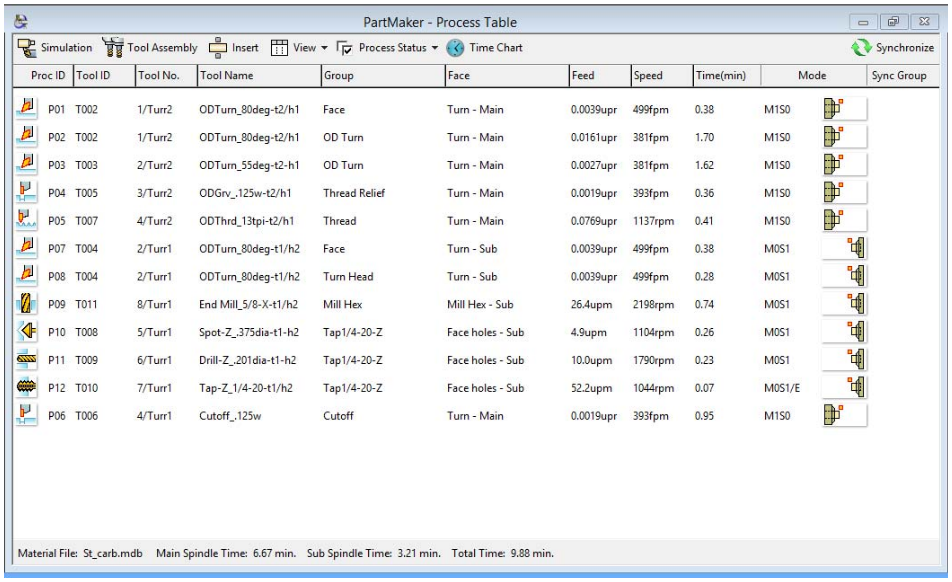


Retain Process Parameters Modifications. Checking this box will retain previous modifications to the process table. Since this is your first time Generating the Process Table it will have no effect.

Move Cut-Off Process to End of Process Table. Checking this box will automatically place your cut-off process at the end of the Process Table.

Eject Part After Last Sub-Spindle Process. Checking this box will automatically place your eject after the last Sub-Spindle process.

The following Process Table will appear:



Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h1	Face	Turn - Main	0.0039upr	499fpm	0.38	M1S0	
P02	T002	1/Turr2	ODTurn_80deg-t2/h1	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1S0	
P03	T003	2/Turr2	ODTurn_55deg-t2-h1	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M1S0	
P04	T005	3/Turr2	ODGrv_.125w-t2/h1	Thread Relief	Turn - Main	0.0019upr	393fpm	0.36	M1S0	
P05	T007	4/Turr2	ODThrd_13tpi-t2/h1	Thread	Turn - Main	0.0769upr	1137rpm	0.41	M1S0	
P07	T004	2/Turr1	ODTurn_80deg-t1/h2	Face	Turn - Sub	0.0039upr	499fpm	0.38	M0S1	
P08	T004	2/Turr1	ODTurn_80deg-t1/h2	Turn Head	Turn - Sub	0.0039upr	499fpm	0.28	M0S1	
P09	T011	8/Turr1	End Mill_5/8-X-t1/h2	Mill Hex	Mill Hex - Sub	26.4upm	2198rpm	0.74	M0S1	
P10	T008	5/Turr1	Spot-Z_.375dia-t1-h2	Tap1/4-20-Z	Face holes - Sub	4.9upm	1104rpm	0.26	M0S1	
P11	T009	6/Turr1	Drill-Z_.201dia-t1-h2	Tap1/4-20-Z	Face holes - Sub	10.0upm	1790rpm	0.23	M0S1	
P12	T010	7/Turr1	Tap-Z_1/4-20-t1/h2	Tap1/4-20-Z	Face holes - Sub	52.2upm	1044rpm	0.07	M0S1/E	
P06	T006	4/Turr1	Cutoff_.125w	Cutoff	Turn - Main	0.0019upr	393fpm	0.95	M1S0	

Visual Process Synchronization and PartMaker Specific Modes

When generating a Process Table, **PartMaker** will default to machining your part in the order in which you have programmed part features. **PartMaker** starts by choosing the processes in the first **Face Window** you have created in the **Setup** dialog and executing all the part features in that **Face Window** in the order they have been created. The software then moves onto the next **Face Window** you have created and executes the processes defined in that **Face Window**.

Of course, in TurnMill, to conserve cycle time, many processes may be happening simultaneously, rather than in the sequential order in which you have created your tool paths in **PartMaker**. The Process Table accommodates this unique feature of TurnMills allowing you define the true order of processes defined in different **Face Windows**. It is on the Process Table that you synchronize operations being performed simultaneously with either different tool posts (i.e. turret 1, turret 2, backworking etc) or on different spindles (main vs. sub) or some combination of the two. Such synchronization is done by pointing and clicking to place the order of processes and then by assigning different Synchronization Modes in the **Set Modes** dialog to link different processes to one another.

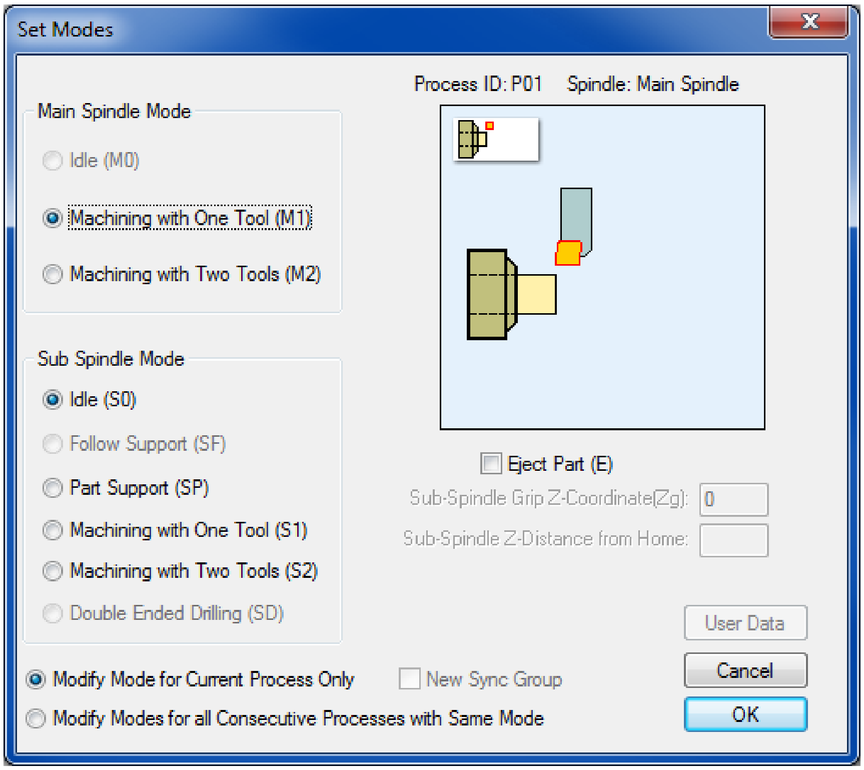
More information on the **Set Modes** dialog can be found in Chapter 2, **PartMaker** Fundamentals. In addition, more information on synchronization techniques can be found in the **Machine Specific** addendum that came with your **PartMaker** software.

Synchronizing Processes on the Main Spindle

To synchronize the Main Spindle operations on the Process Table for the **TurnMill**:

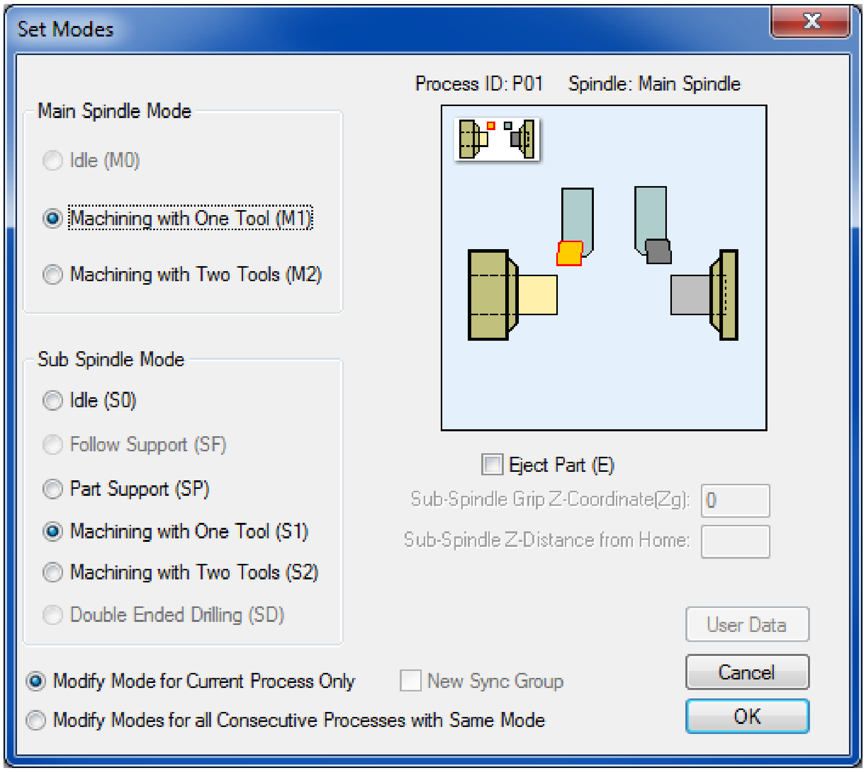
1. Click on the **Set Modes** icon to the right of Process 1 as shown below:

2. After doing so, you will see the **Set Modes** dialog appear as shown below:



The **Set Modes** dialog allows you to visually determine the type of synchronization you want to perform.

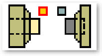
In this case, you will be synchronizing a number of Main Spindle (Turret 1) operations with all of the Sub Spindle (Turret 2) operations. To indicate process 1 will be running simultaneously as a process on the Sub Spindle, click the **Machining with One Tool (S1)** radio button in the **Sub-Spindle Mode** area of the **Set Modes** dialog. After having done so, the **Set Modes** dialog should appear as shown below:



Once you are satisfied your dialog appears as the one above, click the **<OK>** button to return to the Process Table.

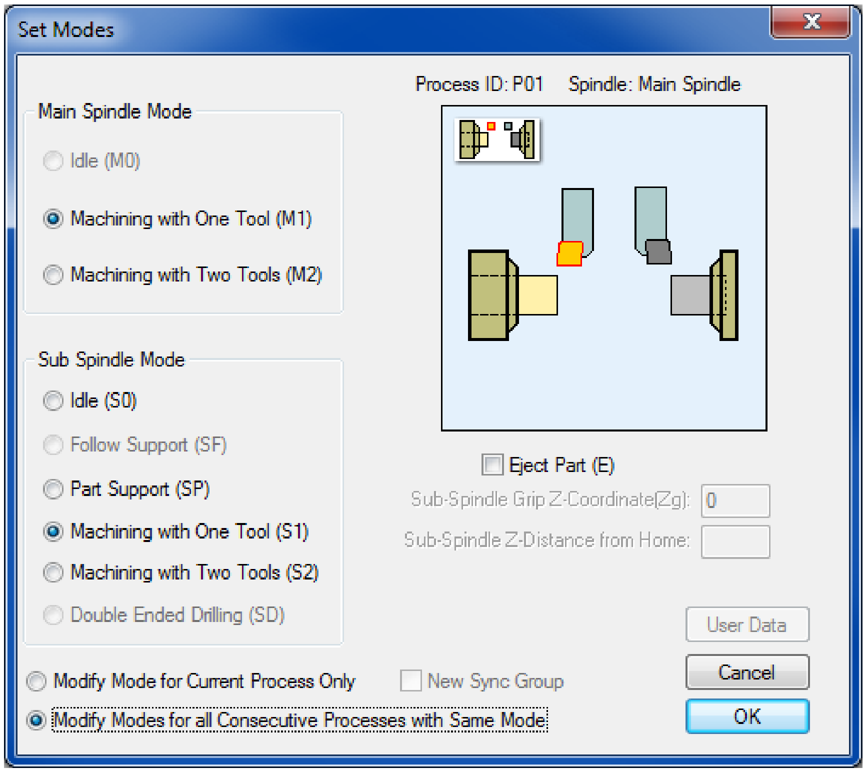
Notice how the **Set Modes** icon now depicts two tools cutting, one on the Main Spindle and one on the Sub Spindle. The Mode column is now filled in with M1S1, indicating that during this process one tool is cutting on the main spindle and one is cutting on the sub spindle at the same time.

M1S1

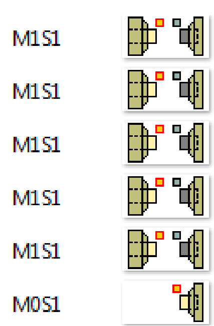


3 In order to quickly set the remaining Main Spindle Processes to **M1S1** (i.e. simultaneous cutting on Main and Sub Spindle) click the **Set Modes** dialog associated with process 2. Again, click the **Machining with One Tool (S1)** radio button in the **Sub-Spindle Mode** area of the **Set Modes** dialog.

4 Next, click the **Modify Modes for all Consecutive Processes with Same Mode** radio button. After having done so, the **Set Modes** dialog should appear as shown below:



5 Click **<OK>** to return to the Process Table. The **Set Modes** icons for all Main Spindle Processes should be automatically assigned and appear as shown below:



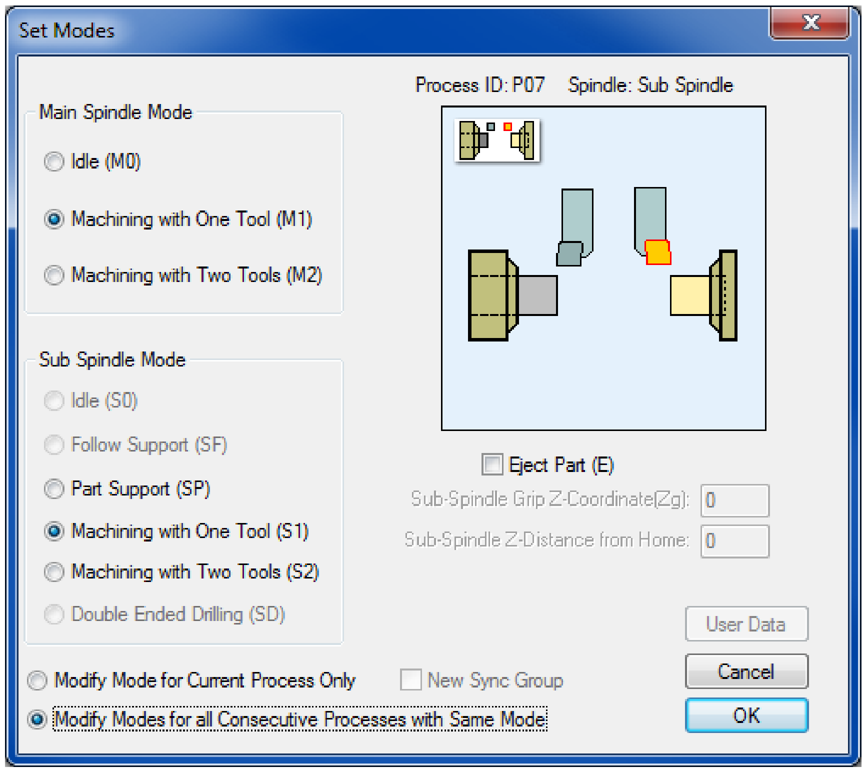
Synchronizing Processes on the Sub Spindle

To synchronize the Sub Spindle operations with the Main Spindle operations discussed above on the Process Table for the **8-axis_demo.job**:

1. Click on Process 7, the first operation programmed on the Sub Spindle. With Process 7 selected click the **Set Modes** icon.

2. In the Set Modes dialog, click the Machining with One Tool (M1) radio button in the Main Spindle Mode area of the Set Modes dialog. To save time, click the Modify Modes for all Consecutive Processes with Same Mode radio button.

   After having done so, the **Set Modes** dialog should appear as shown below:



3. Click the **<OK>** button to return to the Process Table.

Setting the Cut-Off Operation

Having now properly Synchronized the Main Spindle and Sub-Spindle operations, you will now set-up the cut-off operation. In TurnMill, cut-off is typically programmed as the last operation when cutting a part that involves synchronous machining on both spindles.

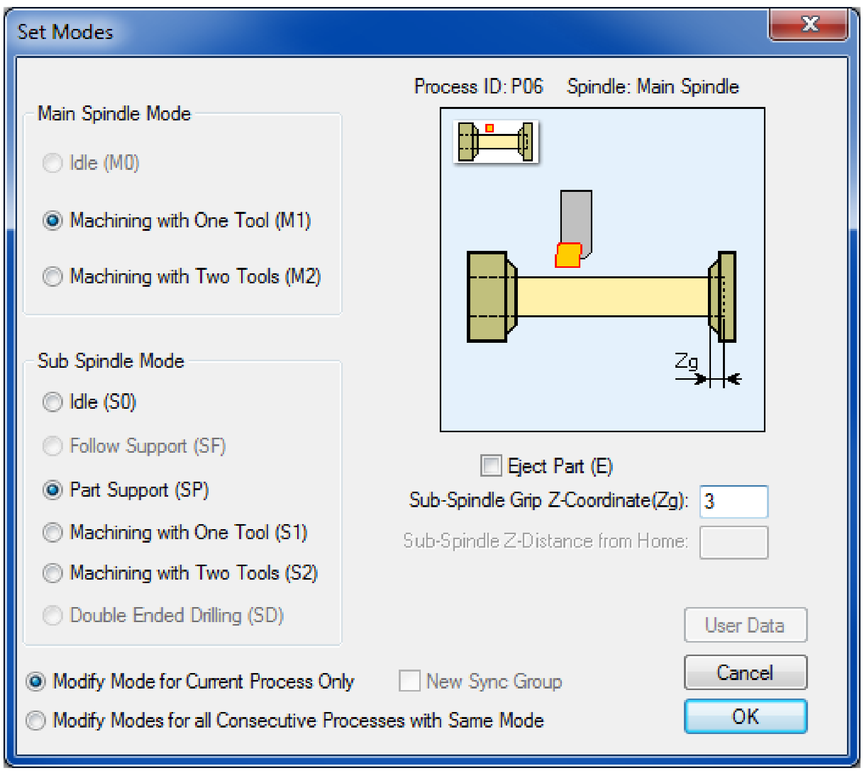
To properly set-up the cut-off and transfer from the main spindle to the sub spindle:

1 Click on the **Set Modes** icon associated with process 6 (the Cut-off operation) to enter the **Set Modes** dialog. The proper mode for the Cut-Off operation is **Cutting with One Tool (M1)** on the Main Spindle while providing **Part Support (SP)** with the sub-spindle, meaning the sub-spindle is supporting while the part is being cut.

To set this mode of operation, check the **Part Support (SP)** radio button in the **Sub-Spindle Mode** area of the **Set Modes** dialog.

When setting this mode, enter a value of 3.0 (80) for the **Sub-Spindle Grip Z-Coordinate (Zg)**, the position at which the sub-spindle is gripping the part during transfer.

After doing so your **Set Modes** dialog should appear as shown below:

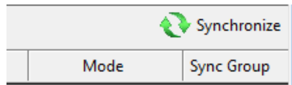


2 Click the **<OK>** button to return to the Process Table.

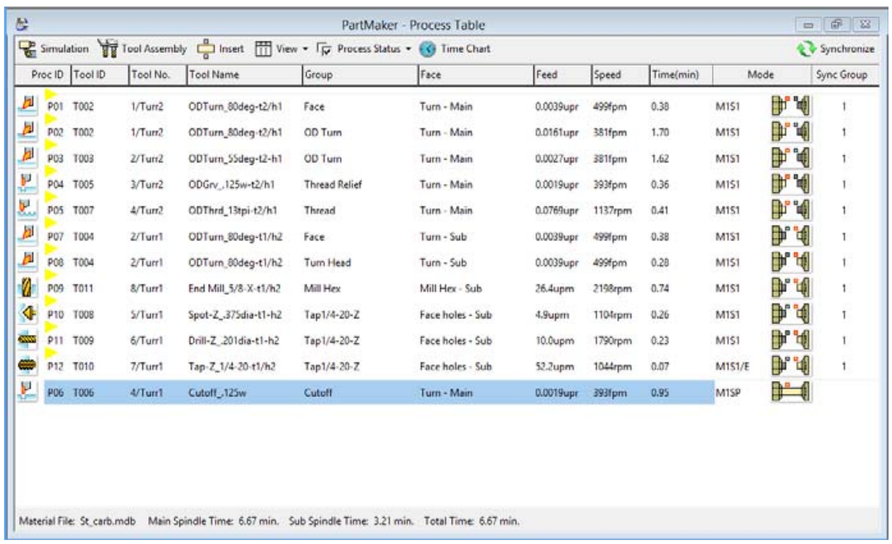
Creating Sync (Synchronization) Groups

With the Process Table generated and the sequence of operations established, you can now verify that your synchronization approach is correct by creating Sync Groups, groups of processes being executed simultaneously. To do so:

1 Click the **Synchronize** button in the upper right hand corner of the Process Table.



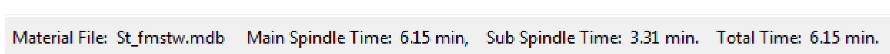
2 If you have made any errors in process synchronization, **PartMaker** will prompt you at this time. If you have not made any errors, your Process Table should appear as shown below:



Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T002	1/Turr2	ODTurn_80deg-t2/h1	Face	Turn - Main	0.0039upr	499fpm	0.38	M1S1	1
P02	T002	1/Turr2	ODTurn_80deg-t2/h1	OD Turn	Turn - Main	0.0161upr	381fpm	1.70	M1S1	1
P03	T003	2/Turr2	ODTurn_55deg-t2-h1	OD Turn	Turn - Main	0.0027upr	381fpm	1.62	M1S1	1
P04	T005	3/Turr2	ODGrv_.125w-t2/h1	Thread Relief	Turn - Main	0.0019upr	393fpm	0.36	M1S1	1
P05	T007	4/Turr2	ODThrd_13tpi-t2/h1	Thread	Turn - Main	0.0769upr	1137rpm	0.41	M1S1	1
P07	T004	2/Turr1	ODTurn_80deg-t1/h2	Face	Turn - Sub	0.0039upr	499fpm	0.38	M1S1	1
P08	T004	2/Turr1	ODTurn_80deg-t1/h2	Turn Head	Turn - Sub	0.0039upr	499fpm	0.28	M1S1	1
P09	T011	8/Turr1	End Mill_5/8-X-t1/h2	Mill Hex	Mill Hex - Sub	26.4upm	2198rpm	0.74	M1S1	1
P10	T008	5/Turr1	Spot-Z_.375dia-t1-h2	Tap1/4-20-Z	Face holes - Sub	4.9upm	1104rpm	0.26	M1S1	1
P11	T009	6/Turr1	Drill-Z_.201dia-t1-h2	Tap1/4-20-Z	Face holes - Sub	10.0upm	1790rpm	0.23	M1S1	1
P12	T010	7/Turr1	Tap-Z_1/4-20-t1/h2	Tap1/4-20-Z	Face holes - Sub	52.2upm	1044rpm	0.07	M1S1/E	1
P06	T006	4/Turr1	Cutoff_.125w	Cutoff	Turn - Main	0.0019upr	393fpm	0.95	M1SP	

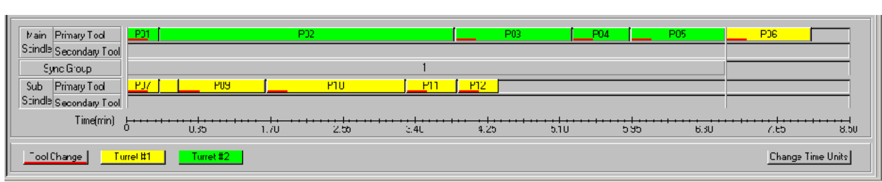
On the Process Table above, processes 1 through 12 are all being executed at the same time (all be it sequentially) and as a result are in the same **Sync Group**. Process 6 is happening independently and thus is not included in a **Sync Group**.

Notice, once **Sync Groups** have been created, **PartMaker** displays a "balanced" cycle time for the entire operation, meaning the **Total Time** displayed is not necessarily a sum of the **Main spindle time** and the **Sub spindle time**, but actually takes the overlapping or synchronized operations into account.



You can also visually verify the synchronization of the **Main Spindle** and **Sub-Spindle**. To do so:

1 Click the **Show Time Chart** button in the lower right hand corner of the Process Table to see a Graphic representation of your Process. You may also select the clock button.



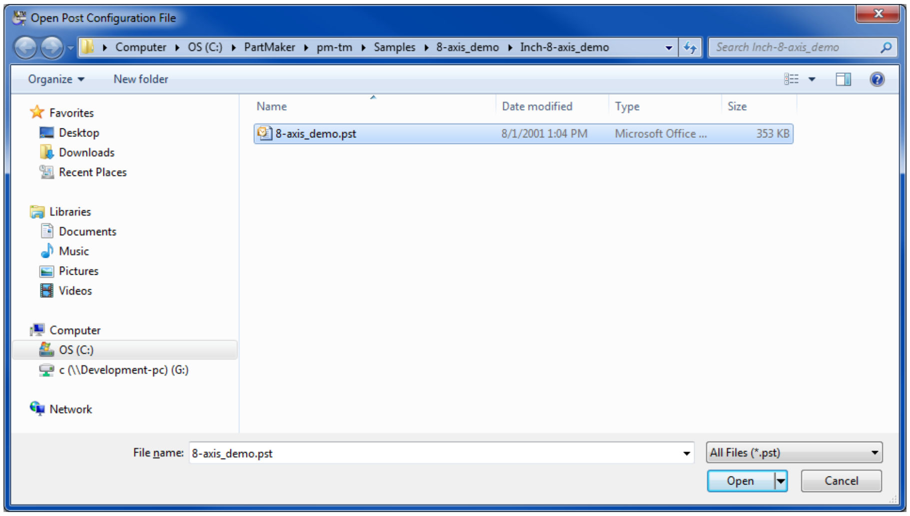
2 Click the **Hide Time Chart** button in the lower right hand corner of the Process Table to hide the time histogram, or click on the clock button again.

Selecting a Post Processor

Having generated a Process Table you must now select a Post Configuration File before you can output an NC progrm. This post configuration file is stored in the pm-tm root directory. If you do not have any other post configuration files loaded in this directory the **8-axis_demo.pst** will be loaded automatically. If you do have other post configuration files in this directory:

1 Choose Post Config File = ? from the Job Optimizer menu.

2 Select the **8-axis_demo.pst** post as shown below. PartMaker for TurnMill comes standard with a post configuration for a Fanuc 18t control which supports, live tooling, c-axis and subspindle operations.

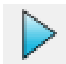


Click **<OK>** to continue.

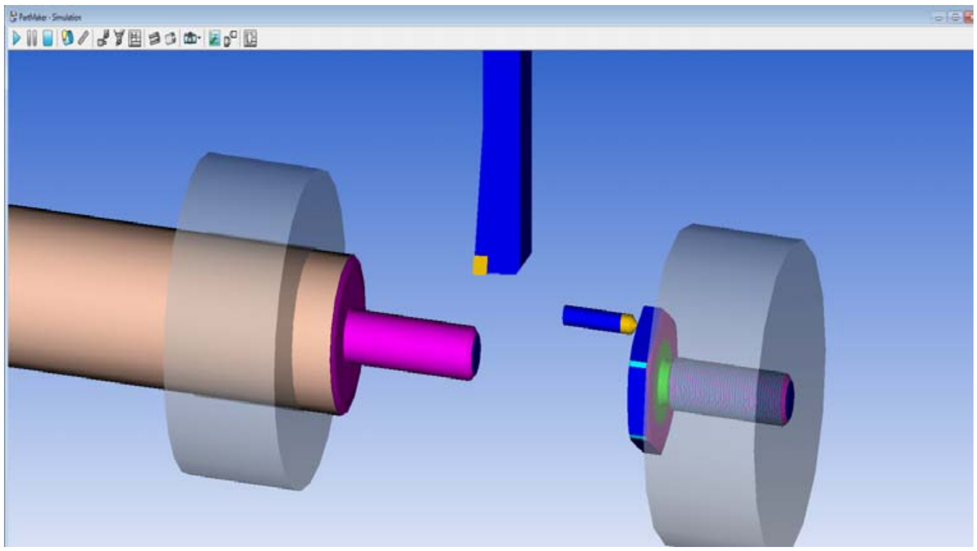
Simulating the Process

Having completed generating the Process Table and synchronizing operations, you can now simulate the processes. The simulation will show your part being cut just how it would on your machine tool. Simulation will show synchronization between your Main and Sub-Spindles. It will also show any collisions that may occur between the tooling, material, and/or Spindles.

1 Click the <**Simulation**> button at the bottom left hand portion of the Process Table, or select the **Simulation** button:



2 Press the Space Bar or choose **Simulation** from the **Simulation** menu to begin the simulation. Alternately, select the icon.



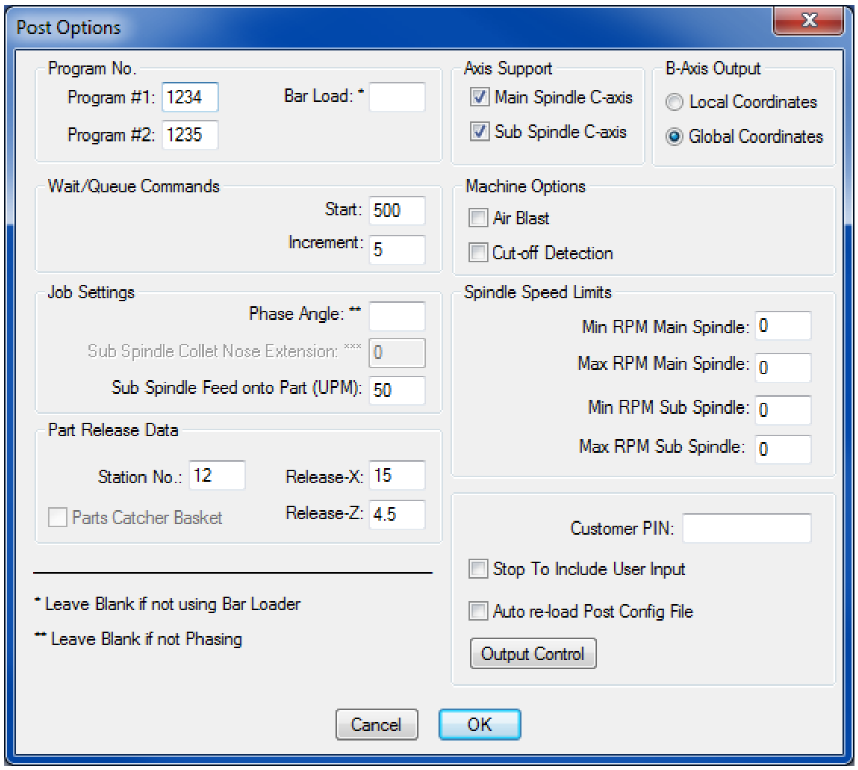
Generating an NC Program

Having selected a Post Configuration file, you are now ready to generate an **NC Program.** To do so:

1 Choose **Generate NC Program** from the Job Optimizer menu.

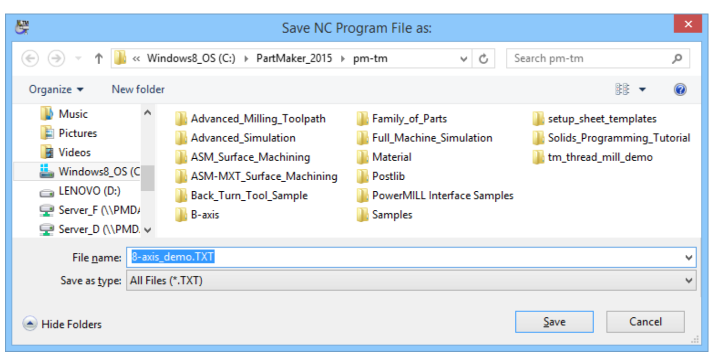
> ***Note:*** *Choosing the* ***Generate NC Program*** *command from the* ***Job Optimizer*** *menu has the same effect as clicking the* ***Sync Group*** *button on the Process Table if* ***Sync Groups*** *have not yet been created.*

2 You will next see the **Post Options** dialog. Set the **Post Options** dialog as shown below:

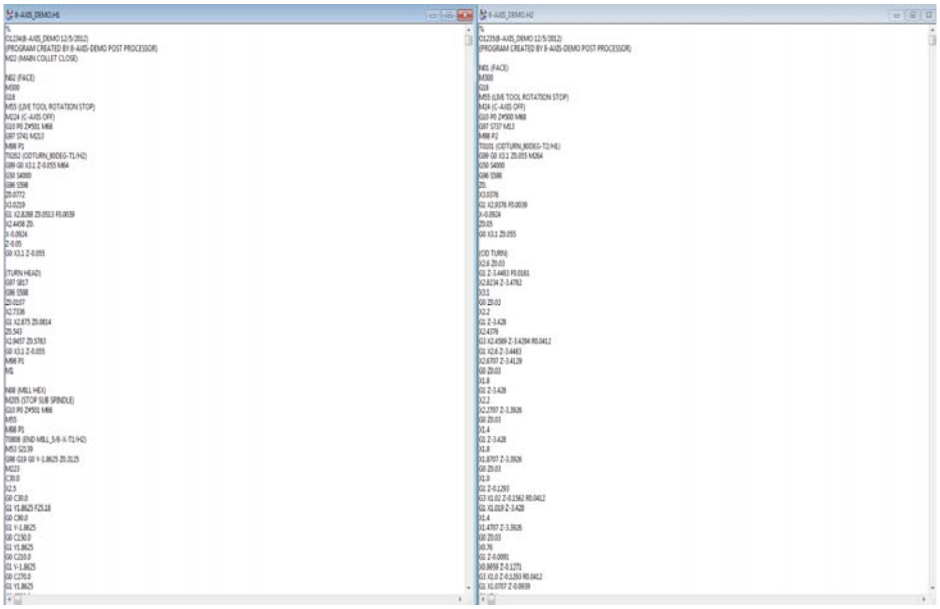


In the **Post Options** dialog you set certain parameters for assignment during post processing. Please refer to your machine specific addendum for more information on how each of the fields in this dialog pertain to your TurnMill equipment.

3 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog. In this dialog, enter a name for your program. Here you should enter **"tutorial"**.



4 Click **<Save>** to generate the NC program. The **NC Programs** will now be displayed.



Chapter 5: 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 three levels: Basic Simulation, Advanced Simulation and Full Machine Simulation. Basic Simulation comes standard with PartMaker Turn-Mill while both Advanced and Full Machine Simulation are available as optional modules with PartMaker Turn-Mill.

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

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

- View a realistic 3D model of your machine and its tool holders and various components based on actual solid models versus just parametric descriptions
- Import solid models for all simulated entities
- Add components to these entities (i.e. turret slides) to model the kinematics of the machine.
- Add tool holders to the simulation and mount multiple tools per holder.
- Interactively specify how the machine is assembled.

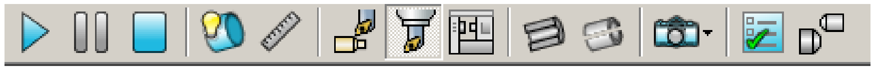
Users who have licensed the Advanced Simulation or Full Machine Simulation modules may still find it beneficial to use the Basic Simulation module to focus on only the part as it is being machined.

You may choose between Basic, Advanced, of Full Machine Simulation by choosing **Basic Simulation**, **Advanced Simulation,** or **Full Machine 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.

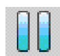
The Simulation Toolbar

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

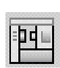
 Resets the simulation

 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

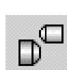
 Sets simulation to operate in Full Machine Simulation mode

 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).

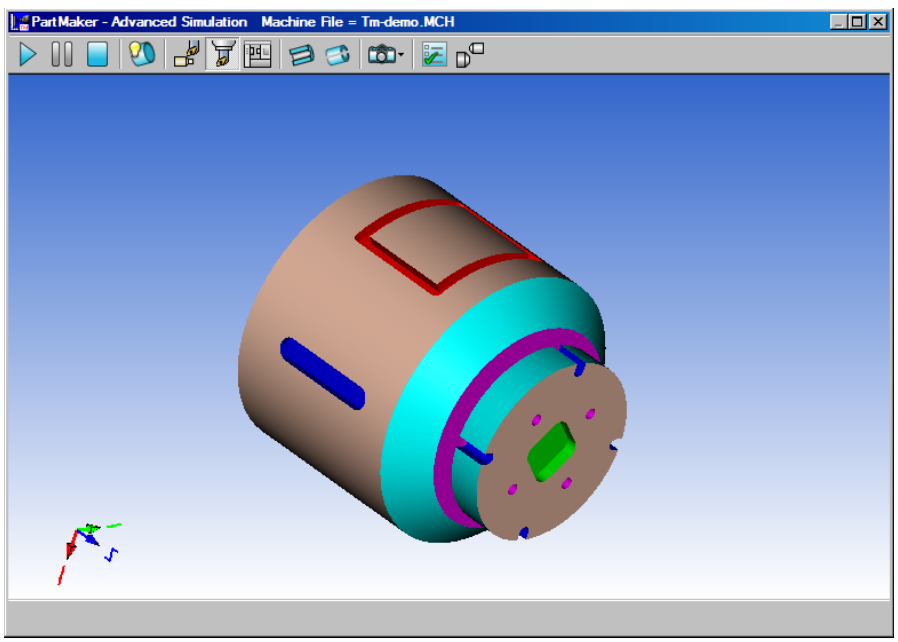
Show Travel Limits

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 Date 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:

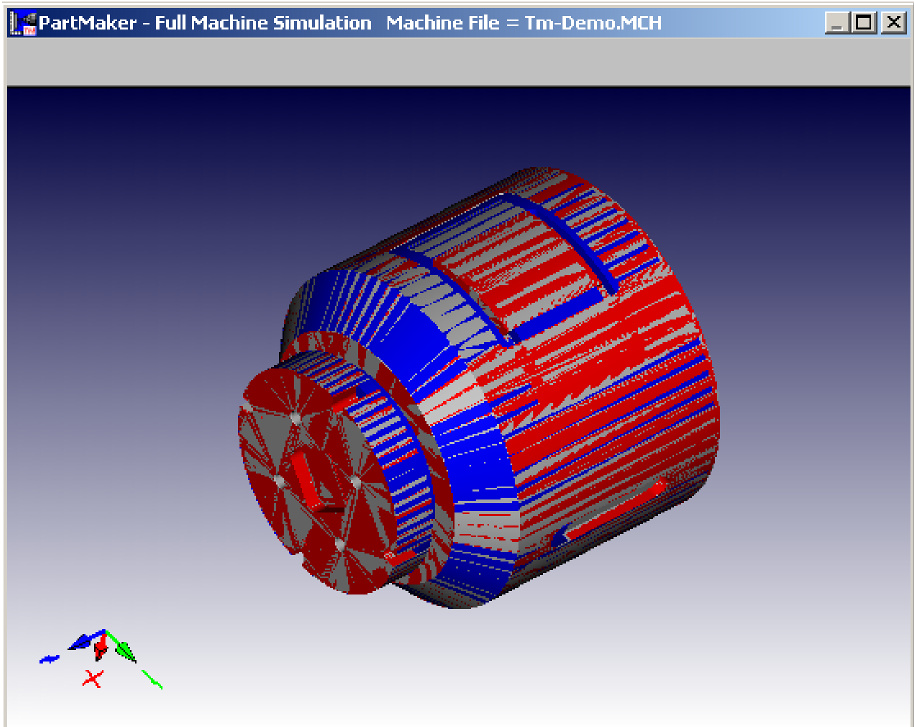


Save Finished Part As STL file

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.

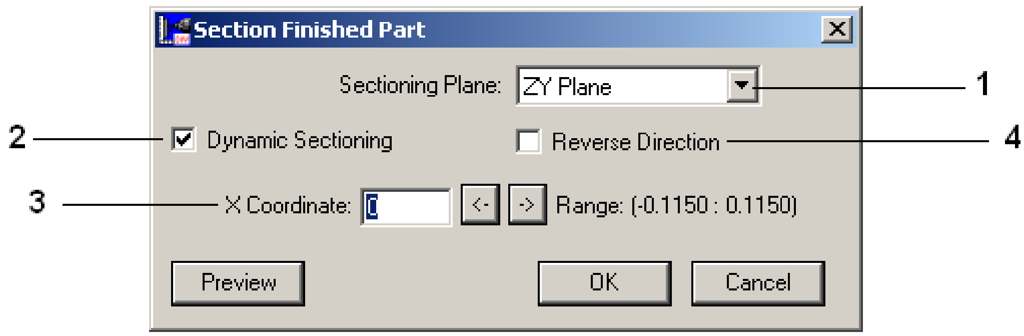
When **Zoom** is active, you can scale the process simulation or finished part by clicking the left hand mouse button and dragging it upward to increase the size of the part and dragging downward to decrease the size of the part.

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 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 use 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.

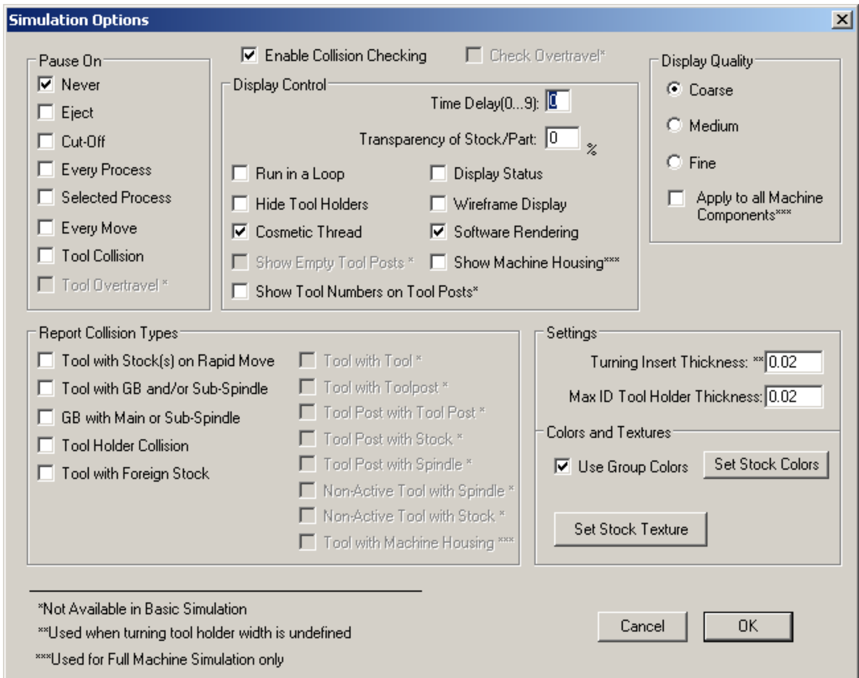
4 **Reverse Direction -** This allows you to reverse the direction of sectional plane shown.

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-Mill 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** checkbox 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 withGB 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 an 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 each tool motion. Negative numbers -9 to -1 may be used to speed simulation more than 0.

Transparency of GB/Sub-Spindle*:* Specifies how transparent the Main Spindle and/or Sub-Spindle appear. 0% means solid appearance, 100% means 'fully transparent' appearance, i.e. Guide Bushing and/or Sub-Spindle will be 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 the current simulation status consisting of the Process ID and 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 stock texture options.

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 upper left hand corner of the Process Table

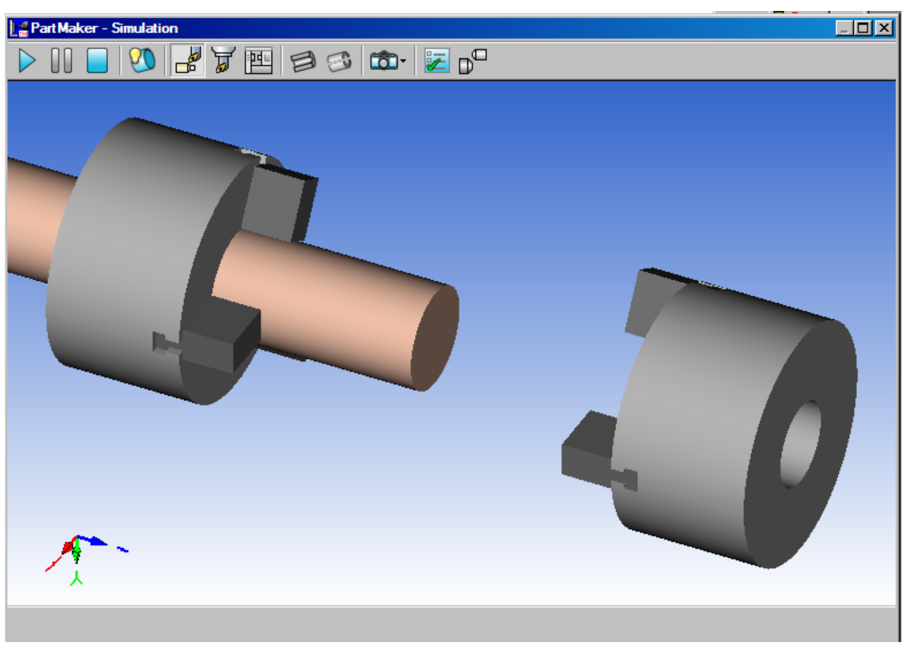
   or

   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, 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:

1 Press the **Start Simulation** button on the Simulation Toolbar

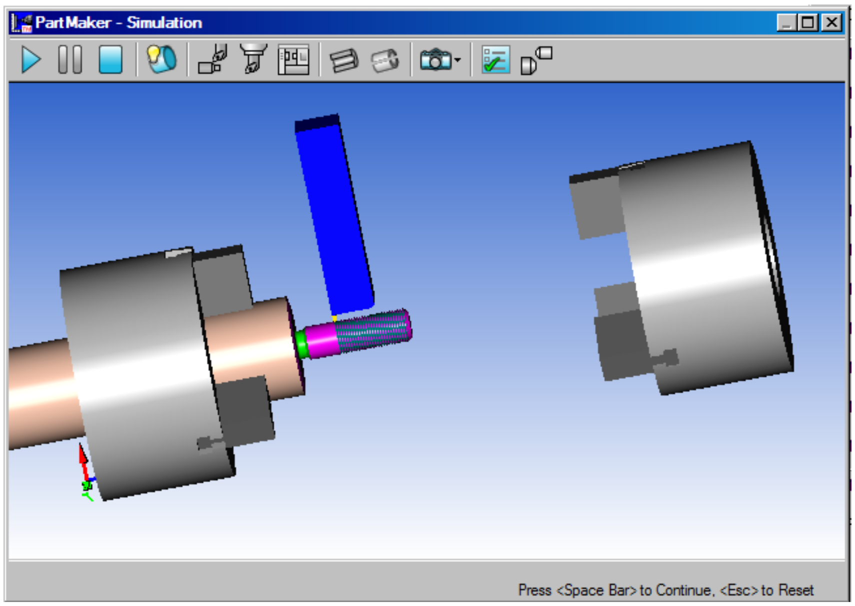
   or

   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.

Irrespective of which Turn-Mill machine you are using, PartMaker installs a number of "generic" machine configurations, which you may view. These machine configurations can be viewed from the C:\PM-TM\Advanced_Simulation directory and consist of the following examples:

Single_Turret_Single_Spindle: A "generic" machine with one turret and a main spindle

Single_Turret_Twin_Spindle: A "generic" machine with one turret and a main and sub spindle

Three_Turret_Twin_Spindle: A "generic" machine with three turrets and a main and sub spindle

Tool_Head_Turret_Twin_Spindle: A "generic" machine with two turrets and a main and sub spindle

Twin_Turret_Twin_Spindle: A "generic" machine with two turrets and a main and sub spindle

None of the examples listed above are meant to represent a specific machine model such as a Hardinge Quest, Haas TL, Mori Seiki ZT, Miyano BNE etc. 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 Twin_Turret_Twin_Spindle.job, Twin_Turret_Twin_Spindle.tdb and Twin_Turret_Twin_Spindle.cdb located in the C:\PM-TM\Advanced_Simulation\ Twin_Turret_Twin_Spindle directory.*

Important! *To launch PartMaker Advanced Simulation you must have an* ****.MCH*** *(machine configuration file) loaded. The MCH file contains a description of the Swiss machine you are using to machine the part. An MCH file will be loaded automatically when a post processor is loaded and the MCH file corresponding to that post processor is loaded in the same directory as the post processor.*

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:

1 Click the **Start Simulation** icon from the Main Toolbar

or

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

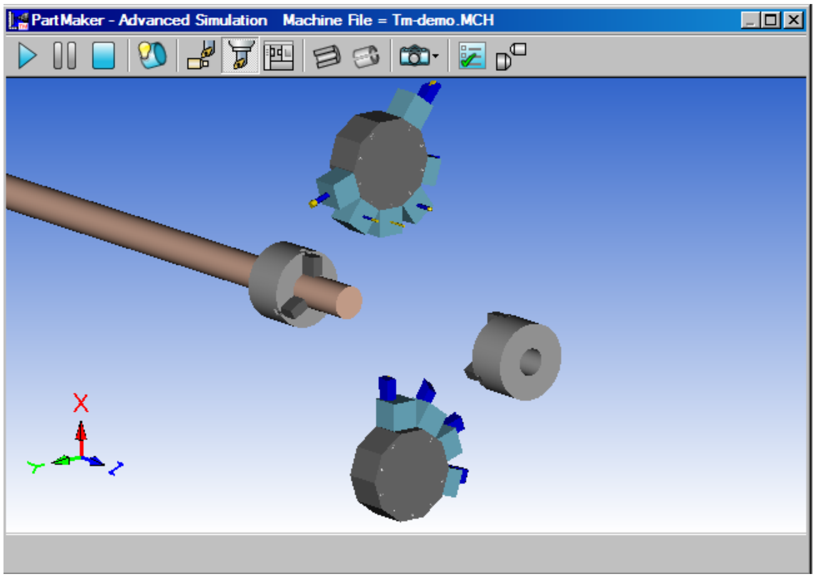
or

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

2 Make sure the **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:

1 Press the Start Simulation button on the Simulation Toolbar

or

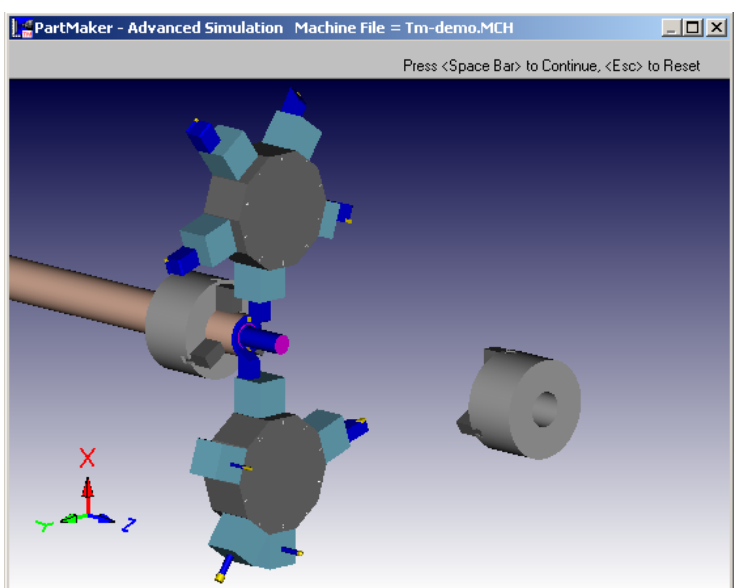
Press the **<Space Bar>** on your keyboard

or

Select Start Simulation from the Simulation Menu

2 Once simulation has started, you can suspend cutting by pressing the **<Esc>** button on your keyboard. Press the **<Space Bar>** to continue the simulation.

Below, is an example of suspending simulation in process:



Using PartMaker Full Machine Simulation

PartMaker Full Machine 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 although it is highly recommended to do so.

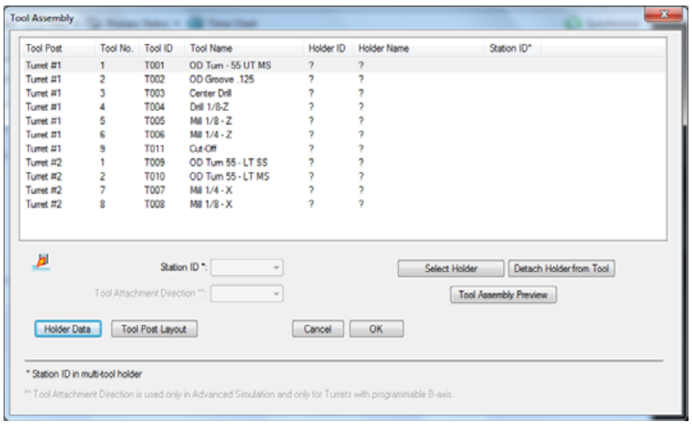
PartMaker installs a “generic” machine configuration, consisting of a Twin Turret Machine for you to view. This machine configuration can be viewed from the C:\PM-TM\Full_Machine_Simulation directory.

Note: *For this tutorial, please load the *.job, *.tdb and *.cdb files called* ***FMS_Turnmill_demo****, from the C:\PM-TM\Full_Machine_Simulation\ directory.*

Important! *To launch PartMaker Full Machine Simulation you must have an* ****.MCH*** *(machine configuration file) loaded. The MCH file contains a description of the Turn-Mill machine you are using to machine the part. This file will be loaded automatically when a post processor is loaded and the MCH file corresponding to that post processor is loaded in the same folder.*

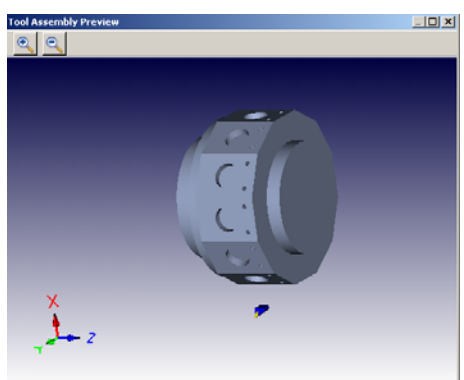
Prior to running Full Machine Simulation, you will attach the holders specified in the Machine Data File to the tools used on the **Process Table**. The tool post and component models will automatically be loaded into simulation.

1. Select **Tool Assembly** from the **Job Optimizer** menu to open the **Tool Assembly** dialog. The tools on the process Table are displayed, sorted by tool post and tool number.



2. Highlight the first tool on list, Tool Number 1 on Turret 1. You will notice that in the **Holder Name** column, there is a “?” which means that a holder is expected to mount to this position.

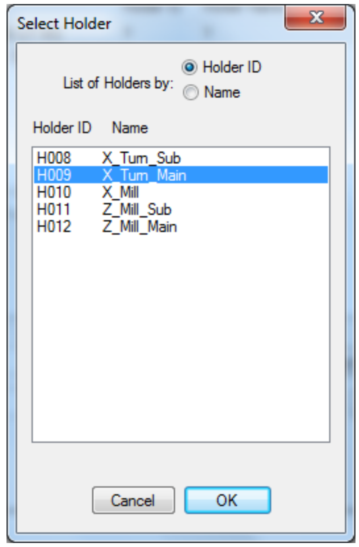
3. Press **<Tool Assembly Preview>** to see how the tool and tool post will look in simulation. Notice that the tool is floating in air away from the Turret.



4 Press the “X” in the top left corner of the window to close the Preview.

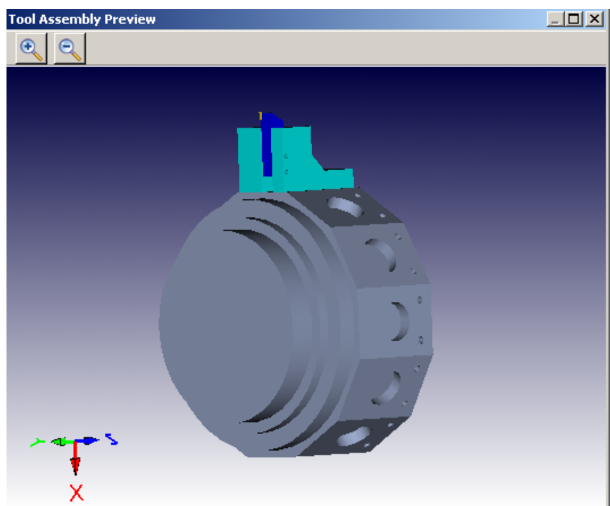
5 Press **<Select Holder>** to open the **Select Holder Dialog** for the highlighted tool.

6 Choose “X_Turn_Main” to specify this holder to be used for simulation.



7 Press <OK> to confirm this holder selection.

8 Press **<Tool Assembly Preview>** to see how the final assembly with the holder will look when attached in simulation. You will see if there are any issues with the holder that you have selected.



9 Press the “X” in the top left corner to close the Preview.

10 Repeat step 2 – step 9 for the remaining tools on Turret 1 and Turret 2. For the following tools, you will use the holders listed bellow:

- **a** Tool 2 on Turret 1: “X_Turn_main”
- **b** Tool 3 on Turret 1: “Z_Mill_Main”
- **c** Tool 4 on Turret 1: “Z_Mill_Main”
- **d** Tool 5 on Turret 1: “Z_Mill_Main”
- **e** Tool 6 on Turret 1: “Z_Mill_Main”
- **f** Tool 9 on Turret 1: “X_Turn_Main”
- **g** Tool 1 on Turret 2: “X_Turn_Sub”

h Tool 2 on Turret 2: “X_Turn_Main”

i Tool 7 on Turret 2: “X_Mill_Sub”

j Tool 8 on Turret 1: “X_Mill_Sub”

11 Press **<OK>** in the **Tool Assembly** dialog to confirm all of your holder selections.

Launching Full Machine Simulation

Once all the holders have been attached to the respective tools on the process table, you can view the Full Machine Simulation. The full machine simulation will include the models of the tool posts and their components, holders for the tools, and custom positioning of the tools and holders on the tool posts.

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

To launch PartMaker Simulation:

1 From the **Simulation** menu, choose **Full Machine Simulation** or right mouse click on the **Process Table** and choose **Full Machine Simulation** from the drop down menu that appears

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

or

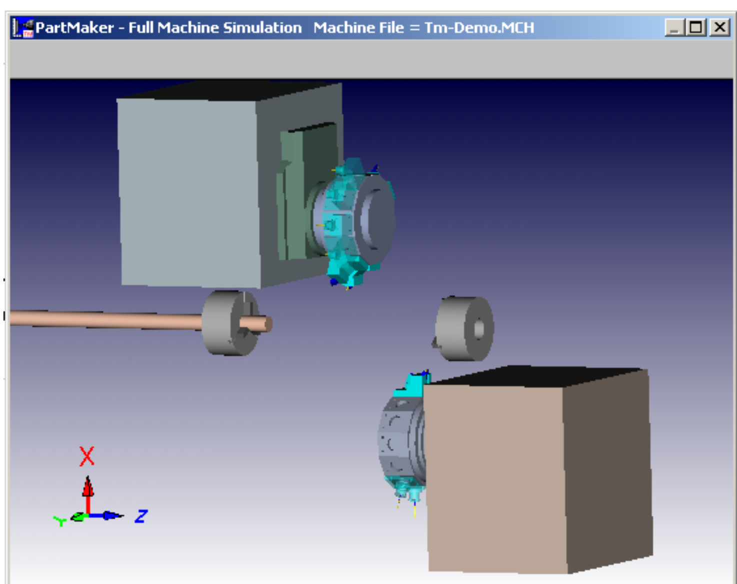
Choose Advanced Simulation from the Simulation menu

or

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

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

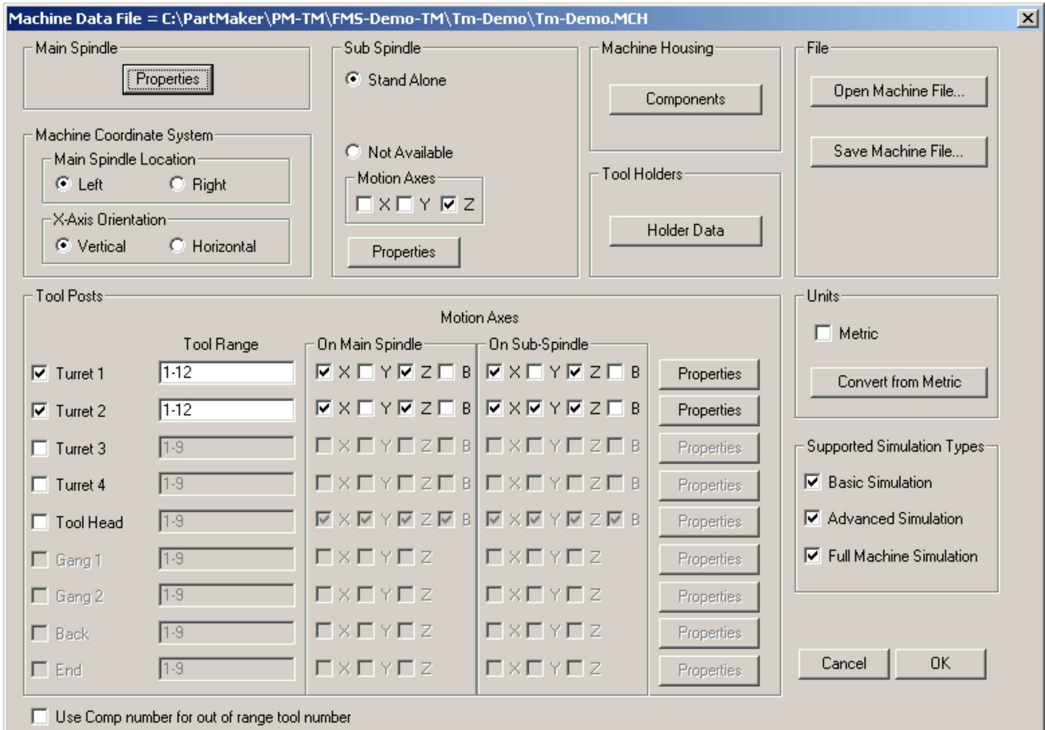
A newly launched simulation window is shown below:



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 mid-process.

Machine File for Advanced and Full Machine 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:



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 then 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 tools 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.

When should you use Full Machine vs. Advanced Simulation?

Full Machine and Advanced Simulation are not mutually exclusive technologies.

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

- Provides a photo realistic representation of the machine, thus enabling robust collision checking of holders, turrets, and other machine components.
- Provides a virtual setup of the machine tool for training purposes.

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

- It will run faster since it is less graphically intensive.
- Models may not be available for a particular machine tool.
- You do not wish attach tools to holders is not required by the user.
- You wish to see tool numbers clearly displayed on the machine model

Although not much more, Full Machine Simulation does take more time to set up for use than Advanced Simulation. It may be beneficial to make sure that the tools are set up correctly in Advanced Simulation, before taking the time to use Full Machine Simulation.

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 Swiss-machines with irregular architectures at this time. Such architectures may include machines with an "L-shaped gang", attached end working and back working arms and machines where the sub-spindle is mounted on a turret.

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

Full Machine Simulation Limitations

The machine model you are seeing in PartMaker's Full Machine Simulation is a realistic model of the machine you are programming. The limitations you may find at this point are:

Material Control Process Simulation

PartMaker Full Machine Simulation does not support the simulation Material Control Processes (MCP) as inserted under the **Job Optimizer** menu. These MCP actions include grippers on the turret and tailstocks.

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

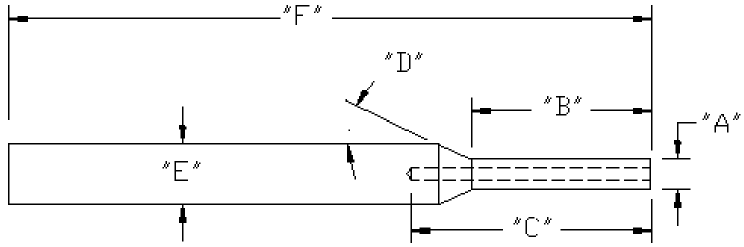
The steps outlined below describe the sequence in which you should approach programming Families of Parts from tabulated drawings. This programming tutorial follows the sequence of steps below in creating a typical family of parts.

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.

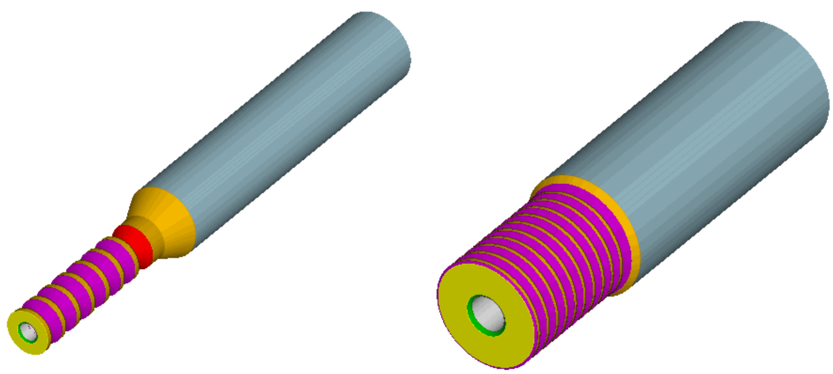
Getting Started

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:



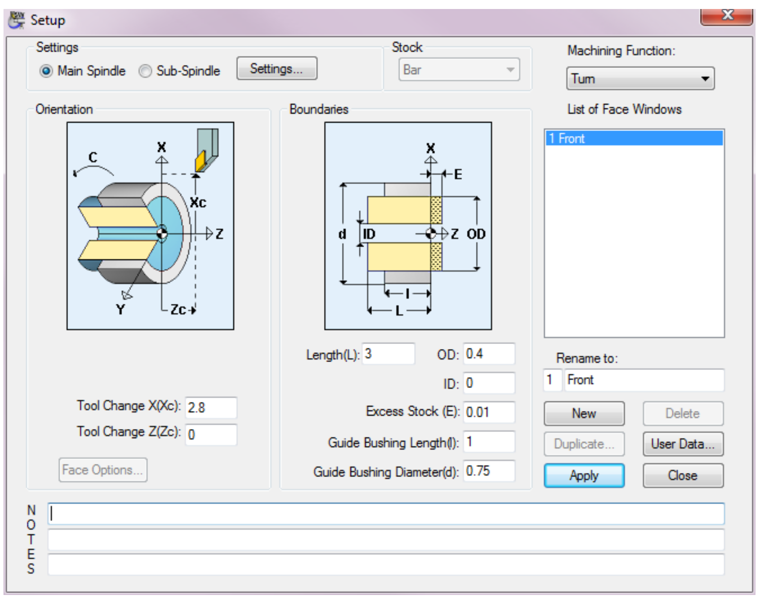
Create Part Geometry

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.

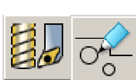
1. Double click on the SwissCAM icon on your desktop or choose **SwissCAM** from the **PartMaker** Group accessed through Programs under the Windows Start button.
2. You will then see the **Setup** definition dialog box.
3. Enter the enter the parameters listed below in the appropriate fields:

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

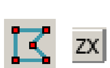
The setup dialog will then appear below.



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**.

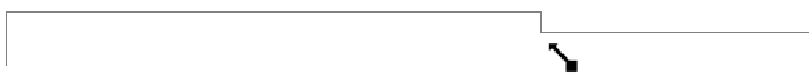
Now enter the ZX coordinates as shown below, pressing **<Enter>** after each:

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.



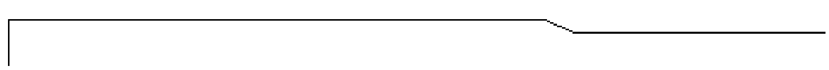
You will now be prompted to enter an angle for this line. At the prompt enter a value of 155 (i.e. 180 – 25) and press **<Enter>**.

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 derformed 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

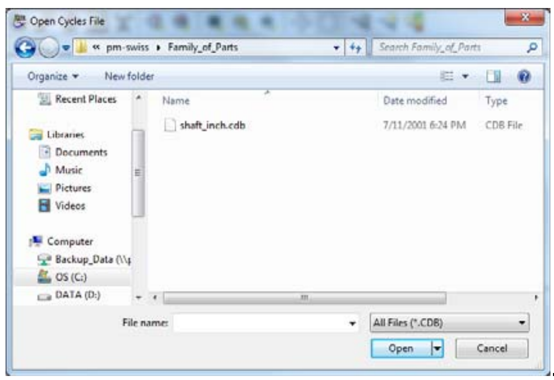
The second step in programming a part in a family of parts is make sure the proper Tools, Materials and Cycles have been created and loaded in PartMaker.

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>**.

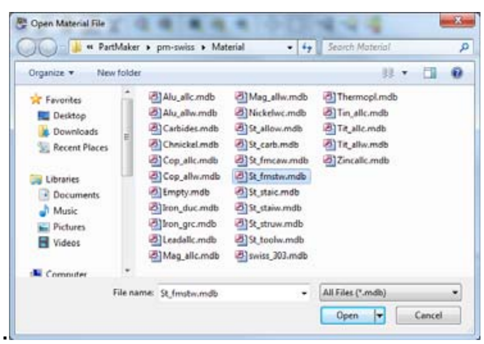
3 Open the Cycles file by choosing **Open Cycles File** from the **File** menu.

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.

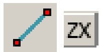
6 Choose st_fmstw.mdb and click **<Open>**



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.

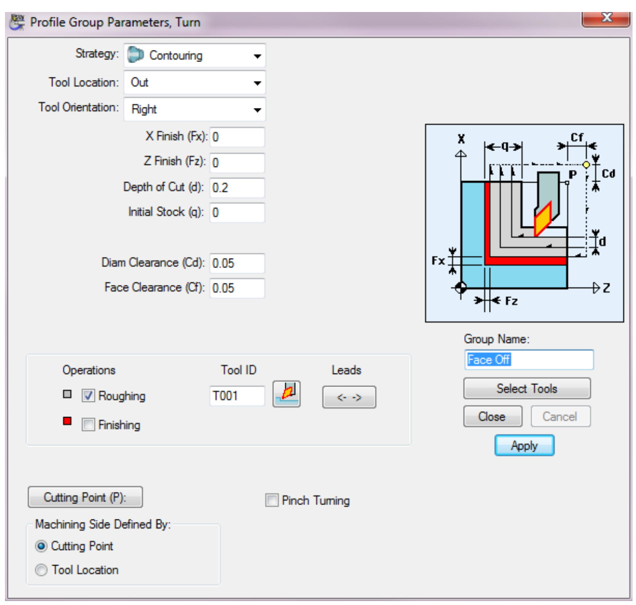
a From the **Strategy** menu, choose **Contouring.**

b From the **Tool Location** menu, select **Out.**

c From the **Tool Orientation** menu, select **Right.**

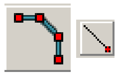
d Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.

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:

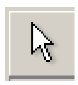


1 From the left-hand side of the screen select the **Define Profile** icon.

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.

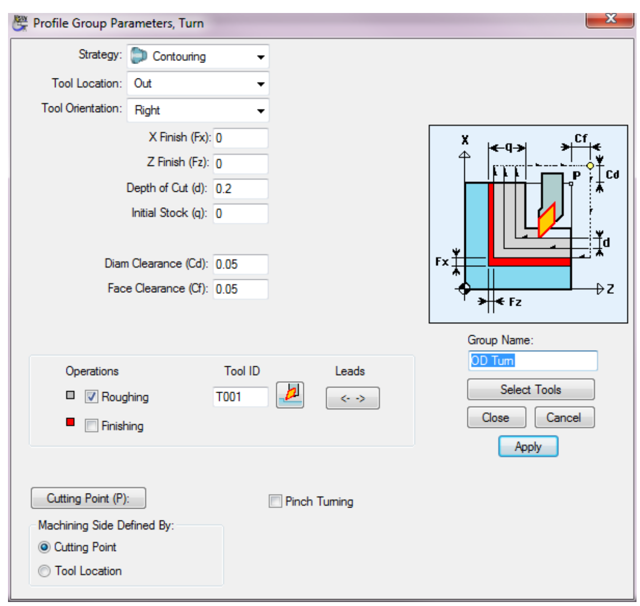
a From the **Strategy** menu, choose **Contouring**.

b From the **Tool Location** menu, select **Out**.

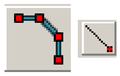
c From the **Tool Orientation** menu, select **Right.**

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:

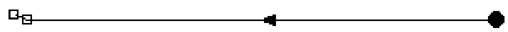


2 From the left-hand side of the screen select the **Define Profile** icon.

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**.

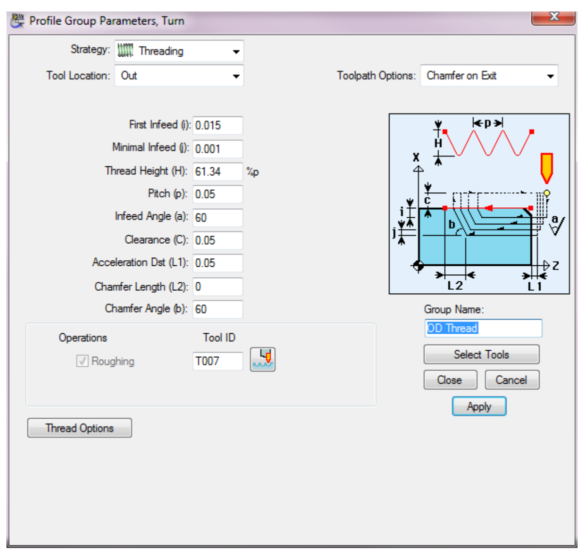
b From the **Tool Location** menu, select **Out**..

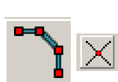
c Select Chamfer on Exit from the Toolpath Options drop dop list.

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:



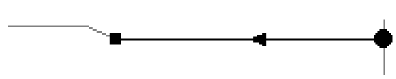


2 From the left-hand side of the screen select the **Define Profile** icon.

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**

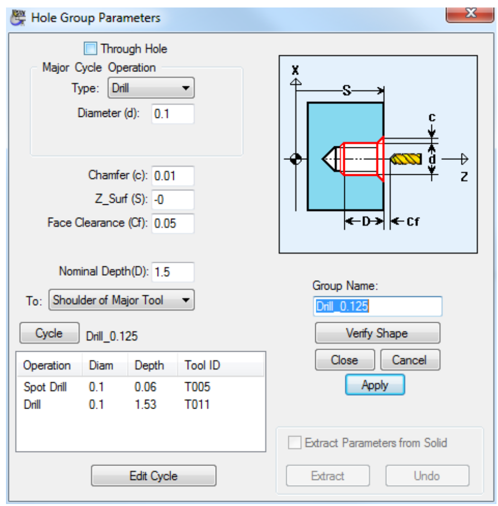
   c Enter .1 in the **Diameter** field

   d Enter .01 in the **Chamfer** field

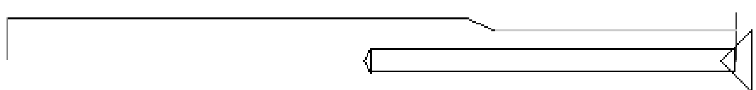
   e Enter 1.5 in the **Nominal Depth** field

   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.

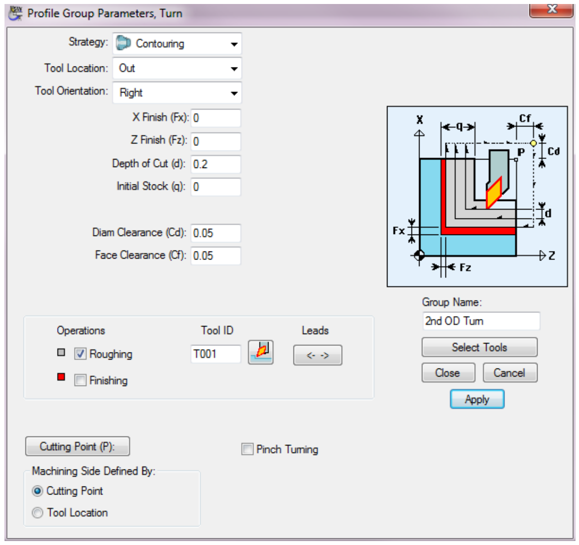
   a From the **Strategy** menu, choose **Contouring**.

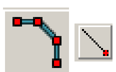
   b From the **Tool Location** menu, select **Out.**

   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:



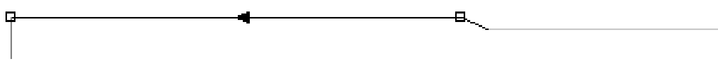


2 From the left-hand side of the screen select the **Define Profile** icon.

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.

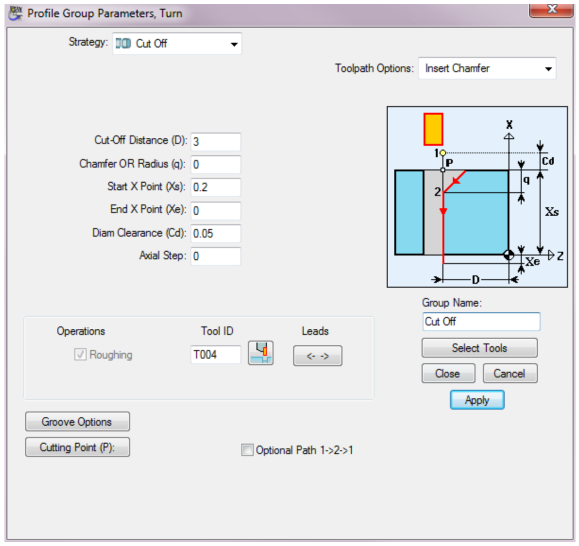
a From the **Strategy** menu, choose **Cut-Off**

b In the **Start X Point (Xs)** enter .4/2

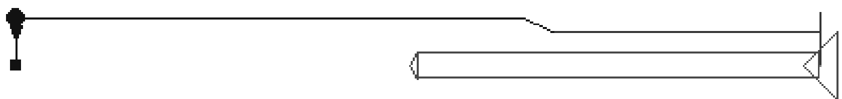
c Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool.

d In the field called **Group Name**, enter **Cut-off**.

Your dialog should now appear as shown below:

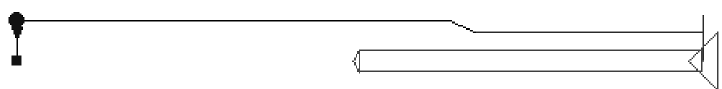


Your completed part should appear as shown below:



6 Before continuing, choose the Selection icon a 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

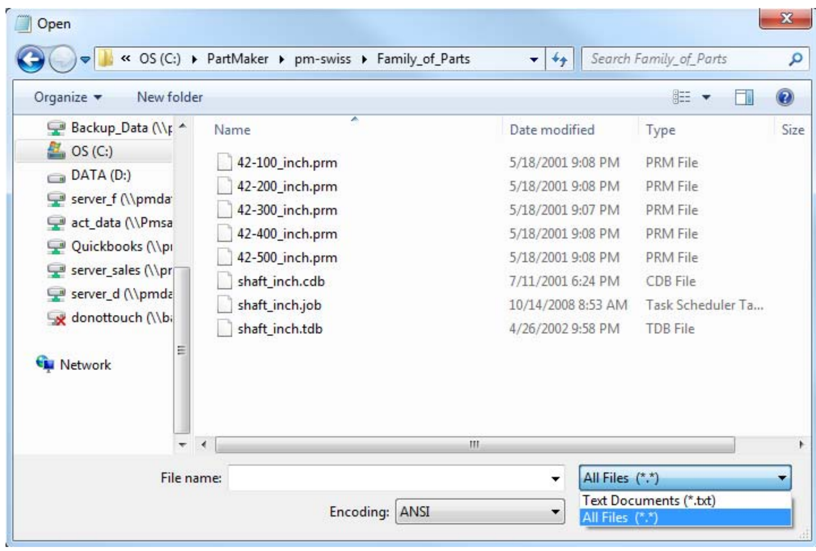
Once you have completed programming the "master" part in the family, you must now take a step back and setup files containing parameter definitions for each part in the family. These files can be created in any word processing program and then saving files with *.prm extension. In this example, **Notepad** (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.

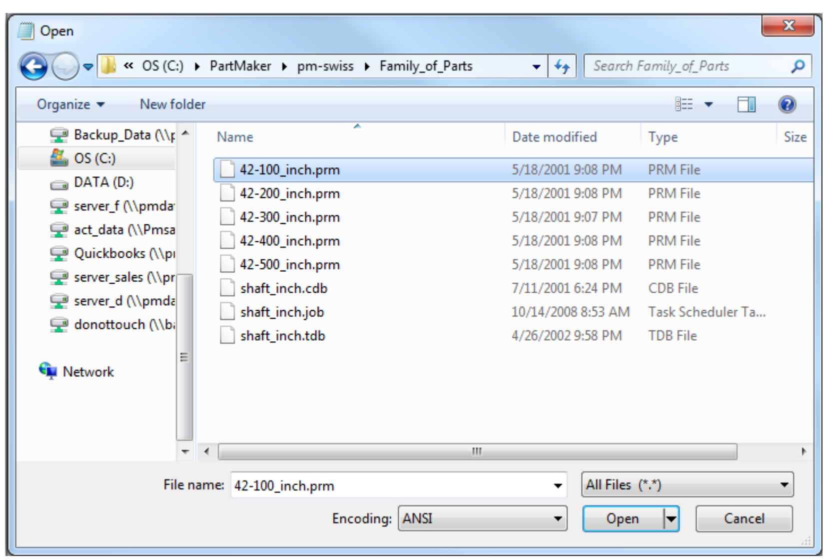
> ***Note:*** *To make creating the first parameter file faster, start by manipulating an existing file, either from a part family you have made before or from the parameter files which came standard with your PartMaker installation.*

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:



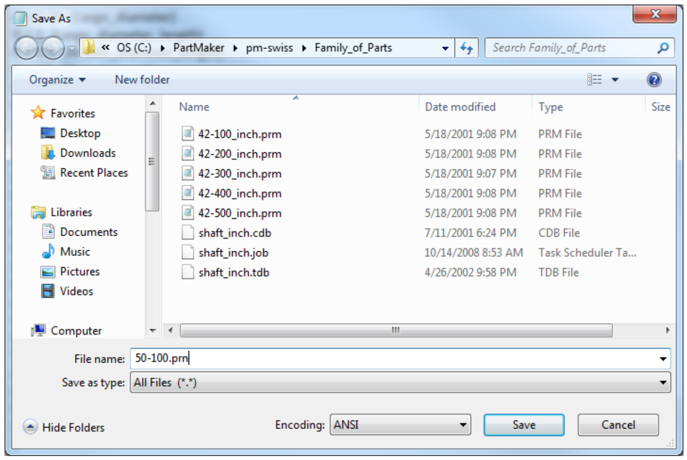
4 Choose an existing parameter file . In this example, open the file titled **42-100_inch.prm** that can be found in the **Family of Parts** directory (located in the root directory of the PartMaker module you are working with.)



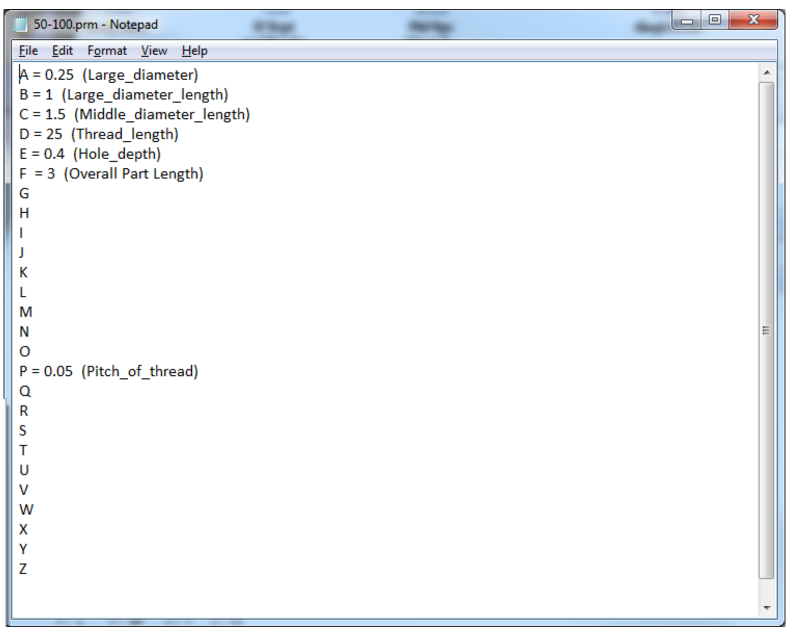
5 Edit the parameter file to reflect the parameters being used in the current Family of Parts. Replace the parameter values with those you have used in the creation of part 50-100 above.

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.*

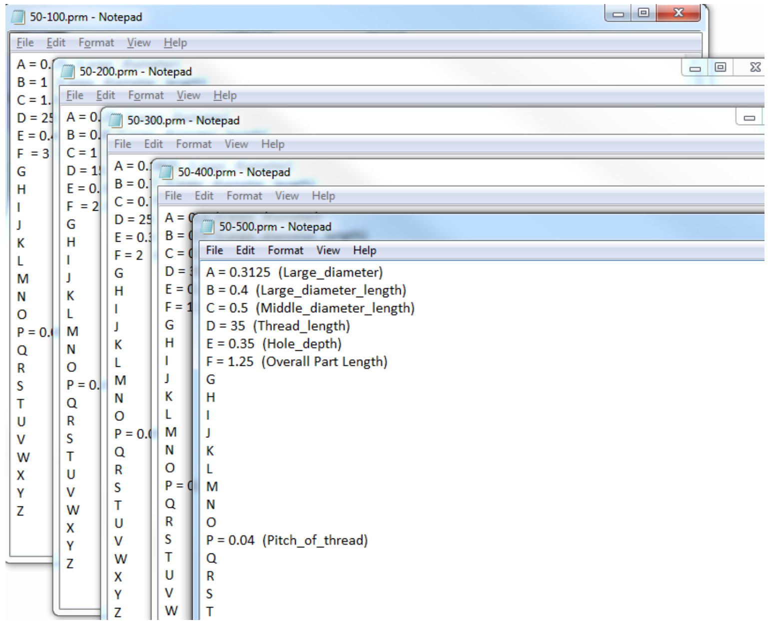
Once you have defined the parameters for the part, save your Parameter File as 50-100.prm by choosing **Save As** from the **File** menu as shown below:



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



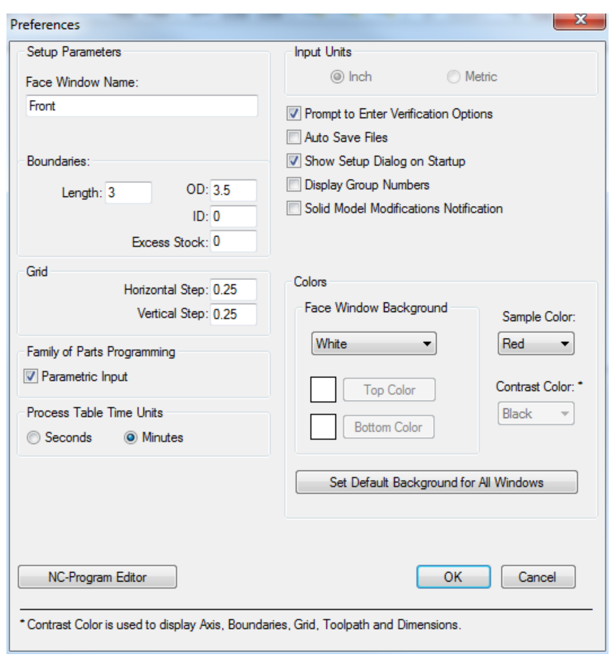
7 Having created the first parameter file, create a file for all other parts in the family by simply replacing parameter values and saving a ***.prm** file for each part in the family. The parameter values for each part for this exercise can be found on page 2 of this appendix. When finished, you should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

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

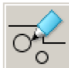


2 Once you have checked **Parametric Input**, click **<OK>**.

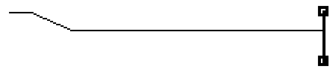
Parameterize Your Part

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.

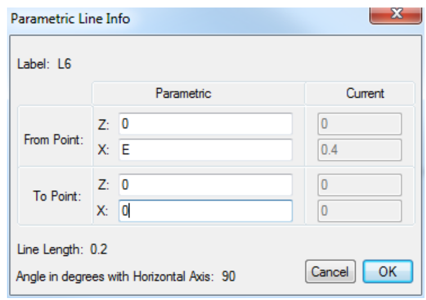
1 Click on the CAD/CAM switch to return to the CAD mode. Any part parameterization exercise should always start in the CAD mode.

2 Double click on the line segment drawn vertically along the face of the part as shown below:

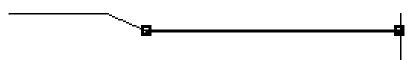


Doing so will allow you to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:

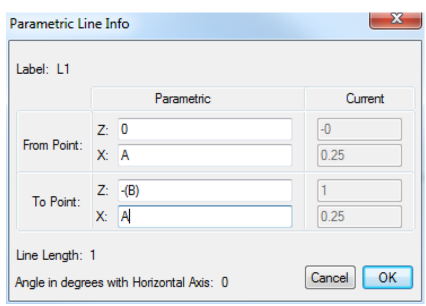


3 Once the dialog appears as above, click **<OK>**.

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



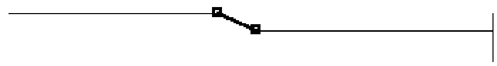
4 Double click on the horizontal line segment along which the thread is applied to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:



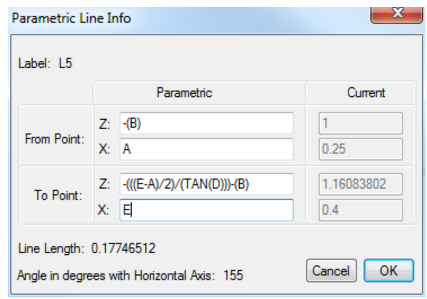
5 Once your dialog appears as above, click **<OK>**.

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:



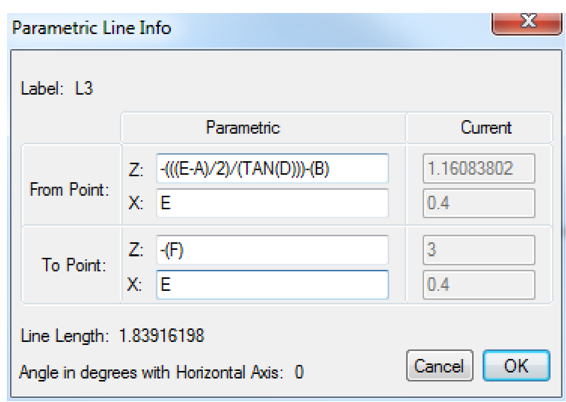
6 Double click on the diagonal line segment of the chamfer to access the **Parametric Line Info** dialog. Define the parametric relationship as shown below:



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>**.

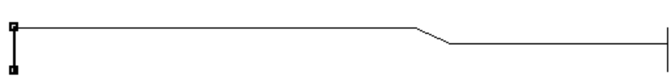
The next geometric element to be parameterized will be the largest horizontal diameter as shown below:

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:

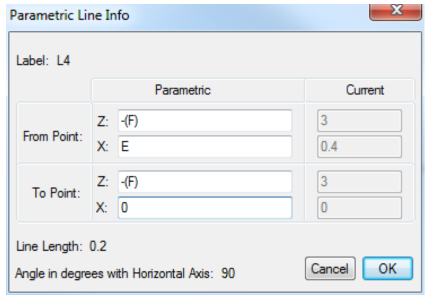


8 Using the **Copy** and **Paste** functions from the **Edit** menu (standard windows accelerator keys **Ctrl + C** and **Ctrl +** V) you can copy and paste parametric relationships from one dialog to another. Once your dialog appears as above, click **<OK>**.

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:



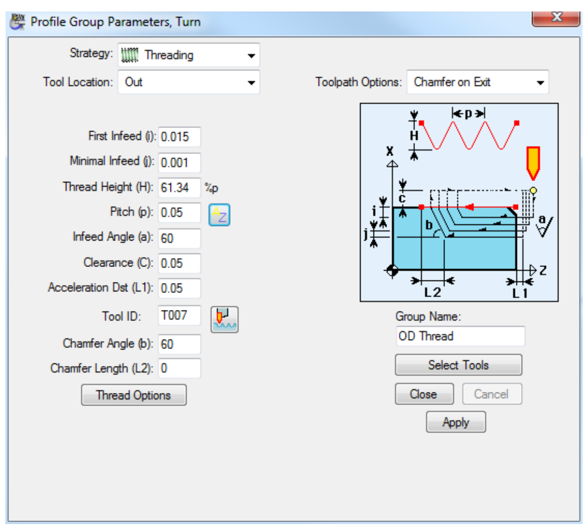
Once your dialog appears as above, click **<OK>**.

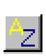
9 Click on the CAD/CAM switch to return to the CAM mode. The parameterization exercise will be completed by applying parametric dimensions to programmed part features.

> ***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.*

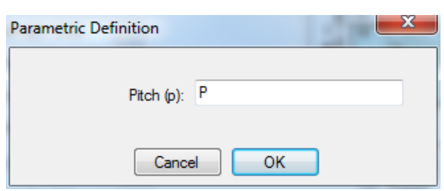
When using Parametric Input you will notice the appearance of **Parametric Input** icons. Clicking on these icons allows you to enter parametric data for part features.

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:





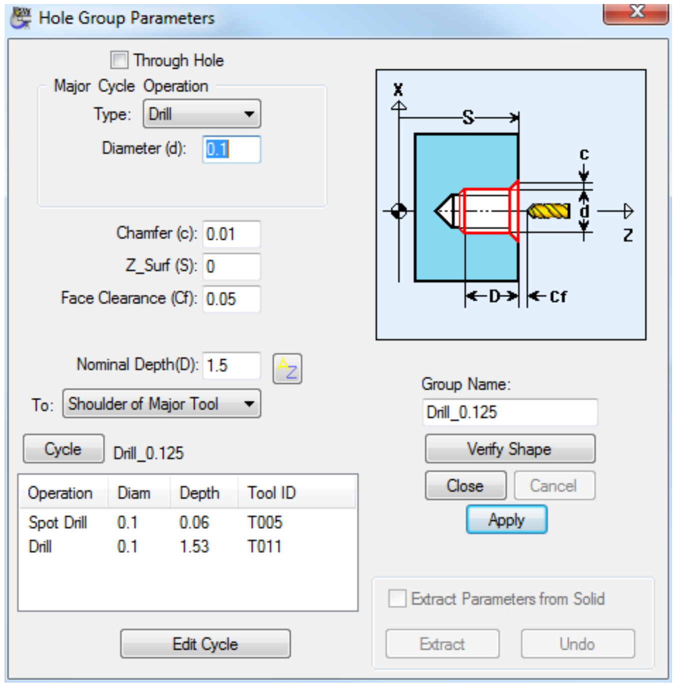
10 Click on the Set **Parametric Pitch** icon to insert a parametric value for Thread Pitch as shown below:



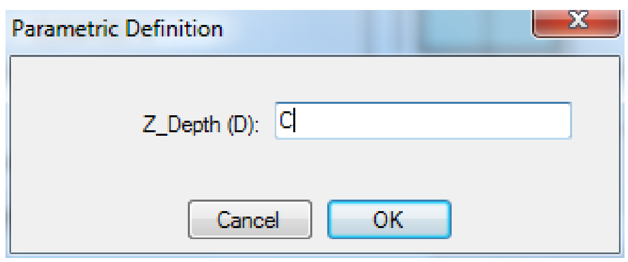
11 Once your dialog appears as above, click **<OK>**. Click **<Close>** in the **Profile Group Parameters** dialog to return to the CAM Face Window.

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:

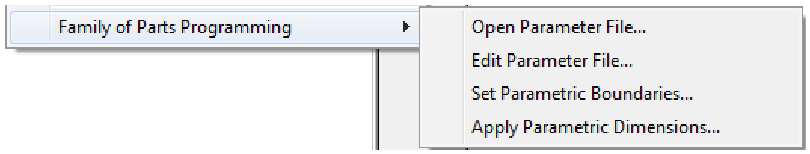


14 Once your dialog appears as above, click **<OK>**. Click **<Close>** in the **Profile Group Parameters** dialog to return to the CAM Face Window.

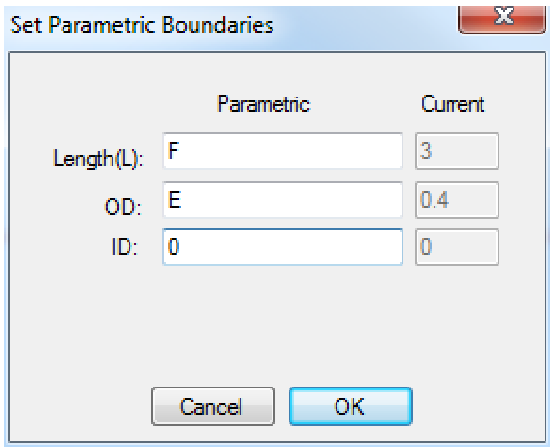
Set Part Boundaries Parametrically

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:



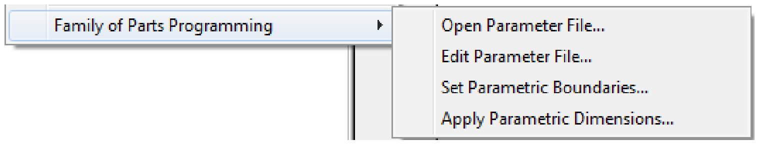
Once the dialog appears as above, click **<OK>** to return to the CAM Face Window.

Update Each Part in the Family

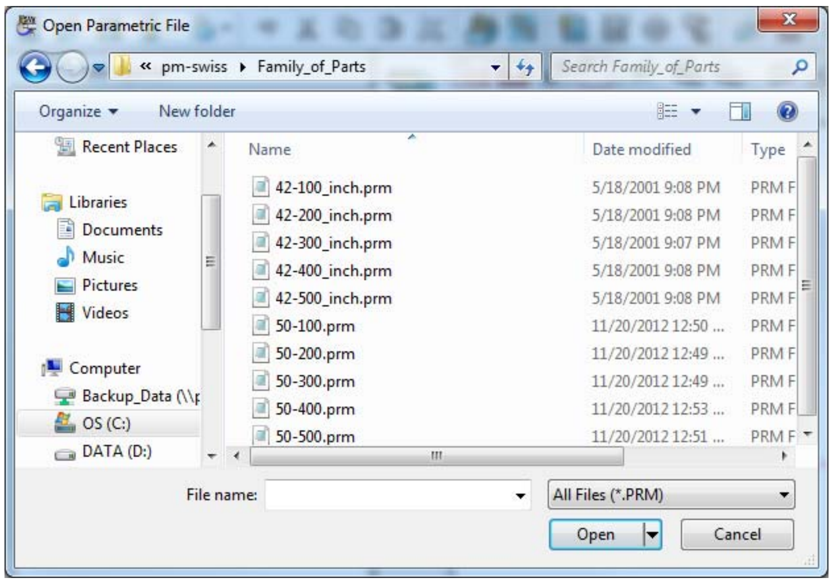
You have now completed setting up your master part and our now ready to automatically program each other part in the family. To automatically reprogram other parts in the family:

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:

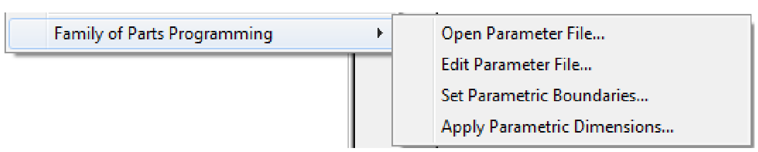


Select the parameter file for the part you would like to program. In this case, choose 50-500.prm as shown below:



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**.

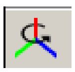
Appendix C: Programming Directly on a Solid Model

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.

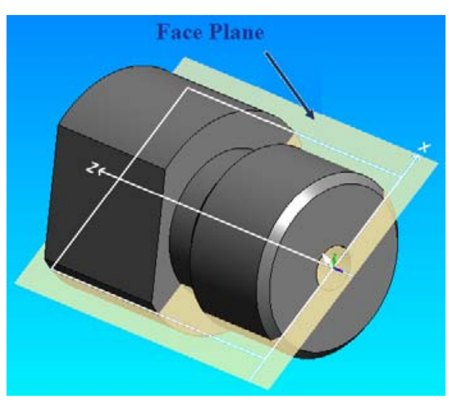
NOTE: *The steps presented in this tutorial are carried out in* ***PartMaker SwissCAM*** *but can be used in any* ***PartMaker*** *application.*

Key Definitions

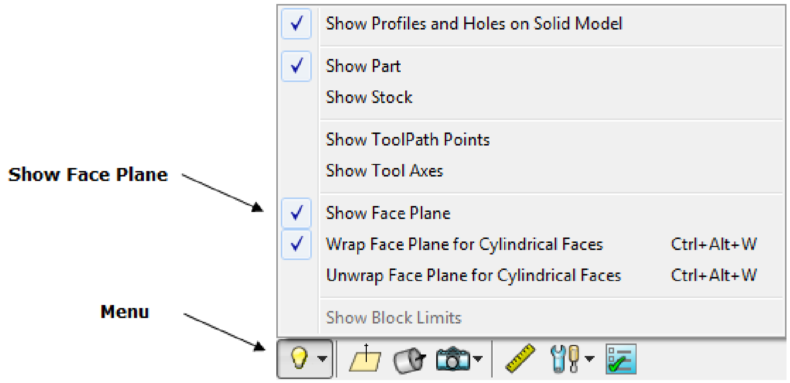


Part Coordinate System (PCS): A part coordinate system (PCS) is the coordinate system in which a solid model is oriented. Before any programming exercise begins, you should always set the PCS to be the same as the coordinate system of the machine you are programming. You can use the **Edit Coordinate 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.

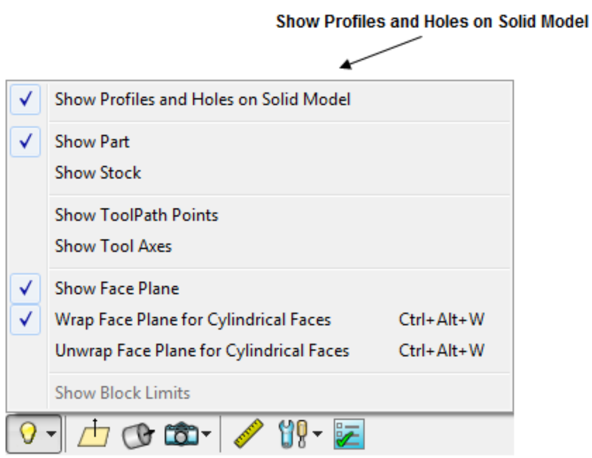
Face Coordinate System (FCS): This is a coordinate system with two axes in the Face Plane and the third axis perpendicular to the Face Plane. The Face Plane is a plane in which the tool paths in the active PartMaker Face Window are created. Each Face Window can have its own Face Plane. The Face Plane 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 incidated:

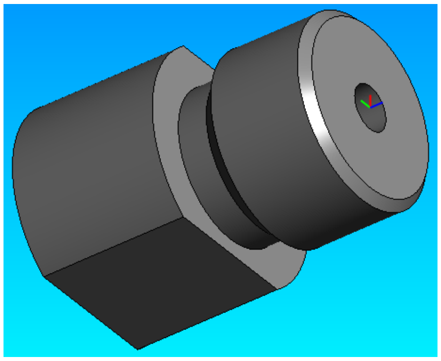
SwissCAM	C:\PartMaker \PM-Swiss\Solids_Programming_Tutorial
Turn-Mill	C:\PartMaker \PM-TM\Solids_Programming_Tutorial
Turn	C:\PartMaker \PM-Turn\Solids_Programming_Tutorial

How you will create the Sample part

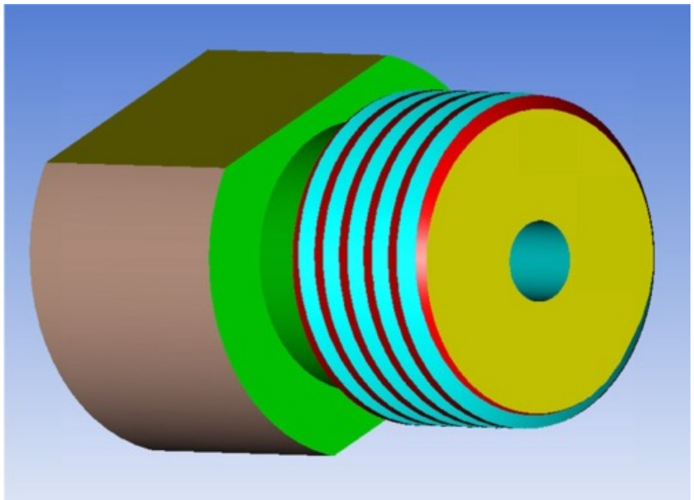
Here are the major steps you will follow to create the Swiss tutorial part:

- √ 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.

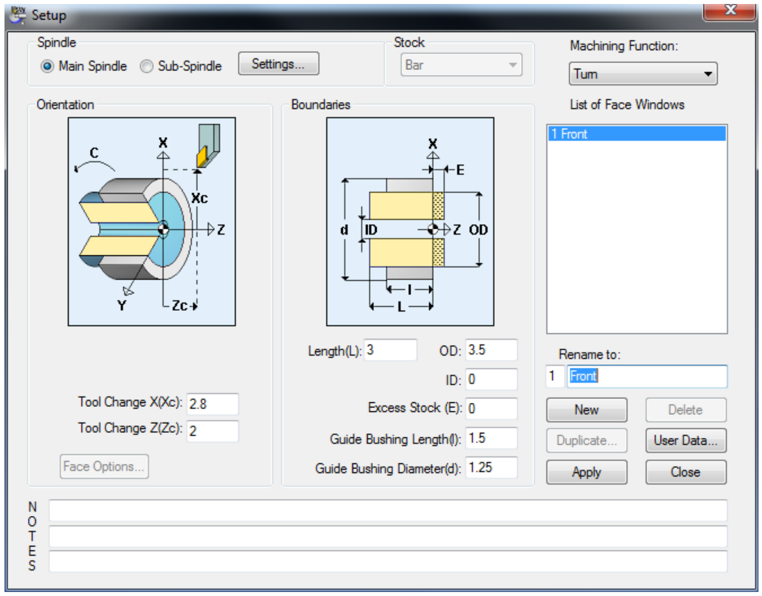
Programming on the Solid Model can be accomplished for almost every Machining Function needed. In most cases you will not need to work in the 2D Face window at all.

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 Follwing 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.*

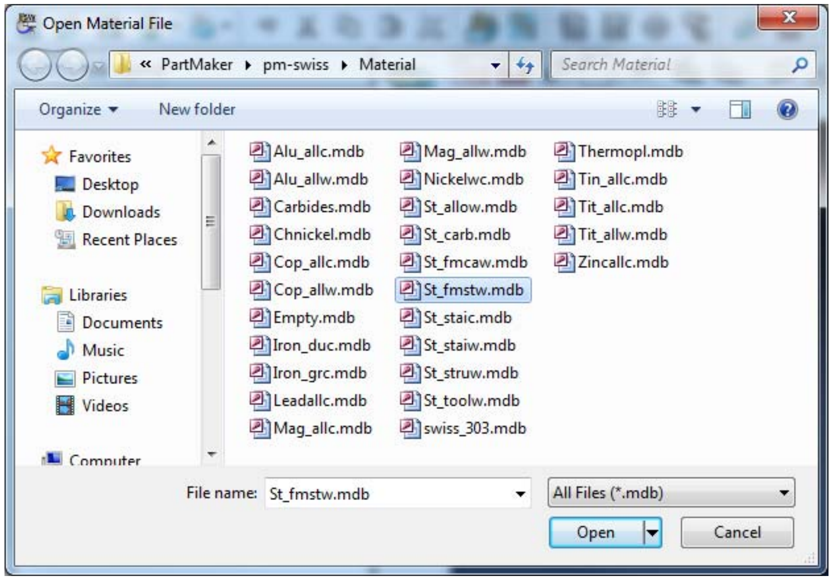
3. Look through the Sample Job file and familize yourself with the Profiles and Hole Groups.
4. When Ready, choose **New Job** from the **File** Menu.

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.

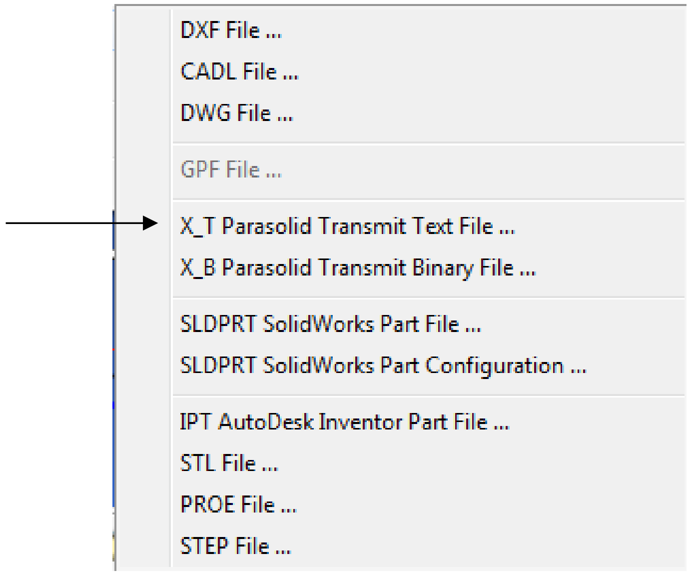
2 Choose **st_fmstw.mdb** as shown below and click **<Open>**.



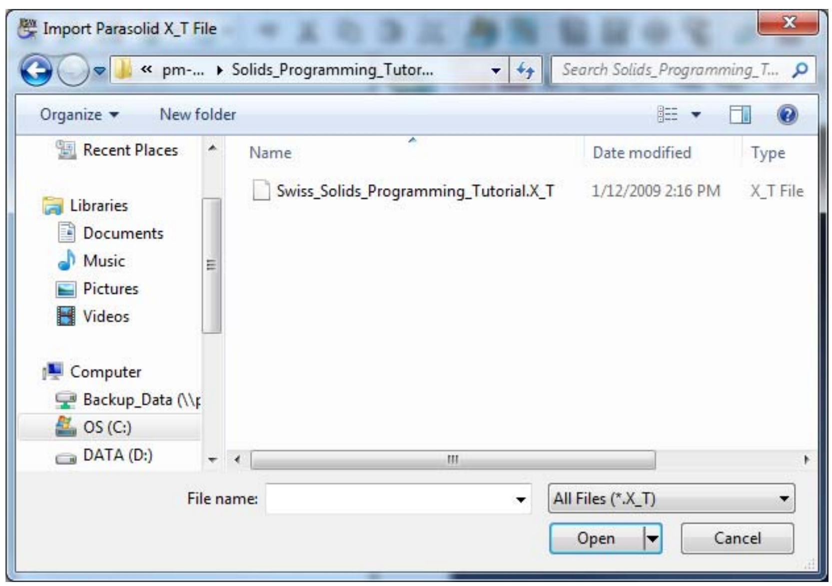
You have now loaded the tools, cycle, and material files that were previously developed for the Solid Model Programming Sample.

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>

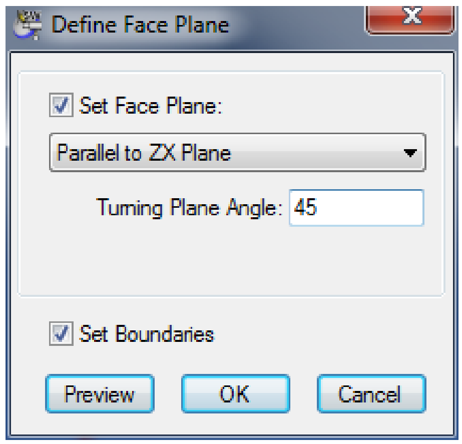


Setting up the First Face Window

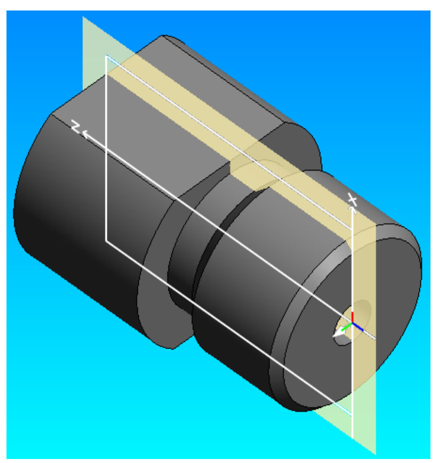
In this section, you will set up the **Face Window** for the turning operations on the main spindle. To do so:

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.

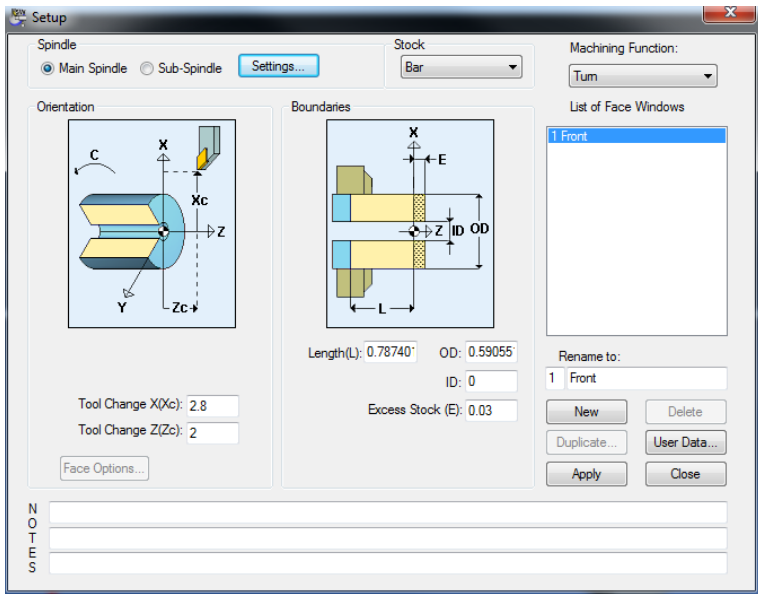


4. Click **<OK>**

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.

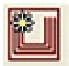
Spindle:	Main Spindle
Stock:	Bar
Machining Function:	Turn
OD:	0.59055118 (For Stock Size)
Excess Stock:	.03

Your Set-up Window should appears as below:



6 Click <Apply> and <Close>

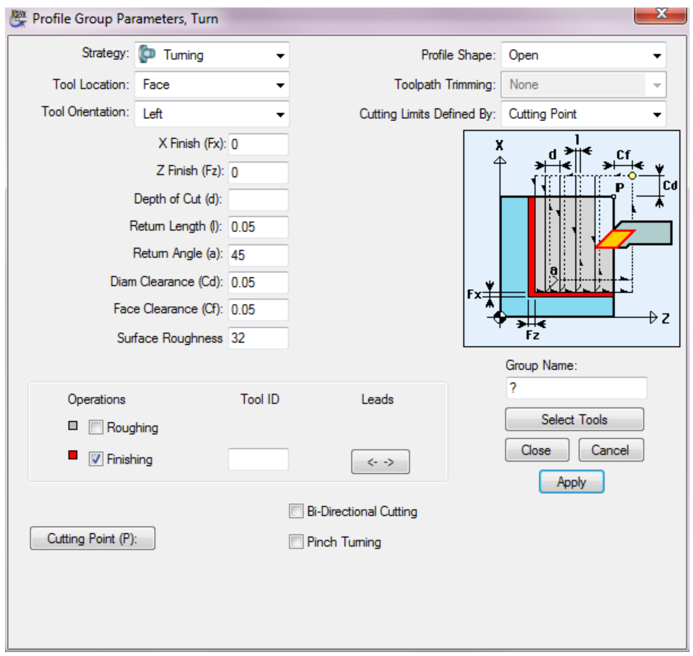
Programming a Face Operation



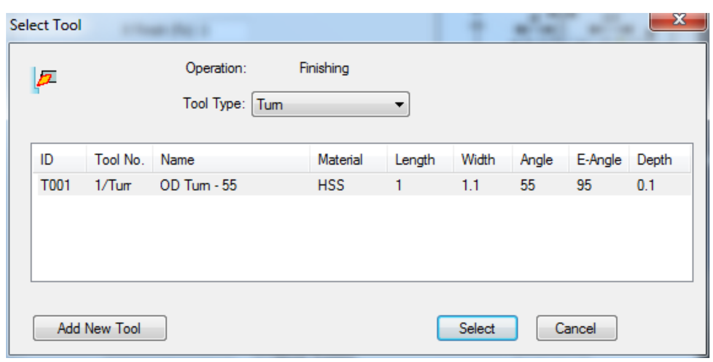
1 Select the **New Profile Group** icon.

2 Change the Following settings in the Profile Group Parameters dialog and Leads Out, Finishing dialog

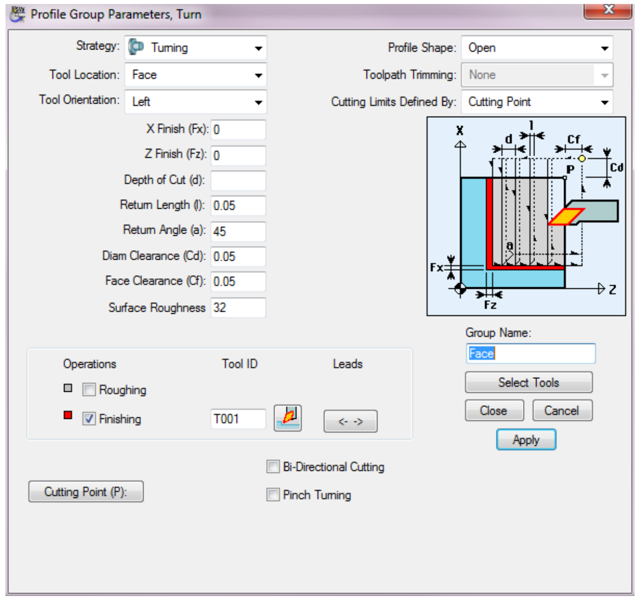
Strategy	Turning
Tool Location	Face
Tool Orientation	Left
Remove Check	Roughing
Check	Finishing
Lead Out Angle (A2)	90 Degrees (click **Leads** button)



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

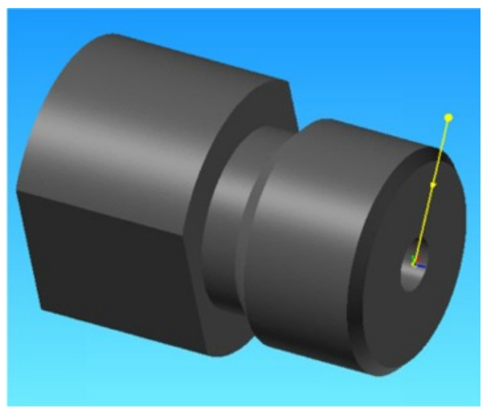


4 Name the Group **Face.** Your dialog should look like the following.



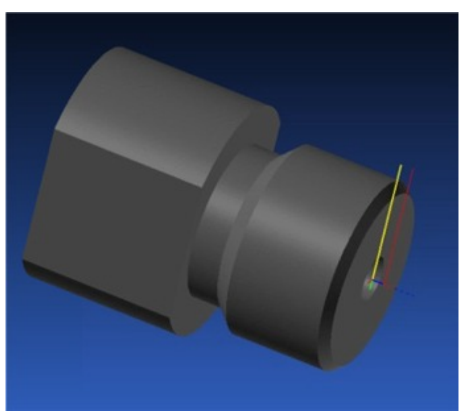
5 Click <Apply> and <Close>

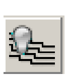
6 You should now see a Facing Profile created on the Solid Model.



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**.

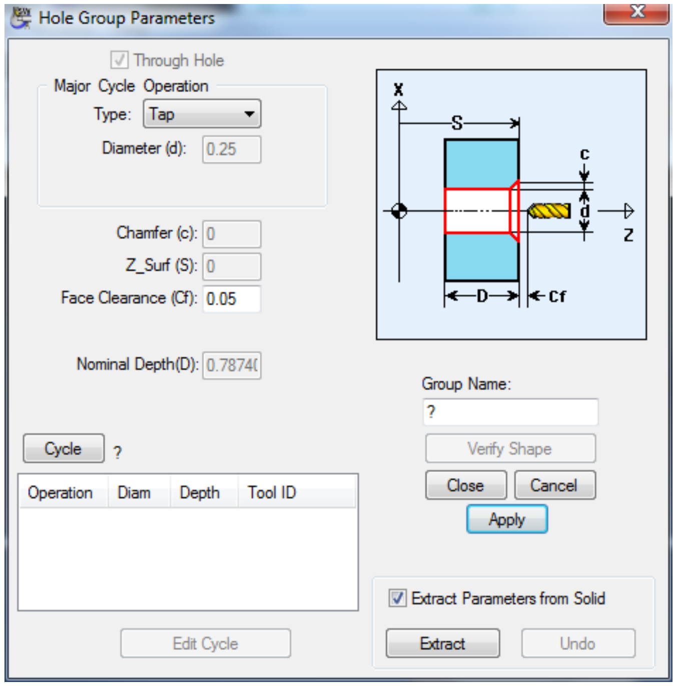




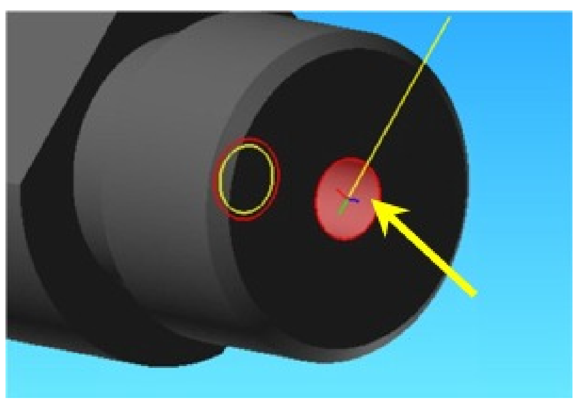
9 Once the tool path has been verified, you can hide by using the **Hide Work Group Toolpath** icon

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.

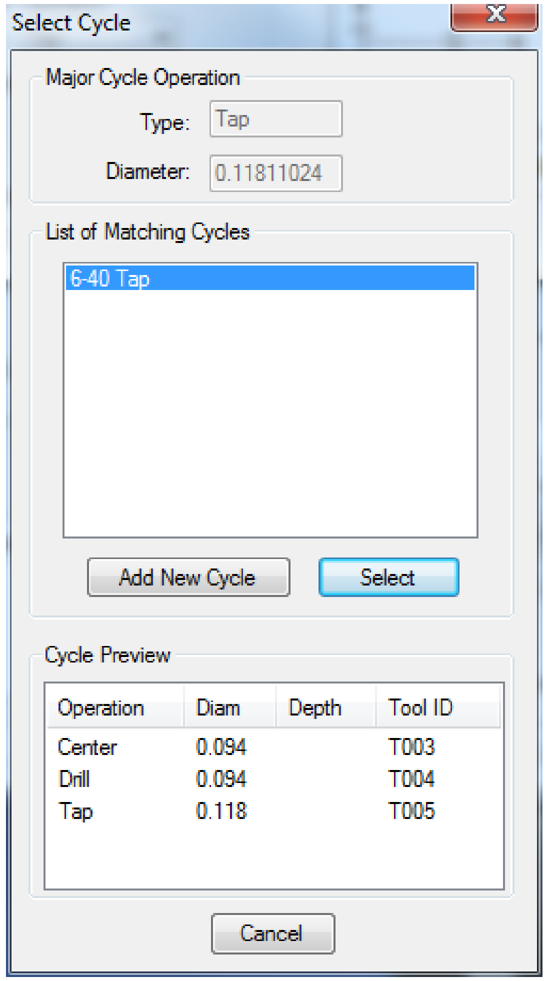


5. Click on the Major Hole Diameter.



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

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



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.

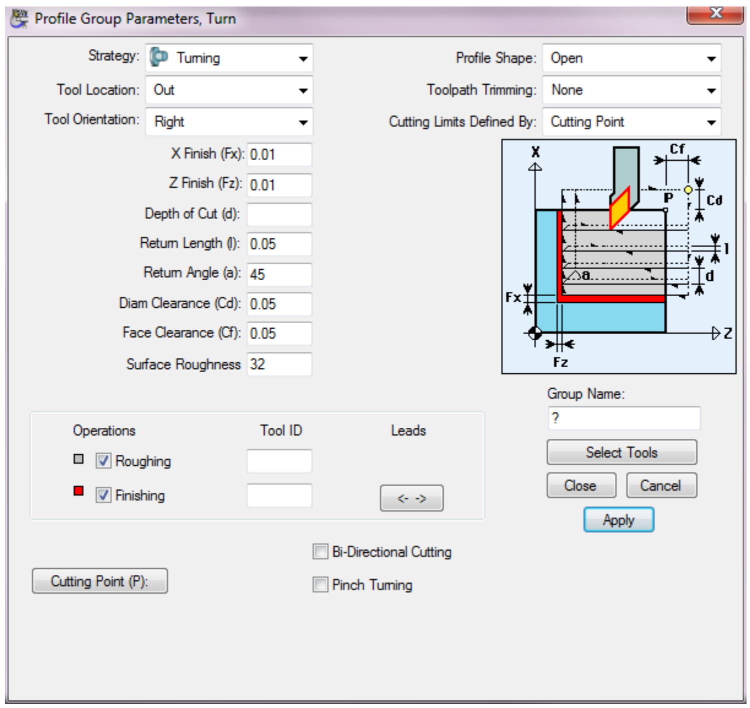
Programming a Turning Operation

1 Choose a new color for this operation in the upper right corner of the screen.

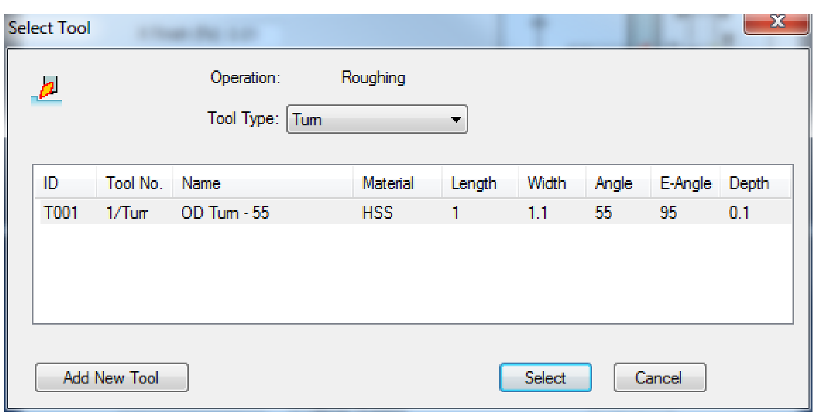
2 Select the **New Profile Group** icon.

3 Change the following settings in the Profile Group.

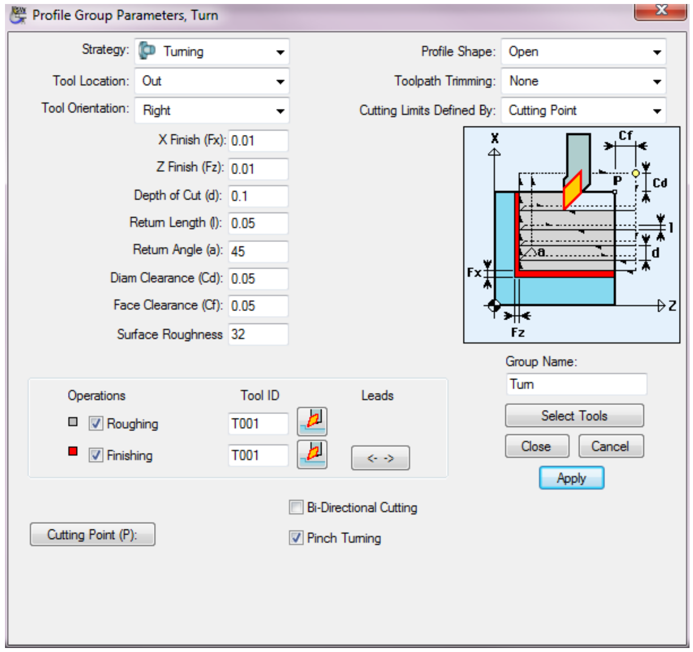
Cycle Type	Turning
X Finish	.01
Z Finish	.01
Check	Finishing



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



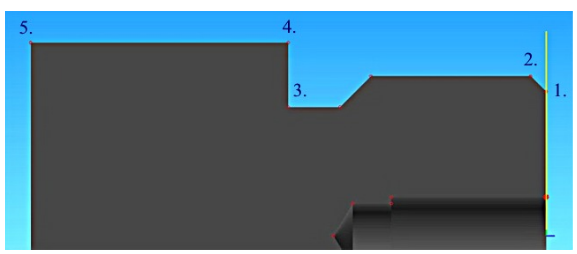
5 Name the Group **Turn**. Your dialog should look like the following.



6 Click <Apply> and <Close>

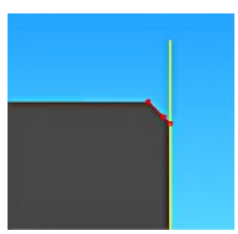
7 Select **Define Profile on Solid Model** icon. The model will automatically section itself at the Angle chosen for the Face Plane.

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.



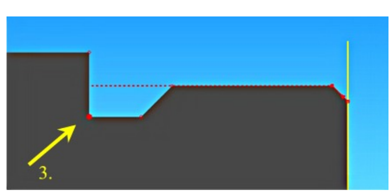
9 Select Point 1

10 Select Point 2



11 Select the Horizontal Constraint icon.

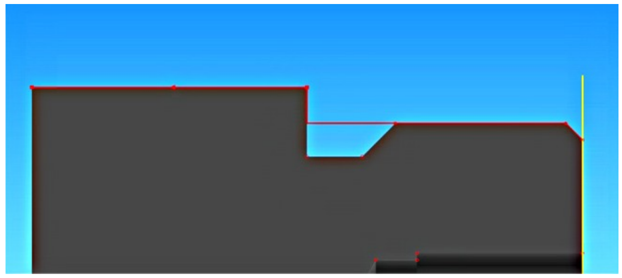
12 Select point 3



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

14 Select point 4

15 Select point 5



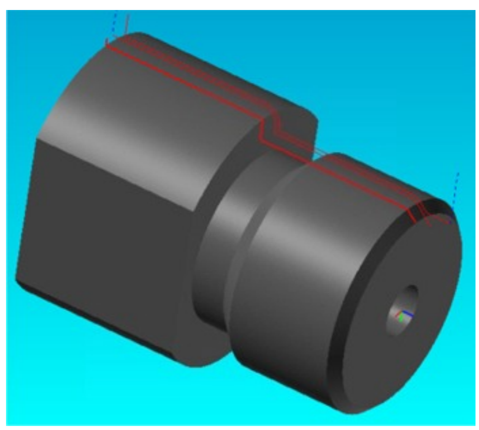
16 Select ZX Coordinates Snap Mode icon.

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.

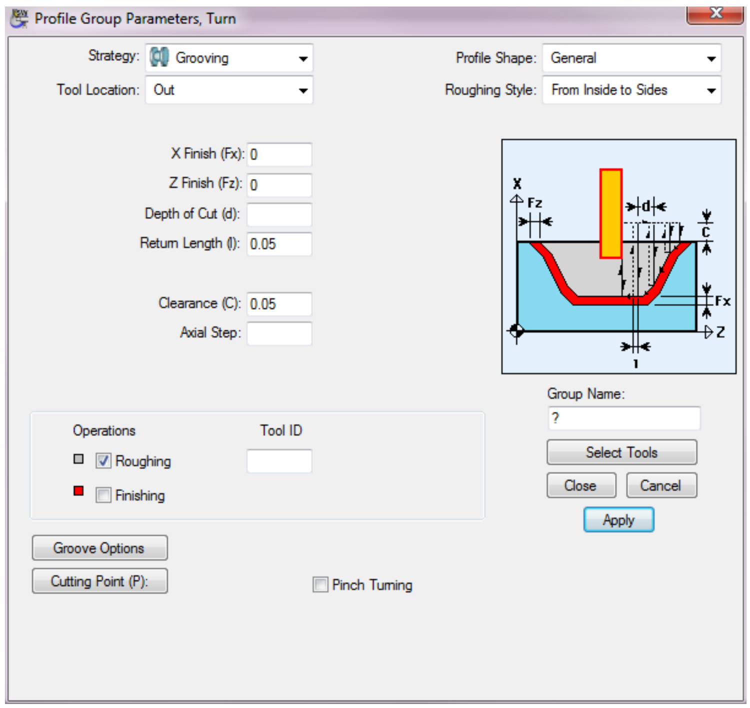
Programming a Grooving Operation

1 Choose a new color for this operation in the upper right corner of the screen.

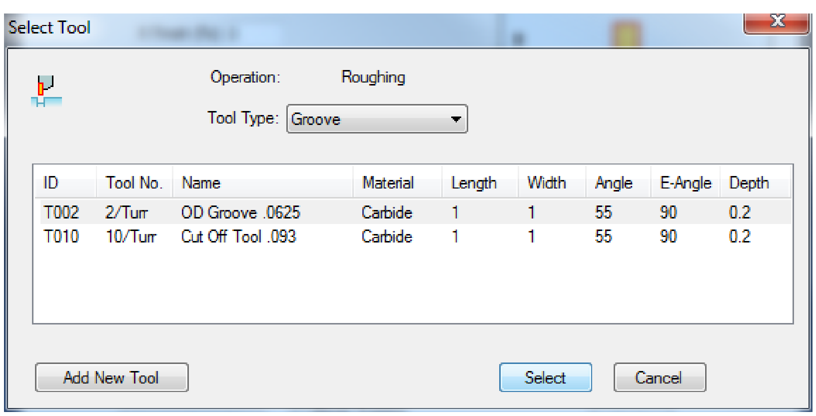
2 Select the **New Profile Group** icon.

3 Change the following settings in the Profile Group.

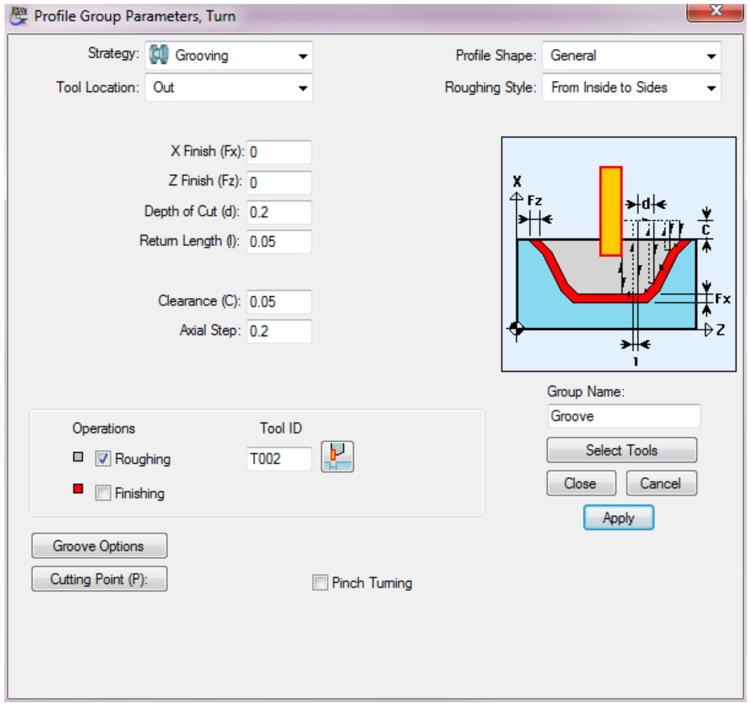
Strategy	Grooving
Profile Shape	General
Check	Roughing



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



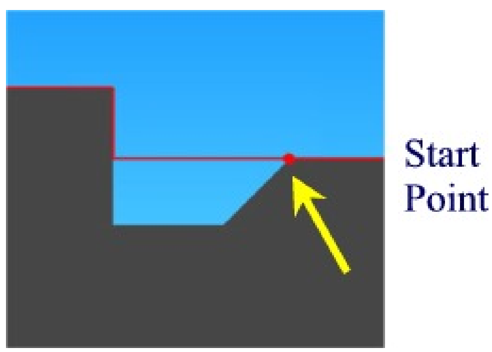
5 Name the Group **Groove**. Your dialog should appear as shown below:



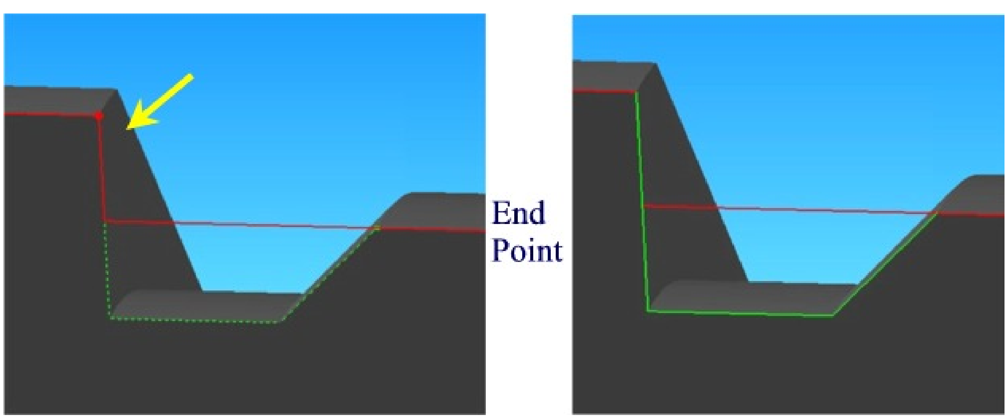
6 Click the <**Apply**> and <**Close**> buttons

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.

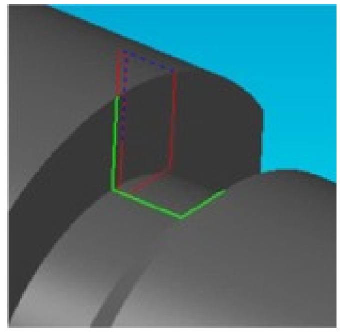


9 Select End Point of the profile.



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.

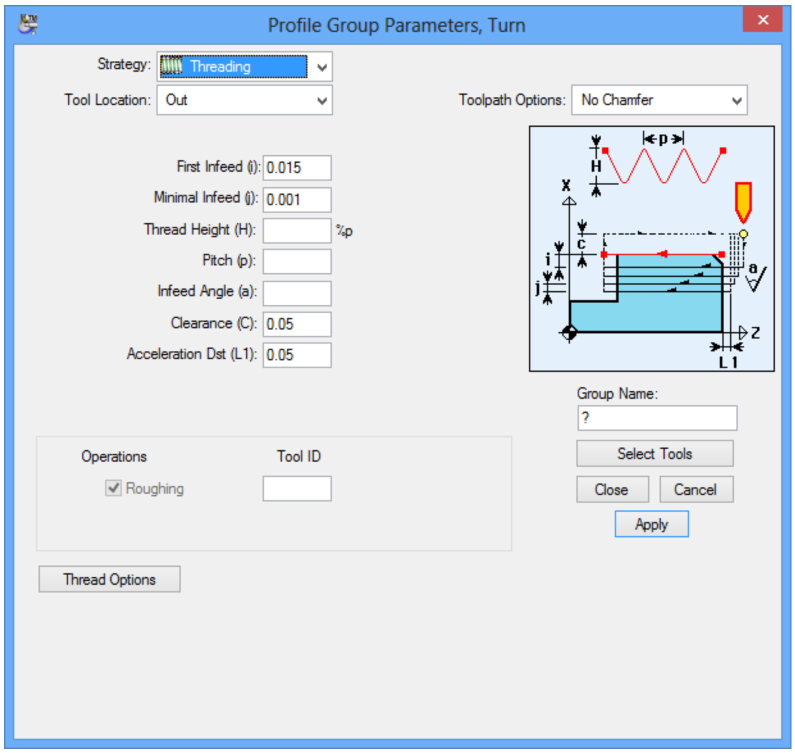
Programming a Threading Operation

1 Choose a new color for this operation in the upper right corner of the screen

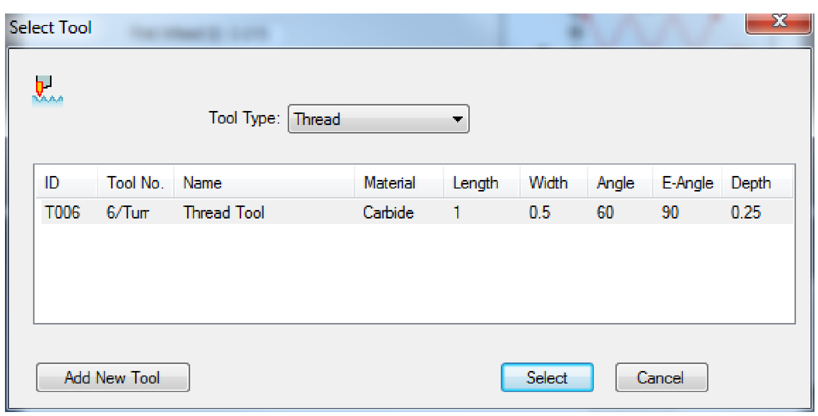
2 Select the New Profile Group icon

3 Change the following settings in the Profile Group.

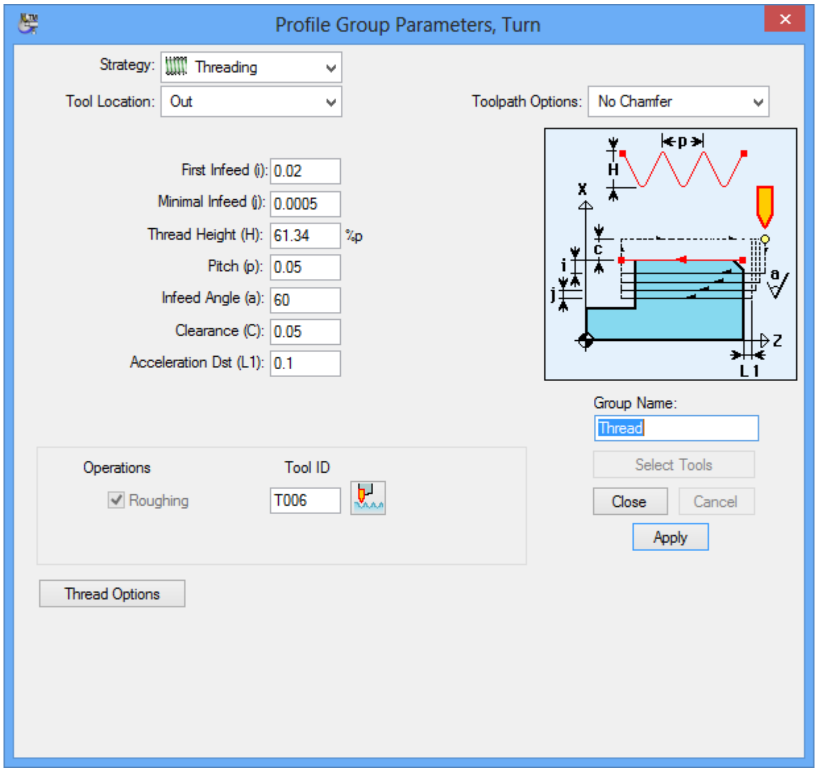
Strategy	Threading
Pitch for 1/20	.05
Acceleration Dst	.1



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



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

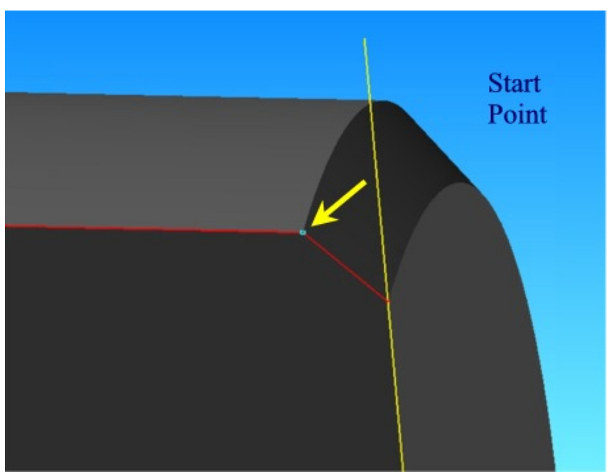


6 Click the **<Apply>** and **<Close>** button


7 Select Define Profile on Solid Model icon.

8 Choose the Edge End Point Snap Mode icon

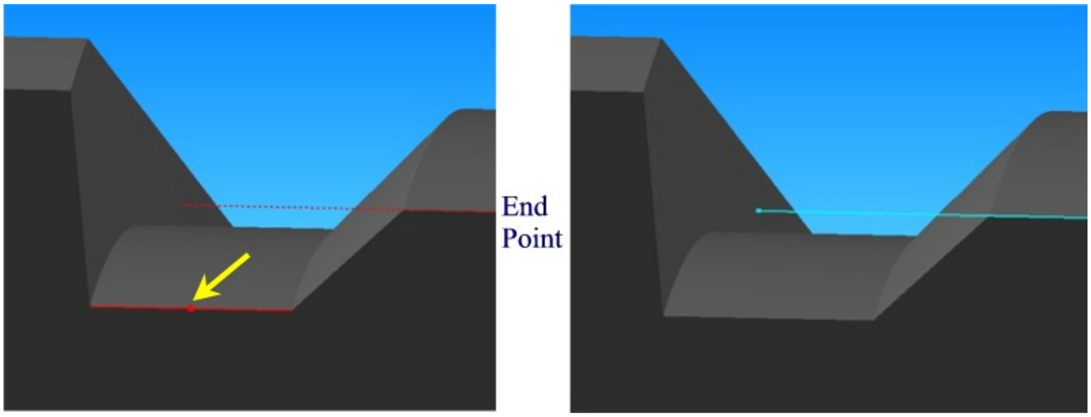
9 Select the Start Point of thread




10 Select Edge Midpoint Snap Mode icon

11 Select the Horizontal Constraint icon

12 Select End Point from bottom of groove area.

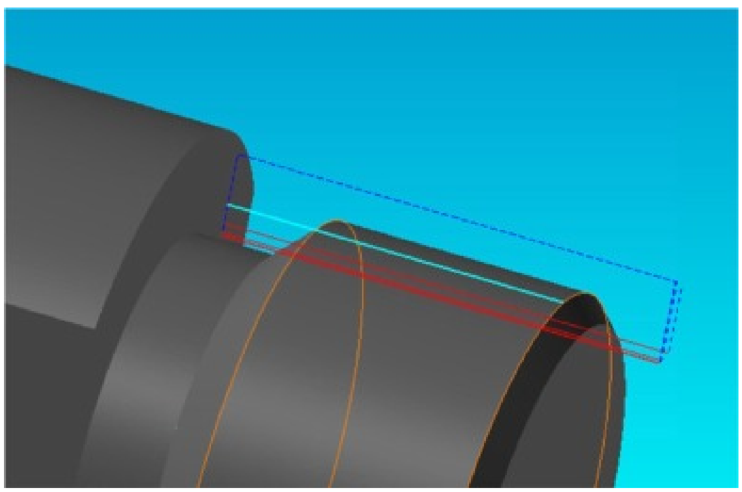


13 De-Select the Horizontal Constraint icon

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.

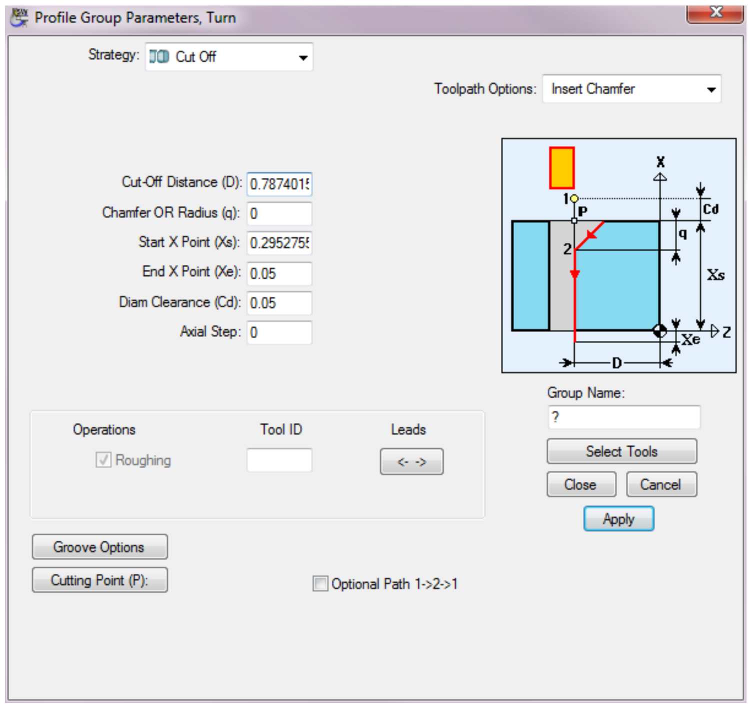
Programming a Cut Off Operation

1 Choose a new color for this operation in the upper right corner of the screen

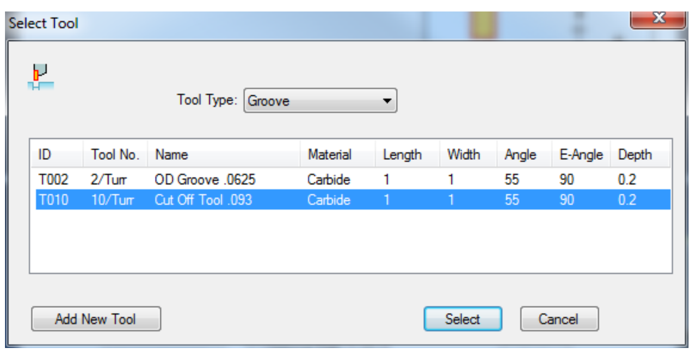
2 Select the New Profile Group icon

3 Change the following settings in the Profile Group dialog and Leads Out, Roughing.

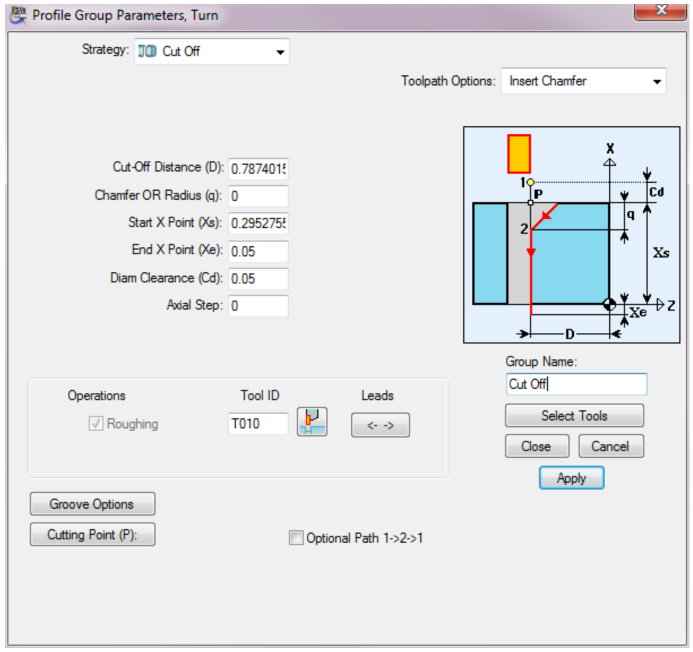
Strategy	Cut Off
End Point X	.05
Lead Out Line Length	0



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

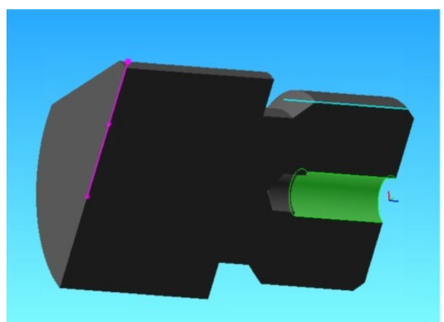


5 Name the Cycle **Cut off**. Your dialog should look like the following.



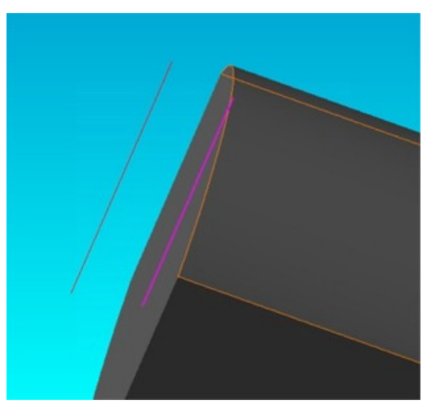
6 Click <Apply> and <Close>

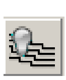
7 The Profile will be created for you.



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**.





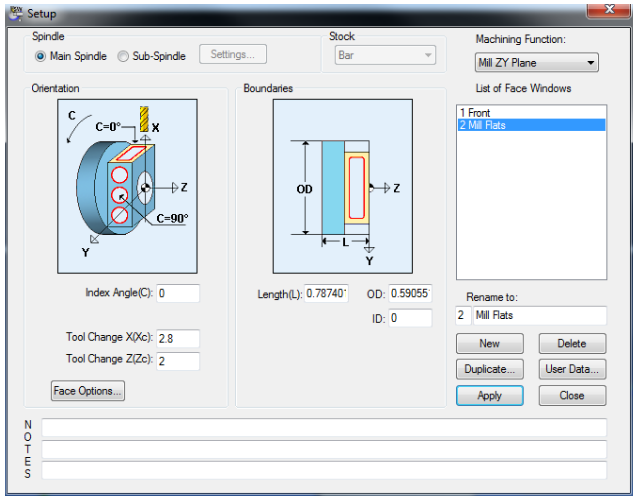
10 After Verifying the Tool Path, click the **Hide Every Tool Path** icon to hide all verifications.

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**.

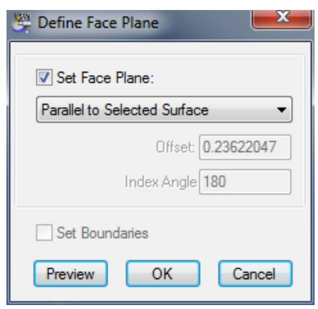


3 Click the **<Apply>** and **<Close>** button

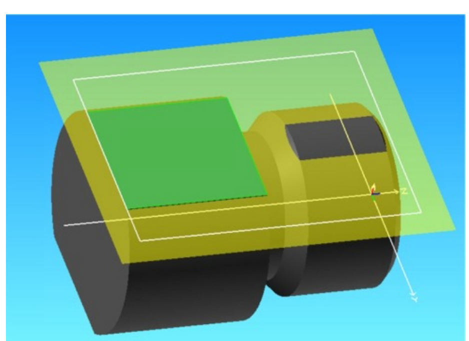
4 If you have not done so already, choose the **Full Part View** icon from the **Solids** toolbar to make sure you have a complete solid model shown and not a sectional view

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.



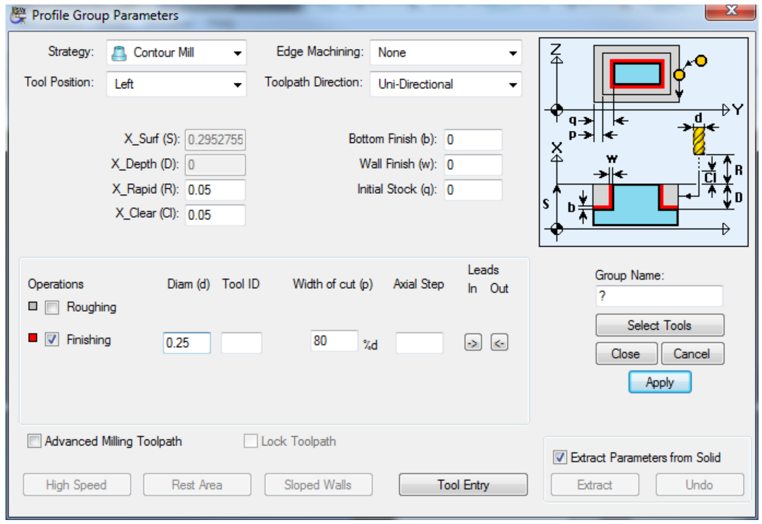
8 Click **<OK>**

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

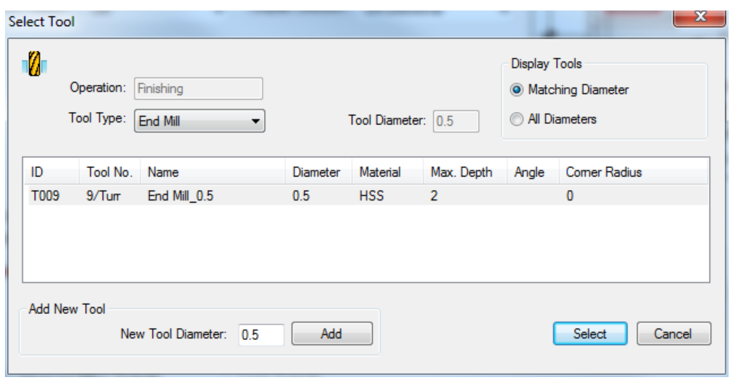
11 Change the **Tool Position** to Left

12 Check the **Extract Parameters from Solid** check box as shown below:

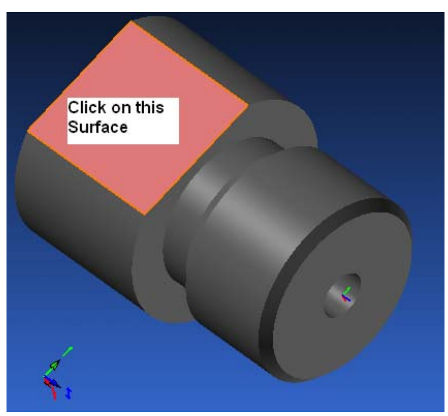


13 Uncheck the **Roughing** box so that **Finishing** is checked

14 Enter 0.5 in the **Diam(d)** field and click **<Select Tools>**



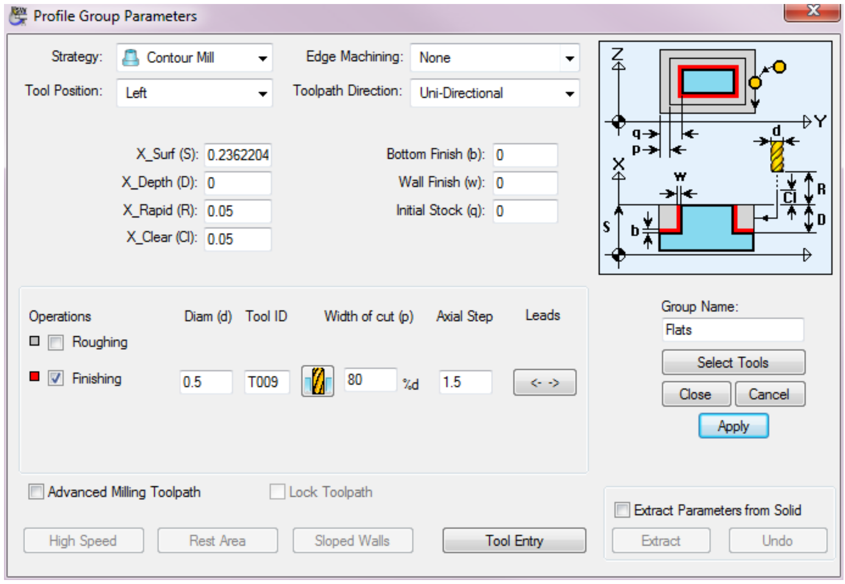
15 With the **Profile Group Parameters** dialog open, click on the plane as shown below:



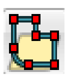
16 Click the <**Extract**> button to extract the machining parameters directly from the model.

17 Rename the Group to **Flats**

18 The completed dialog will appear as shown below:



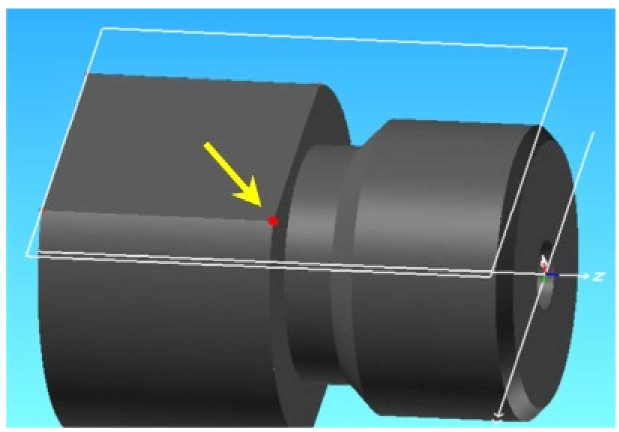
19 Click the **<Apply>** and **<Close>** button.



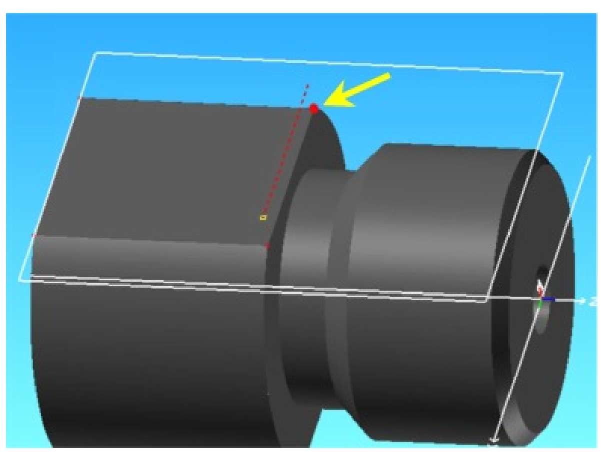
20 Choose the Define Profile on Solid Model icon

21 Select the Edge Endpoint Snap Point icon

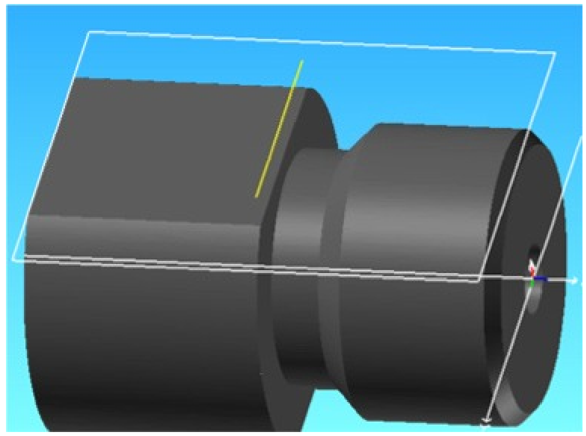
22 Select the Start Point of the profile



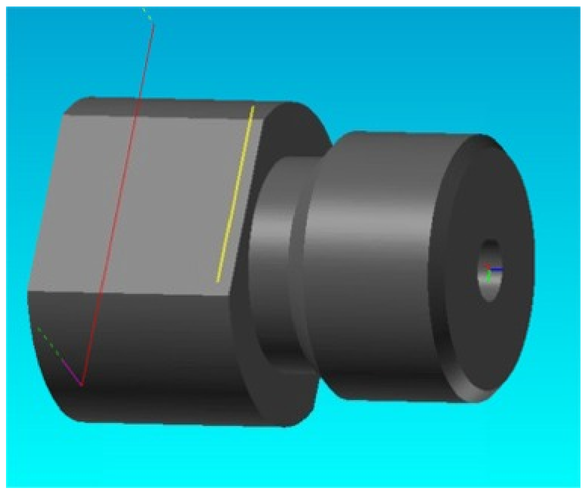
23 Select the End Point of the profile



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

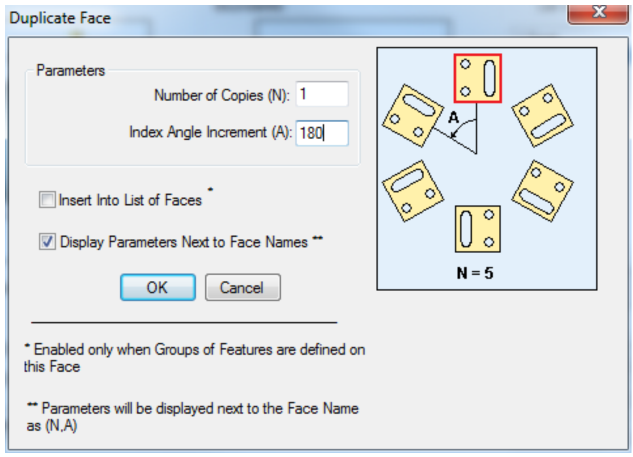
Having programmed this milling path, you can now duplicate the window to create the second flat. This can be done as follows:

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:



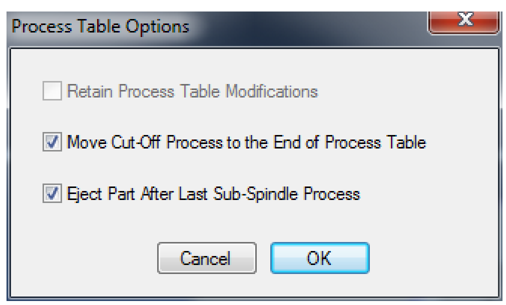
3 Click **<OK>**

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

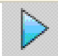
Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T001	1/Turr	OD Turn - 55	Face	Front	0.0028upr	208fpm	0.07	M1S0	
P02	T003	3/Turr	Center Drill	6-40 Tap	Front	0.0022upr	4875rpm	0.03	M1S0	
P03	T004	4/Turr	Drill .098	6-40 Tap	Front	0.0023upr	4875rpm	0.05	M1S0	
P04	T005	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 .0(	Groove	Front	0.0015upr	399fpm	0.21	M1S0	
P08	T006	6/Turr	Thread Tool	Thread	Front	0.0500upr	896fpm	0.09	M1S0	
P10	T009	9/Turr	End Mill_0.5	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_fmstw.mdb Main Spindle Time: 1.43 min, Sub Spindle Time: 0.00 min. Total Time: 1.43 min.

Simulating Machining in 3D

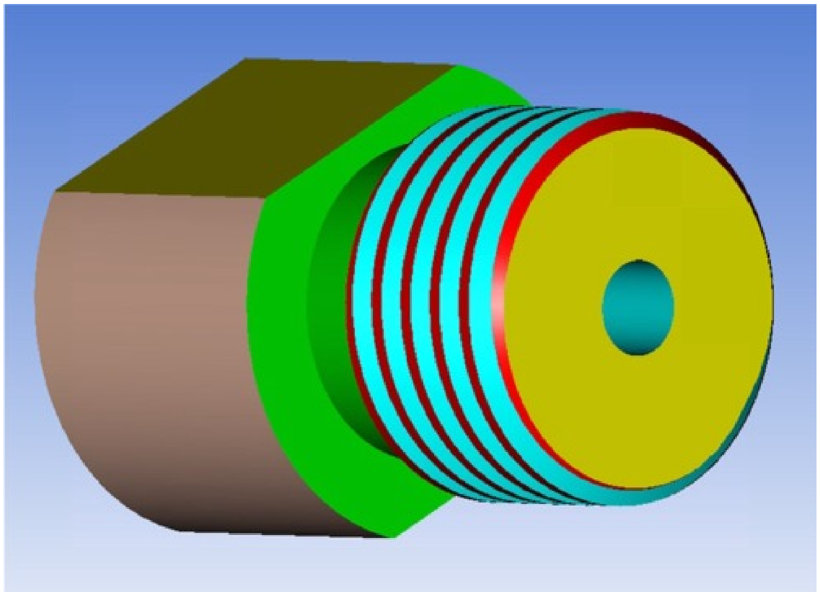
Now that you have generated the Process Table, you can simulate the cutting of the part in 3D. To do so:

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:



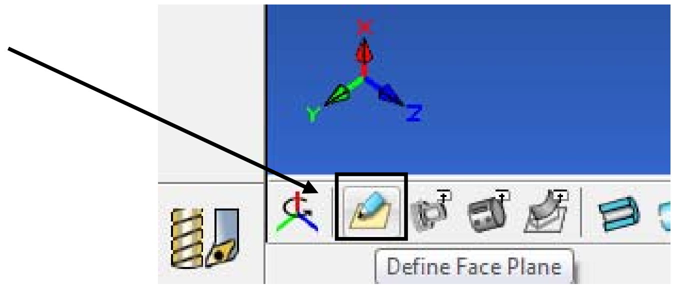
Other Samples

Turn Function Face Window Programming

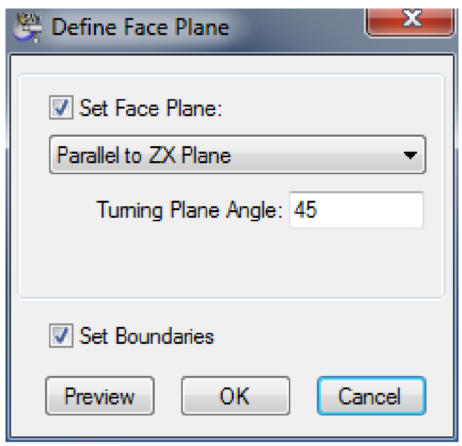
Describes Programming on a Solid Model in a Turn Function Face Window. This applies to PartMaker SwissCAM, TurnMill and Turn modules.

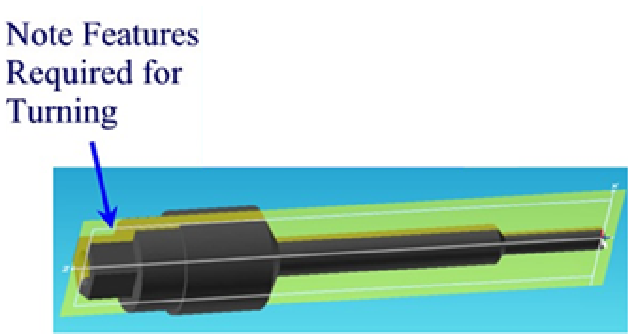
Setting the Face Plane

1 Open the Define Face Plane Dialog.



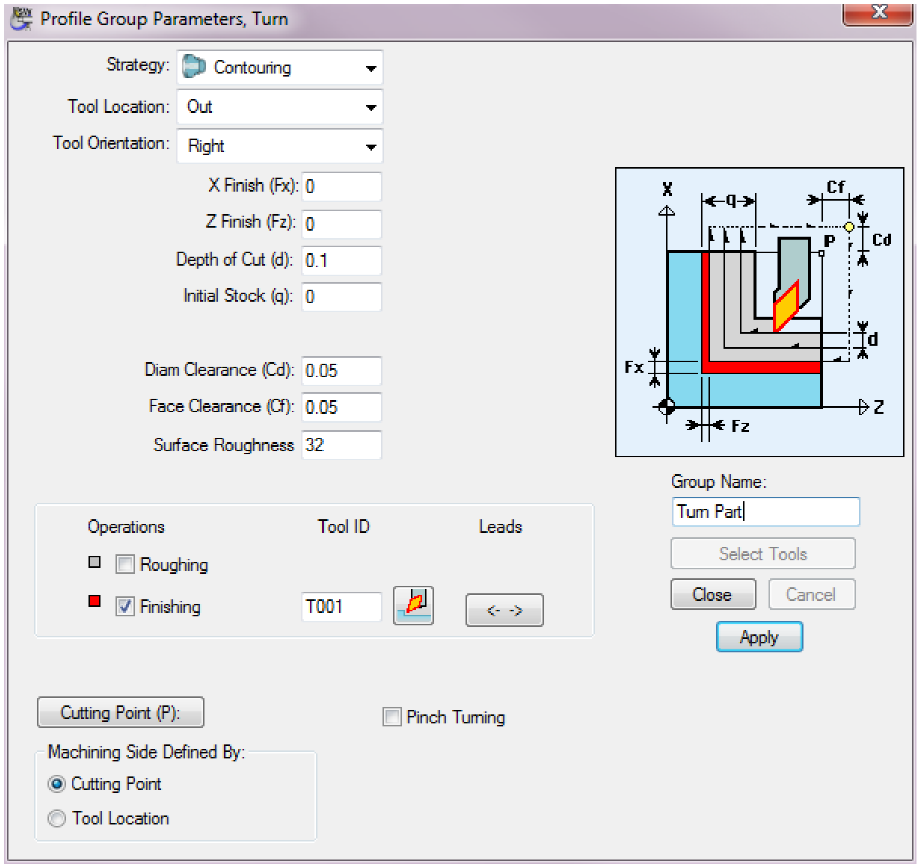
2 Set Face Plane as required to be able to Select Points on this plane for the Profile. Note the features on the Part when setting and use the Preview button to preview the changes.





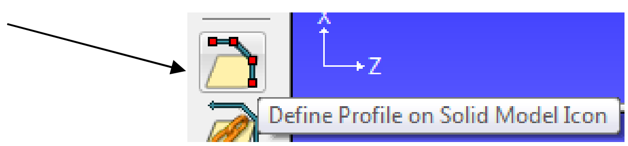
Creating the Profile Group

Create Profile Group as Normal

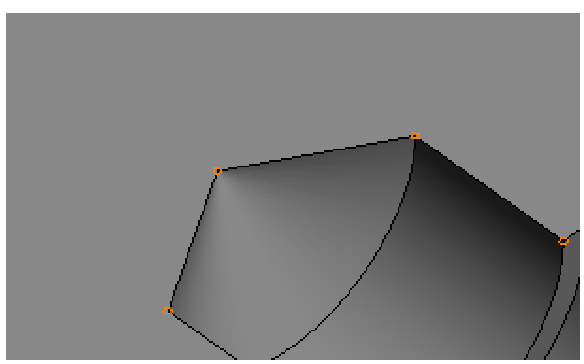


Selecting the Profile

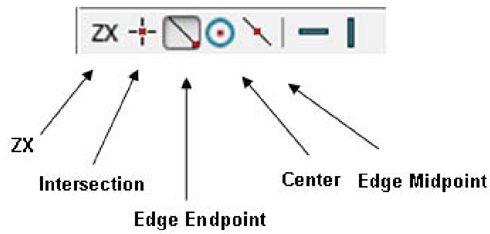
1. Select by choosing the Profile Points
2. Choose the **Define Profile on Solid Model Icon**



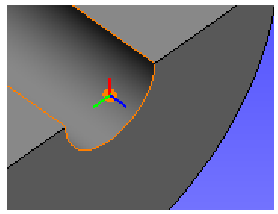
3. At this point you will see Red Points show on the Solid Model at the intersection of the Face Plane. Select these points to create your Profile.



4 Use the Snap Modes to choose the Profile Points on the Solid Model.

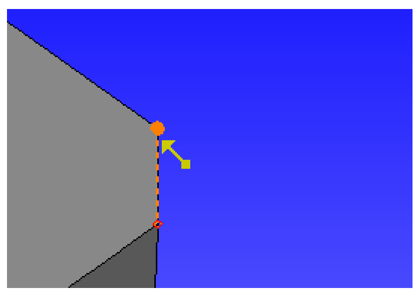


5 Click on the Point to Start the Profile.



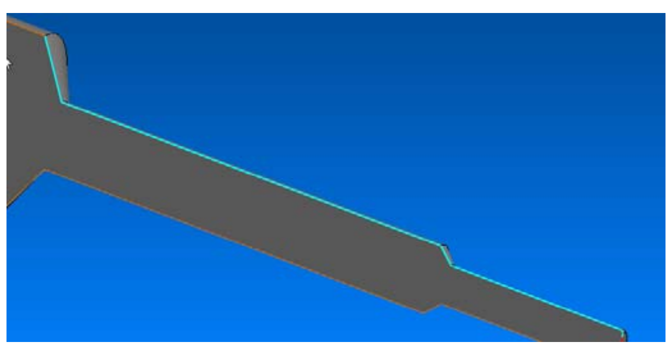
Starting Point at Center Using Edge Midpoint

6 Move the Mouse Curser 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.

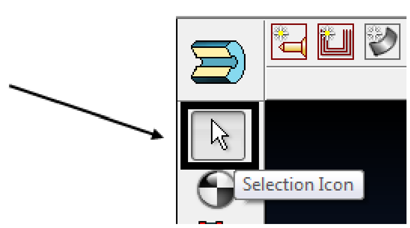


Next Point Select using Edge End Point

7 Continue selecting points for the Profile.



8 Click the **Selection** icon when done.

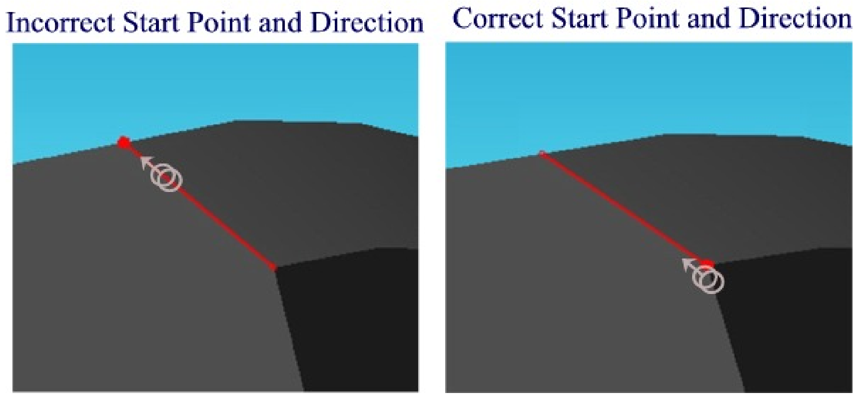


Selecting by Chain

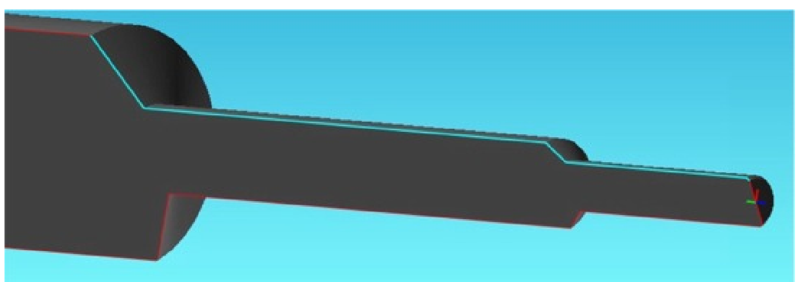
You may chain a profile on a Solid Model by selecting the **Chain Profile on Solid Model** icon.

To use this icon, you need to select the Starting Point and Direction.

Note: Depending on where you place the mouse curser, the starting point changes and the Direction Reverses.



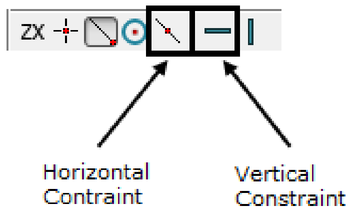
Select Endpoint of Chain. During Selection, a Dotted Line will indicate the Profile. Click the End Point to Finish Profile.



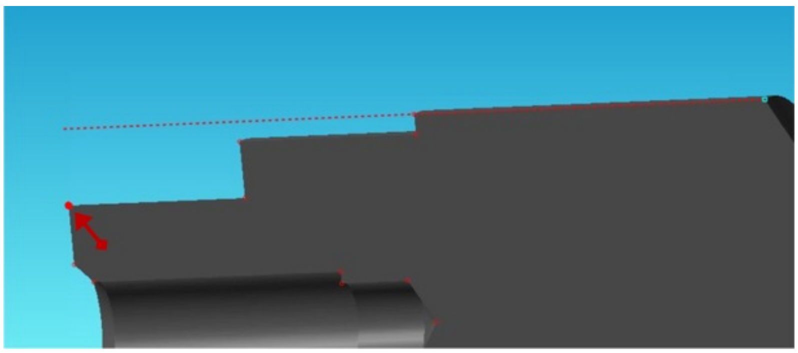
Using Horizontal and Vertical Constraints

You can use the Horizontal and Vertical Constraints to Extend Profiles that may not have a Point to choose from on the Model.

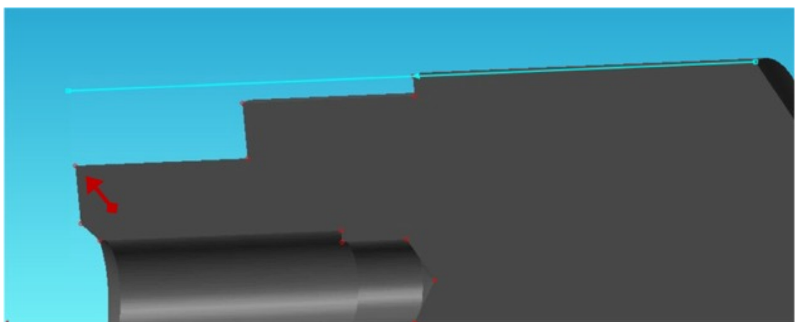
When selecting your Profile using the **Define Profile on Solid Model** icon, select either **Horizontal** or **Vertical Constraint** to lock the Profile in a Horizontal or Vertical move.



1. Move the mouse curser 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.



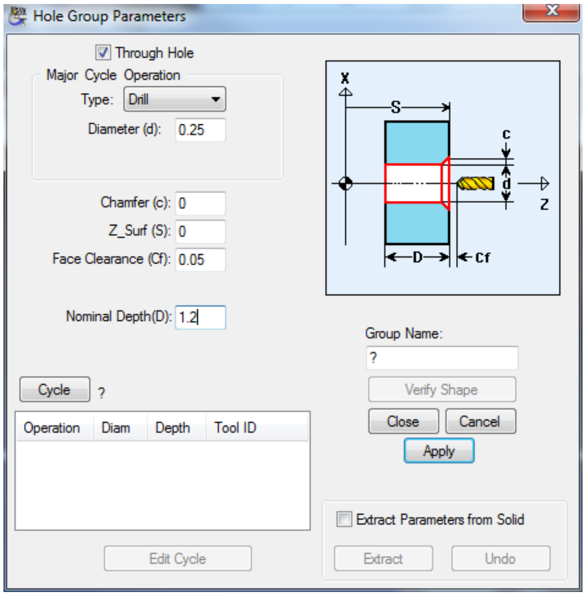
2. Click the Point to Extend the Profile to this Point.



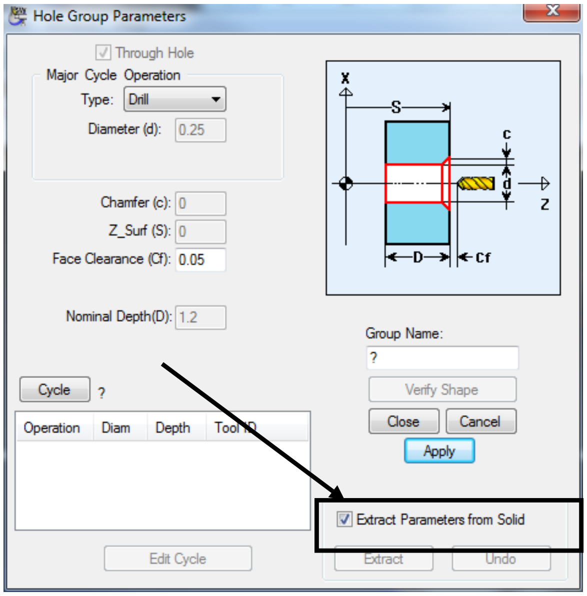
3. Depress the Horizontal or Vertical Constant icon to turn off the constraint.

Creating a Hole Group

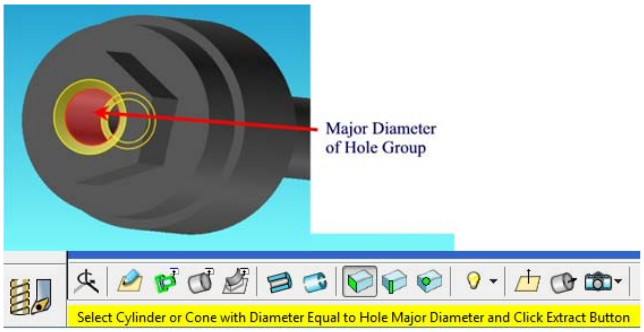
1 Create New Hole Group Parameters dialog



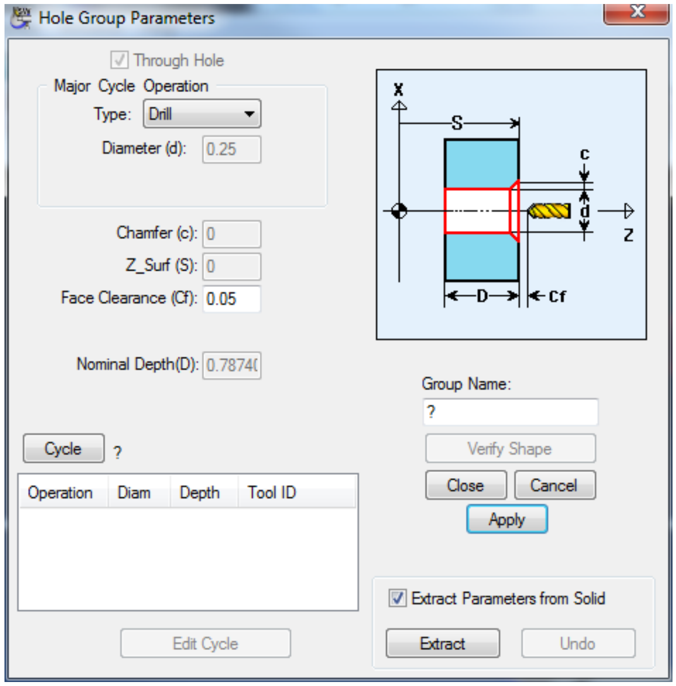
2 Check **Extract Parameters from Solid**.



3 Select Major Cylinder for Hole on Solid Model. (Note: See Explanation at Bottom of Screen)



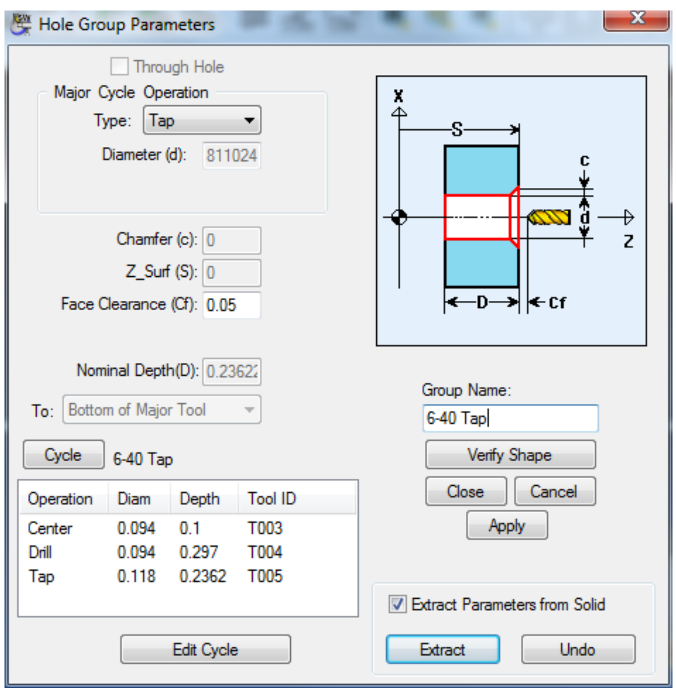
4 Click **<Extract>**



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>**

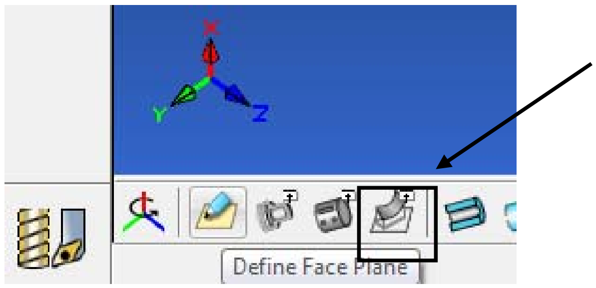
Milling Functions Face Window Programming

Describes Programming on a Solid Model in Milling Functions Face Windows. This applies to PartMaker SwissCAM and Turn-Mill.

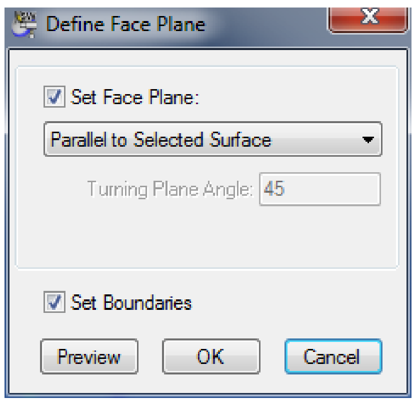
Creating a New Profile Group (Mill ZY window shown)

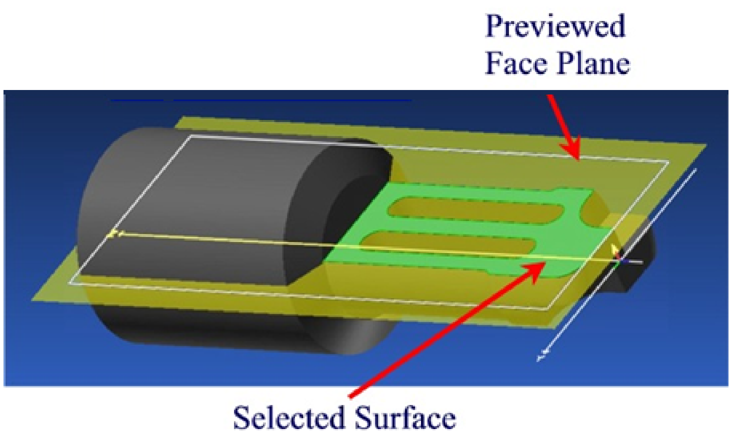
Setting the Face Plane

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.

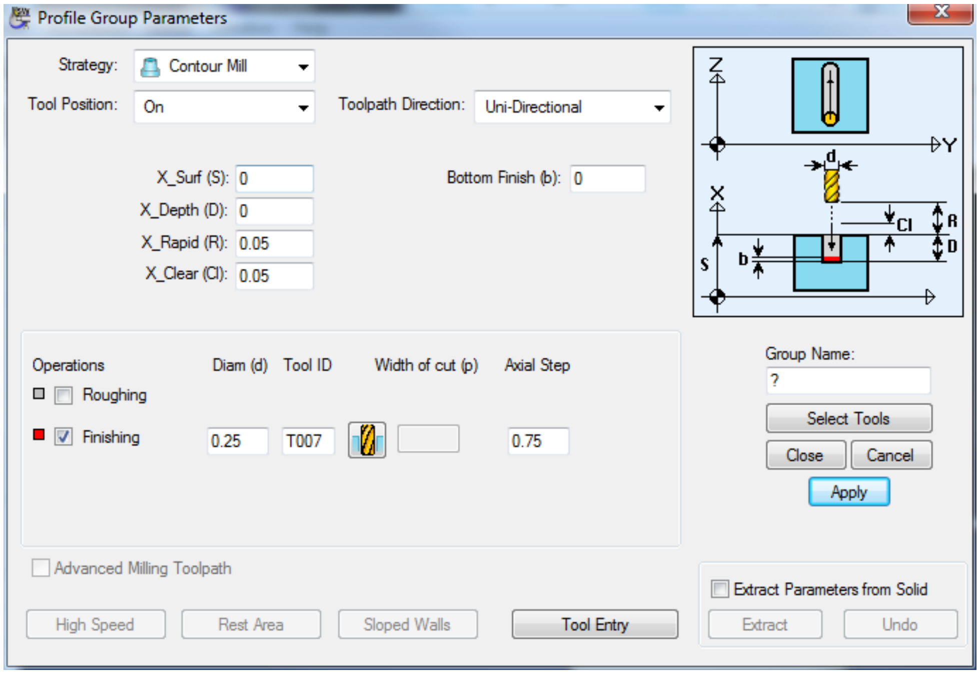




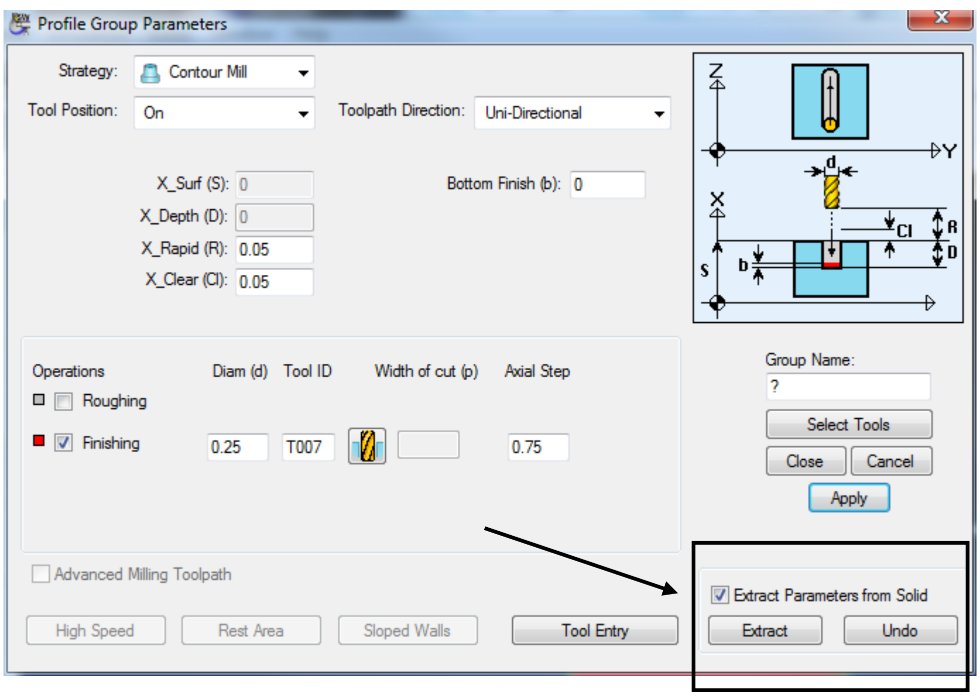
3 Click **<OK>** to accept Face Plane Setting. This will fill in the Set-up Window with the correct settings for the Face Plane.

Creating the Feature

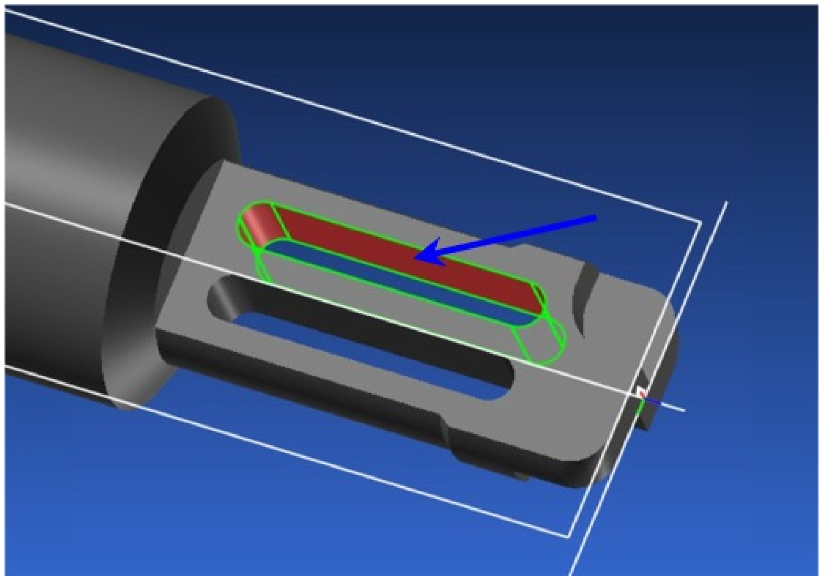
1 Create Profile Group as Normal



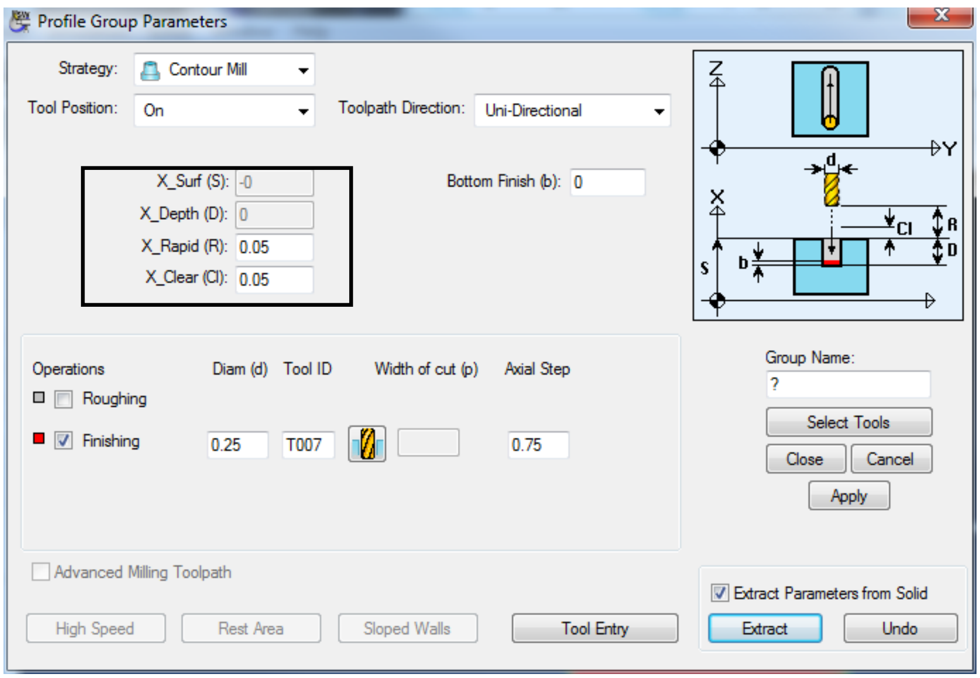
2 Check **Extract Parameters from Solid**



3 Select Feature to Extract Properties from.

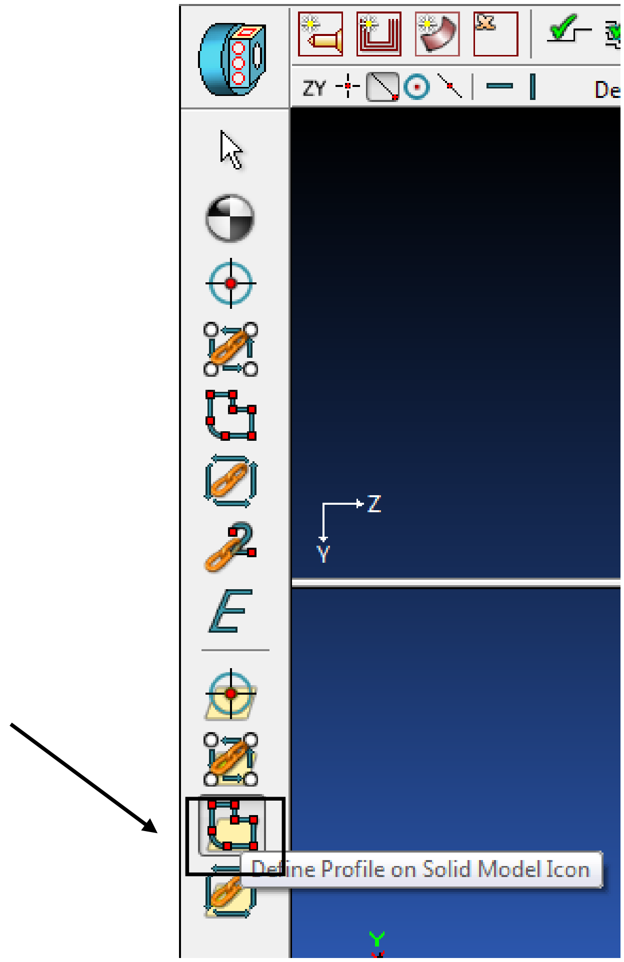


4 Click **<Extract>.** This will extract the information to the Profile Group.

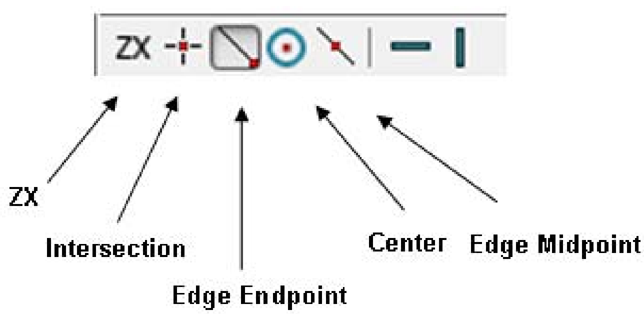


Creating the Profile

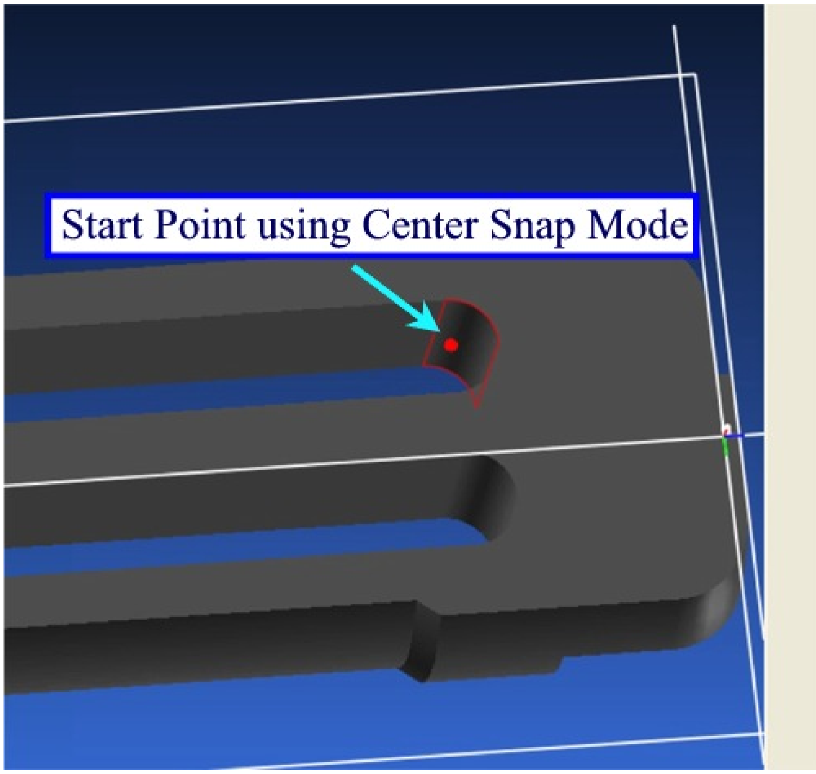
1 Choose the **Define Profile on Solid Model Icon**



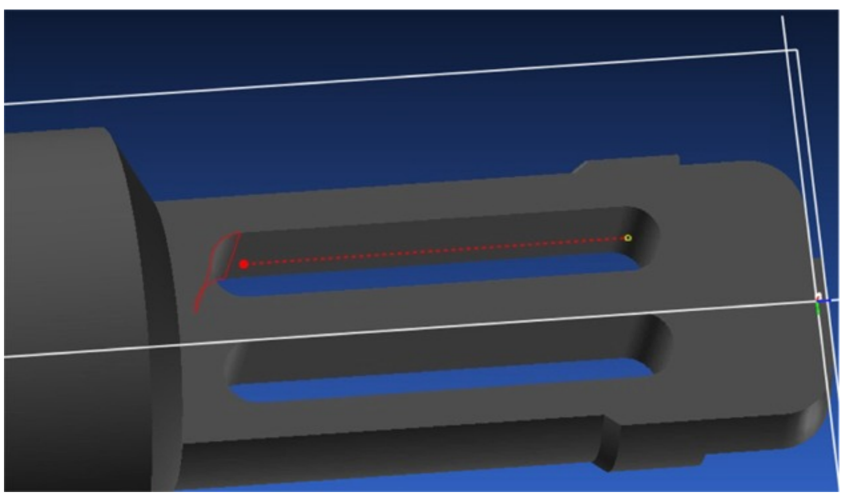
2 Use the Snap Modes to choose the Profile Points on the Solid Model.



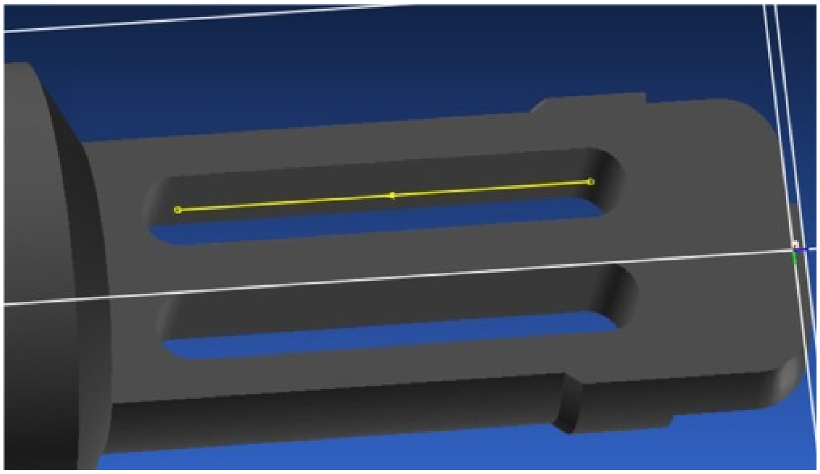
3 Click on the Cylinder Surface to select the center point as the Start of the Profile.



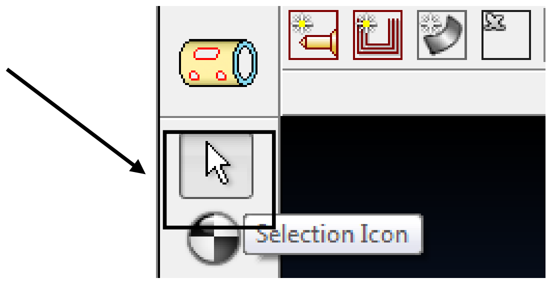
4 Move the Mouse Curser to the Next Point of the Profile. There will be a Dotted Line indicating Profile and a Dot where next Point will be. Click to connect the profile to this Point.



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 Curser, the Starting Point changes and the Direction Reverses.*

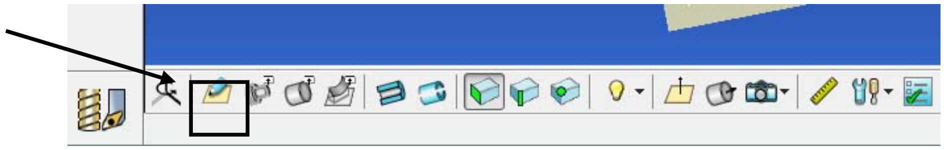
2 Select Endpoint of Chain. During Selection, a Dotted Line will indicate the Profile. Click the End Point to Finish Profile

3 You may use Horizontal and Vertical Constraints if required.

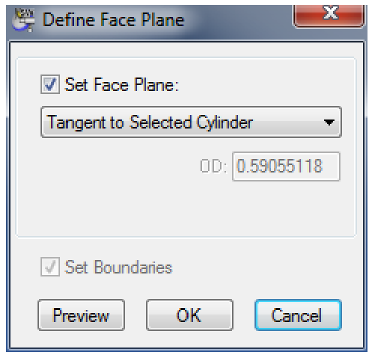
Creating a Hole Group (Mill Diameter Index Face window shown)

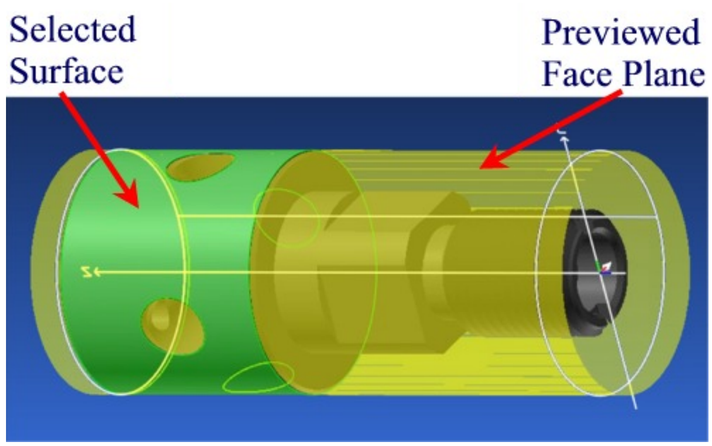
Setting the Face Plane

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.

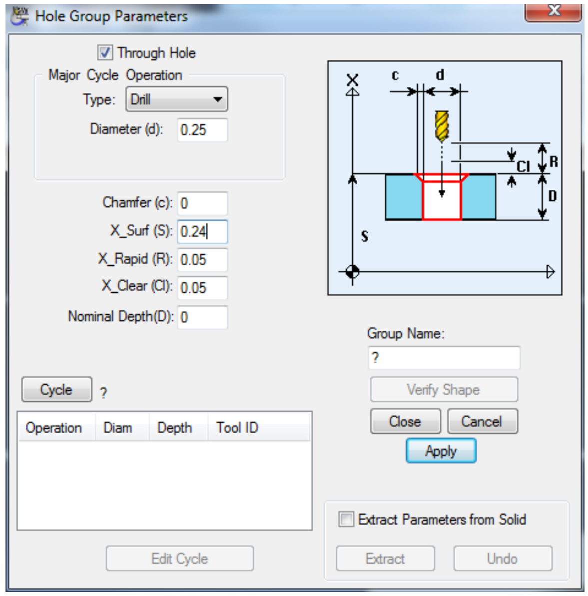




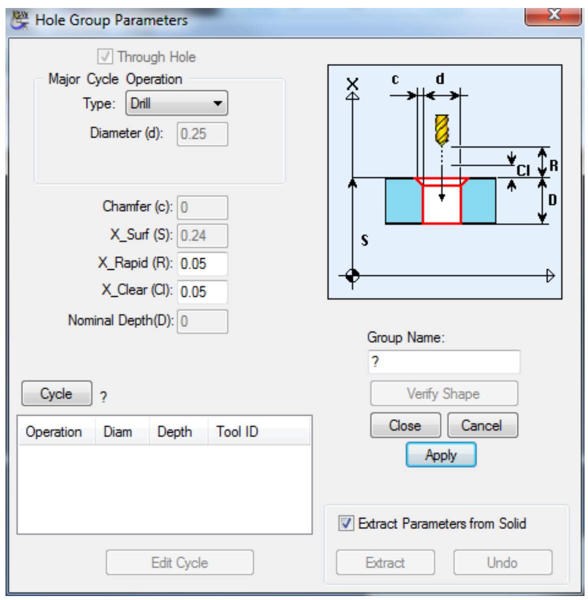
3 Click **<OK>** to accept Face Plane Setting. This will fill in the Set-up Window with the correct settings for the Face Plane.

Creating the Feature

1 Create New Hole Group Parameters.

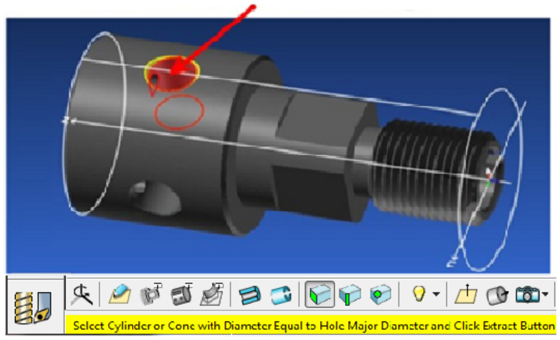


2 Check **Extract Parameters from Solid**

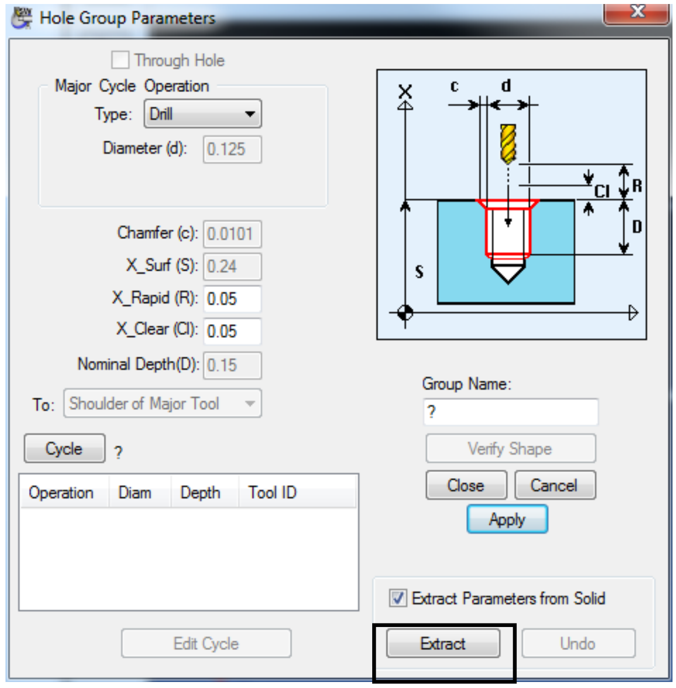


3 Select **Major Cylinder for Hole on Solid Model**. (Note: See Explanation at Bottom of Screen)

Major Diameter of Hole Group



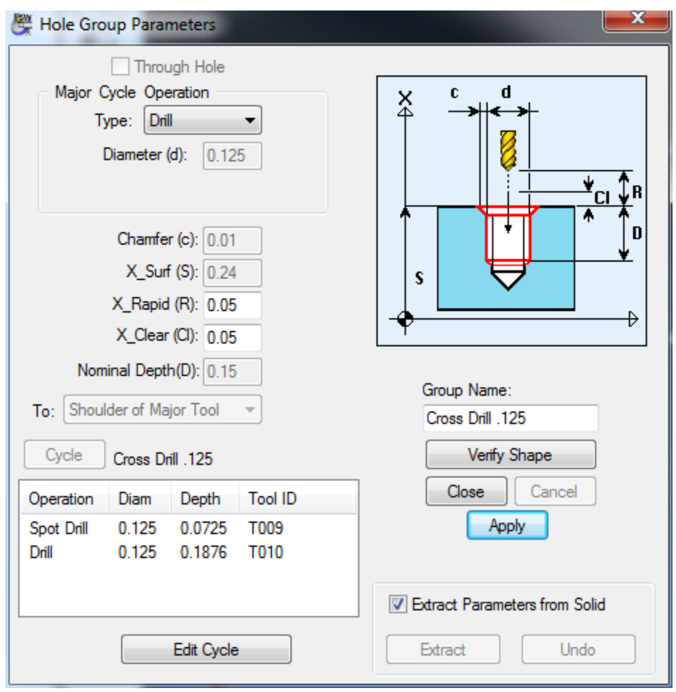
4 Click **<Extract>**



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

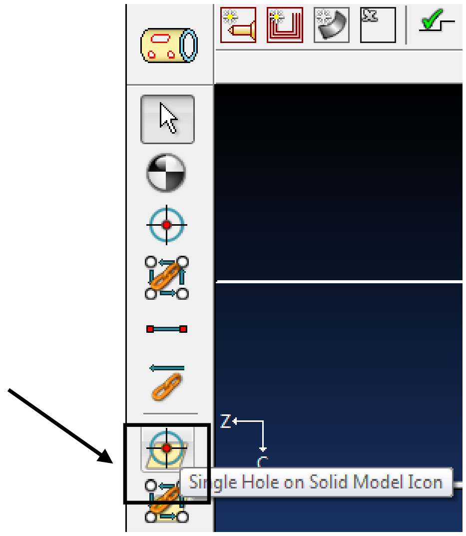
7 Click **<Cycle>** and select a cycle



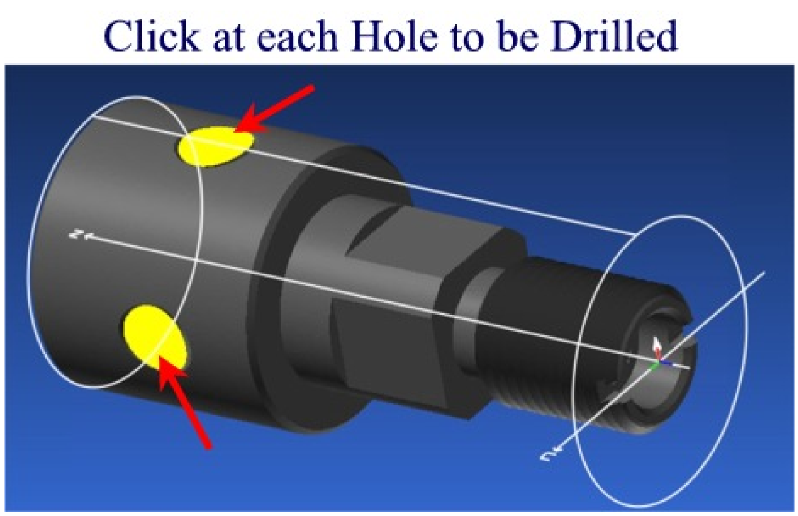
8 Click **<Apply>** then **<Close>**

Selecting Single Holes

1 Choose the **Single Hole on Solid Model Icon**.

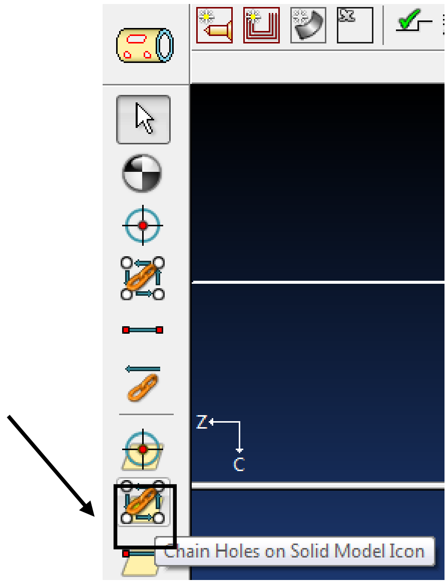


2 Select each hole that will be drilled.



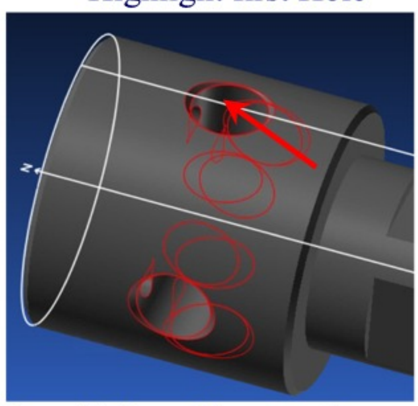
Selecting by Chain

1 Choose **Chain Holes on Solid Model icon**

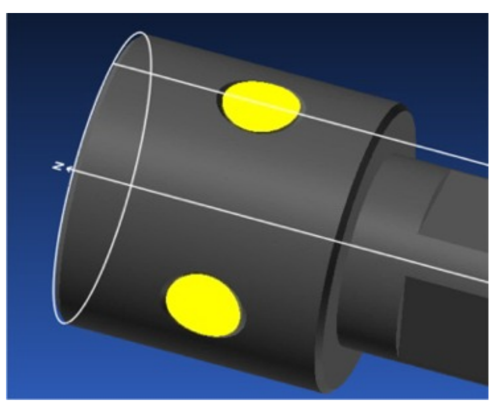


2 Select First Hole, PartMaker will then automatically select holes with the same feature in the Face Plane.

Highlight first Hole



Note: Holes with same features automatically become Highlighted



Appendix D: Practical Applications of Turn-Mill Machining Function Face Windows for Live Tooling Operations

Introduction

This appendix to the PartMaker Turn-Mill User Manual is intended to give you, the user, a more in-depth and practical understanding of applying PartMaker's Machining Function Windows for use with Live Tooling Operations. This Appendix is meant to supplement the information provided in Chapter 2 of your PartMaker Turn-Mill Programming Manual titled "PartMaker Fundamentals."

This chapter will explain the following:

- What types of Part Features Should be created in each Machining Function Face Window
- Which Hole Group and Profile Groups Parameters options are available in each Machining Function Face Window
- Which Graphics Icons are available in each Machining Function Face Window
- How Part Features should be created in each Machining Function Face Window

Many of the part examples provided in the pages to follow can be found in the various sub-directories labeled with machine names in the Sample Directory found in the Pm-Swiss directory in which PartMaker Turn-Mill is installed. The file name and path for each file and specific *.job, *.tdb and *.cdb are provided where relevant for each example.

Machining Function Selection

Turn-Mill Machines are capable of performing a full set of turning operations: facing, turning, grooving and threading. Many Turn-Mills are also capable of performing various types of milling as well as drilling holes on different surfaces. Multiple spindles in which such operations can be performed simultaneously further increase the machines' productivity and complexity.

Milling capabilities of a Turn-Mill vary significantly and can include:

- Index milling/drilling
- Milling on a selected ZX plane
- Polar milling on the face of the part
- XZC continuous milling on cylindrical surfaces
- Milling interpolation in the YZ plane

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.

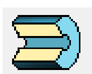
A specific machining function such as turning, polar milling or cylindrical milling is assigned to each face. **PartMaker** lets you create a dedicated window that contains a workspace for the graphic representation of face features.

A dialog box associated with each **Face Window** shows you the type of machining to be performed on a face, the location of the face boundaries and parameters defining the positioning of the face relative to the machine coordinate system.

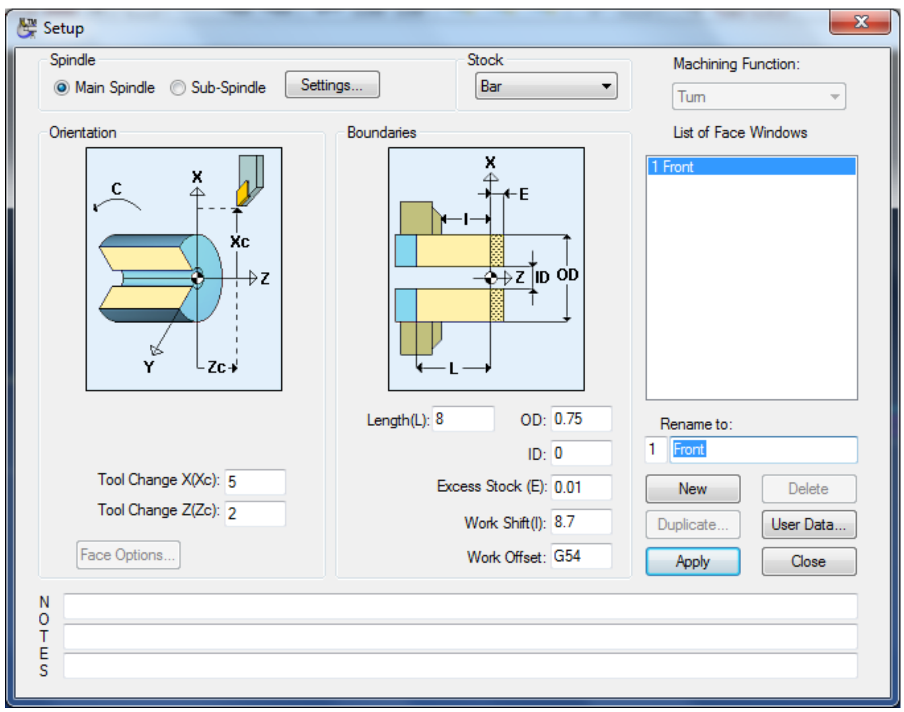
Getting Started: The Setup Dialog

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.

Once divided into much smaller, similar motion components, programming of a multi-axis Swiss-type lathe becomes a much easier task. To best explain how **PartMaker** handles this task, it is necessary to first explain how each of these individual-machining functions is defined.



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:



1 The Main Spindle and Sub Spindle radio buttons allow you to select whether the programming carried out in a particular **Face Window** is being machined on the Main or Sub-Spindle.

2 The Tool Change X and Z positions for a particular machine should be entered here.

3 The Machining Function drop down menu allows you to select the type of machining motion being executed in a particular **Face Window**. The machining motions that can be defined in each selection from the Machining Function drop down menu are explained below.

4 The List of **Face Windows** displays the list of **Face Windows** currently existing for a given job.

5 The Rename To: field allows you to name the **Face Window** you are working with. Existing **Face Windows** can be selected from the Window menu based on the names assigned to them in this field of the **Setup** dialog.

6 Clicking the function buttons shown in the **Setup** dialog with the left-hand mouse button does the following:

<Apply>	Accepts or applies any new parameters or data entered into the various fields in the Setup dialog, and the dialog remains open.
<New>	Creates new face windows of different machining function types or on different spindles, adding to the List of Face Windows
<Close>	Accepts or applies any new parameters or data entered into the various fields in the Setup dialog, and closes the dialog.
<Delete>	Deletes the current selection from the List of Face Windows

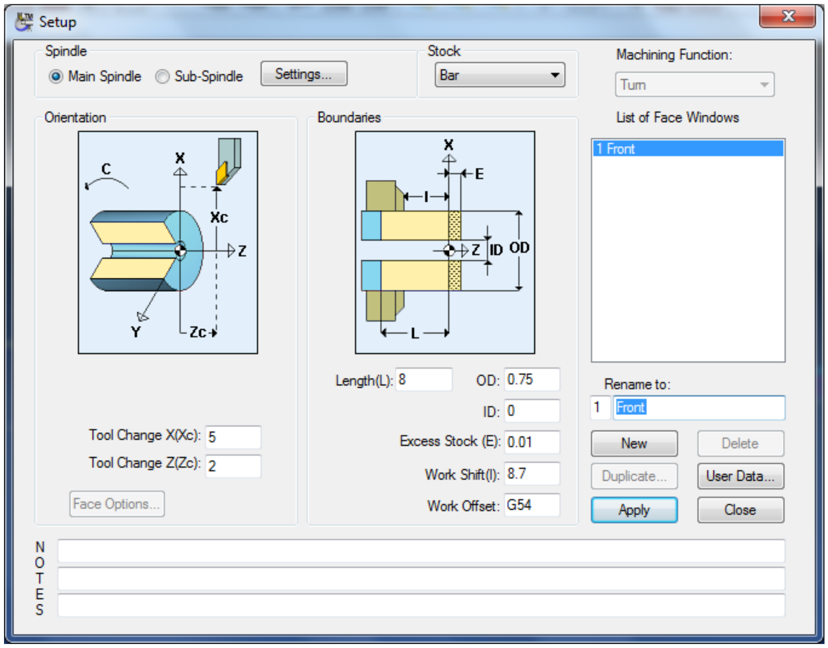
The various selections under the Machining Function drop down menu are explained on the following pages.

7 The Notes fields allow the user to enter notes about a specific job. These notes can be directly output into the NC program to improve documentation.

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.

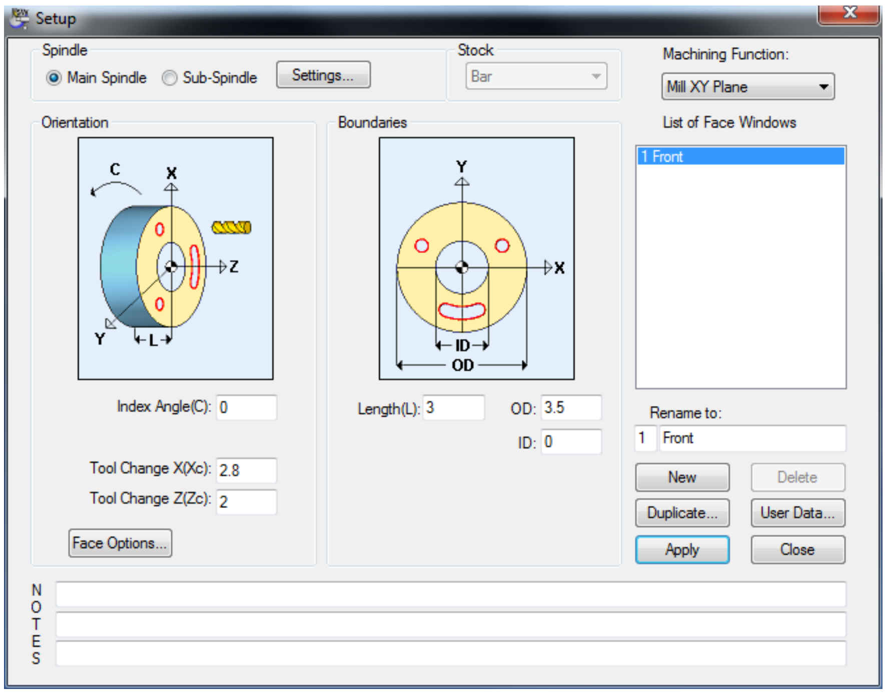


Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining
Excess Stock (E)	Rough stock left for facing on the front of the part
Work Shift (I)	The amount the material is exposed from the chuck face
Work Offset	The work offset assignment for the spindle

Machining Function: Mill XY Plane

The Mill XY Plane machining function supports machining operations on the face of the part using the Y-axis capability of the machine. G-code output will be generated in the form of X and Y coordinates. Features that can be created by using this Face Window include off-center holes on the face of the part, pockets on the face of the part, and contours (such as CAM shapes) on the face of the part.

Programming of this machining function requires that your Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



Index Angle (C)	Orientation angle of the stock to achieve the desired face position
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

Types of Features Created in Mill XY Machining Function Window

The Mill XY window uses a Z-oriented “live” tool. There is no polar C-axis movement (the C-axis indexes and locks).

***Note:** Your machine must have Y-axis capability and also needs to have ample travel to machine in the negative quadrant of the part.*

The features shown on the parts below are representative examples of features that can be created in the Mill XY Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found in the Samples directory which was installed when you initially installed PartMaker Turn-Mill. Where relevant, the locations of these files have been provided.

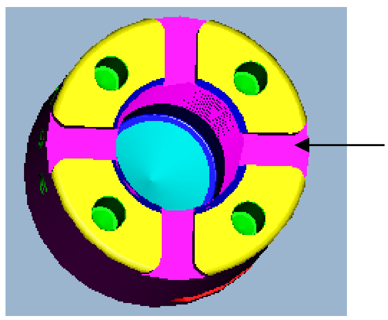


Figure 1: Milled Slots

(Relevant files: C:\PartMaker\Pm-TM\Samples\Hardinge_TT-65\ TT-65.job, TT-65.cdb, TT-65.tdb)

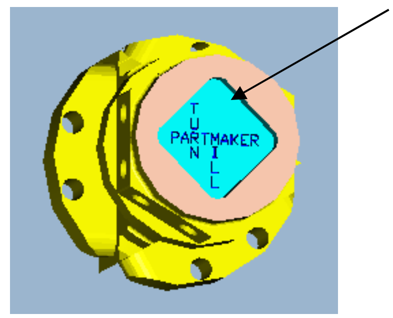


Figure 2.: Milled Pocket and Engraving

(Relevant files: C:\PartMaker\Pm-TM\Samples\Hitachi-Seiki_HiCell\HiCell.job, HiCell.cdb, HiCell.tdb)

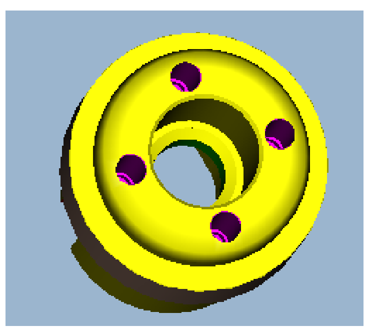


Figure 3: Off-Center Holes

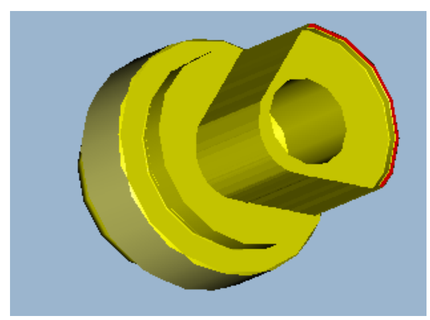
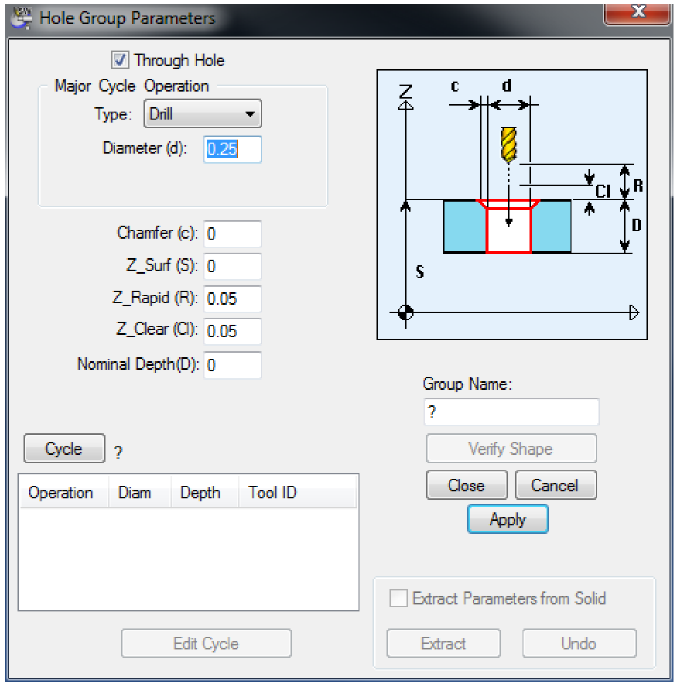


Figure 4: Milled Cam

Part Features Dialogs Available in Mill XY Machining Function

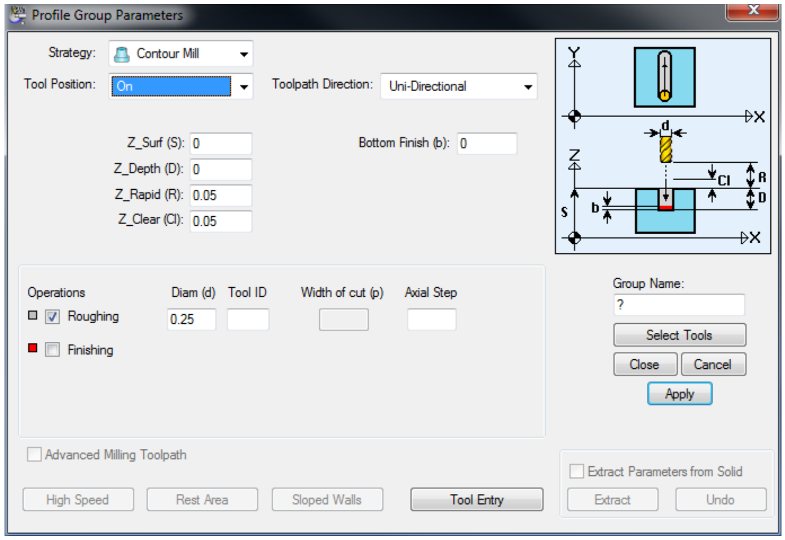
Both the **New Hole Group** and **New Profile Group** commands from the **Part Features** menu can be used in the Mill XY Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in Z.



Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycles database
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
Z_Surf (S)	Establishes your Z-zero work plane
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf
Z_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Nominal Depth (D)	The depth of the operation to be performed

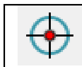
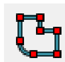
The Mill XY machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with **Tool Position** On centerline is shown. To create a pocket, choose **Pocket Mill** from Strategy drop down menu.



Strategy	Indicates the type of feature to be milled, either a **Contour Mill** or **Pocket Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling)
Z_Surf (S)	Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part
Z_Depth (D)	The depth of the operation to be performed
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf.
Z_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove when tool position set to Left or Right {Not shown above}
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Tool Entry	Indicates the manner of entry of end mill into a feature, either **Vertical, Helical, Sloped** or **Sloped Down and Return**

Graphics Icons Available in the Mill XY Plane Machining Function

The Mill XY Plane Machining Function Face Window uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

	**Single Hole Icon** enables you to use Snap Modes to define the location on the geometry where a currently active Hole Group Symbol is to be executed.
	**Chain Circle Icon** enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.
	**Define 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 Geometry 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 with an initial line or arc clicked.
	**2-Point Chain Icon** enables you to specify a begin point and an end point to be chained together.
	**Engrave Icon** enables you to use the Engrave Data dialog box to create a profile containing numbers and letters.

Programming Example using Mill XY Plane Machining Function

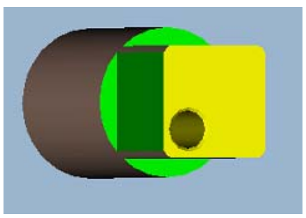
Square and Off-Center Hole on Face

A practical programming application of the Mill XY Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a ½ Inch Square with a ¼ inch End Mill and create a .1 diameter off-center hole. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill XY Plane Face Window should have an orientation of **Z Tool**)

- .25 Inch Diameter End Mill
- .1 Inch Diameter Drill
- Spot Drill with a diameter larger than .1 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .1 diameter drill and a spot drill.



Features to be created:

1. *Mill a .5 inch square with a .25 dia end mill*
2. *Spot Drill and Drill an off center hole with a .1 diameter drill*

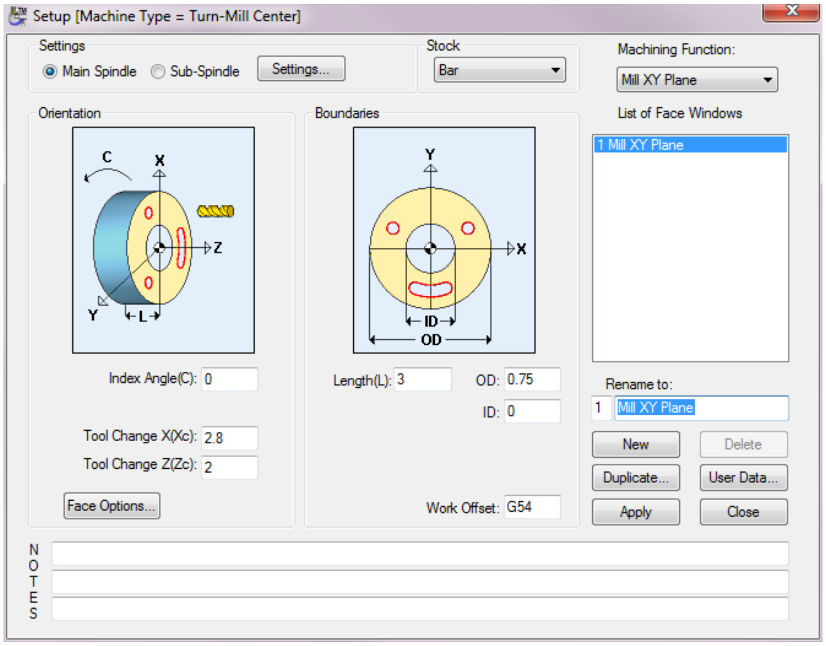
Step 1. Create a Mill XY Plane Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill XY Plane Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill XY Plane**

4 Enter the setup parameters as shown below:



5 Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill XY Plane Window. Draw the following features:

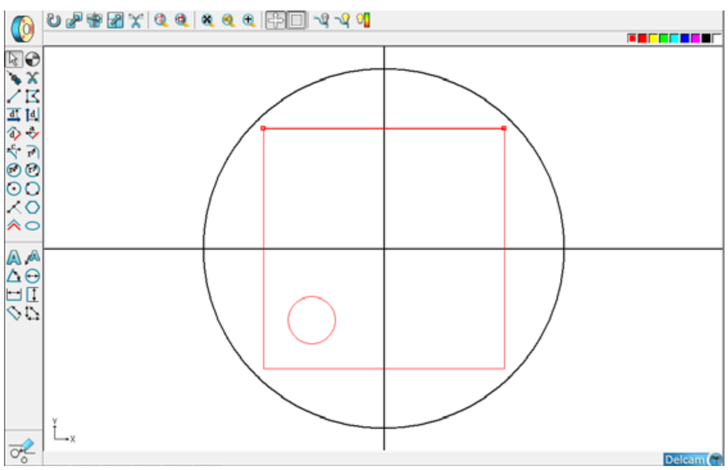
1 Choose **Show Axes** from the View Menu

2 Choose **Show Boundaries** from the View Menu

3 Draw a ½ inch square

4 Draw a .05 inch radius circle with a center location at X<-.15> and Y<-.15>

Your completed geometry and CAD Face Window should appear as shown below:



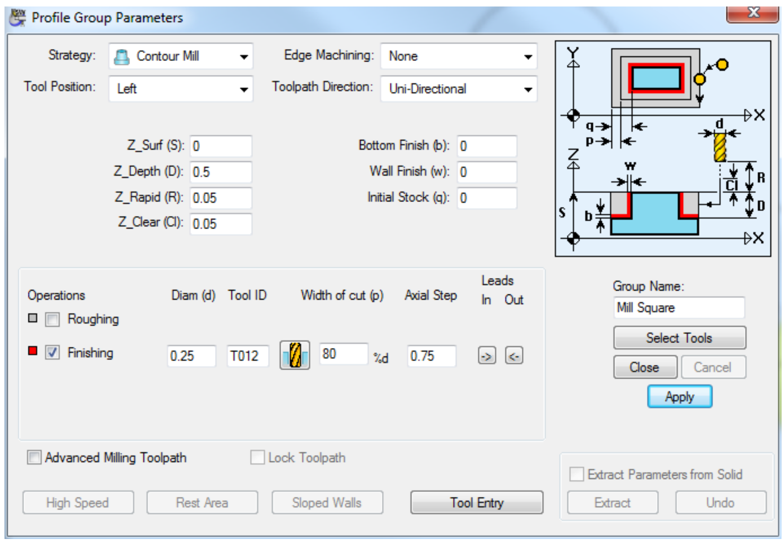
Note: *You can always go to a different Face Window in your *.job file by selecting from the* ***Window*** *menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing* ***Show Windows Icons*** *from the* ***View*** *menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

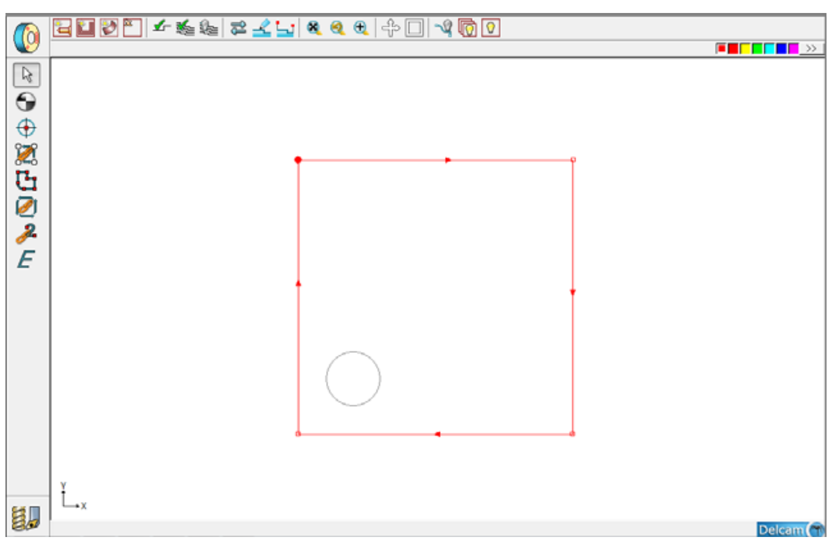
Start by entering the CAM mode in the Mill XY Plane Window. First you will create the tool path to mill the square. To do so:

1. From the Part Features menu, choose New Profile Group
2. Choose Contour Mill under Strategy
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter a **Z_Depth** of .5
5. Enter a Tool Diameter of .25 and click <Select Tools>
6. In the Group Name field, type **Mill Square**

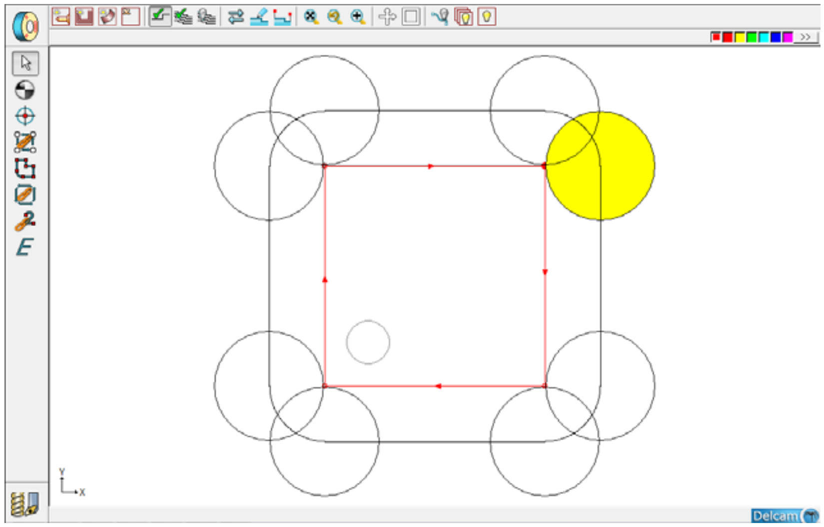
   Your completed **Profile Group Parameters** dialog should appear as shown below:



7. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



8 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:



9 Choose **Hide Work Tool Path** to hide the tool path.

Next, you will create the off-center hole. To do so:

1 From the Part Features menu, choose New Hole Group

2 Uncheck the **Through Hole** checkbox

3 Choose a Major Cycle Operation of **Type: Drill**

4 Enter a Major Cycle Operation **Diameter** of .1

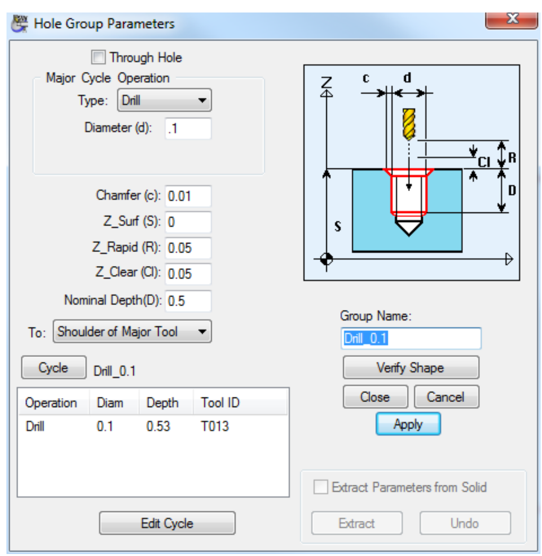
5 Enter a **Chamfer (c)** of .01

6 Enter a **Nominal Depth (D)** of 0.5

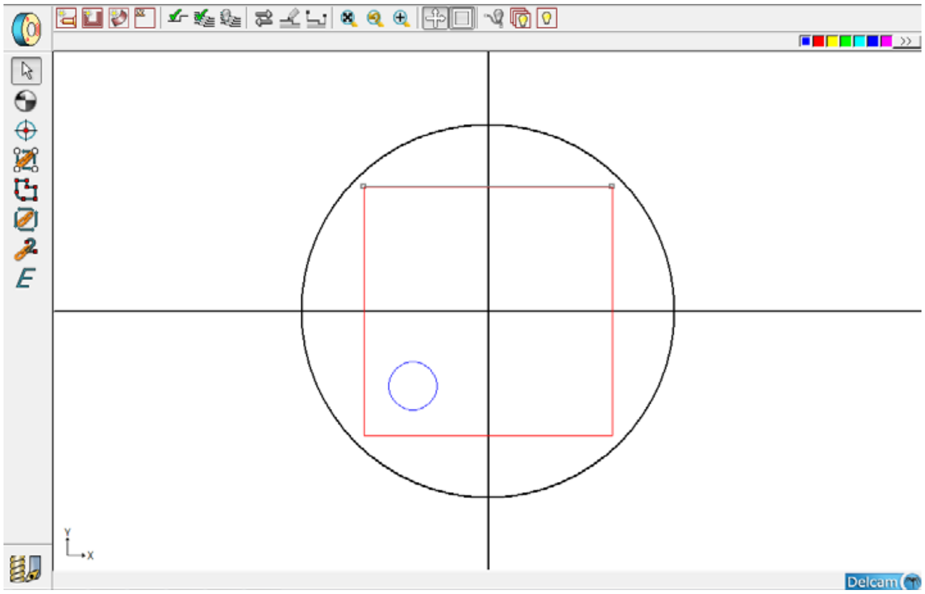
7 Click the **<Cycle>** button.

8 In the Group Name field, type **Drill_0.1**

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



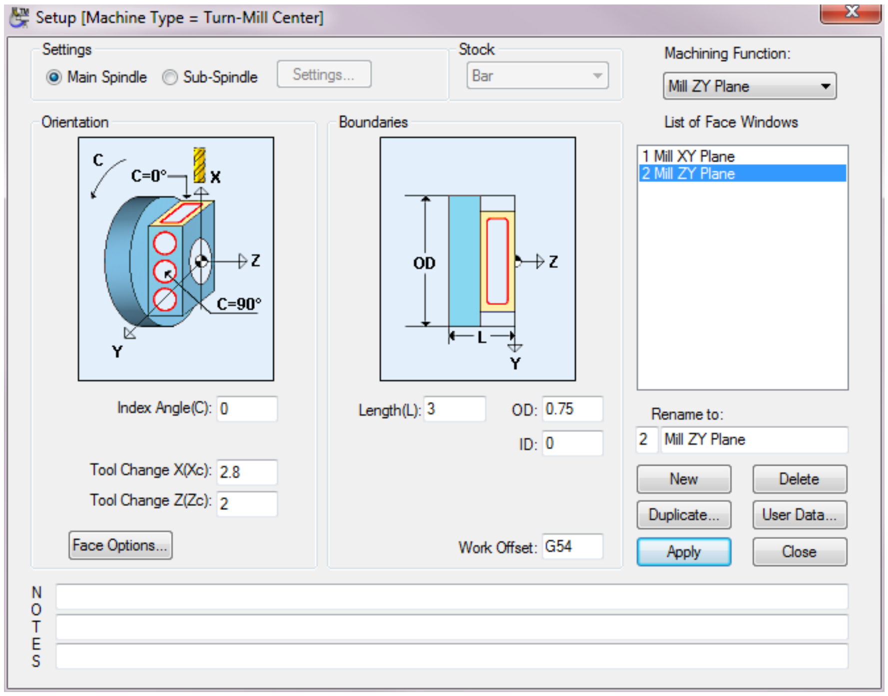
Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circle representing the hole to create the tool path. It should appear as shown below:



Machining Function: Mill ZY Plane

The Mill ZY Plane machining function is used to perform milling interpolation in the ZY plane when the spindle is locked at a discreet angular position. All motions are performed with a tool oriented along the machines X-axis.

Programming of this machining function requires that your Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



Index Angle (C) Orientation angle of the stock to achieve the desired face position

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

Types of Features Created in Mill ZY Machining Function Window

The Mill ZY window uses an X-oriented “live” tool. There is no polar C-axis movement (the C-axis indexes and locks). This window can be used for creating off-center cross-holes, flats on the diameter of the part and interpolated features. Saw Slots should also be created in this Face Window.

The features shown on the parts below are representative examples of features that can be created in the Mill ZY Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker Turn-Mill. Where relevant, the locations of these files have been provided.

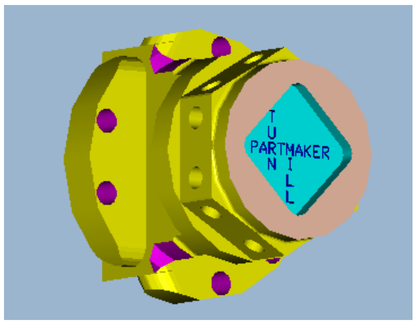


Figure 1.: Milled Flats and Cross Holes

(Relevant files: C:\PartMaker\Pm-TM\Samples\Hitachi-Seiki_HiCell\HiCell.job, HiCell.cdb, HiCell.tdb)

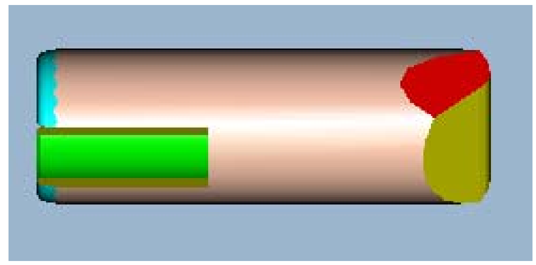


Figure 2.: Milled Radius ,Flats and Slot

(Relevant files: C:\PartMaker\Pm-TM\Samples\Mazak_SQT\SQT-MSY.job, SQT-MSY.cdb, SQT-MSY.tdb)

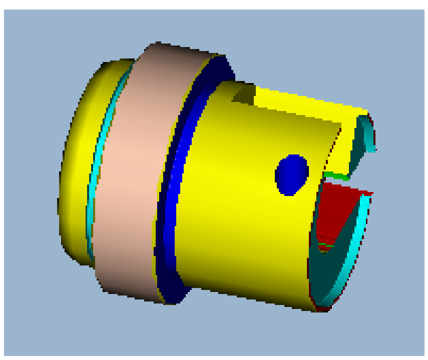


Figure 3: Off-Center Cross Hole

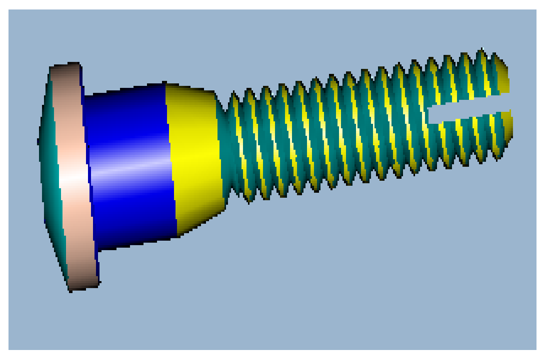
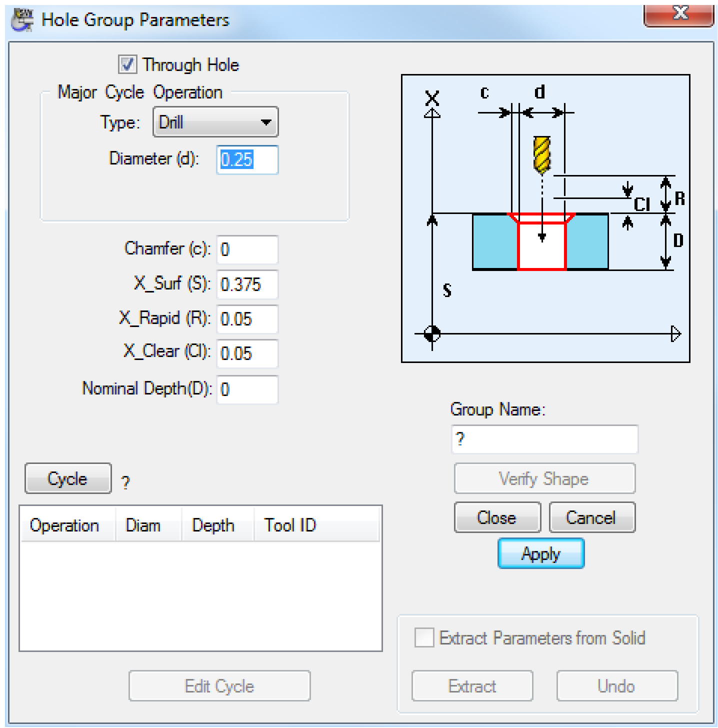


Figure 4: Milled slot

Part Features Dialogs Available in Mill ZY Machining Function

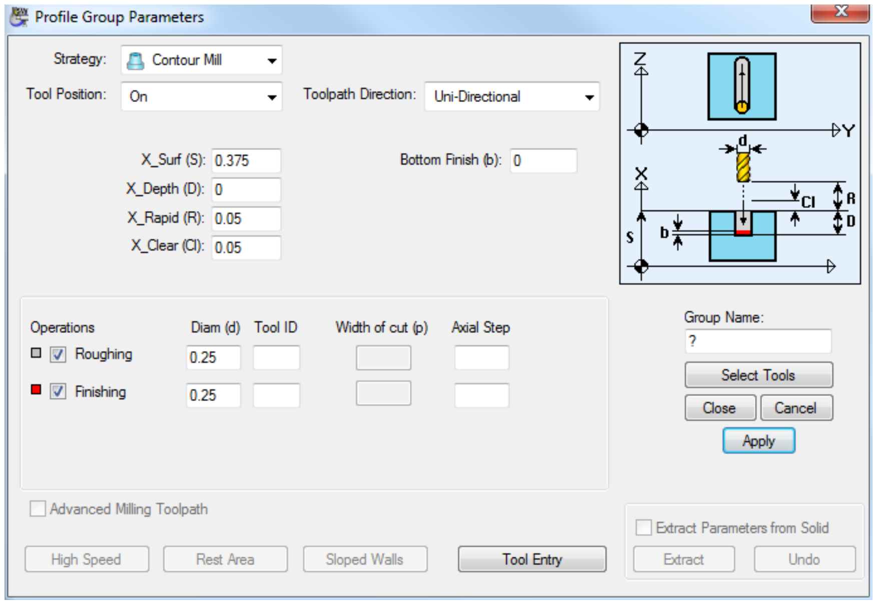
Both the **New Hole Group** and **New Profile Group** commands from the Part Features menu can be used in the Mill ZY Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in X.



Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycles database
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
X_Surf (S)	Establishes your X-zero work plane on the OD of the part
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf
X_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Nominal Depth (D)	The depth of the operation to be performed

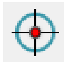
The Mill ZY machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.



Strategy	Indicates the type of feature to be milled, either a Contour Mill or Pocket Mill
Tool Position	For Contour Mill indicates the tool position during cutting, such as Left (climb milling), Right (conventional milling) or On (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above of your X_Surf.
X_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove when tool position set to Left or Right {Not shown above}
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Tool Entry	Indicates the manner of entry of end mill into a feature, either Vertical, Helical, Sloped or Sloped Down and Return

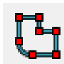
Graphics Icons – Machining Functions for Mill ZY Plane

The Machining Functions Mill ZY Plane use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.



Single Hole Icon enables you to use Snap Modes to define the location on the geometry where a currently active Hole Group Symbol is to be executed.

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.



Define 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 Geometry 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 with an initial line or arc clicked.

2-Point Chain Icon enables you to specify a begin point and an end point to be chained together.

Engrave Icon enables you to use the Engrave Data dialog box to create a profile containing numbers and letters.

Programming Examples using Mill ZY Plane Machining Function

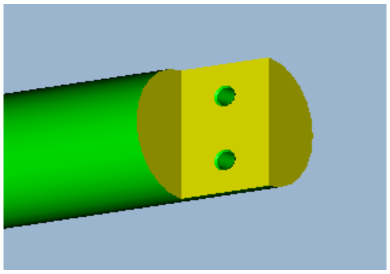
Practical programming applications of the Mill ZY Plane machining function have been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

Example A: Creating a single flat with 2 off-center holes

In the example given below, you will mill a flat with a ½ inch End Mill and create two .1 diameter off-center holes. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZY Plane Face Window should have an orientation of **X Tool**)

- .500 Inch Diameter End Mill
- .1 Diameter Drill
- Spot Drill with a diameter larger than .1 inch

Also make sure you have a cycle in PartMaker's Cycles Database created to include both a .1-diameter drill and a spot drill.



Features to be created:

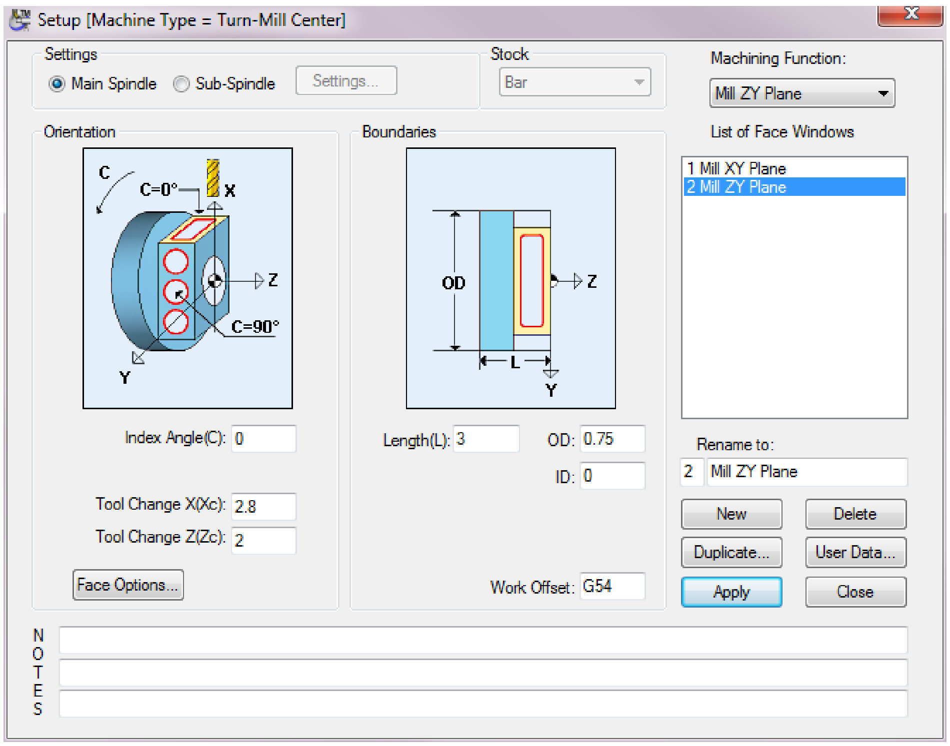
1. *Mill a flat with a .500 dia end mill*
2. *Spot Drill and Drill two off center holes with a .1 diameter drill*

Step 1. Create a Mill ZY Plane Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill ZY Plane Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill ZY Plane**
4. Enter the setup parameters as shown below:



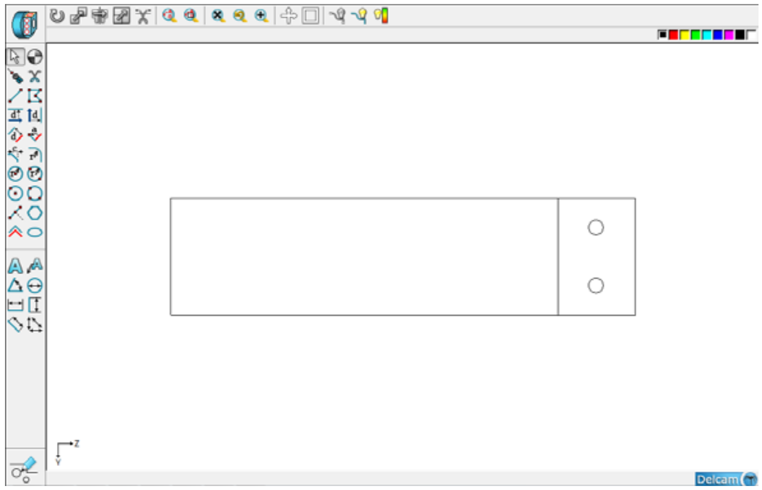
5. Once the **Setup** dialog appears as above, click on **Close.**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZY Plane Window. Draw the following features:

1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw one line at Z<-0.5> to represent the end of the flat.
4. Draw two .05-inch radius circles with a center location at Z<-0 25> and Y<-0.1875> and at Z<-0 25> and Y <0.1875>.

Your completed geometry and CAD Face Window should appear as shown below:



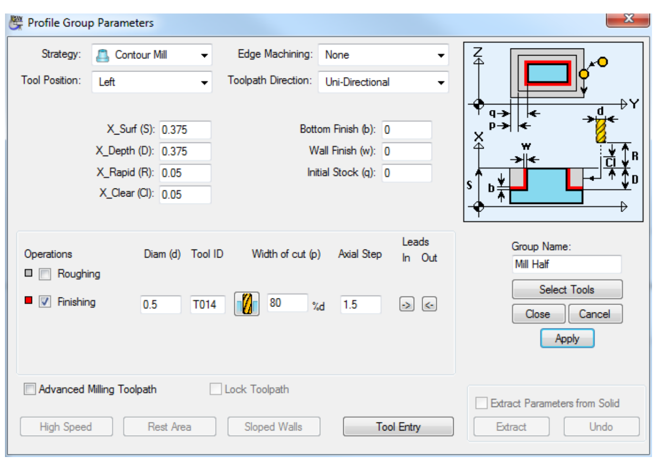
Note: *You can always go to a different Face Window in your *.job file by selecting from the* ***Window*** *menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing* ***Show Windows Icons*** *from the* ***View*** *menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and Assign Tool Paths

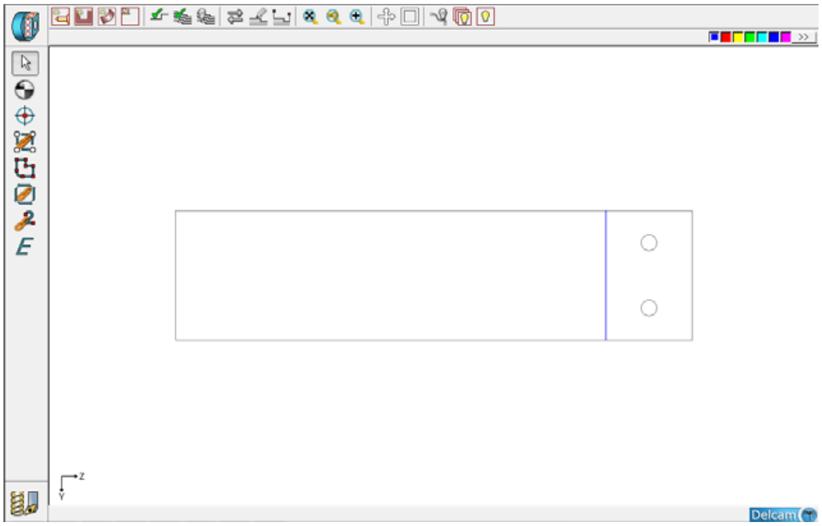
Start by entering the CAM mode in the Mill ZY Plane Window. First you will create the tool path to mill the flat. To do so:

1. From the Part Features menu, choose New Profile Group
2. Choose Contour Mill under Strategy
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter an X_Surf of .375
5. Enter an X_Depth of .375 (we want the tool to go to center)
6. Enter a Tool Diameter of .5 and click on **Select Tools**.
7. In the **Group Name** field, type **Mill Half.**

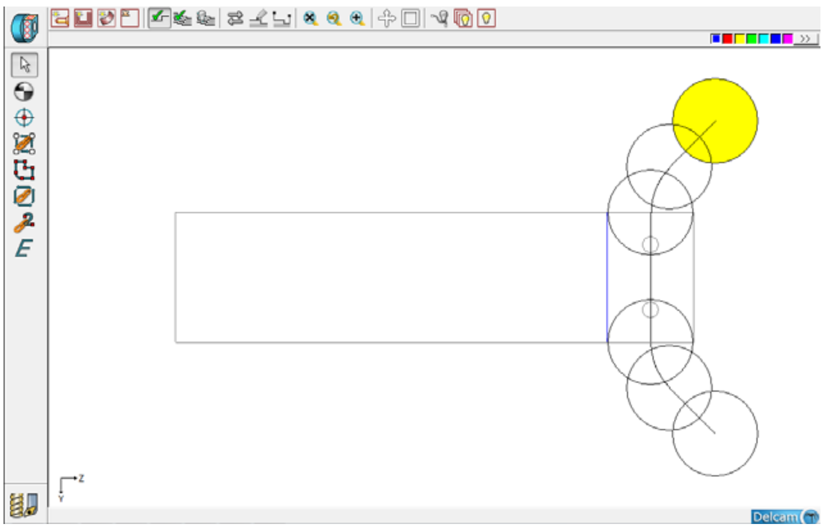
Your completed **Profile Group Parameters** dialog should appear as shown below:



8 Assign the part feature to the flat drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



9 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:



10 Choose **Hide Work Tool Path** to hide the tool path.

Next, you will create the off-center holes. To do so:

1 From the Part Features menu, choose New Hole Group

2 Uncheck the **Through Hole** Box

3 Choose a Major Cycle Operation of **Type: Drill**

4 Enter a Major Cycle Operation **Diameter** of .1

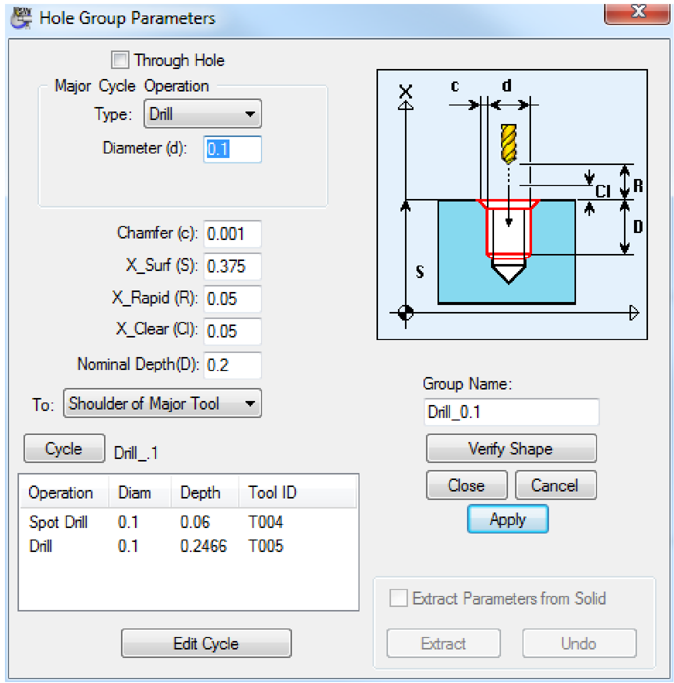
5 Enter a **Chamfer** of .01

6 Enter a Nominal Depth (D) of 0.2

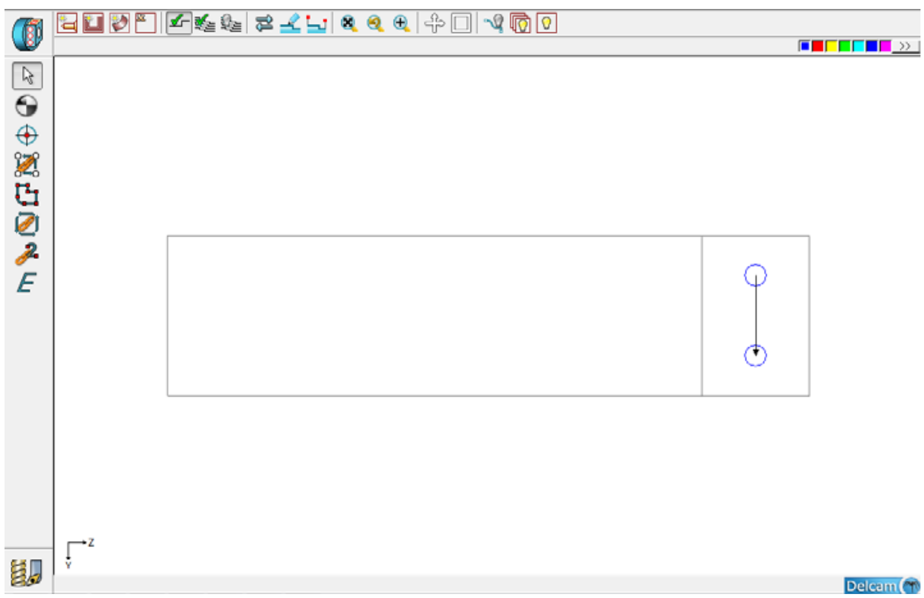
7 Click the **Cycle** button.

8 In the **Group Name** field, type **Drill_0.1**

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



Assign the part feature to the circles by clicking the **Chain Circle** icon and clicking on the circle representing the holes to create the tool path. It should appear as shown below:

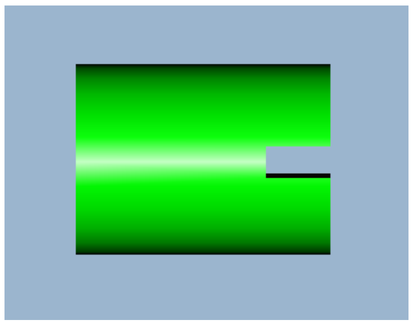


Example B: Creating a Saw Slot

In the example given below, you will mill a .050 wide slot with a 1-inch Diameter Slot Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZY Plane Window should have an orientation of **X Tool**)

***Note**: You will have needed to purchase the advanced tools option to create a Slotting Saw in your tools directory.*

- 1 Inch Diameter Slot Mill



Features to be created:

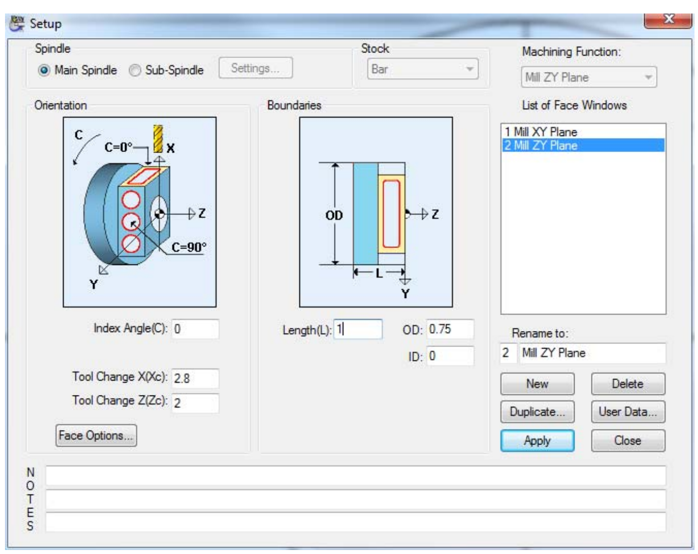
Mill a .125 thick slot x ¼ deep along the face of the part.

Step 1. Create a Mill ZY Plane Machining Function Face Window

***Note:** When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill ZY Plane Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill ZY Plane**
4. Enter the setup parameters as shown below:



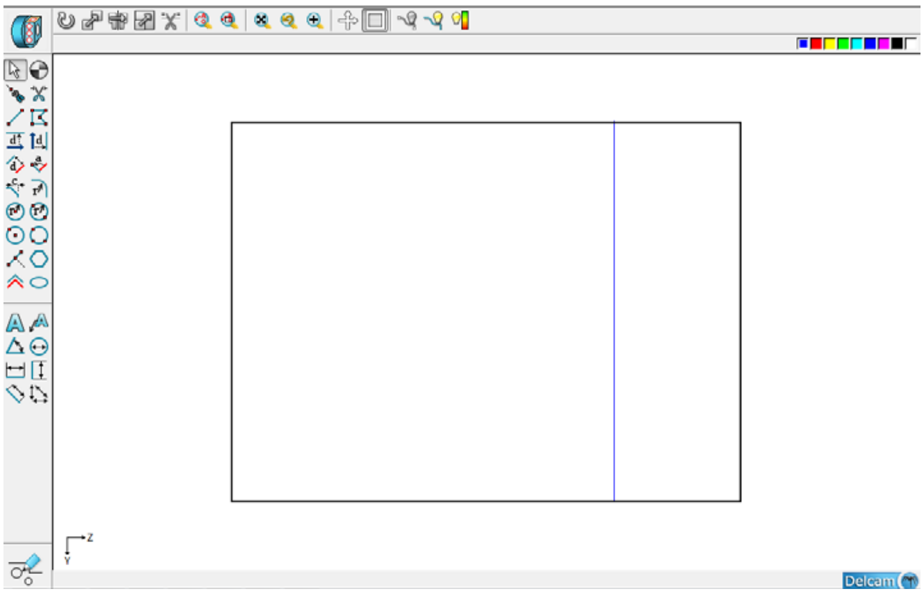
5. Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZY Plane Window. Draw the following features:

1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a line that represents the .250 depth in Z for the end of the mill slot.

   Your completed geometry and CAD Face Window should appear as shown below:



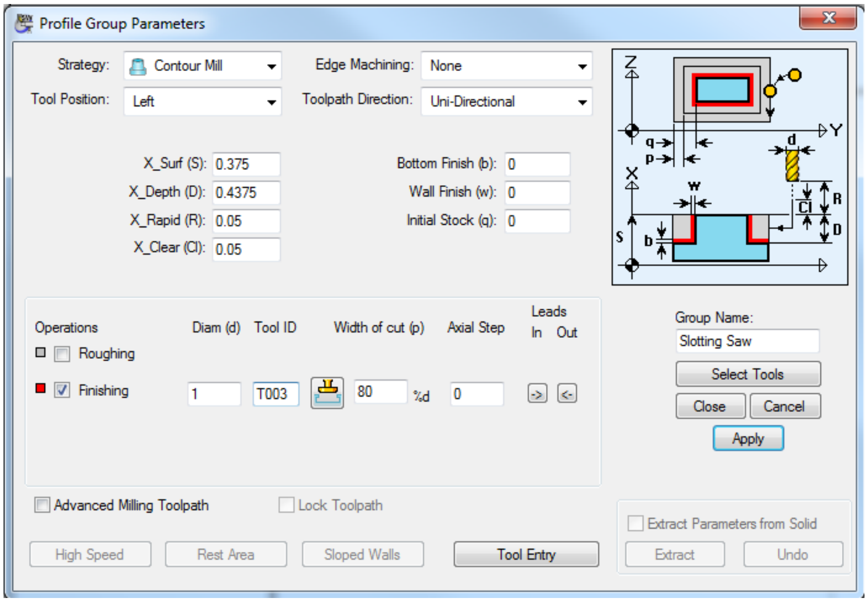
Note: *You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

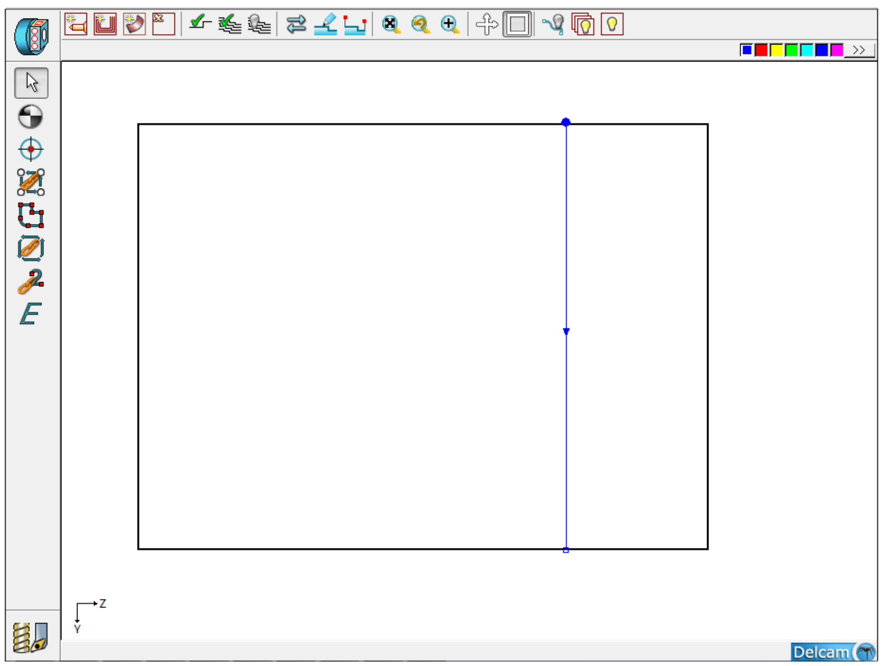
Start by entering the CAM mode in the Mill ZY Plane Window. First you will create the tool path to mill the square. To do so:

1. From the Part Features menu, choose New Profile Group
2. Choose Contour Mill under Strategy
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter an **X_Depth** of .4375 (X_Surf + ½ thickness of the Slotting Saw.)
5. Enter the Tool ID number of the Slotting Saw in the **Tool ID** box and then hit **Apply**.
6. In the Group Name field, type **Slotting Saw**.
7. Click on the **Close** Button

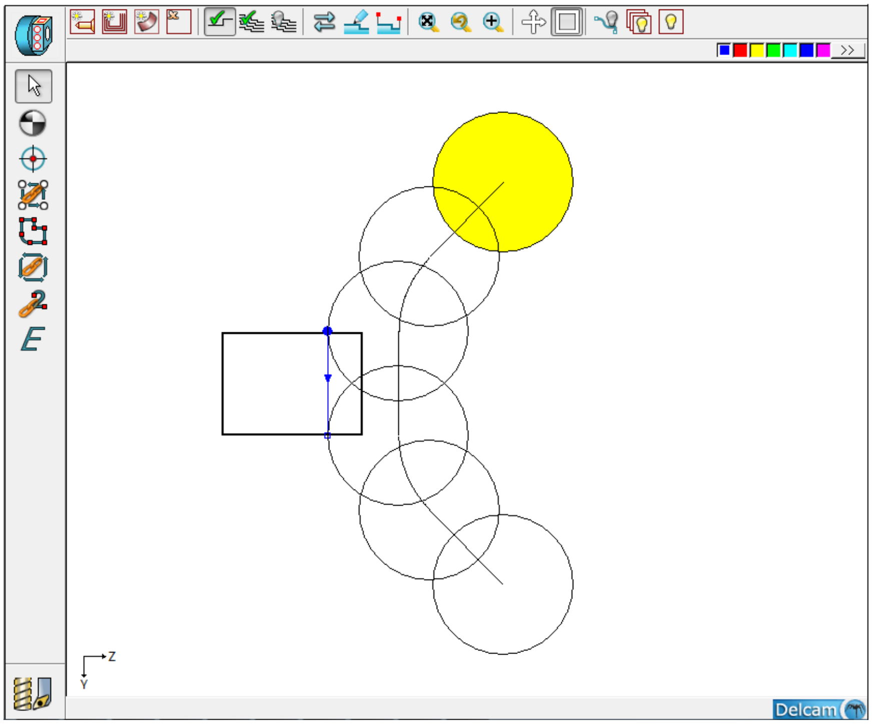
Your completed **Profile Group Parameters** dialog should appear as shown below:



8 Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



9 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:

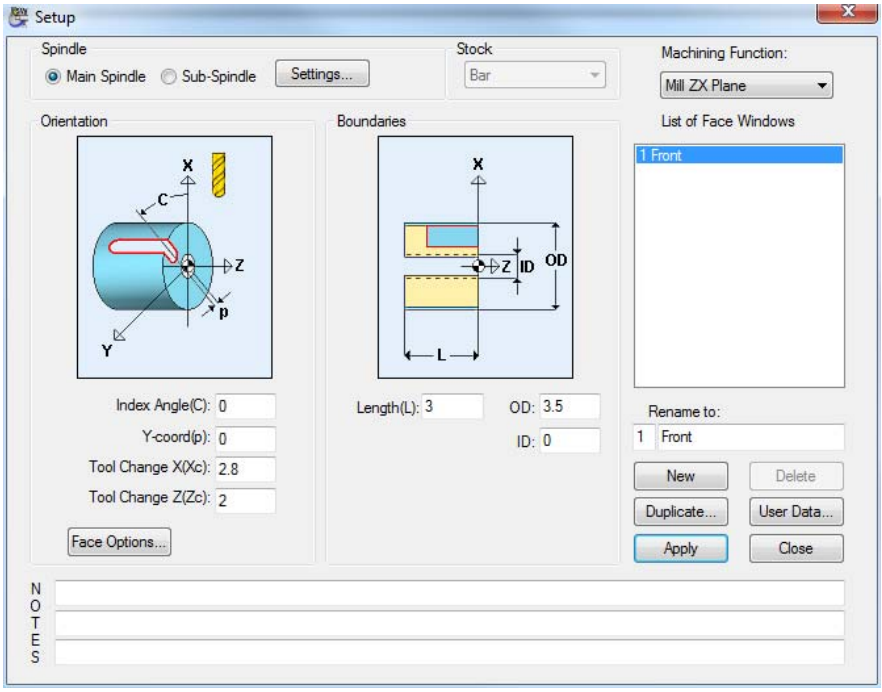


10 Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill ZX Plane

The Mill ZX Plane machining function is used to perform milling in the Z and X planes only. Such operations typically involve following a turning profile with an end mill (often ball nose) with the machine spindle in a locked position.

Programming of this machining function requires that your Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



Index Angle (C)	Orientation angle of the stock to achieve the desired face position
Y-coord (p)	Y coordinate value measuring the distance from the centerline of the part the tool is offset during machining. This value should always be positive.
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

Types of Features Created in Mill ZX Machining Function Window

The Mill ZX window uses an X-oriented “live” tool. There is no polar C-axis movement (the C-axis indexes and locks). This window is mainly used for doing key slots and operations that require simultaneous Z-axis and X-axis moves.

The features shown on the parts below are representative examples of features that can be created in the Mill ZX Window.

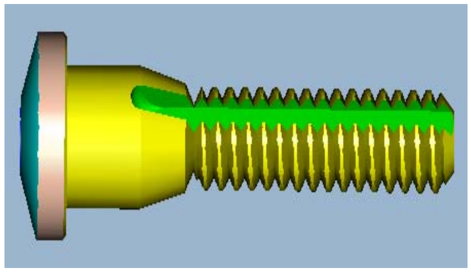


Figure 1. Milled Slot

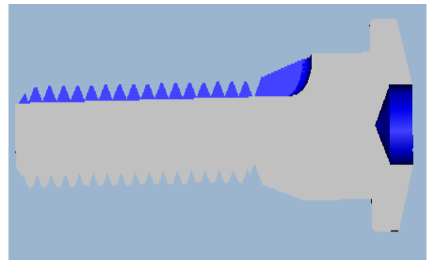


Figure 1a. (Cross section of milled slot)

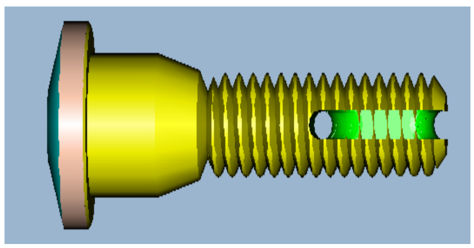


Figure 2. Milled slot

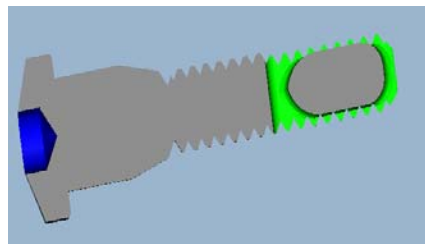
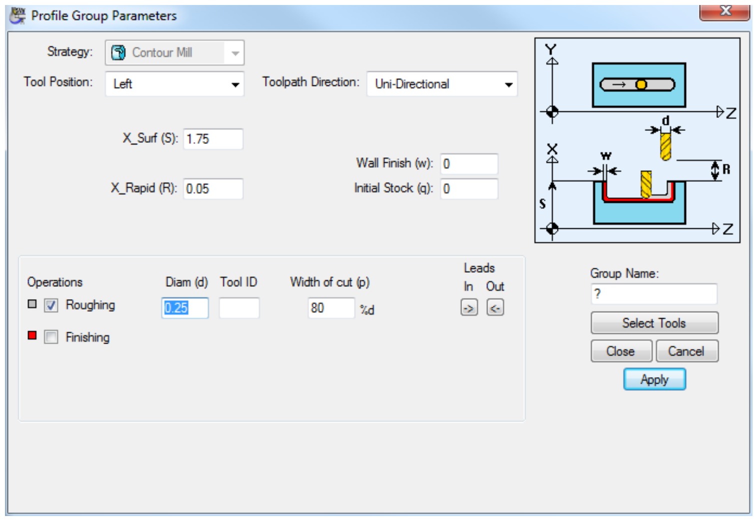


Figure2a. (Cross section of milled slot)

Part Features Dialogs Available in Mill ZX Machining Function

The Mill ZX machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing Left or Right from the Tool Position drop down menu.

Note: *In the Mill ZX window you are only allowed to use a New Profile Group. A New Hole group and Pocket Milling are not accessible.*



Strategy Indicates the type of feature to be milled. In the Mill ZX Face Window this must always be set to Contour Mill.

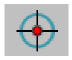
Tool Position For Contour Mill indicates the tool position during cutting, such as Left (climb milling), Right (conventional milling) or On (on centerline milling).

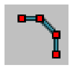
X_Surf (S) Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.

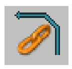
X_Rapid (R) Specifies how much clearance the tool will rapid above your X_Surf

Graphics Icons – Machining Function Mill ZX Plane

The Machining Function Mill ZX Plane uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

 **Single Hole Icon** Not applicable

 **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.

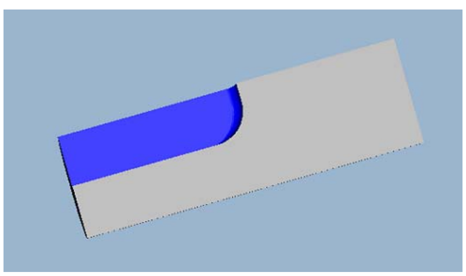
 **Chain Icon** enables you to specify that the currently active group symbol is to be applied to a profile that is created when **PartMaker** automatically connects all the elements that form an unbroken chain with an initial line or arc clicked.

Programming Example using Mill ZX Plane Machining Function

A practical programming application of the Mill ZX Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a .125 wide slot down the length of the part and radius out. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZX Plane Face Window should have an orientation of **X Tool**)

- .125 Inch Diameter End Mill



Features to be created:

Slot running down the length of the part then radius out in X.

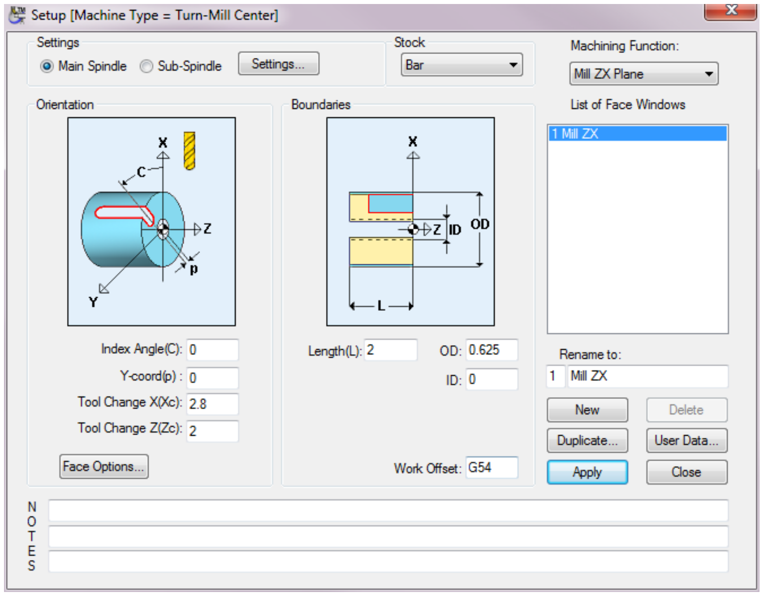
Step 1. Create a Mill ZX Plane Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill ZX Plane Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill ZX Plane**

4 Enter the setup parameters as shown below:



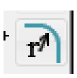
Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZX Plane Window. Draw the following features:

5 Choose **Show Axes** from the View Menu

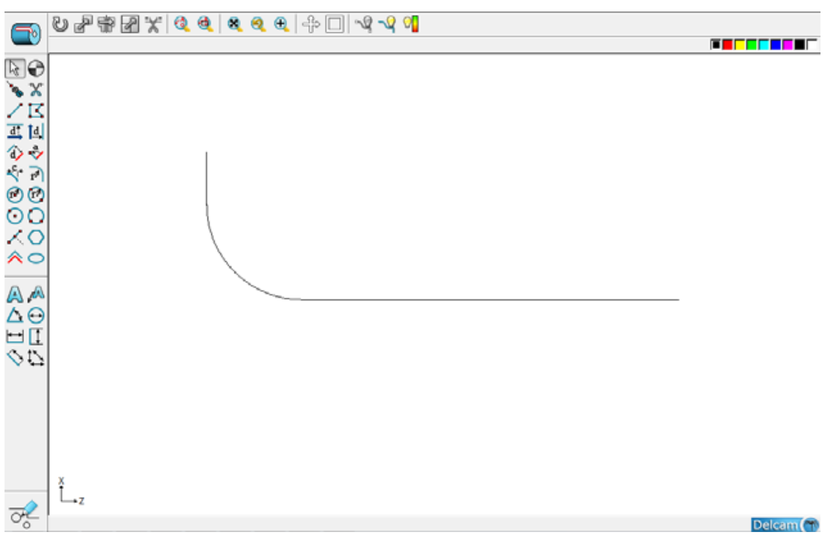
6 Choose **Show Boundaries** from the View Menu

7 Draw a horizontal line down the center of the part. Draw a vertical line 1 inch parallel to the face of the part.



8 Choose the **Fillet Icon** and enter a value of .2 and click on the inside of the intersection you just created to generate a radius.

Your completed geometry and CAD Face Window should appear as shown below:



Note: *You can always go to a different Face Window in your *.job file by selecting from the* ***Window*** *menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing* ***Show Windows Icons*** *from the* ***View*** *menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

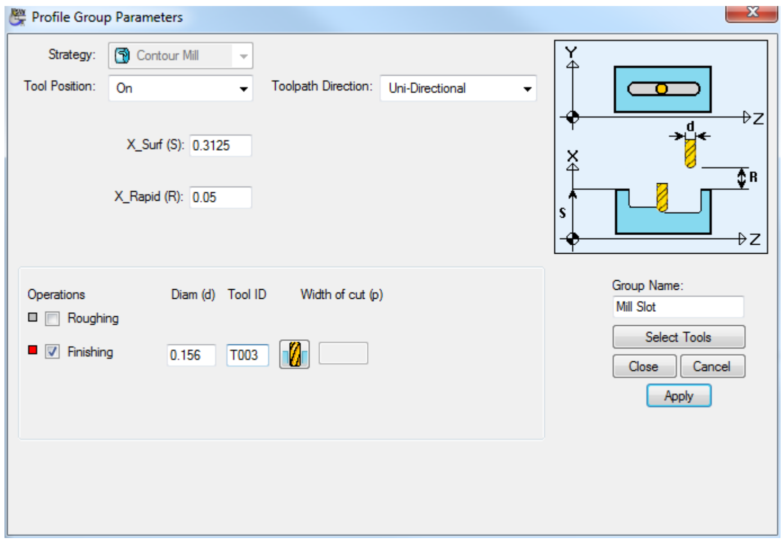
Start by entering the CAM mode in the Mill ZX Plane Window. First you will create the tool path to mill the slot. To do so:

1. From the Part Features menu, choose New Profile Group

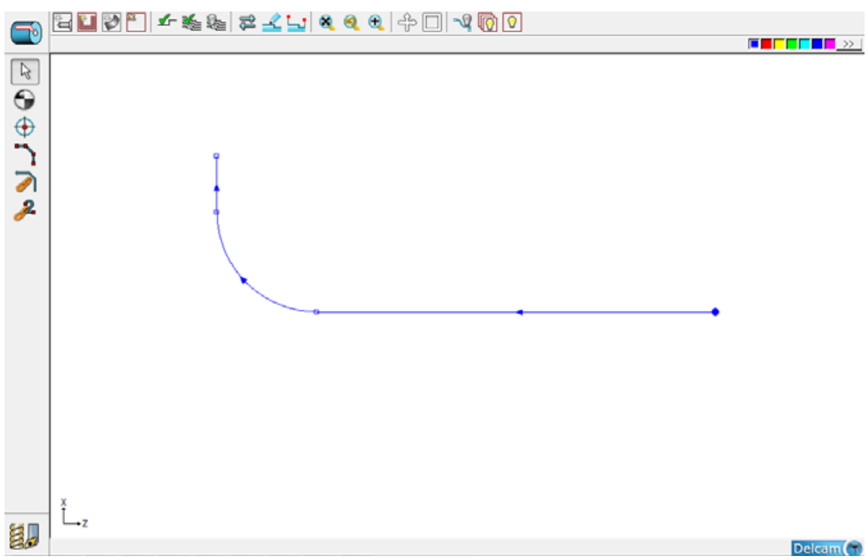
   Strategy is set to **Contour Mill** by default.

2. Choose **On** from **Tool Position** to indicate that we will be milling with the centerline of the tool.
3. Enter **X_Surf** of 0.3125
4. Enter a Tool Diameter of 0.125 and click **Select Tools**
5. In the **Group Name** field, type **Mill Slot**

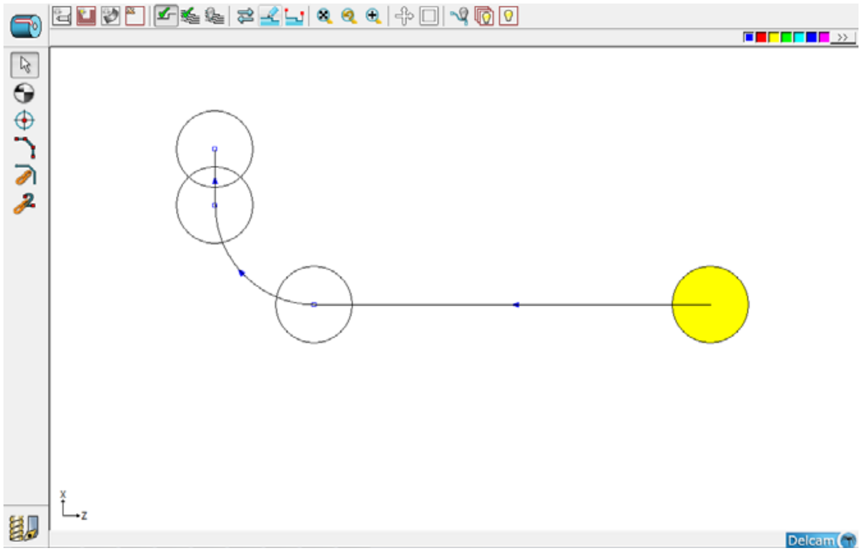
Your completed **Profile Group Parameters** dialog should appear as shown below:



6. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



7 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:

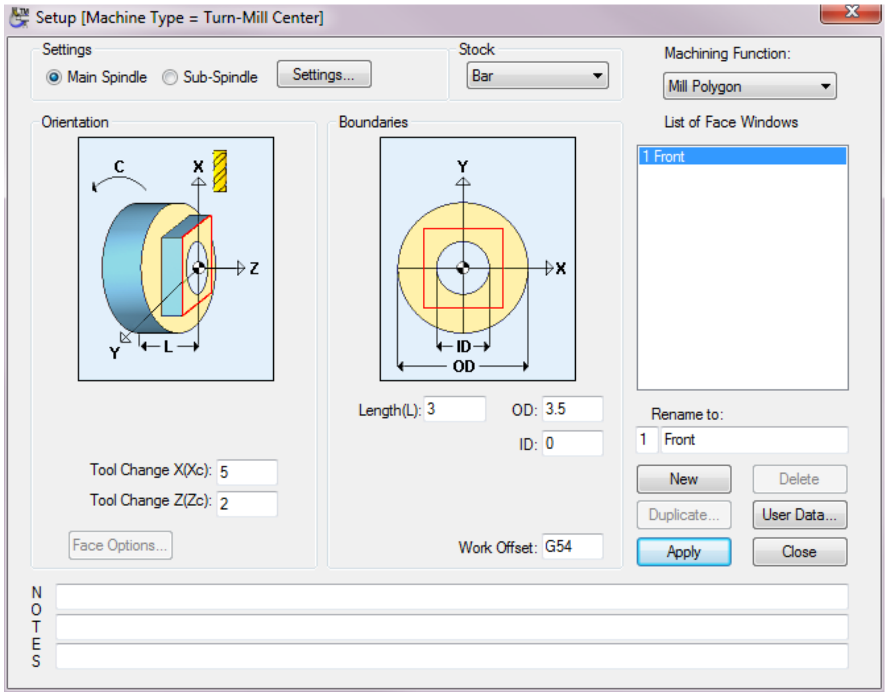


8 Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill Polygon

The Mill Polygon machining function allows you to create polygon shapes on the OD of a part using the Y-axis capability of your machine and a tool oriented along the X-axis of the machine.

Programming of this machining function requires that your Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.



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

Types of Features Created in Mill Polygon Machining Function Window

The Mill Polygon window uses an X-oriented “live” tool. There is no polar C-axis movement (the C-axis indexes and locks). This window is mainly used when creating multiple flats like a polygon, square or a hex. This operation does use bi-directional cutting therefore it cuts in one direction and indexes then cut in the opposite direction.

The features shown on the parts below are representative examples of features that can be created in the Mill Polygon Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker Turn-Mill. Where relevant, the locations of these files have been provided.

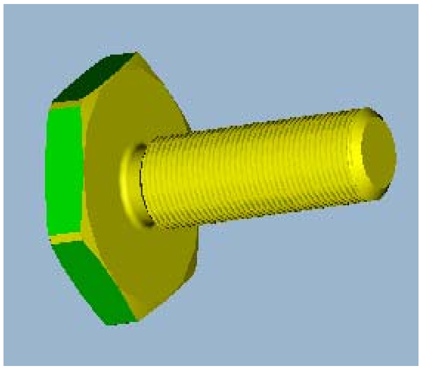


Figure 1.: Milled Hex

(Relevant files: C:\PartMaker\Pm-TM\Samples\8-Axis_Demo\8-Axis_Demo.job, 8-Axis_Demo.cdb, 8-Axis_Demo.tdb)

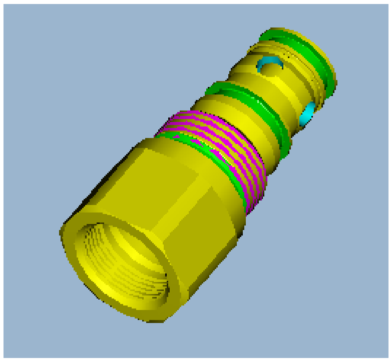


Figure 2.: Milled Hex

(Relevant files: C:\PartMaker\Pm-TM\Samples\Nakamura_TW\TW.job, TW.cdb, TW.tdb)

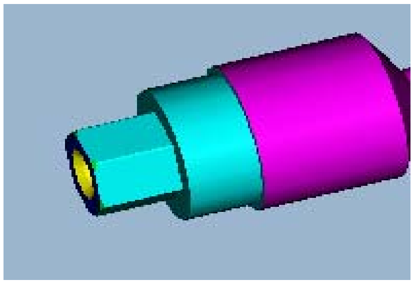


Figure 3.: Milled Hex

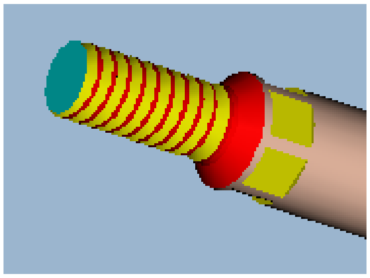
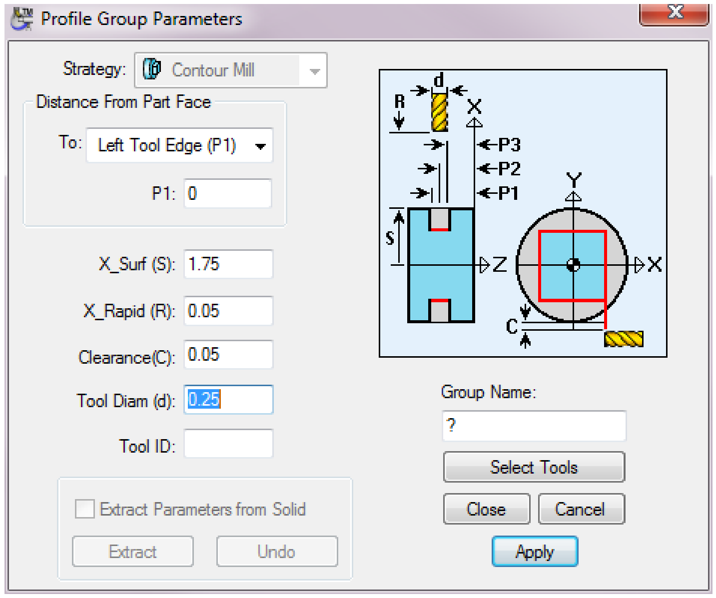


Figure 4.: Milled Polygon

Part Features Dialogs Available in Mill Polygon Machining Function

The Mill Polygon machining function can be used to create flats only. Below, a Profile Group parameter window is shown. In the drop down menu you pick the tool edge you are going to use with respect to the part.

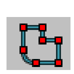
Note: *In the Mill Polygon window you are only allowed to use New Profile Group. A New Hole Group is not accessible.*



Distance From Part Face, To:	Indicates placement of tool with respect to the part face. The tool can be positioned with respect to the Left Tool Edge (P1), the Tool Center (P2) or the Right Tool Edge (P3).
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf
Clearance(C)	Specifies the clearance amount the tool will position to past the barstock diameter.

Graphics Icons – Machining Function Mill Polygon

The machining functions for Mill Polygon uses the following icons to allow you to assign part features to created geometry in order to create a tool path.



Define 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 Geometry 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 with an initial line or arc clicked.

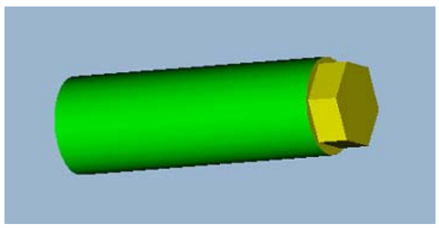
Programming Example using Mill Polygon Machining Function

Milling a Hexagon

A practical programming application of the Mill Polygon Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database.

In the example given below, you will mill a hexagon with a ¼ inch End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Polygon Face Window should have an orientation of **X Tool**)

- .25 Inch Diameter End Mill



Features to be created:

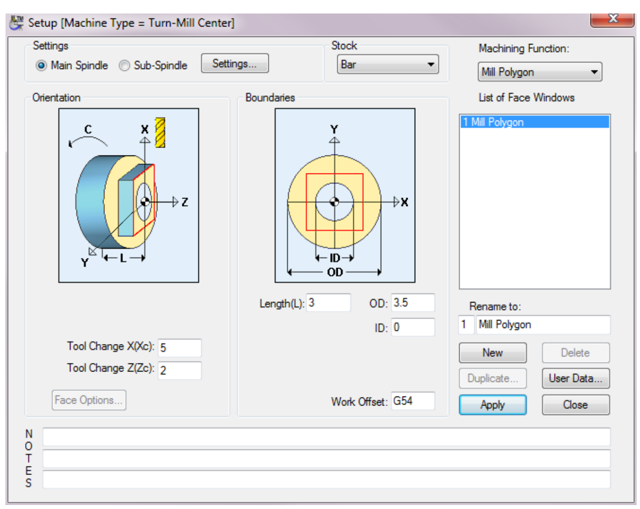
Mill a hexagon with a .25 dia end mill

Step 1. Create a Mill Polygon Plane Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill Polygon Plane Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill Polygon**
4. Enter the setup parameters as shown below:



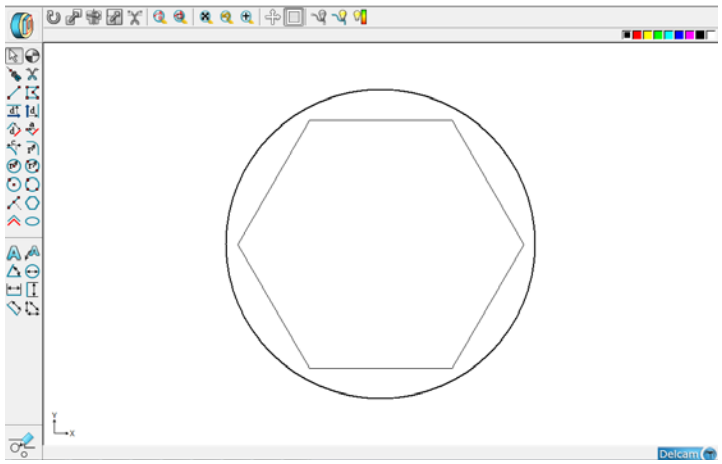
5. Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill Polygon Window. Draw the following features:

1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Choose the **Hexagon Icon** and enter .25 for the **Center to Flat** Distance

   Your completed geometry should appear as shown below:



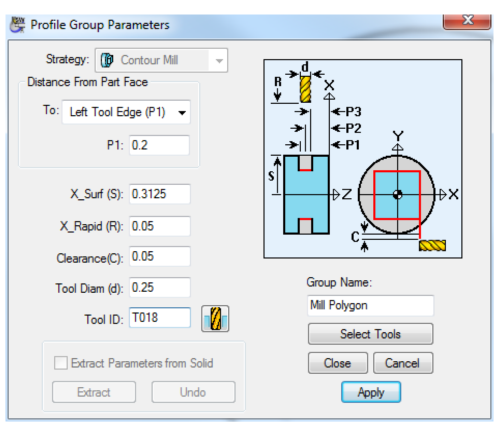
***Note:** You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

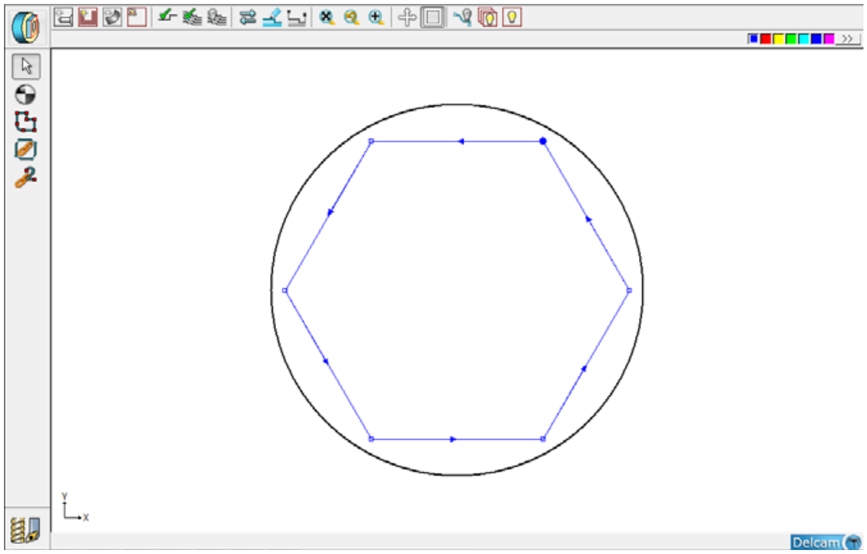
Start by entering the CAM mode in the Mill Polygon Window. First you will create the tool path to mill the hexagon. To do so:

1. From the Part Features menu, choose New Profile Group
2. Choose Left Tool Edge (P1) under Distance From Part Face
3. Enter 0.2 For the **P1** value
4. Enter a Tool Diameter of .25 and click **Select Tools**
5. In the Group Name field, type Mill Polygon

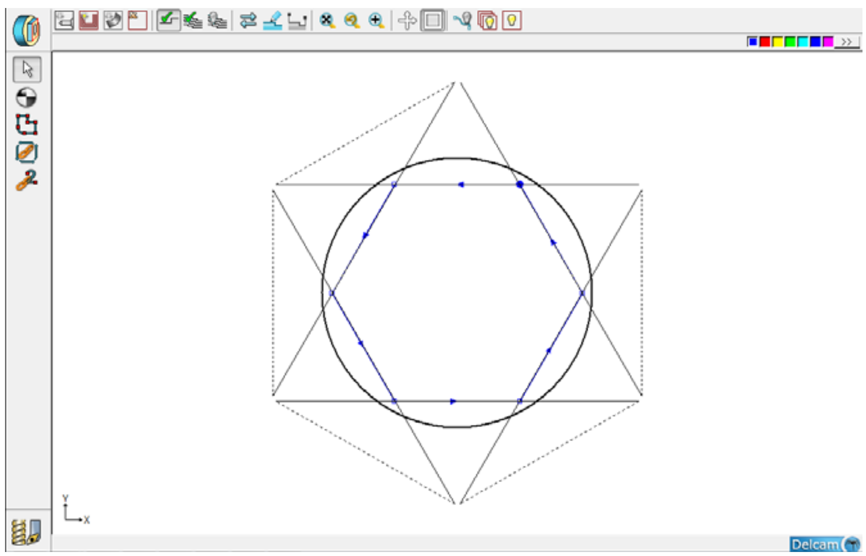
   Your completed dialog should appear as shown below:



6 Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



7 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:



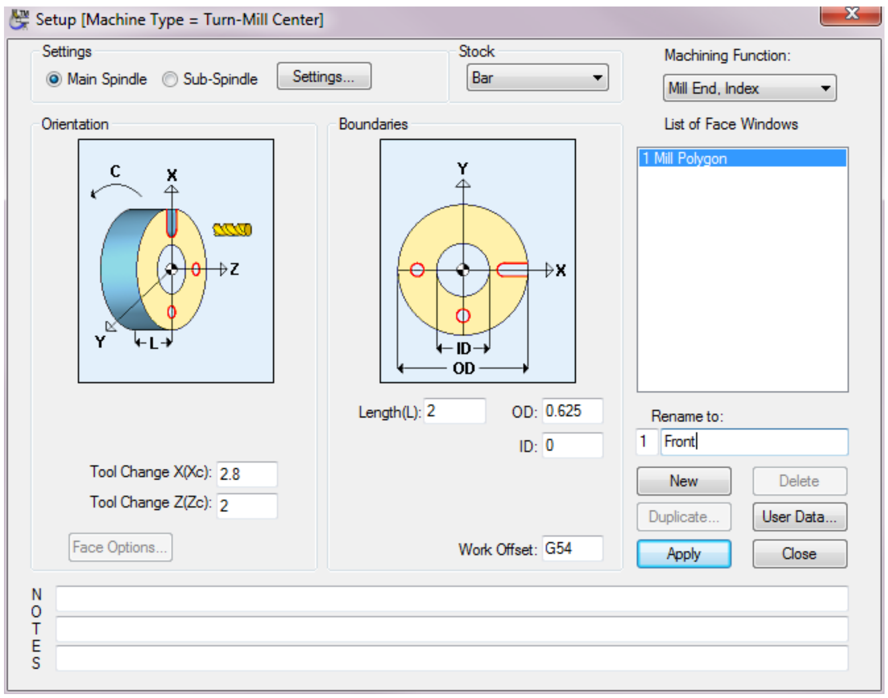
8 Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill End, Index

The Mill End, Index machining functon is used when drilling holes and milling slots on the end face of the part. The holes are located in the XY plane and oriented along the Z-axis.

The shapes that can be created with this machining function are:

- Holes in the end of the work created by Z-axis feed. This motion type supports off-center drilling on the end of the part, using C-axis indexing and X-axis positioning to locate the hole position.
- Radial slots which are milled using C-axis indexing to position and hold the part while X-axis motion is used to cut the part. The tool would be a milling tool oriented along the Z-axis.



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

Types of Features Created in Mill End Index Machining Function Window

The MILL END INDEX window uses a Z-oriented “live” tool. There is no polar C-axis movement (the C-axis indexes and locks). This window is mainly used for doing any number of holes or slots using a Z-oriented tool.

The features shown on the parts below are representative examples of features that can be created in the MILL END INDEX window

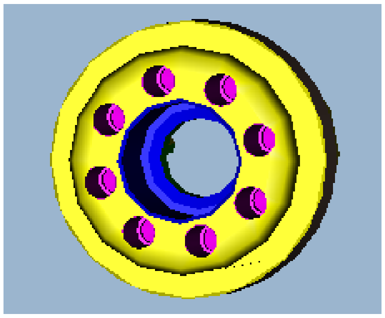


Figure 1. Face holes

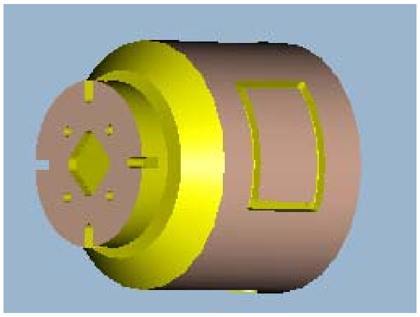


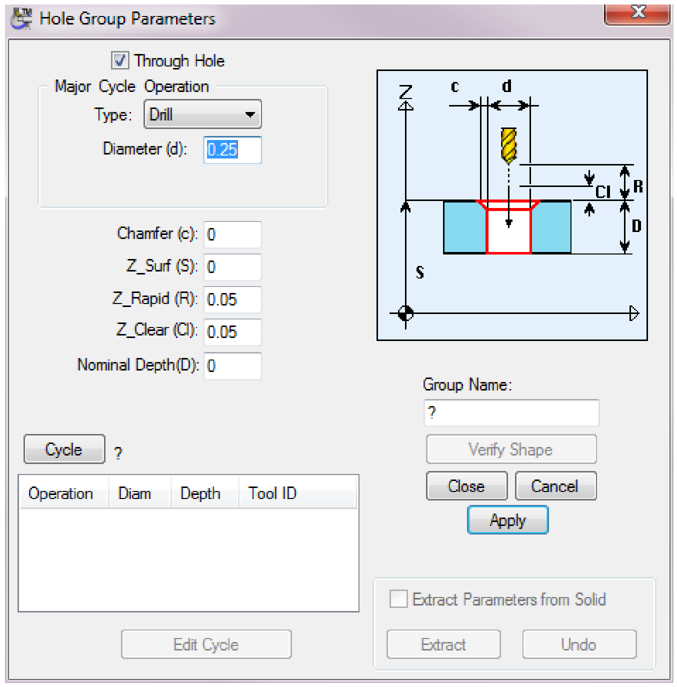
Figure 2. Face holes and Slots

(Relevant files: C:\PartMaker\Pm-TM\Samples\Tm-demo\Tm-demo.job, Tm-demo.cdb, Tm-demo.tdb)

Part Features Dialogs Available in Mill End Index Machining Function

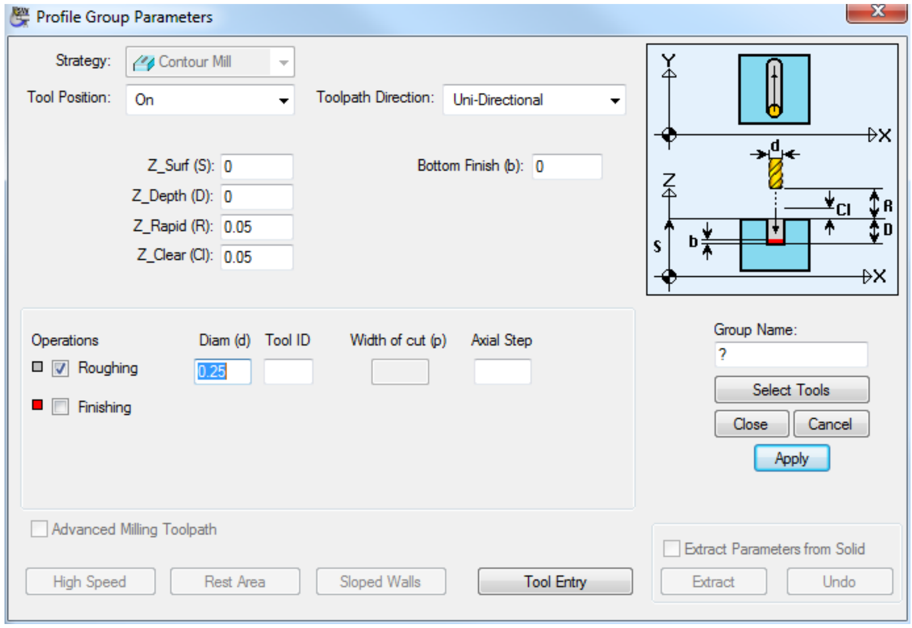
Both the **New Hole Group** and **New Profile Group** commands from the **Part** Features menu can be used in the **Mill End Index** Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in Z.



Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycle's database
Diameter (d)	Diameter of the tool, which is the major cycle operation
Chamfer(c)	Specifies how much chamfer per side is needed
Z_Surf (S)	Establishes your Z-zero work plane
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf
Z_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Inclination Angle	Specifies the angle of the tool when making inclined holes (you must have a "B" axis or an angled attachment.)
Nominal Depth (D)	The depth of the operation to be performed

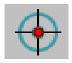
The Mill End Index machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown.



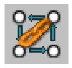
Strategy	Indicates the type of feature to be milled is a Contour Mill
Tool Position	For Contour Mill indicates the tool position during cutting, On (on centerline milling)
Z_Surf (S)	Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part
Z_Depth (D)	The depth of the operation to be performed
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf.
Z_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Tool Entry	Does not apply to the Mill End Index machining function

Graphics Icons – Machining Functions Mill End Index

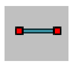
The Machining Functions Mill End Index uses the following icons to allow you to assign part features to created geometry in order to create a tool path.



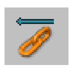
Single Hole Icon enables you to use Snap Modes to define the location on the geometry where a currently Hole Group Symbol is to be executed.



Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.



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. When performing indexing operations, the Profile Icon may only join two points during its usage.



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 with an initial line or arc clicked. When performing indexing operations, the Chain Icon may only join two points during its usage.

Programming Example using Mill End Index Machining Function

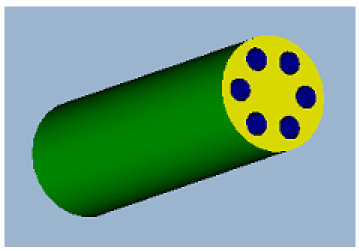
Bolt Hole Pattern on the Face

A practical programming application of the Mill End Index machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will drill six .125 diameter holes on the face of the part. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill End Index Face Window should have an orientation of **Z Tool**)

- .125 Inch Diameter Drill
- Spot Drill with a diameter larger than .125 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .125-inch drill and a spot drill.



Features to be created:

Spot Drill and Drill (6) holes with a .125 diameter drill

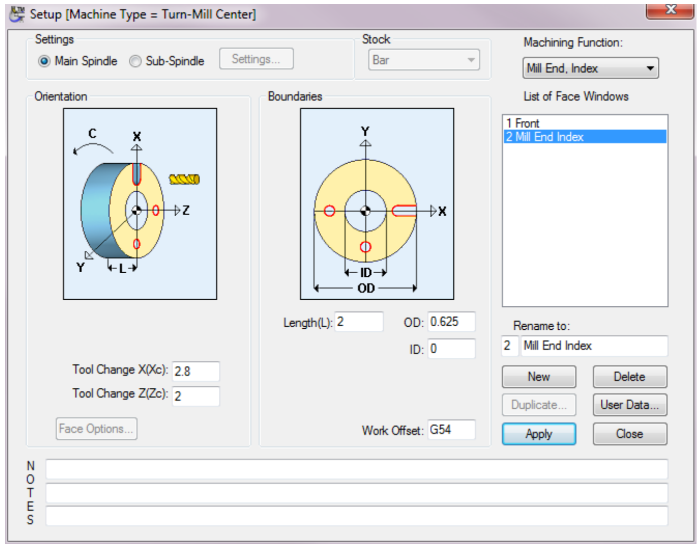
Step 1. Create a Mill End Index Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill End Index Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3. From the Machining Function drop down menu, choose Mill End Index

4 Enter the setup parameters as shown below:



5 Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill End Index Window. Draw the following features:

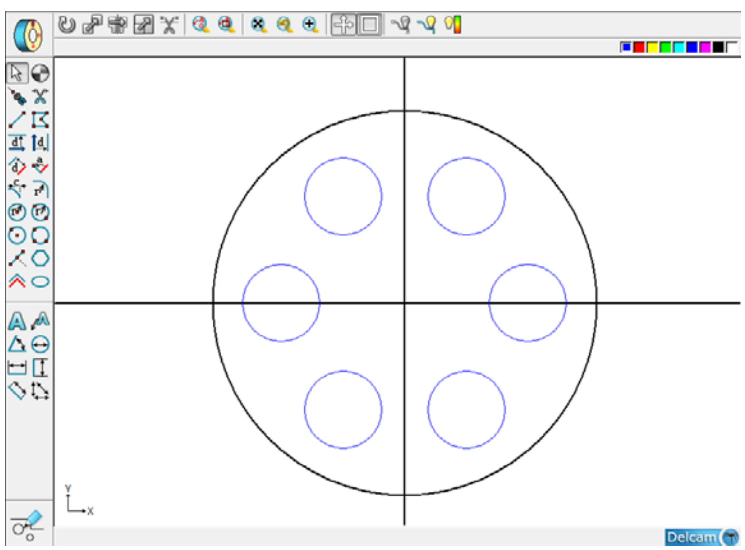
1 Choose **Show Axes** from the View Menu

2 Choose **Show Boundaries** from the View Menu

3 Draw a .0625 inch radius circle with a center location at X<-.2> and Y<0>

4 Choose the **Rotate** command and make 5 copies and rotate about the X<0> and Y<0> and with **Angle (A)** of 60.

Your completed geometry and CAD Face Window should appear as shown below:



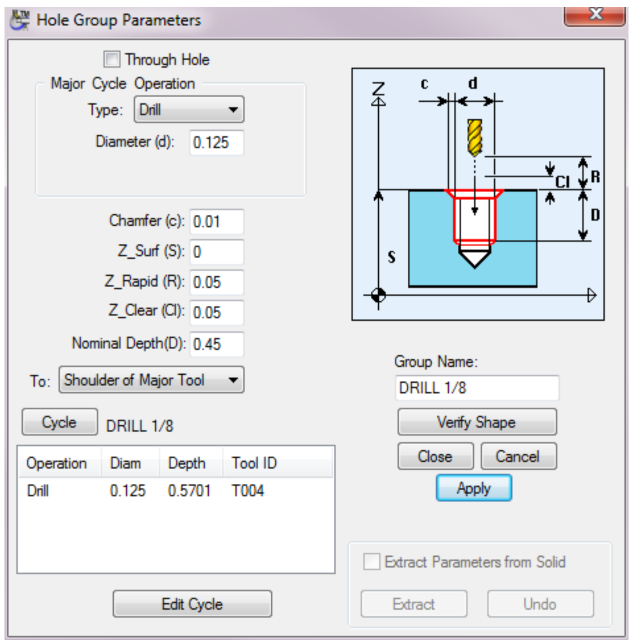
Note: *You can always go to a different Face Window in your *.job file by selecting from the* ***Window*** *menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing* ***Show Windows Icons*** *from the* ***View*** *menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

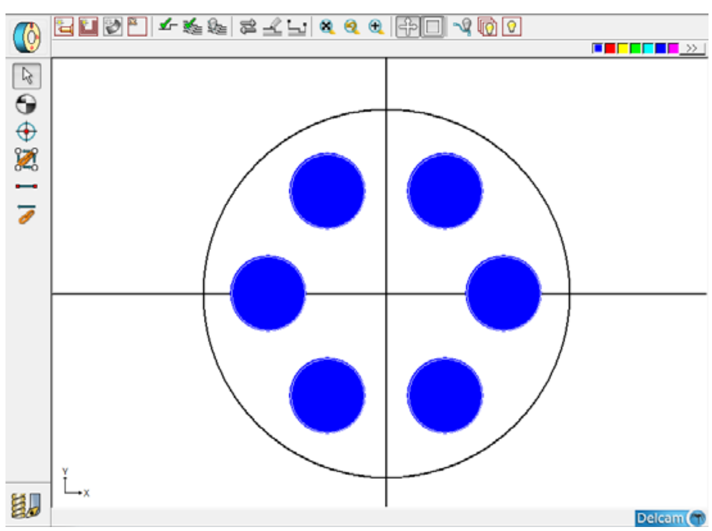
Start by entering the CAM mode in the Mill End Index Window. First you will create the 0.125 Boltholes, to do so:

1. From the Part Features menu, choose New Hole Group
2. Uncheck the **Through Hole** Box
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .125
5. Enter a **Chamfer** of .01
6. Enter a Nominal Depth (D) of 0.45
7. Click the **Cycle** button.

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



8. Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circle representing the hole to create the tool path. It should appear as shown below:

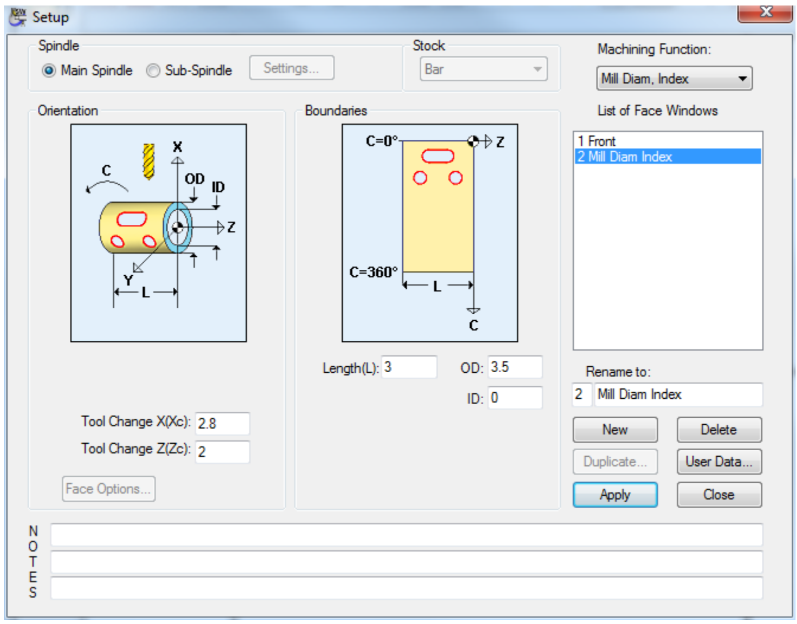


Machining Function: Mill Diam, Index

The Mill Diam, Index machining function allows you to create "cross holes" or radial slots on the OD of the part. In order to support this machining function, your Swiss-type lathe must be able to support C-axis indexing, which may not be performed while the tool is in contact with the work.

This machining function supports the following features cut on the outside diameter of the work using a tool oriented along the machine's X-axis:

- Slots oriented along the length (Z-axis) of the stock.
- Holes drilled anywhere along the cylinder except the end of the stock.



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

Types of Features Created in Mill Diam, Index Machining Function Window

The Mill Diam, Index window uses a X-oriented "live" tool. There is no polar C-axis movement (the c-axis indexes and locks). This window is mainly used for doing any number of holes or slots using a X-oriented tool.

The features shown on the parts below are representative examples of features that can be created in the Mill Diam, Index window.

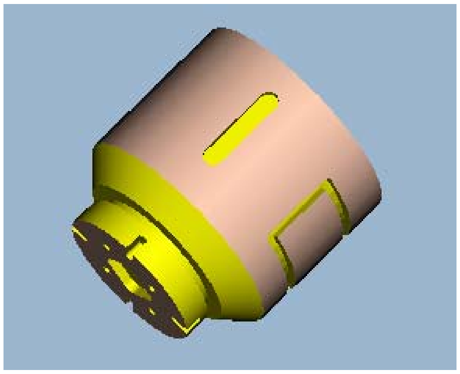


Figure 1. Milled Slot

(Relevant files: C:\PartMaker\Pm-TM\Samples\Tm-demo\Tm-demo.job, Tm-demo.cdb, Tm-demo.tdb)

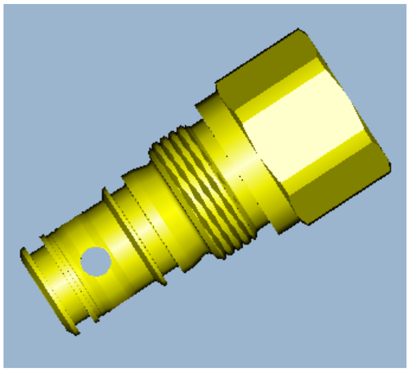


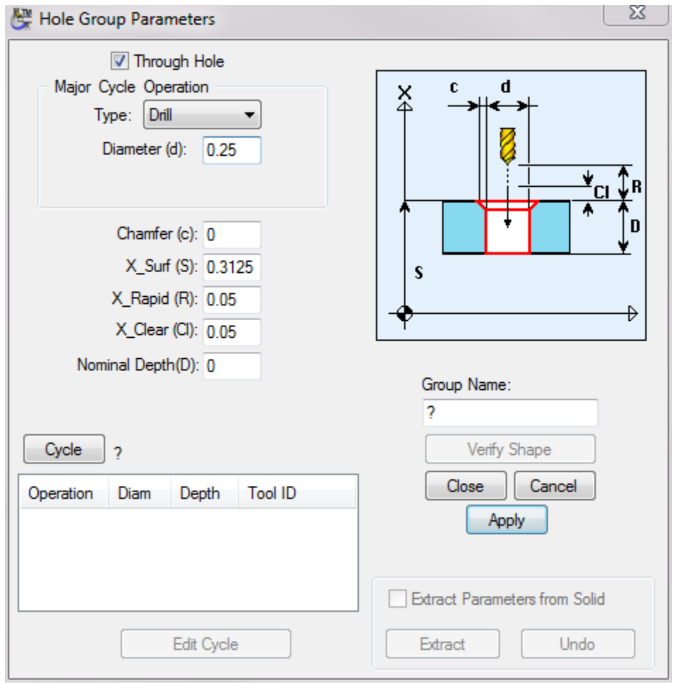
Figure 2. Cross holes

(Relevant files: C:\PartMaker\Pm-TM\Samples\Nakamura_TW\TW.job, TW.cdb, TW.tdb)

Part Features Dialogs Available in Mill Diam, Index Machining Function

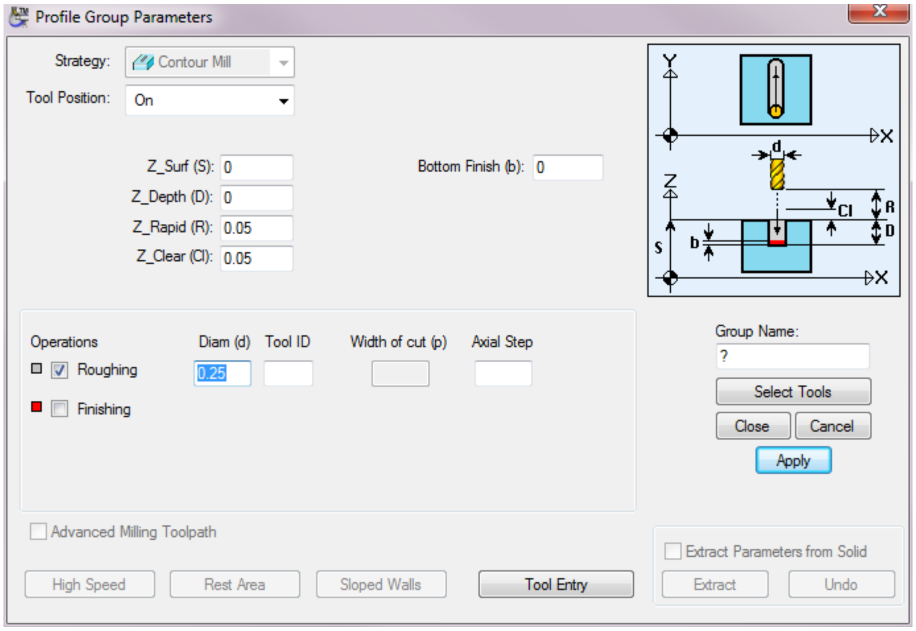
Both the **New Hole Group** and **New Profile Group** commands from the **Part** Features menu can be used in the Mill Diam, Index Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in X.



Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycle's database.
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
X_Surf (S)	Establishes your X-zero work plane.
X_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your X_Surf.
X_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Nominal Depth (D)	The depth of the operation to be performed

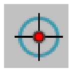
The Mill Diam,Index machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown.



Strategy	Indicates the type of feature to be milled is a **Contour Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, **On** (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above your X_Surf.
X_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Tool Entry	Does not apply to the **Mill Diam, Index** machining function

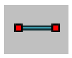
Graphics Icons – Machining Functions for Mill Diam, Index

The Machining Functions for Mill Diam, Index uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

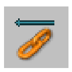


Single Hole Icon enables you to use Snap Modes to define the location on the geometry where a currently Hole Group Symbol is to be executed.

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.



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. When performing indexing operations, the Profile Icon may only join two points during its usage.



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 with an initial line or arc clicked. When performing indexing operations, the Chain Icon may only join two points during its usage.

Programming Example using Mill Diameter Index Machining Function

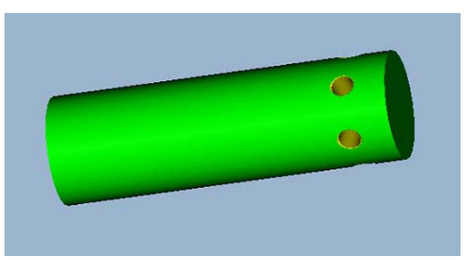
Creating Cross holes

A practical programming application of the Mill Diameter Index machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will Drill six holes with a .125 diameter drill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Diameter Index Face Window should have an orientation of **X Tool**).

- .125 Inch Diameter Drill
- Spot Drill with a diameter larger than .125 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .125-inch drill and a spot drill.



Features to be created:

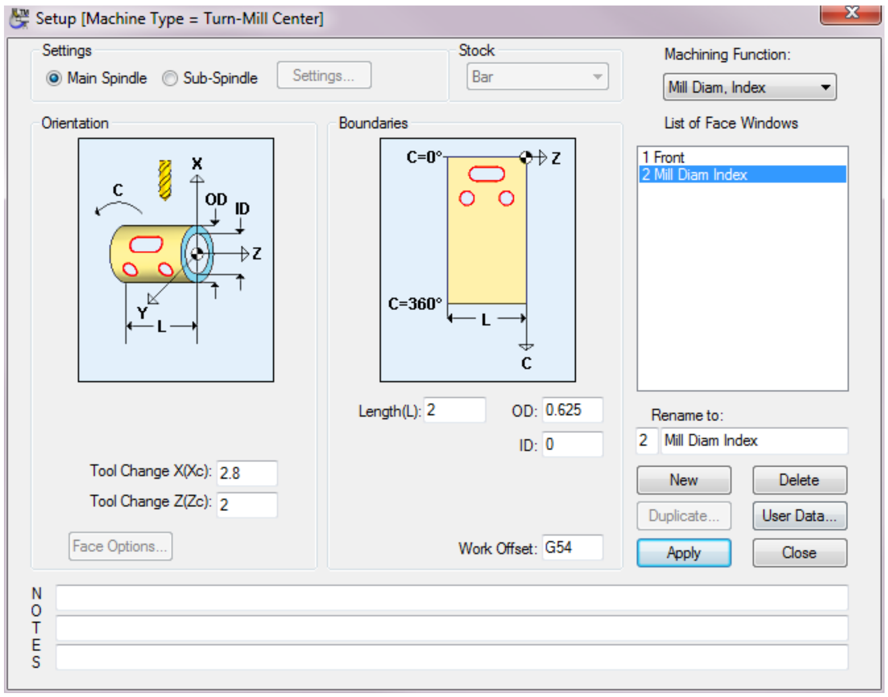
Spot Drill and Drill six holes on diameter with a .125 diameter drill

Step 1. Create a Mill Diameter Index Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill Diam, Index Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the Machining Function drop down menu, choose Mill Diam, Index
4. Enter the setup parameters as shown below:

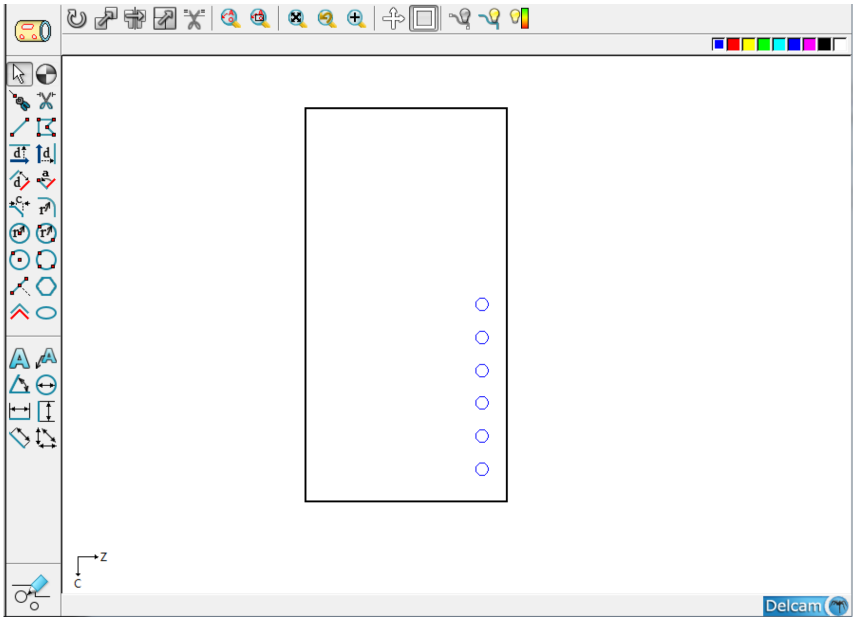


Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill Diam, Index Window. Draw the following features:

1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a .0625-inch radius circle with a center location at Z<. 25. > and C<0>.
4. Click on the **Selection Icon** and click on the circle you just created.
5. Choose **Translate** from the **Edit Menu** and make 5 more copies with **Horizontal Distance (H)** of 0 and **Vertical Distance (V)** of 60 degrees.

Your completed geometry and CAD Face Window should appear as shown below:



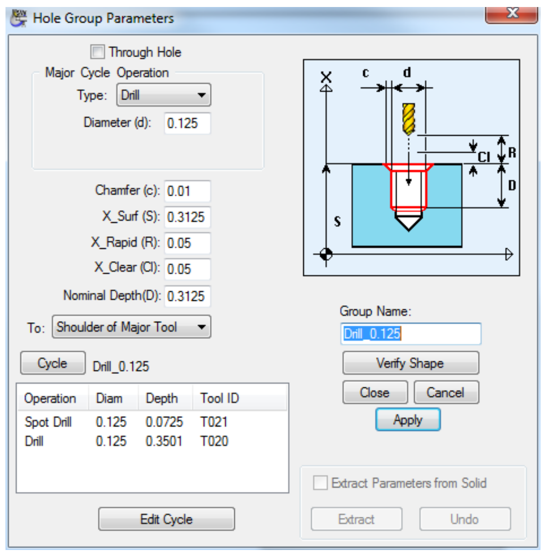
***Note:** You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

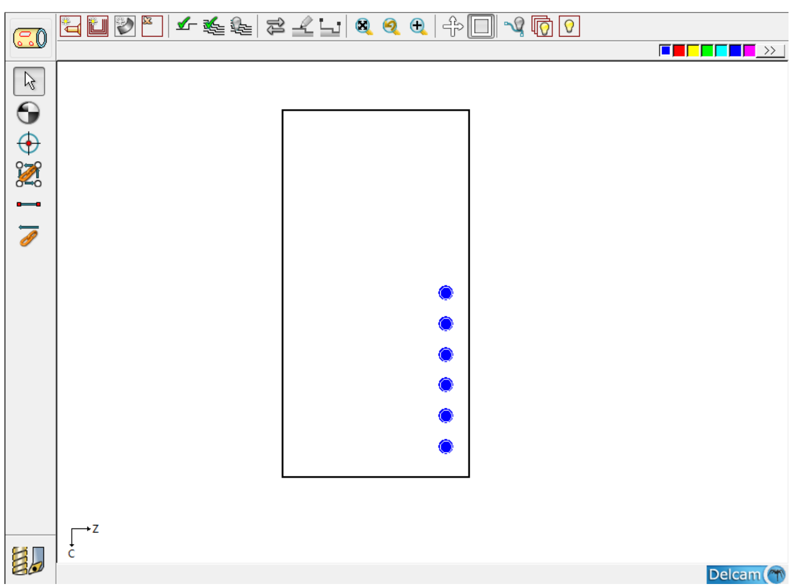
Start by entering the CAM mode in the Mill Diam, Index Window. First you will create the tool path to drill the cross holes. To do so:

1. From the Part Features menu, choose New Hole Group
2. Uncheck the **Through Hole** checkbox
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .125
5. Enter a **Chamfer (c)**of .01
6. Enter a **Nominal Depth (D)** of 0.3125
7. Click the **Cycle** button.

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



8. Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circles representing the holes to create the tool path. It should appear as shown below:



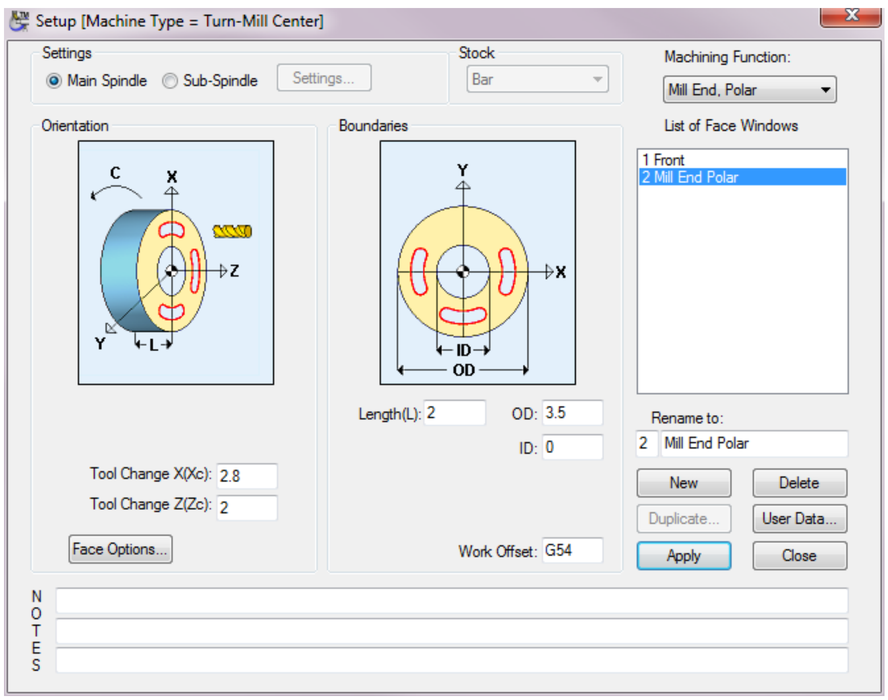
Machining Function: Mill End, Polar

The Mill End, Polar machining function allows you to perform polar interpolation on the face of the part. This machining function assumes that the Swiss-type lathe has the ability to perform simultaneous feed motions with the X, Z and C-axes.

The following rules apply to Mill End, Polar:

- Tools must be oriented along the machine's Z-axis.
- Shapes created must not pass, or if extended, pass through the center of rotation of the stock (X=0). For example, the radial slots that are programmable in Mill End, Index may NOT be programmed using Mill End, Polar. Also, any arc move that intersects the X=0 center is **NOT** allowed to be programmed using this machining function.

*Programming of this machining function requires that your Swiss-type lathe has a programmable C-axis and that you have purchased the **PartMaker** C-axis milling option.*



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

Types of Features Created in Mill End, Polar Machining Function Window

The MILL END, POLAR window uses a Z-oriented “live” tool. There is polar C-Axis movement (the C-Axis does not lock). This window is mainly used for milling profiles and pockets on the face of the part.

The features shown on the parts below are representative examples of features that can be created in the MILL END, POLAR window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker Turn-Mill. Where relevant, the locations of these files have been provided.

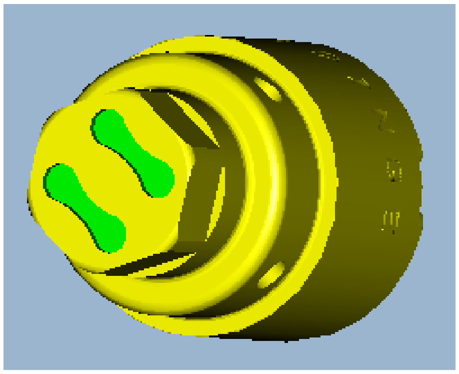


Figure 1. Milled Pockets
(Relevant files: C:\PartMaker\Pm-TM\Samples\Hardinge_TT-65\TT-65.job, TT-65.cdb, TT-65.tdb)

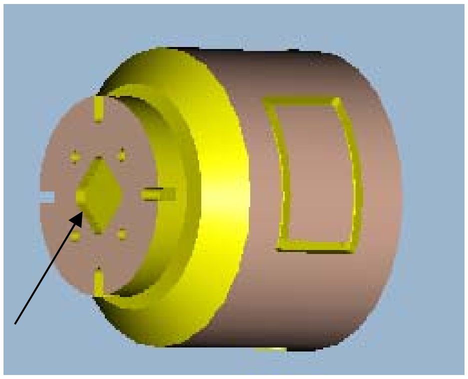


Figure 2. Face Pocket
(Relevant files: C:\PartMaker\Pm-TM\Samples\Tm-demo\Tm-demo.job, Tm-demo.cdb, Tm-demo.tdb)

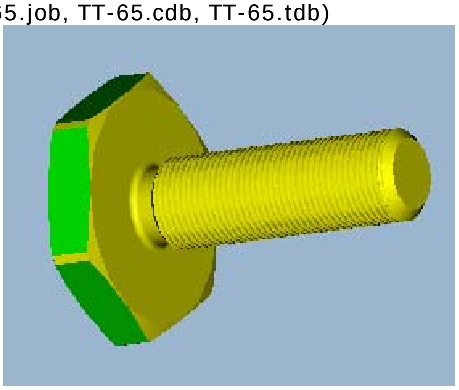


Figure 3. Milled Cam
(Relevant files: C:\PartMaker\Pm-TM\Samples\Citizen_FL\Citizen_FL_demo.job, Citizen_FL_demo.cdb, Citizen_FL_demo.tdb)

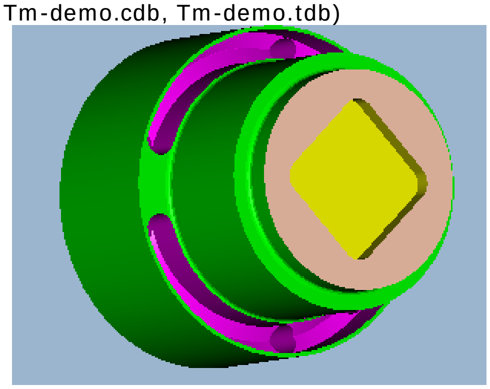
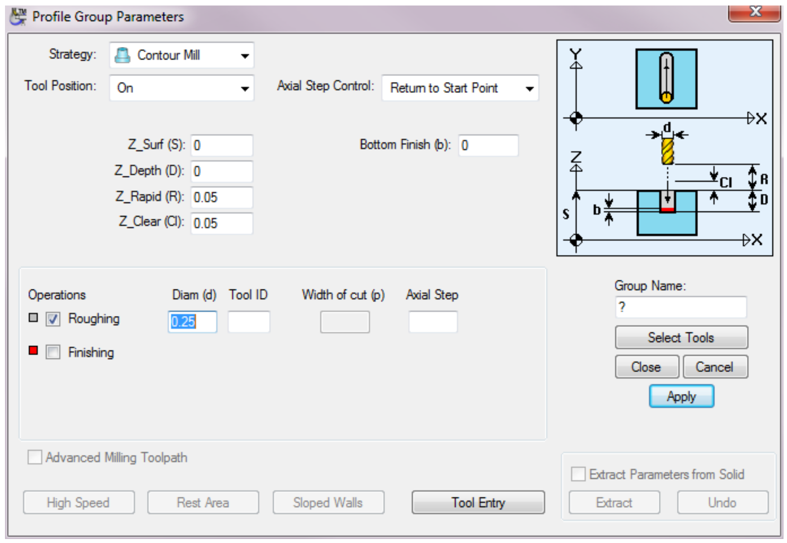


Figure 4. Milled Pocket on the Face

Part Features Dialogs Available in Mill End, Polar Machining Function

The Mill End, Polar machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with tool motion on centerline is shown. If you wish to create a pocket, choose Pocket Mill from Strategy drop down menu. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.

Note: *In the Mill End, Polar window you are only allowed to use a New Profile Group. New Hole group is not accessible.*



Strategy — Indicates the type of feature to be milled, either a Contour Mill or Pocket Mill

Tool Position — For Contour Mill indicates the tool position during cutting, such as Left (climb milling), Right (conventional milling) or On (on centerline milling)

Z_Surf (S) — Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part

Z_Depth (D) — The depth of the operation to be performed

Z_Rapid (R) — Specifies how much clearance the tool will rapid to in front of your Z_Surf.

Z_Clear (C) — Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.

Bottom Finish (b) — Amount of material left on the bottom of the feature for the finishing end mill to remove

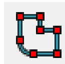
Wall Finish (w) — Amount of material left on the side (or wall) of the feature for the finishing end mill to remove when tool position set to Left or Right {Not shown above}

Roughing/Finishing — Indicates whether a Roughing and/or Finishing Tool will be used

Tool Entry — Indicates the manner of entry of end mill into a feature, either Vertical, Helical, Sloped or Sloped Down and Return

Graphics Icons Available in the Mill End, Polar Machining Function

The Machining Function Mill End, Polar, uses the following icons to allow you to assign part features to created geometry in order to create a tool path.



Define 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 Geometry 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 with an initial line or arc clicked.

2-Point Chain Icon enables you to specify a begin point and an end point to be chained together.

Engrave Icon enables you to use the Engrave Data dialog box to create a profile containing numbers and letters.

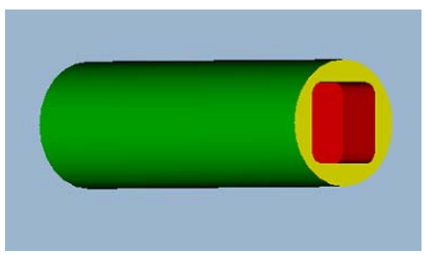
Programming Example using Mill End, Polar Machining Function

Creating a Pocket on the Face of the Part

A practical programming application of the Mill End, Polar machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a Square Pocket .400 from side to side with a .08 diameter End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill End, Polar Face Window should have an orientation of **Z Tool**)

- .080 Diameter End Mill



Features to be created:

Mill a square pocket with a .080 dia. end mill

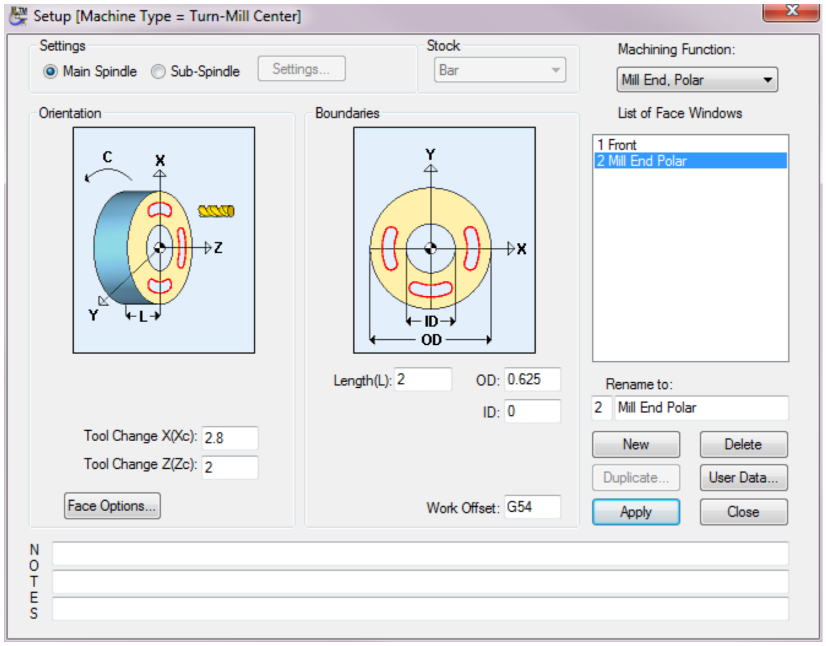
Step 1. Create a Mill End, Polar Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill End, Polar Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window

3 From the **Machining Function** drop down menu, choose **Mill End, Polar**



4 Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

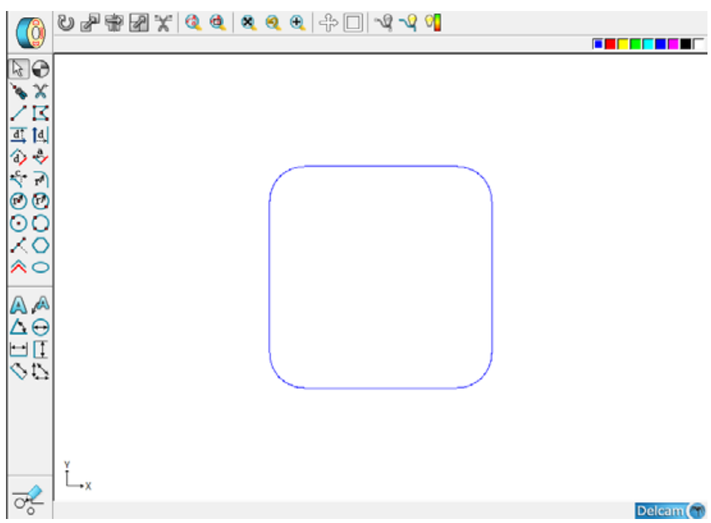
Start by entering the CAD mode in the Mill End, Polar Window. Draw the following features:

1 Choose **Show Axes** and **Show Boundaries** from the View Menu

2 Draw a square that is .400 from flat to flat.

3 Draw a .0625 inch radius using the **Fillet Icon** to round off each corner

Your completed geometry and CAD Face Window should appear as shown below:



Note: *You can always go to a different Face Window in your *.job file by selecting from the* ***Window*** *menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing* ***Show Windows Icons*** *from the* ***View*** *menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

Start by entering the CAM mode in the Mill End, Polar Window. First you will create the tool path to mill the square. To do so:

1 From the Part Features menu, choose New Profile Group

2 Choose Pocket Mill under Strategy

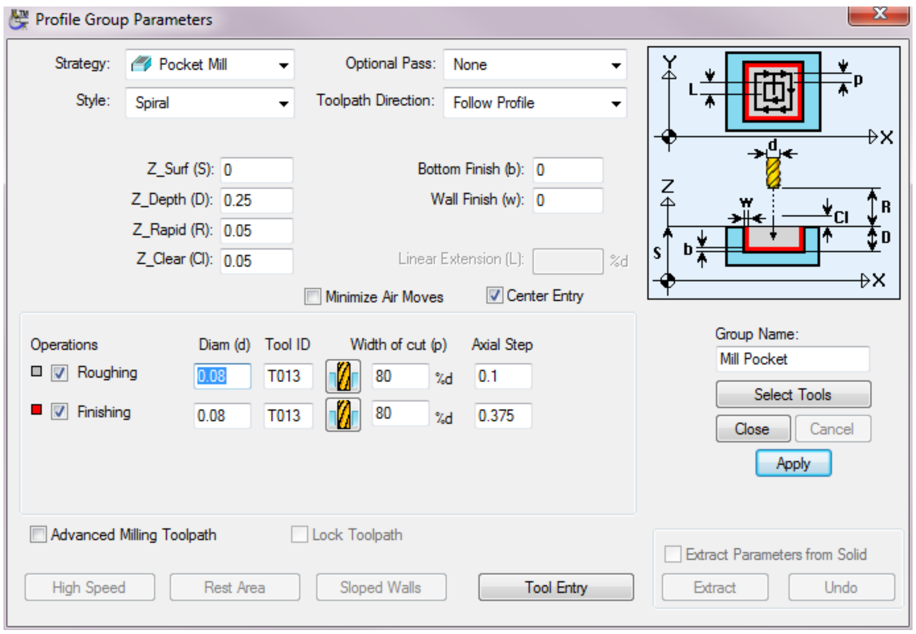
3 Choose **Spiral** for the type of pocket to be performed

4 Enter a **Z_Depth** of 0.25

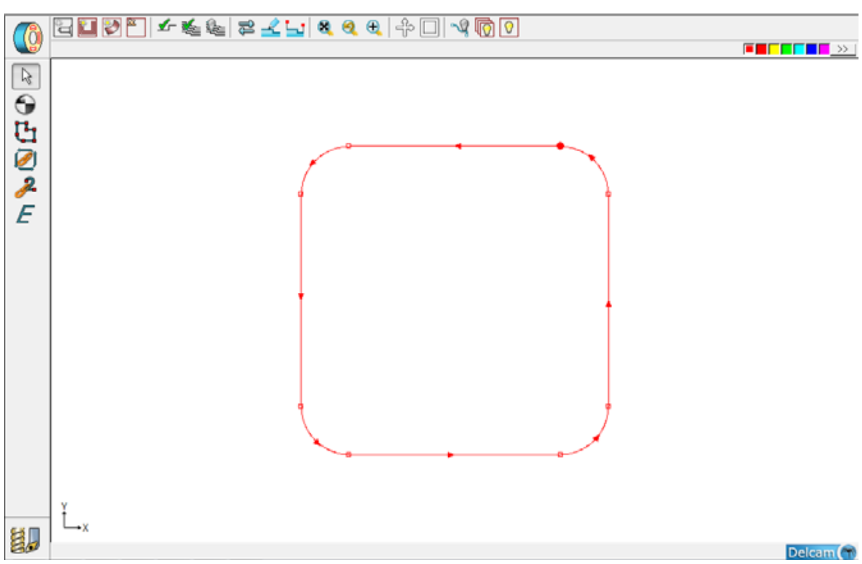
5 Enter a Tool Diameter of .08 for the Roughing and Finishing and click **Select Tools**

6 In the Group Name field, type Mill Pocket

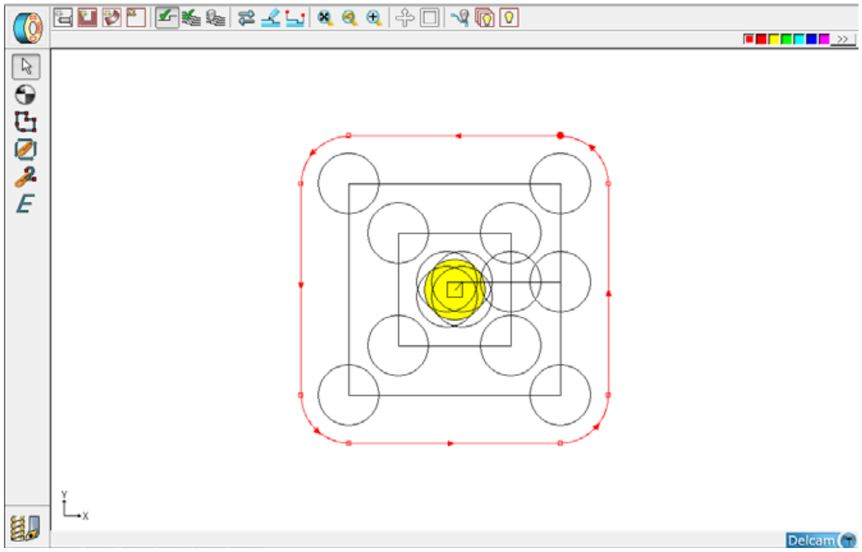
Your completed **Profile Group Parameters** dialog should appear as shown below:



7 Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



8 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path similar to the one below:

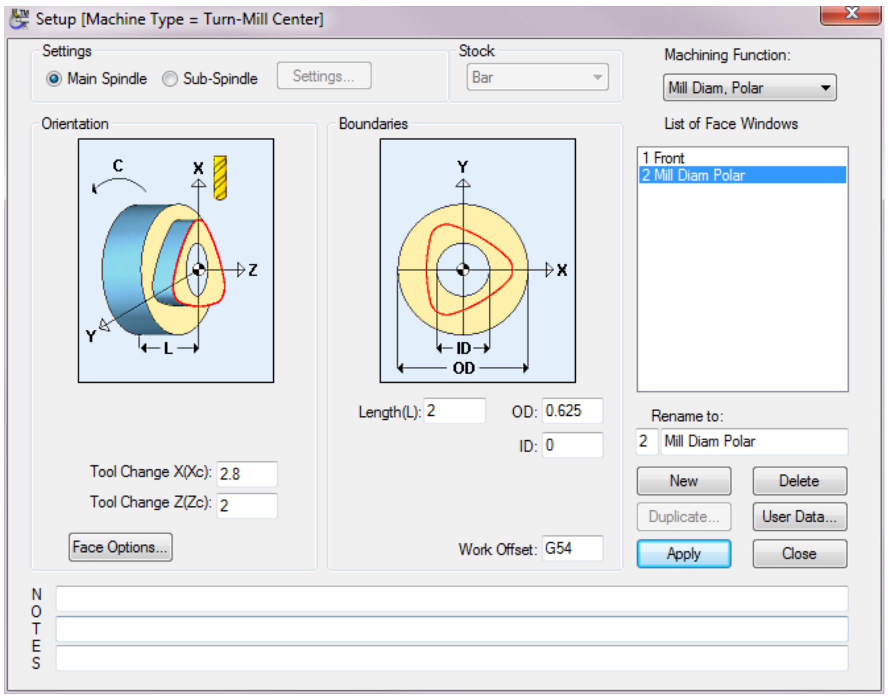


9 Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill Diam, Polar

The Mill Diam, Polar machining function supports polar milling anywhere on the stock using an X-axis oriented milling tool. This machining function assumes that your Swiss-type lathe has the ability to perform simultaneous feed motions with the X, Z and C-axes. The Mill Diam, Polar machining function is most commonly used to machine polygon features on the OD of the part using continuous C-axis feed. Programming of this type in **Face Windows** is carried out in much the same manner as the Mill Polygon machining function.

*Programming of this machining function requires that your Swiss-type lathe has a programmable C-axis and that you have purchased the **PartMaker** C-axis milling option.*



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

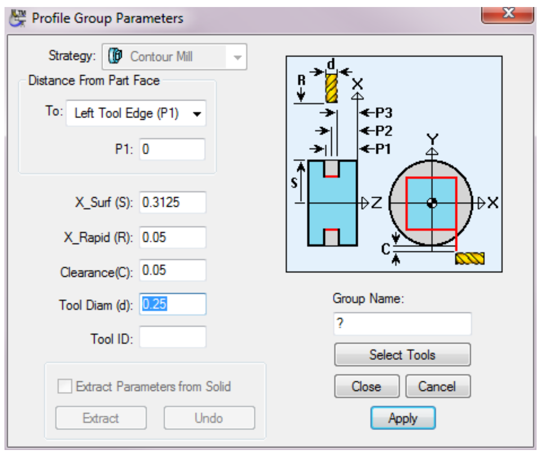
Types of Features Created in Mill Diam Polar Machining Function Window

The Mill Diam Polar window uses an X-oriented “live” tool. There is polar C-axis movement (the C-axis does not lock). The Mill Diam Polar window is not a commonly used window; it is typically used in Surface Machining applications. Please refer to the Advanced Surface Machining manual for more information.

Part Features Dialogs Available in Mill Diam Polar Machining Function

The Mill Diam Polar machining function can be used to create flats. Below, a Profile Group parameter window is shown.

Note: *In the Mill Diam Polar window you are only allowed to use New Profile Group. A New Hole Group is not accessible,*



Distance From Part Face, To: Indicates placement of tool with respect to the part face. The tool can be positioned with respect to the **Left Tool Edge (P1),** the **Tool Center (P2)** or the **Right Tool Edge (P3)**.

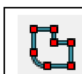
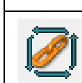
X_Surf (S) Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.

X_Rapid (R) Specifies how much clearance the tool will rapid above your X_Surf

Clearance(C) Specifies the clearance amount the tool will position past the barstock diameter.

Graphics Icons – Machining Functions for Mill Diam Polar

The Machining Functions, Mill Diam Polar use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.

	**Define 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 Geometry 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 with an initial line or arc clicked.
	**2-Point Chain Icon** enables you to specify a begin point and an end point to be chained together.

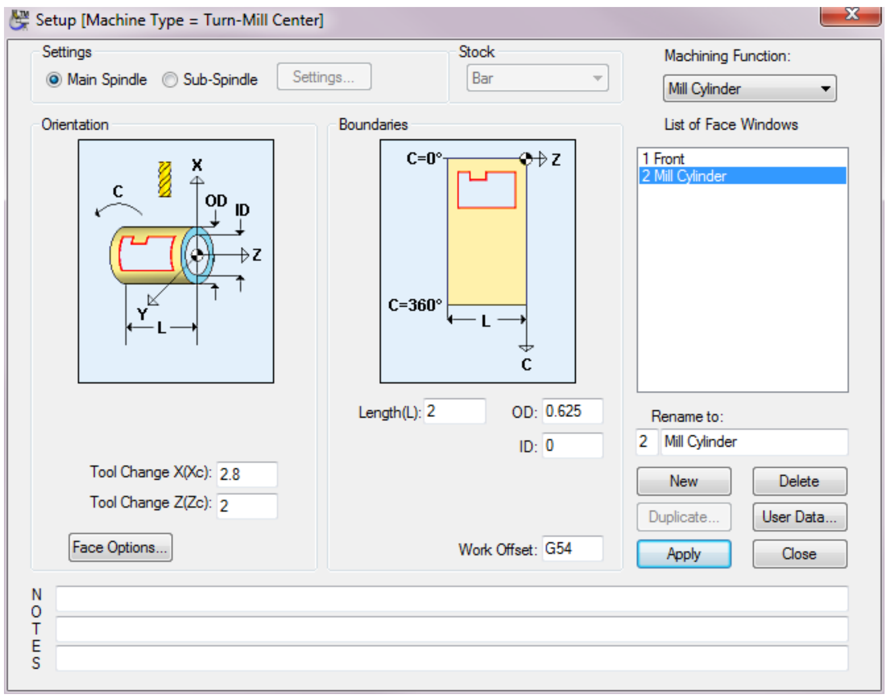
Machining Function: Mill Cylinder

The Mill Cylinder machining function supports continuous milling on the outside diameter of the part. Cylindrical milling assumes that your Swiss-type lathe has the ability to simultaneously feed in the X, Z and C axes.

Using motions of this type, combined with a tool oriented along the X axis of the machine, profiles can be created on the outside diameter of the stock.

Tool positioning is programmed using X, Z, and C axis coordinates. The C position is programmed in terms of degrees of rotation relative to machine C=0 which is clearly shown in the graphic below.

Programming of this machining function requires that your Swiss-type lathe has a programmable C-axis and that you have purchased the ***PartMaker*** *C-axis milling option.*



Length (L)	Length of the finished part
OD	Diameter of the surface to be machined
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill Cylinder Machining Function Window

The Mill Cylinder window uses an X-oriented “live” tool. There is polar C-axis movement (the C-axis does not lock). This window is mainly used for doing contoured pockets and or slots as well as engraving.

The features shown on the parts below are representative examples of features that can be created in the Mill Cylinder window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker Turn-Mill. Where relevant, the locations of these files have been provided.

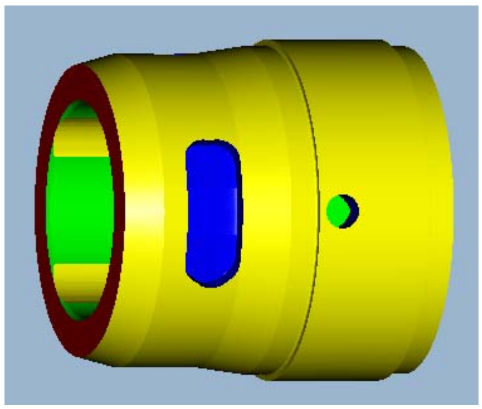


Figure 1. Milled Slot

(Relevant files: C:\PartMaker\Pm-TM\Samples\Mori-Seiki_ZL\ZL.job, ZL.cdb, ZL.tdb)

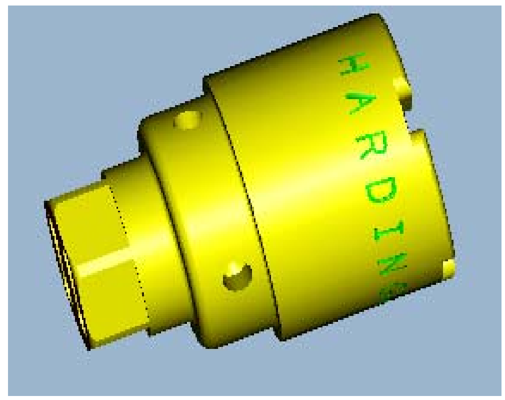
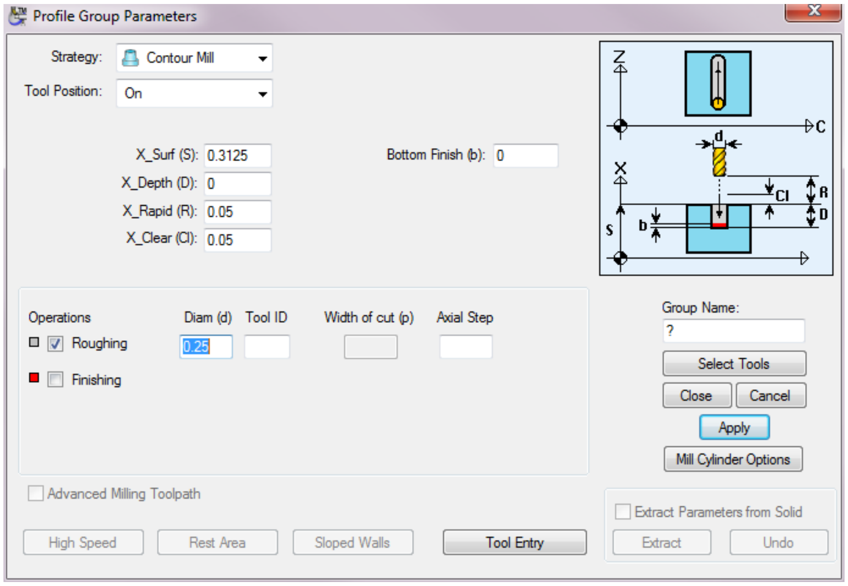


Figure 2. Engraving

(Relevant files: C:\PartMaker\Pm-TM\Samples\Hardinge_TT-65\TT-65.job, TT-65.cdb, TT-65.tdb)

Part Features Dialogs Available in Mill Cylinder Machining Function

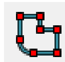
The Mill Cylinder machining function can be used to create Contours, Pockets and Lettering on the outside diameter. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.



Strategy	Indicates the type of feature to be milled, either a Contour Mill or Pocket Mill
Tool Position	For Contour Mill indicates the tool position during cutting, such as Left (climb milling), Right (conventional milling) or On (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above your X_Surf.
X_Clear (C)	Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove when tool position set to Left or Right
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Tool Entry	Indicates the manner of entry of end mill into a feature, either Vertical, Helical, Sloped or Sloped Down and Return

Graphics Icons-Machining Functions for Mill Cylinder

The Machining Functions for Mill Cylinder use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.

	**Define 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 Geometry 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 with an initial line or arc clicked.
	**2-Point Chain Icon** enables you to specify a begin point and an end point to be chained together.
	**Engrave Icon** enables you to use the Engrave Data dialog box to create a profile containing numbers and letters.

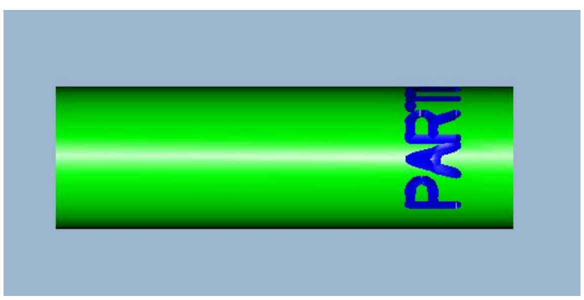
Programming Examples using Mill Cylinder Machining Function

A practical programming application of the Mill Cylinder machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

Example A: Engraving on the Outside Diameter

In the example given below, you will engrave the words "PartMaker" with a ¼ inch Spot Drill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Cylinder Face Window should have an orientation of **X Tool**)

- .25 Inch Spot Drill



Features to be created:

Engraving with the words "PARTMAKER" on the Outside Diameter

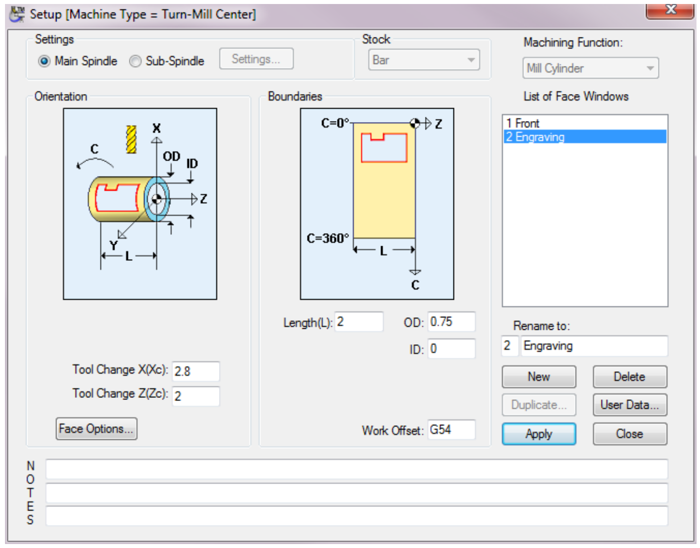
Step 1. Create a Mill Cylinder Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill Cylinder Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill Cylinder**

4 Enter the setup parameters as shown below:



5 Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Features

Note: *When performing engraving it is not necessary to create part geometry.*

Start by entering the CAM mode in the Mill Cylinder Window.

1 Choose **Show Axes** from the View Menu

2 Choose **Show Boundaries** from the View Menu

3 From the Part Features menu, choose the New Profile Group.

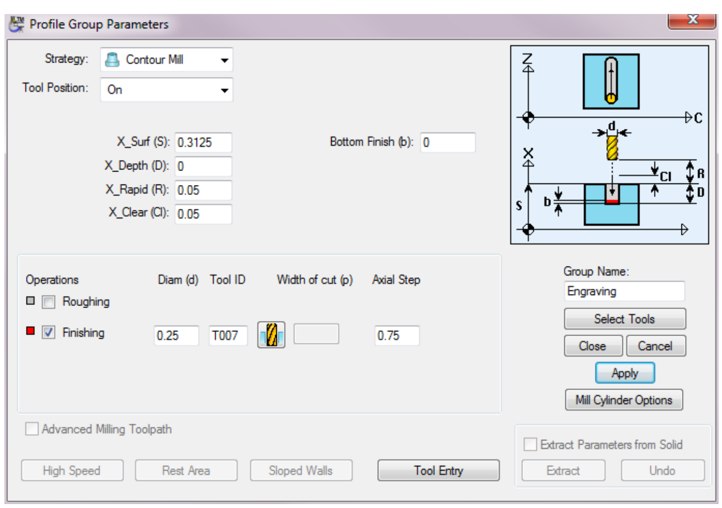
4 Choose **Contour Mill** under **Strategy**

5 Choose **ON** from **Tool Position** to indicate that we will be working on centerline with the tool.

6 Enter an **X_Surf** of 0.3125

7 Enter an **X_Depth** of 0.025

8 Enter the Tool ID number of the 0.25 diameter Spot Drill created and click on **Select Tools**

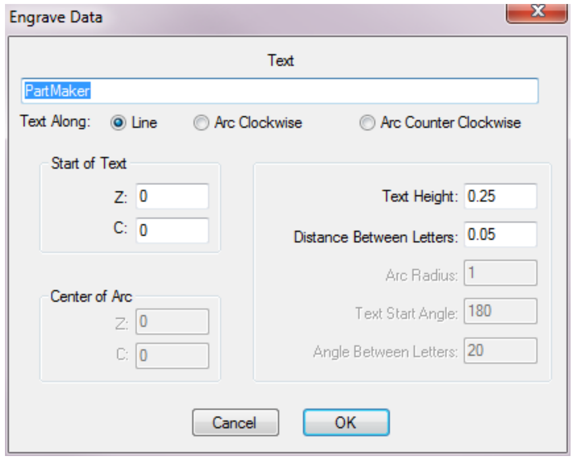


9 When all of the necessary data has been entered click on **Close.**

Step 3. Create Text

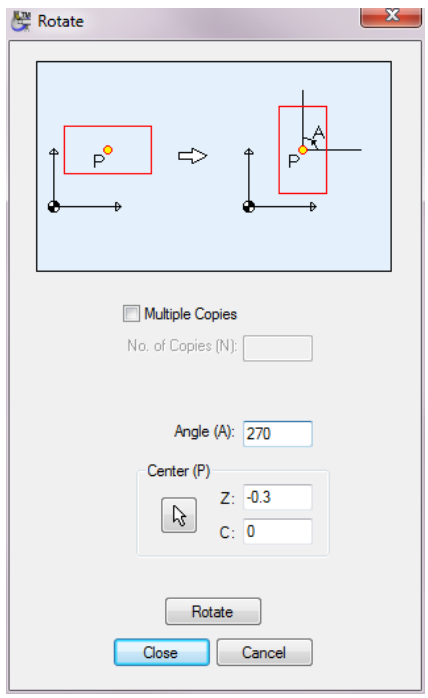
1 Click on the **Engraving Icon** and the Engrave Data dialog box will display.

2 Type the word “Partmaker” in the box shown below then click on **OK**.



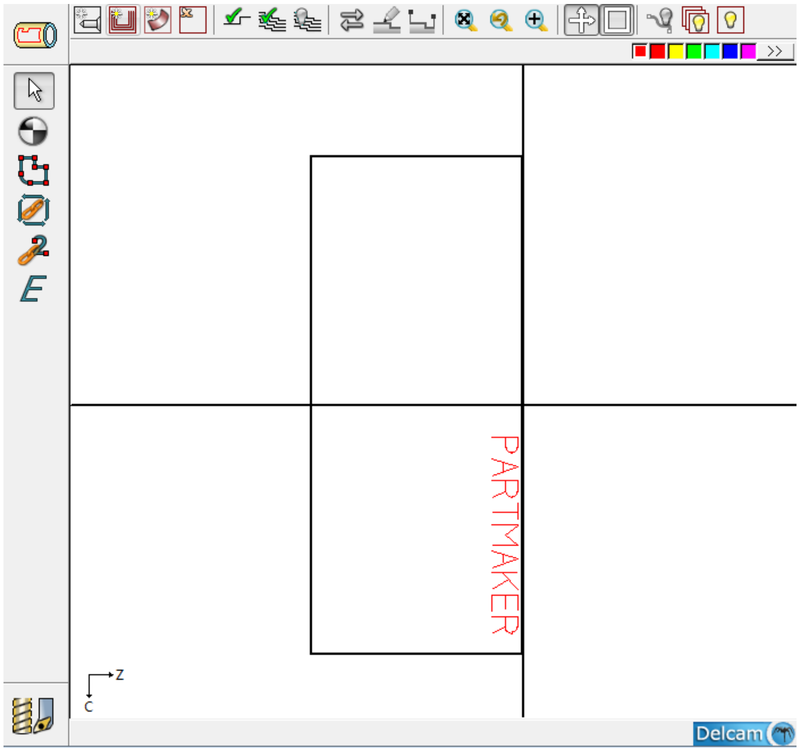
3 Drag a window around the text “PartMaker” and choose the **Rotate** command under **Edit** menu you will get the following dialog box shown below.

4 Fill in all the necessary information from the dialog box below.

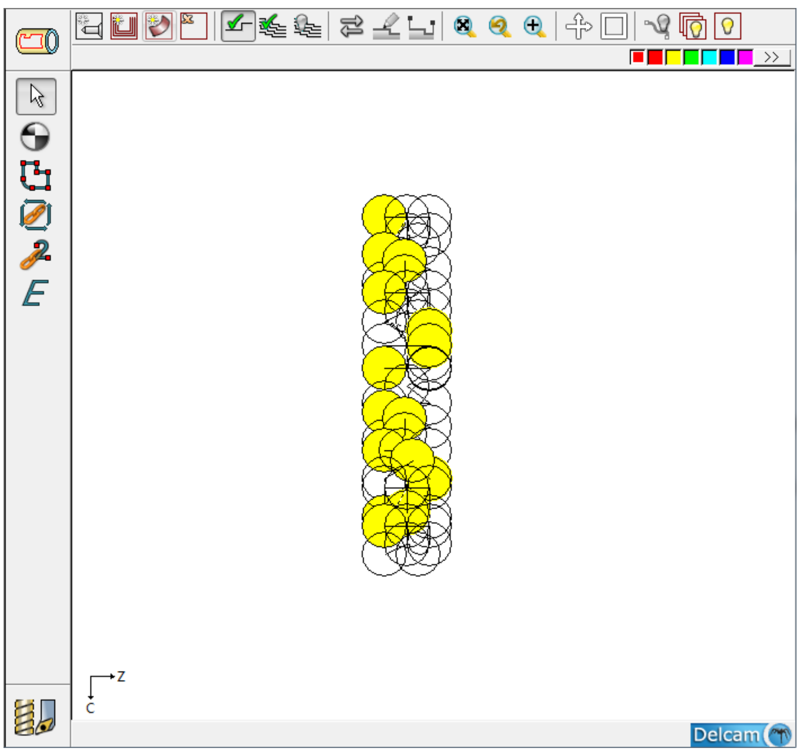


5 When all of the necessary data has been entered click on **OK**

Your completed geometry should appear as shown below:



6 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:

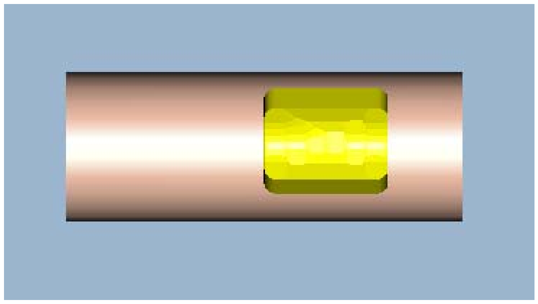


7 Choose **Hide Work Tool Path** to hide the tool path.

Example B: Milling a Contoured Pocket

In the example given below, you will mill a contoured pocket on the diameter with a 1/8-inch End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Cylinder Face Window should have an orientation of **X Tool**)

- .125 Inch Diameter End Mill



Features to be created:

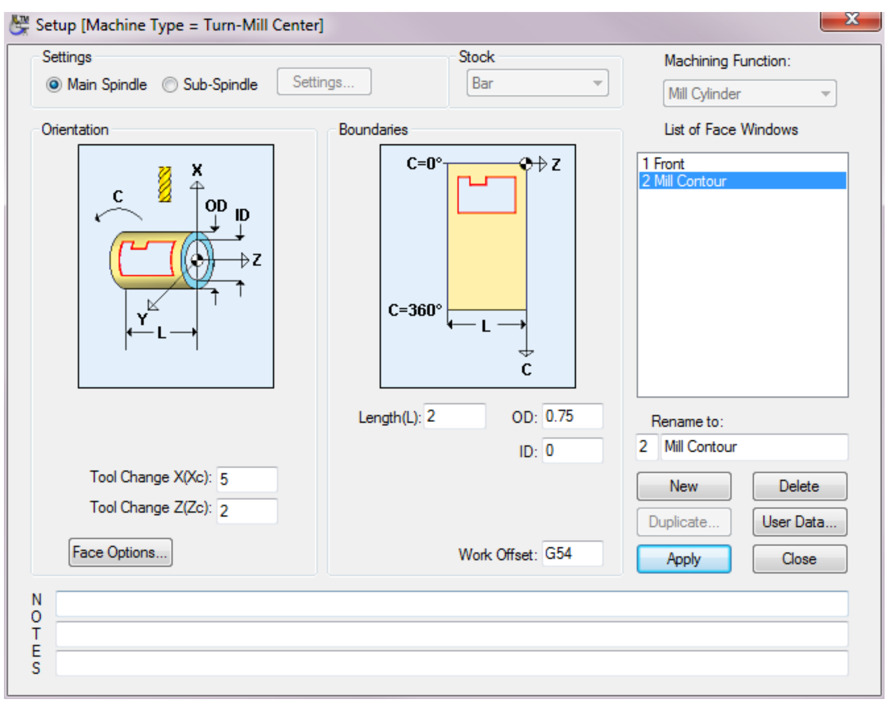
Mill a Contoured Pocket with a .125 diameter End Mill.

Step 1. Create a Mill Cylinder Machining Function Face Window

Note: *When creating a milled feature in PartMaker Turn-Mill, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).*

Start by creating a Mill Cylinder Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill Cylinder**
4. Enter the setup parameters as shown below:



Step 2. Create Part Geometry

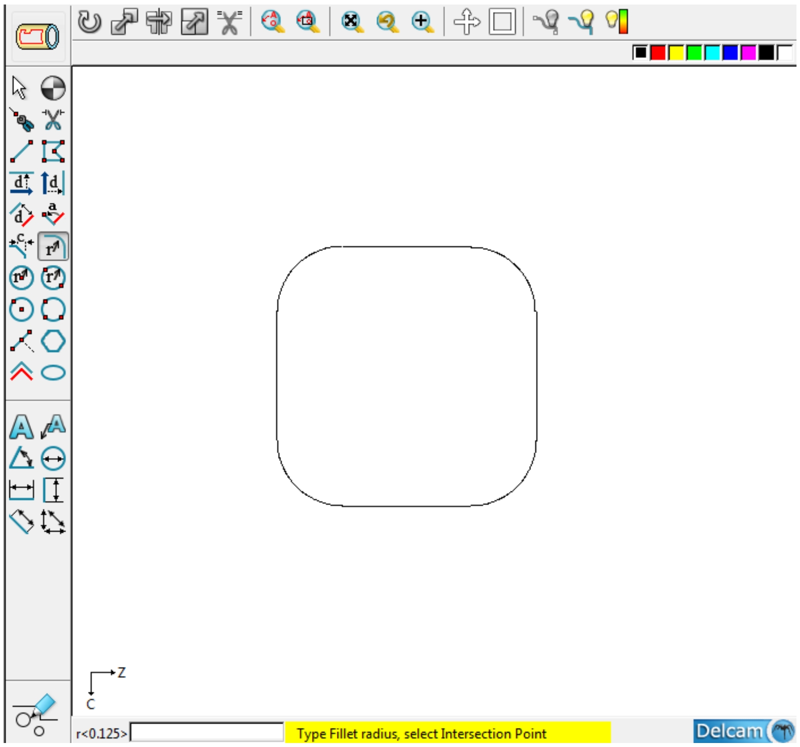
Start by entering the CAD mode in the Mill Cylinder Window. Draw the following features:

5 Choose **Show Axes** from the View Menu

6 Choose **Show Boundaries** from the View Menu

7 Draw a ½ square with .125 radius corners anywhere in the boundaries.

Your completed geometry and CAD Face Window should appear as shown below:



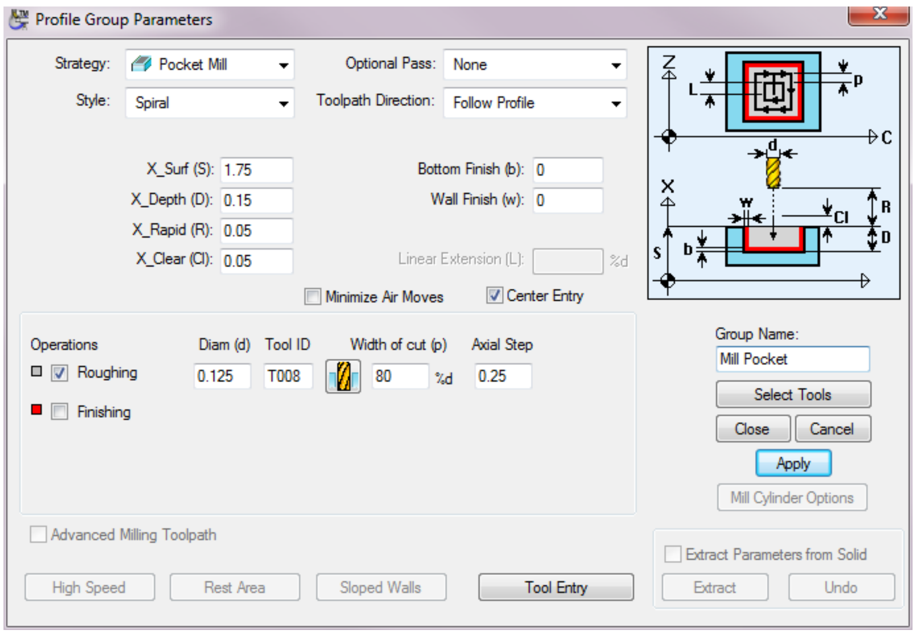
*__Note:__ You can always go to a different Face Window in your *.job file by selecting from the __Window__ menu. Additionally, you can toggle between different Face Windows within the active Window by choosing __Show Windows Icons__ from the __View__ menu and double clicking on the icon representing the Window you want to view.*

Step 3. Create Part Features and assign tool paths

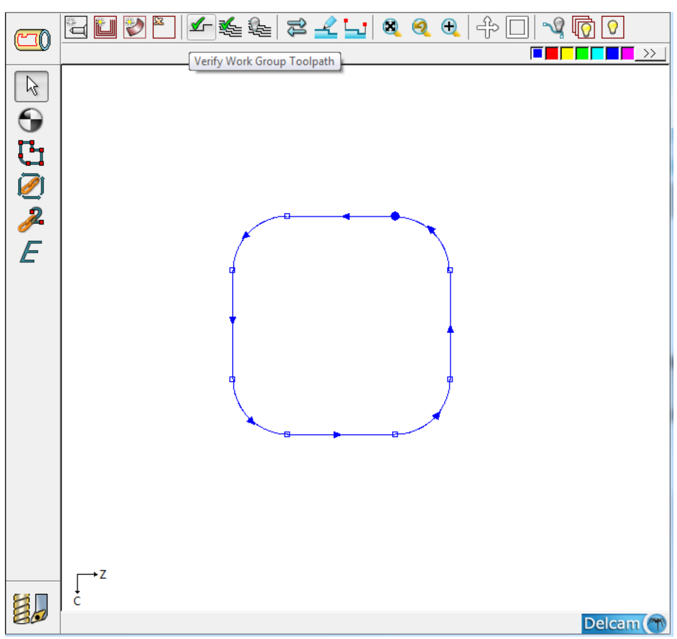
Start by entering the CAM mode in the Mill Cylinder Window. First you will create the tool path to mill the square. To do so:

1. From the Part Features menu, choose New Profile Group
2. Choose **Pocket Mill** under **Strategy**
3. Enter a **X_Depth** of 0.15
4. Enter a **Tool Diameter** of 0.125 and click **Select Tools**
5. In the **Group Name** field, type **Mill Pocket**

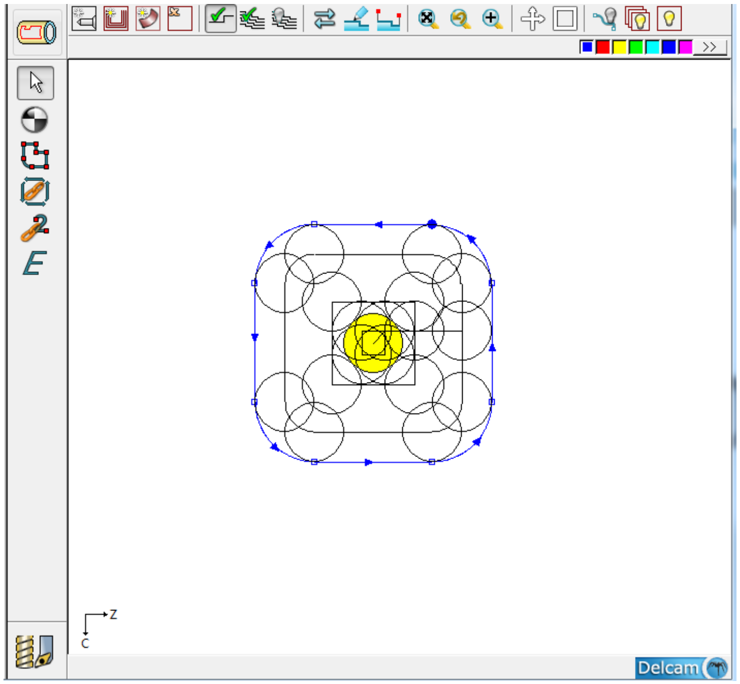
   Your completed **Profile Group Parameters** dialog should appear as shown below:



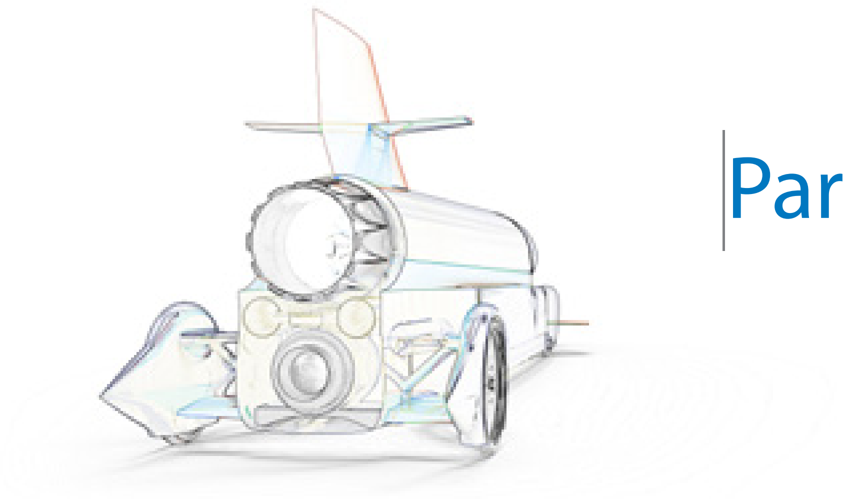
6. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:



7 From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:



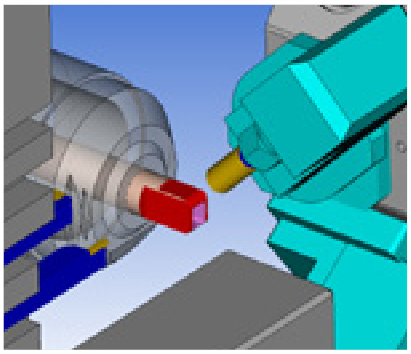
8 Choose **Hide Work Tool Path** to hide the tool path.



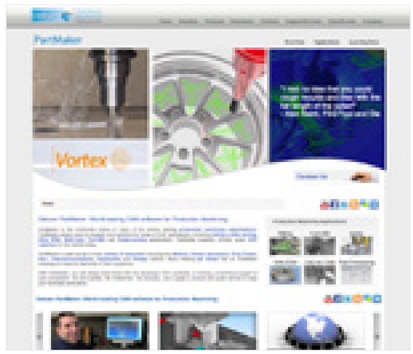
PartMaker 2015

World-leading CAM software for production machining

www.partmaker.com



PartMaker 2015



www.partmaker.com

www.delcam.tv

Unlock the power of Delcam software

LEARNING ZONE

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

Delcam | Advanced Manufacturing Solutions

Powering your productivity

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

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

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

AUTODESK.