

Delcam 

Advanced  
Manufacturing  
Solutions

# PartMaker 2015

PartMaker Mill  
Version 2015 or Higher



User Guide

## Sequence of Programming Steps for PartMaker Mill

1. Complete the Setup dialog to determine the stock, table type and part boundaries you will be working with.
2. Create geometry in CAD mode or choose the Import command from the File menu if you want to import geometry from a CAD system in either a 2D or 3D Solid Model Format. Working with a Solid Model may allow your part boundaries to be determined automatically.



**Note:** To enter CAD Mode click the CAD/CAM switch at the bottom left hand corner of the screen.

The CAD/CAM switch appears as



in CAM Mode and as



in CAD Mode.



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



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

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

6. Choose the **Defaults** command from the **Job Optimizer** menu to set up your defaults.

7. Switch back to CAM Mode clicking the CAD/CAM switch.



8. Choose **New Hole Group** from the **Part Features** menu to add a new hole group.

9. Specify hole locations using the following icons:



Single Hole Icon



Chain Circle Icon



Hole Pattern Icon

10. Choose **New Profile Group**  (or **New Surface Group**  if performing 3D Surface Machining) from the **Part Features** menu to define a new profile group.

11. Create profiles using the following icons:



Define Profile Icon



Chain Geometry Icon



2-Point Chain Icon

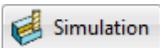
12. Repeat steps 8 through 11 to create groups of all part features required for the job.



13. Choose **Post Config File=?** command from the **Job Optimizer** menu and select the desired post configuration file.



14. Choose the **Generate Process Table** command from the **Job Optimizer** menu to generate a process table.



15. Choose **Simulation** from the **Simulation** menu to watch the part cut in 3D.



16. Choose the **Generate NC Program** command from the **Job Optimizer** menu to generate an NC Program.

---

**PartMaker 2015**

# **User Manual**

**User Guide/PartMaker Mill**



## **Important User Notices**

Copyright 1991-2014 Delcam Ltd. All rights reserved.

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

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

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

### **Acknowledgements**

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

### **Patents**

PartMaker software is subject to the following patents:

Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces

Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

*PartMaker 2015. Published on 08 December 2014*

# Contents

|  |          |
|--|----------|
| <b>Introduction .....</b>  | <b>1</b> |
| Using CAM Software to Program NC Machines .....                              | 1        |
| How PartMaker Improves Productivity .....                                    | 1        |
| Intuitive Windows GUI Assures Quick Learning.....                            | 1        |
| Tools Data Base Keeps Track of Tool Inventory.....                           | 2        |
| Cycles Data Base Stores Sequences of Repetitive Operations .....             | 2        |
| Materials Data Base Facilitates Automatic Feeds and Speeds Calculations..... | 2        |
| Multi Face Programming Simplifies Machining in Different Planes .....        | 2        |
| Optimized Process Plan Generation Saves Programming Time .....               | 3        |
| User Customizable Output Format Eliminates Manual Editing.....               | 3        |
| Verification by Cutting Simulation Catches Programming Errors .....          | 3        |
| About This Guide .....   | 4        |
| <b>Chapter 1: Getting Started With PartMaker .....</b>                       | <b>1</b> |
| In Your Package .....  | 1        |
| Technical Support.....   | 1        |
| Before Using PartMaker... ..   | 1        |
| System Requirements .....  | 2        |
| Starting PartMaker.....  | 3        |
| Installing PartMaker 4 & 5-Axis Mill Post Processors .....                   | 4        |
| Exiting PartMaker .....  | 8        |
| Using On-Line Help .....   | 8        |
| Installing Updates .....   | 9        |
| Working with Databases.....  | 10       |
| Working with Files.....  | 10       |
| <b>Chapter 2: PartMaker Fundamentals .....</b>                               | <b>1</b> |
| Introduction .....   | 1        |
| The Setup Dialog .....   | 1        |
| Table, Stock and Machining Function Setting.....                             | 2        |
| Table Type: Standard .....   | 3        |
| Mill XY Plane .....  | 8        |

|   |    |
|---|----|
| Table Type: Rotary 4 Axis .....                                 | 12 |
| Machining Functions for Block Stock: Mill 4 Axis Plane .....    | 14 |
| Machining Functions for Cylinder Stock: Mill 4 Axis Plane ..... | 15 |
| Machining Functions for Cylinder Stock: Mill Polygon .....      | 16 |
| Machining Functions for Cylinder Stock: Mill Diam, Index .....  | 17 |
| Machining Functions for Cylinder Stock: Mill Cylinder .....     | 18 |
| Table Type: Rotary 5 Axis .....                                 | 19 |
| Machining Functions for Block Stock: Mill XY Plane .....        | 21 |
| Machining Functions for Block Stock: Mill 4 Axis Plane .....    | 22 |
| Machining Functions for Block Stock: Mill 5 Axis Plane .....    | 24 |
| Machining Functions for Cylinder Stock: Mill XY Plane .....     | 26 |
| Machining Functions for Cylinder Stock: Mill 4 Axis Plane ..... | 28 |
| Machining Functions for Cylinder Stock: Mill 5 Axis Plane ..... | 30 |
| Machining Functions for Cylinder Stock: Mill End Index .....    | 32 |
| Machining Functions for Cylinder Stock: Mill End Polar .....    | 33 |
| Machining Functions for Cylinder Stock: Mill Polygon .....      | 34 |
| Machining Functions for Cylinder Stock: Mill Diam, Index .....  | 35 |
| Machining Functions for Cylinder Stock: Mill Cylinder .....     | 36 |
| Machining Functions for Cylinder Stock: Turning .....           | 37 |
| Table Type: Vertical Rotary .....                               | 40 |
| Machining Functions for Block Stock: Mill XY Plane .....        | 42 |
| Machining Functions for Block Stock: Mill 5 Axis Plane .....    | 45 |
| Machining Functions for Cylinder Stock: Mill XY Plane .....     | 47 |
| Machining Functions for Cylinder Stock: Mill 5 Axis Plane ..... | 49 |
| Machining Functions for Cylinder Stock: Mill End Index .....    | 51 |
| Machining Functions for Cylinder Stock: Mill End Polar .....    | 52 |
| Machining Functions for Cylinder Stock: Mill Diam, Index .....  | 53 |
| Machining Functions for Cylinder Stock: Mill Cylinder .....     | 54 |
| Machining Functions for Cylinder Stock: Turning .....           | 55 |
| Table Type: Tombstone .....                                     | 57 |
| Machining Functions for Block Stock: Mill XY Plane .....        | 60 |
| Machining Functions for Cylinder Stock: Mill XY Plane .....     | 61 |
| Working in a Face Window .....                                  | 65 |
| Using the CAD/CAM Switch .....                                  | 66 |
| Using the Color Bar .....                                       | 66 |
| Using the Icon Toolbars .....                                   | 66 |
| Main Toolbar .....  | 66 |
| Face Window Toolbar – CAM Mode .....                            | 66 |
| Face Window Toolbar – CAD Mode .....                            | 67 |
| Creating Groups of Part Features .....                          | 67 |
| Using the Graphics Icons in PartMaker CAD mode .....            | 68 |
| Geometry Drawing Icons .....                                    | 69 |
| Dimensioning Icons .....  | 70 |
| Snap Modes .....  | 71 |
| Using the Graphics Icons in PartMaker CAM mode .....            | 72 |

|  |          |
|--|----------|
| <b>Chapter 3: Creating a Sample Part</b> .....             | <b>1</b> |
| Introduction .....   | 1        |
| How You Will Create Milldemo .....                         | 1        |
| Start PartMaker.....                                       | 4        |
| Generate a Process Table.....                              | 43       |
| Simulate the Process.....                                  | 46       |
| Generate an NC Program.....                                | 47       |
| <br>   |          |
| <b>Chapter 4: PartMaker® Simulation</b> .....              | <b>1</b> |
| Introduction .....   | 1        |
| The Simulation Toolbar .....                               | 3        |
| Simulation Features.....                                   | 4        |
| Reset .....  | 4        |
| Show Travel Limits .....                                   | 4        |
| Show Finished Part .....                                   | 4        |
| Save Finished Part As STL file .....                       | 4        |
| Compare Finished Part to Solid Model .....                 | 5        |
| Comparison Options .....                                   | 5        |
| View Setting Features.....                                 | 5        |
| View Manipulation Features: Rotate, Move and Zoom .....    | 7        |
| Sectioning Features.....                                   | 8        |
| Simulation Options.....                                    | 9        |
| Pause On .....   | 9        |
| Settings .....   | 10       |
| Enable Collision Checking/Report Collision Types .....     | 10       |
| Display Control .....                                      | 11       |
| Display Quality.....                                       | 12       |
| Colors and Textures .....                                  | 12       |
| <br>   |          |
| <b>Appendix A:Materials</b> .....                          | <b>1</b> |
| PartMaker Material Library .....                           | 1        |
| <br>   |          |
| <b>Appendix B: Included Post Configuration Files</b> ..... | <b>1</b> |
| PartMaker Post Library .....                               | 1        |
| Inch Post Configuration Files .....                        | 1        |
| Metric Post Configuration Files.....                       | 8        |
| <br>   |          |
| <b>Appendix C: GPF Files</b> .....                         | <b>1</b> |
| Why GPF Files.....   | 1        |

**Appendix D: Programming Directly on a Solid Model..... 1**

- Introduction ..... 1
- Key Definitions ..... 1
- Solids Programming Tutorial ..... 3
- How you will create the Sample part ..... 3
- Getting Started..... 5
- Starting PartMaker..... 5
- View the Finished Sample ..... 6
- Importing a Solid Model..... 6
- Select a Material for the Part..... 7
- Editing the Coordinate System ..... 7
- Setting up the Face Window..... 8
  - Programming a Pocket Operation.....11
  - Programming a Profile Operation .....15
  - Programming a Reamed Hole Group .....19
  - Programming a Tapped Hole Group.....22
  - Programming a Bore Hole Group .....26
- Generating the Process Table..... 30
- Simulating Machining in 3D ..... 30
- Generating the NC Program..... 31

**Appendix E: The 4th-Axis Tutorial Part ..... 1**

- Introduction ..... 3
- Getting Started..... 3
- Starting PartMaker Mill ..... 4
  - Setting Preferences for the 4th-Axis Exercise.....4
- Creating the Flats Using Mill Polygon ..... 8
  - Setting Up the Face Window .....8
  - Creating Geometry in CAD Mode .....11
  - Creating the Hexagonal Profile.....13
  - Creating the Octagonal Profile.....15
  - Creating the Octagonal Profile.....18
  - Creating Indexed Holes Using Mill Diameter Index.....20
  - Creating the Indexed Holes .....24
  - Creating Helical Channels Using Mill Cylinder .....26
  - Creating the Helical Channels .....30
  - Creating Offset Holes Using Mill 4 Axis Plane .....33
  - Creating the Off Center Holes.....36
- Creating an NC Program ..... 39
  - Generating a Process Table .....39
  - Selecting a Post Processor.....40
  - Simulating the Process .....40
  - Generating an NC Program .....41

|  |          |
|--|----------|
| <b>Appendix F: The 5-Axis Milling Tutorial Part.....</b> | <b>1</b> |
| How you'll create the 5 Axis Mill tutorial part.....     | 3        |
| Introduction .....                                       | 3        |
| Getting Started.....                                     | 3        |
| Starting PartMaker.....                                  | 4        |
| Setting Up the Face Window .....                         | 9        |
| Setting Up the 5-axis Table .....                        | 10       |
| Transfer geometry for a 5 axis plane .....               | 11       |
| Perform Milling Operations for the Chosen Plane.....     | 14       |
| Creating the Contour Mill Profile .....                  | 14       |
| Develop the Tool Path .....                              | 16       |
| Creating the Milled Pocket.....                          | 19       |
| Duplicating the Face Window .....                        | 22       |
| Creating the NC Program .....                            | 23       |
| Generating a Process Table .....                         | 23       |
| Selecting a Post Processor.....                          | 24       |
| Simulate the Machining of the Part .....                 | 24       |
| Generate the NC program .....                            | 25       |

*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 improvement in productivity 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, 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 their knowledge into the software. Using **PartMaker**, your parts can 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, which includes part numbers and reordering information among others. The tool descriptions are kept in a tool file. The tools used on different machines may be kept in multiple tool files.

## Cycles Data Base Stores Sequences of Repetitive Operations

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

## Materials Data Base Facilitates Automatic Feeds and Speeds Calculations

A *materials database* allows the user to store materials data that is used in the automatic calculation of feedrates 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.

## Multi Face Programming Simplifies Machining in Different Planes

You can program machining operations in different faces (planes) and specify up to eight different faces per part. A separate **Face Window** is associated with each face of a part. **PartMaker** determines the order of machining based on the angle you specify for each face from the smallest angle to the largest. You can choose to finish all machining operations on one face before proceeding to the next or perform all operations with a tool on all faces, then change tools.

## 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.

## 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 obstacle 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 flexible *ConfigPost* application that allows the user to create and save custom postprocessor configuration files for virtually any CNC 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** is sold bundled with a graphic verifier that reads a part program and graphically simulates the entire cutting process utilizing solid modeling techniques. This add-in allows the programmer to catch errors before expending machining time.

## About This Guide

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

- Chapter 1 “Getting Started With PartMaker”** explains how to get started with **PartMaker**. This chapter includes a description of the hardware and software you need to run **PartMaker** and tells you how to install **PartMaker** and related files. It also shows you how to access **PartMaker** on-line context sensitive help.
- Chapter 2 “PartMaker Fundamentals”** discusses the unique programming approach **PartMaker** employs for programming CNC mills. 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 “Mill Programming Tutorial”** shows you how to create a sample part using some of the most popular features of **PartMaker**. We recommend you go through this chapter to familiarize yourself with the **PartMaker** application.
- Chapter 4 “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 parameter. For information on specific PartMaker dialogs, use the context sensitive help that comes with PartMaker. This context sensitive help can be found by pressing the F1 key on your keyboard.

---

# Chapter 1: Getting Started With PartMaker

## In Your Package

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

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

## Technical Support

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

- Phone:** 215-643-5077  
**Fax:** 215-653-0105  
**e-mail:** support@partmaker.com  
**Web Site:** <http://www.PartMaker.com>

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:

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

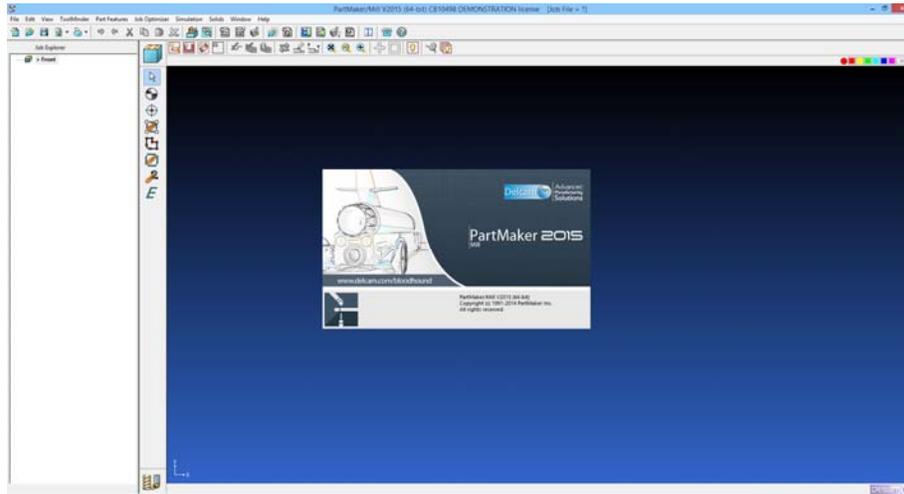
64-bit and 32-bit versions of PartMaker are available and are supported on Windows 7. Only the 64-bit version of PartMaker is compatible with Windows 8 PCs.

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

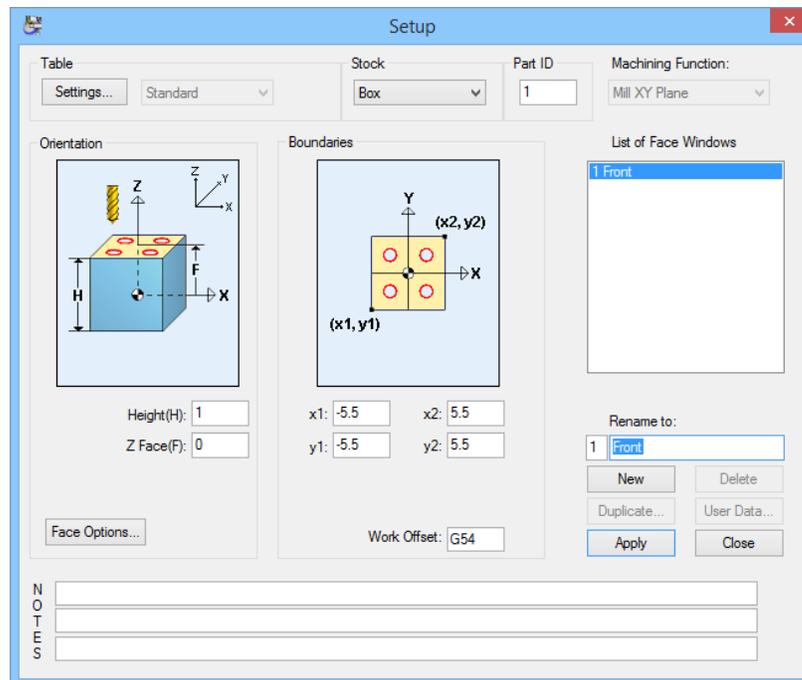
## Starting PartMaker

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

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



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



## Installing PartMaker 4 & 5-Axis Mill Post Processors

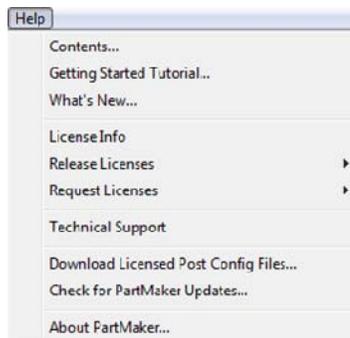
PartMaker Mill Version 2014 and higher features a unique capability for downloading your licensed Mill post processors and related files directly through the software's user interface. This feature is helpful for both new users installing the software for the first time and existing users who wish to 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 as described below.

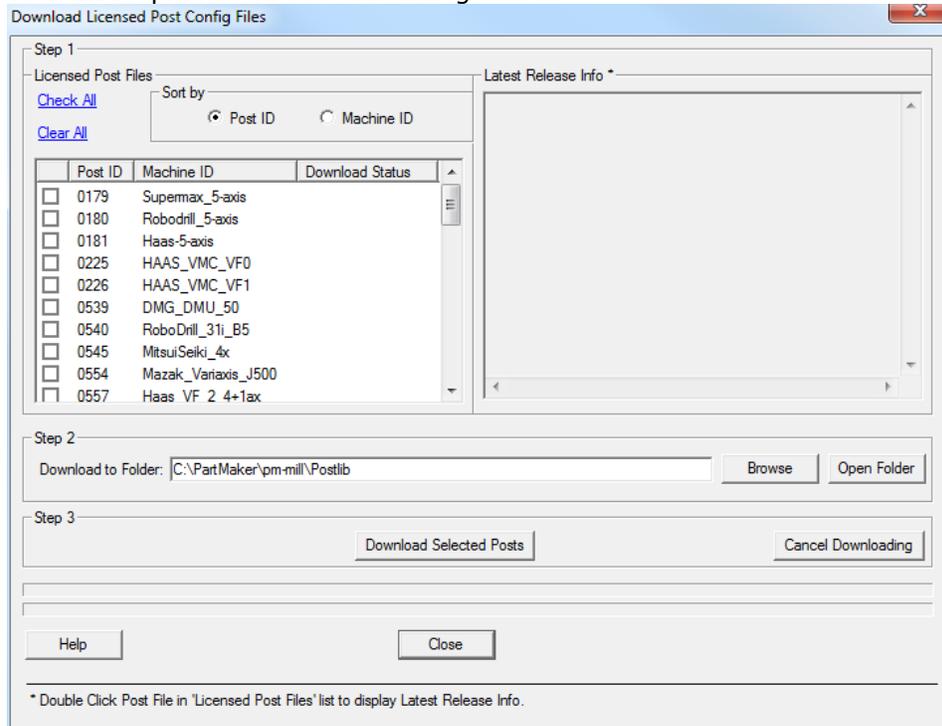
Once you have successfully launched the Mill application, you can install the Mill post processors that you have licensed directly from the PartMaker user interface.

To do so:

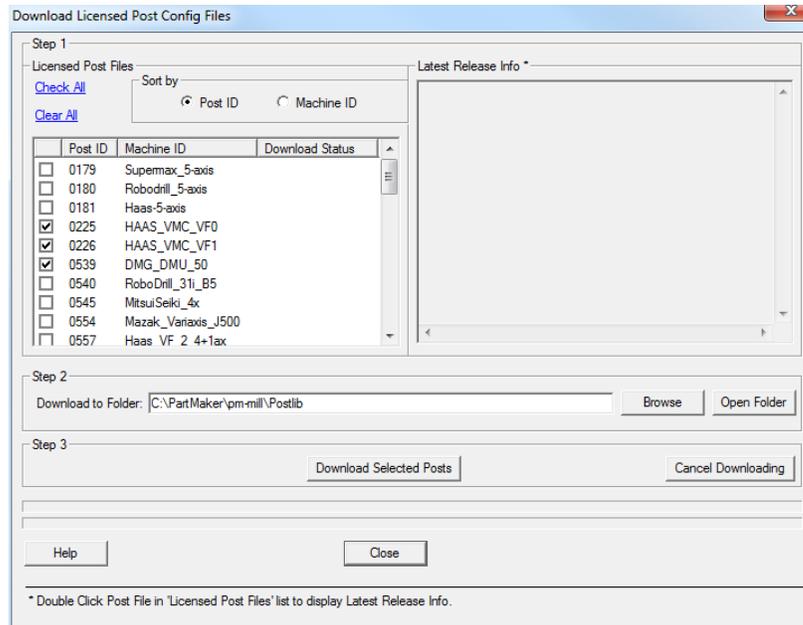
- 1 From the Help menu, choose Download Licensed Post Config Files...



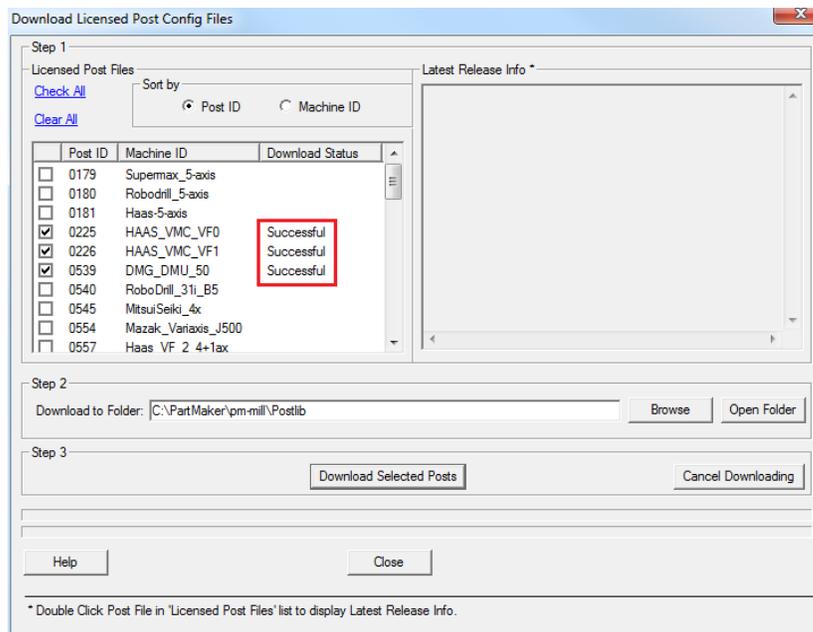
- 2 You will be presented with the dialog below:



- 3 Check the boxes to the left corresponding to 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 the specified 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.



If you receive any message other than "Successful" after this process, contact PartMaker technical support at [support@partmaker.com](mailto: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 previous steps, you will have downloaded what are known as “distribution post processors,” the library post processor support by PartMaker Inc. for particular machine-control combinations. PartMaker post processors 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 CNC Mill machines utilizing the full capability of your machine tool. All PartMaker post processors are customizable, by you, the user.

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

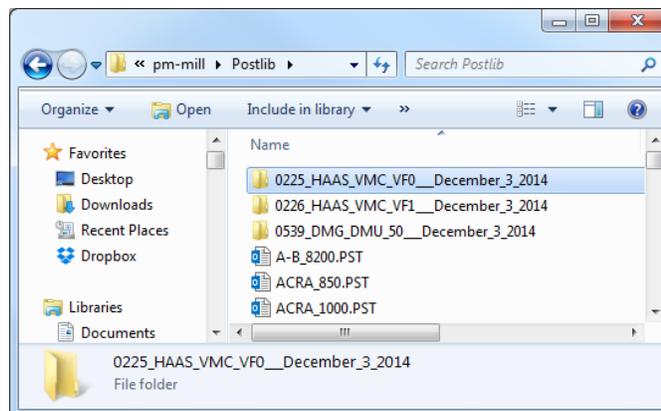
- 1 The Post Processors your company has licensed** – This includes the \*.pst file. PartMaker defines a post processor as a machine, control combination.
- 2 Related Simulation Files** – PartMaker’s Advanced and Full Machine simulation (FMS) modules use additional files to simulate the architecture of your machine. You will need to use the \*.MCH corresponding to your post processor to view your machine’s working envelope when using Advanced Simulation. If you have licensed FMS, 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 4 of this manual for information on using PartMaker’s 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 download in this example:

- 1** The picture below shows the result of the download above:

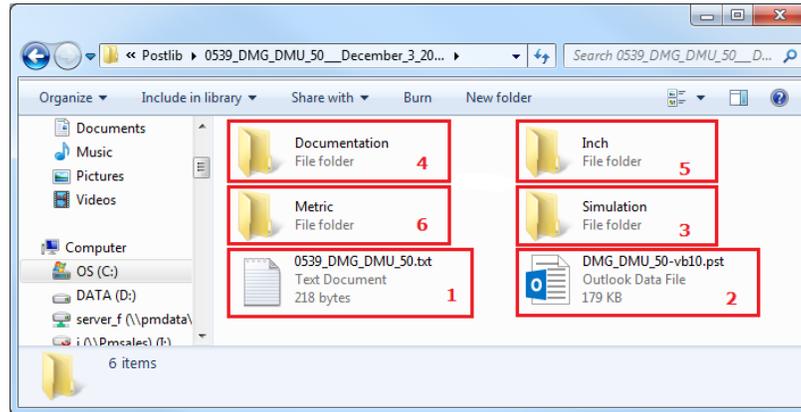


**Note:** PartMaker Mill comes with an extensive library of 3-Axis post processors as seen in the image.



- 2** Double clicking the folder, 0539\_DMG\_DMU\_50\_\_December\_3\_2014, above reveals the following where:
  - 1** A \*.TXT is a readme file explaining changes made in the most recent version of the post processor

- 2 The \*.pst, the post processor file
- 3 A folder containing the simulation, \*.MCH and \*.FMS referred to above
- 4 A folder containing the machine specific addendum and FMS reference kit referred to above
- 5 A folder containing tutorial files in the inch system corresponding to the tutorial in the user guide
- 6 A folder containing tutorial files in the metric system corresponding to the tutorial in the user guide



*Tip: If using Advanced and/or Full Machine Simulation, placing the \*.FMS and \*.MCH in the same directory as the \*.PST file will allow your simulation files to open automatically when loading the post processor.*

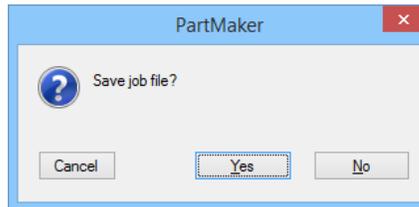
### More on Mill Post Processors

PartMaker Inc's application staff maintains a library of well documented well supported Mill post processors. Once you start to customize your own post processor using the ConfigPost application, you create a "non-distribution" or ND post processor.

Please also note, PartMaker is constantly updating and improving its 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.

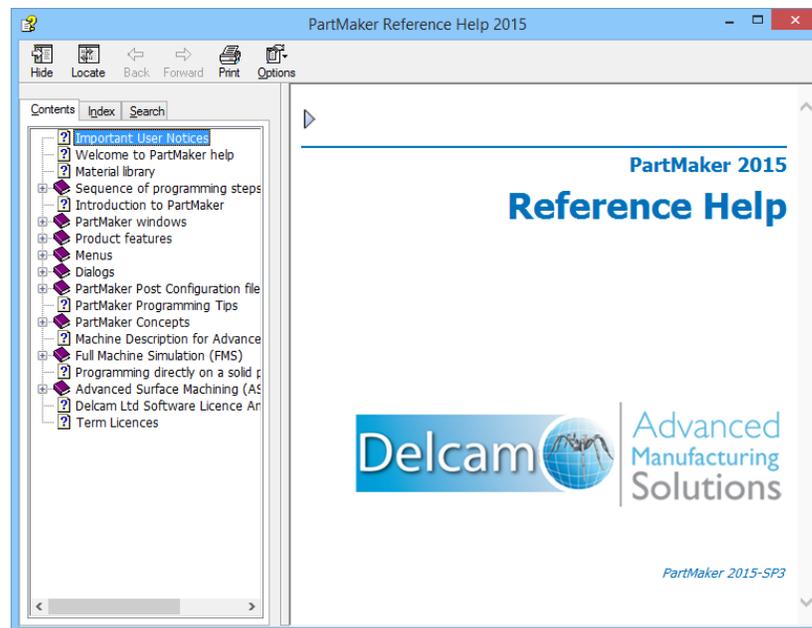
## Using On-Line Help

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** (shown on the following page).

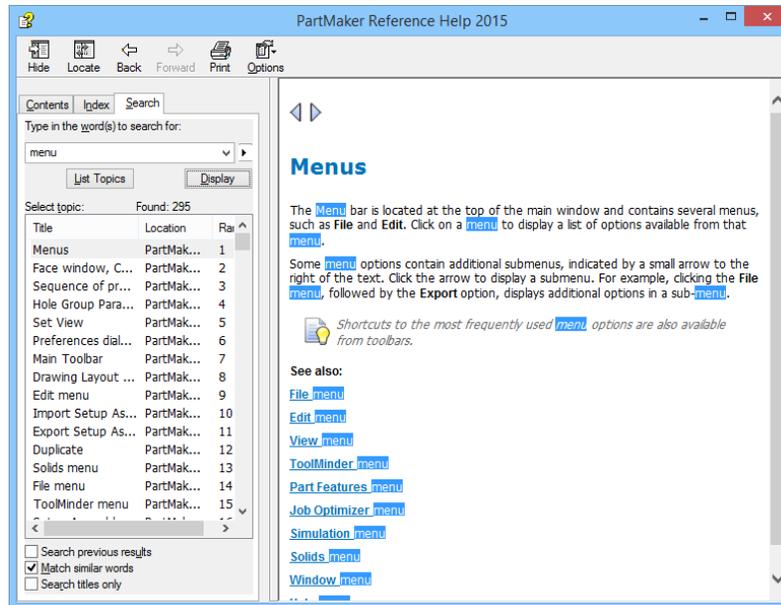


- 2 Click an index to view a list of help topics.
  - 3 Click the topic you want to read about.
- Information about the selected topic appears in the window.



**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 different file types for storing tools, cycles, and material information as well as your jobs and the postprocessor files used to create NC programs for machining.

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

# Chapter 2: PartMaker Fundamentals

## Introduction

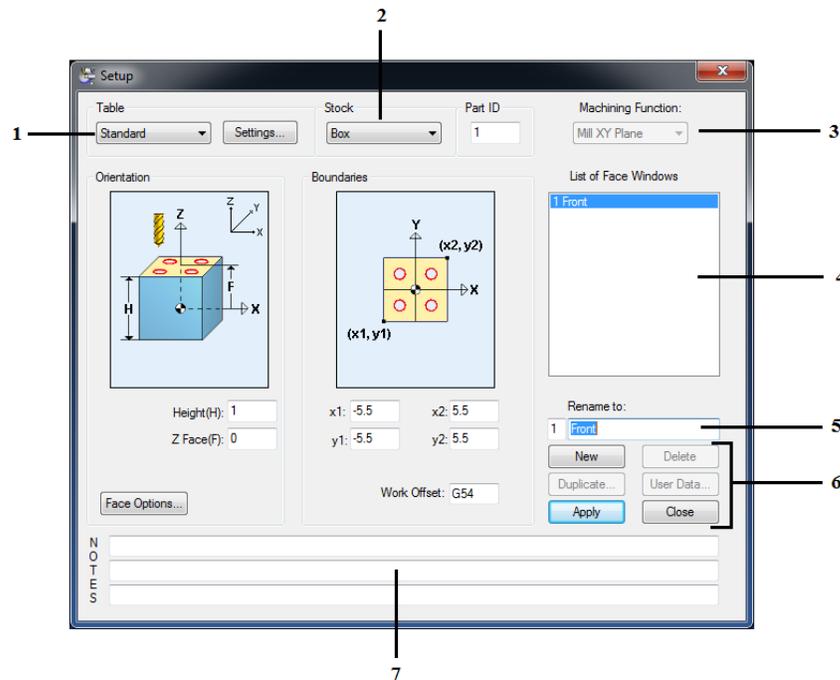
This chapter provides an overview of the **PartMaker** environment. You will learn basic concepts such as how to work in a face window to define part geometry in CAD Mode and locations of holes and profiles in CAM mode.

## The Setup Dialog

Any programming exercise in PartMaker begins with the **Setup** dialog.

The **Setup** dialog is displayed when launching the PartMaker application. It can also be accessed at any time during the use of PartMaker by choosing **Setup** from the **View** menu. It can also be accessed by clicking on the short cut to the Setup dialog icon in the upper left corner of any PartMaker **Face Window**.

The diagram below highlights the various aspects of the **Setup** dialog.



- 1 Table:** Determines the type of machining table you will be making a part on. Choices here are **Standard**, **Tombstone**, **Rotary 4-axis**, **Rotary 5-axis** and **Vertical Rotary**. Each of these items will be discussed in detail in the pages to follow.
- 2 Stock:** Determines the type of stock being machined, either a prismatic **Block** or round **Cylinder**. Once the setup dialog has been completed, you can use PartMaker's **Define Custom Stock Shape** command to create a custom, user defined stock (see Chapter 5 for information on this feature).
- 3 Machining Function:** Specifies the machining function to be used with in a given Face Window. Options under this drop down menu include Mill XY Plane, Mill 4-Axis Plane, Mill 5-Axis Plane, Mill End, Index, Mill End, Polar, Mill Polygon, Mill Diam, Index, and Mill Cylinder. The ability to access and appearance of each of these items will change depending on the selections made for Table and Stock.

- 4 **List of Face Windows:** Presents the list of Face Windows created for a particular job file.
- 5 **Rename To:** States the name of a given Face Window. You should give each Face Window you are working with an appropriate name based on the machining occurring within that Face Window. The number of the left of this field determines the order of the Face Window when you have a number of Face Windows in a particular job file.
- 6 **Function Buttons:** These buttons allow you to perform the following operations in the **Setup** dialog:
 

|                          |   |
|--------------------------|---|
| <b>&lt;New&gt;</b>       | Creates new face windows of different machining function types, adding to the List of Face Windows.   |
| <b>&lt;Delete&gt;</b>    | Deletes the currently selection from the List of Face Windows. This button will only be active when you have more than one Face Window and DO NOT have a Process Table generated. |
| <b>&lt;Duplicate&gt;</b> | 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.  |
| <b>&lt;User Data&gt;</b> | Allows the user view the User Data for a given post processor. Only active if a post processor is loaded.   |
| <b>&lt;Apply&gt;</b>     | Accepts or applies any new parameters or data entered into the various fields in the <b>Setup</b> dialog, and the dialog remains open.  |
| <b>&lt;Close&gt;</b>     | Accepts or applies any new parameters or data entered into the various fields in the <b>Setup</b> dialog and closes the dialog.   |
- 7 **Notes:** Allows you to specify notes about a particular Job or Face Window. Depending on the configuration of the post processor you choose, the contents of these fields can be output into your NC program. See the ConfigPost User Guide that came with your PartMaker Software for more information on post processor configuration.

## Table, Stock and Machining Function Setting

Whenever starting a programming exercise in PartMaker Mill, the first step is to determine the Table type you will be using for a part. You can only set one type of Table in a given job file. The table type chosen in the *first* Face Window will determine the type of machining Table used throughout the job. It will also determine the appearance of the **Face Options** dialog, which is explained in greater detail in the pages to follow.

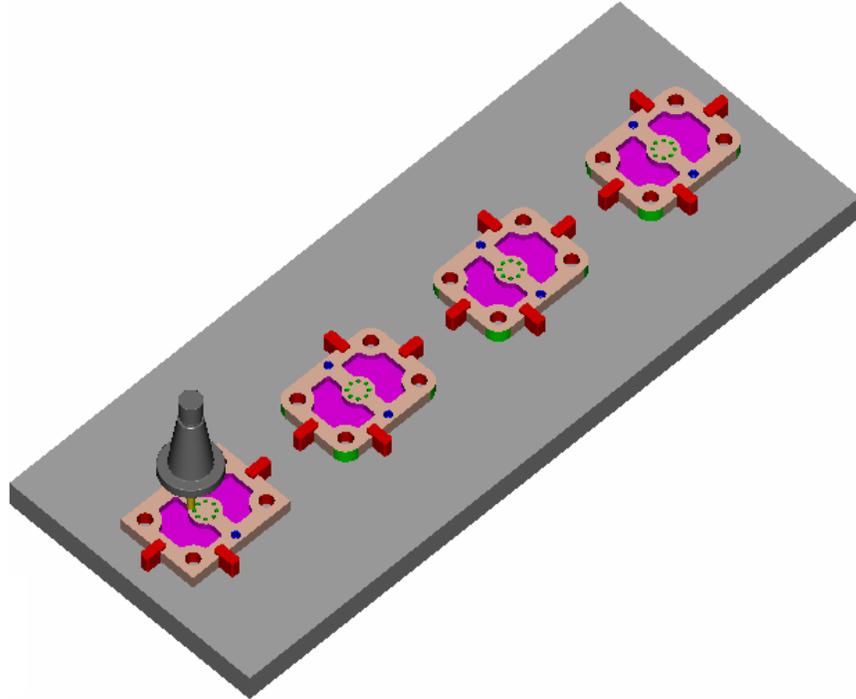
Once a job file has been created, the Table type can be changed by changing the selection for the Face Window in the list of Face Windows.

Once you have determined the type of machining Table you will be using in a given job, you will proceed to determine the type of **Stock** being used. These two selections will then determine the types of machining functions available to you.

The various combinations of Table, Stock and Machining Function types will be explained to follow.

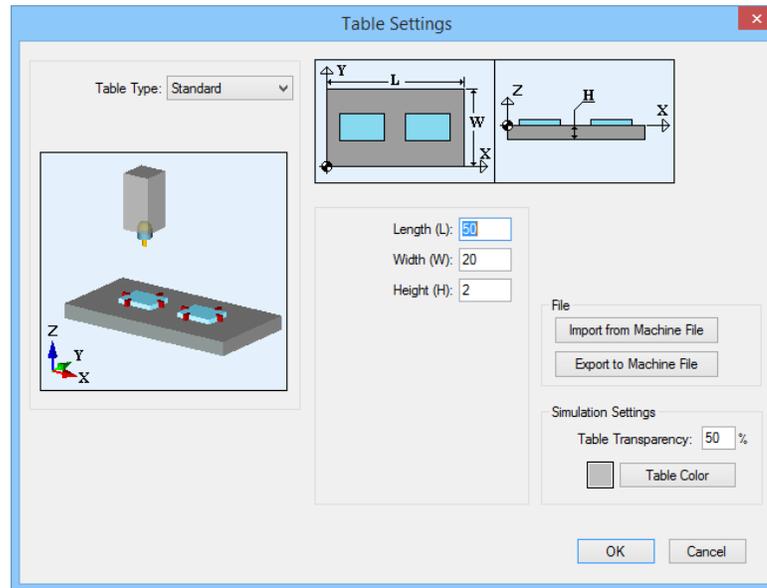
## Table Type: Standard

Setting the Table Type to **Standard** assumes you are working on a vertical machining center with a flat machining table. This selection should generally be set when programming for a vertical machining center and you wish to see the stock being machined relative to the machining table. This Table Type is especially helpful when machining multiple pieces in a single setup such as in the picture shown below:



## Settings

The **Settings** dialog is accessed for the **Table Type: Standard** by clicking the **Settings** button. The **Settings** dialog is used to define the dimension and appearance of the machining table in PartMaker's 3D machining simulation.



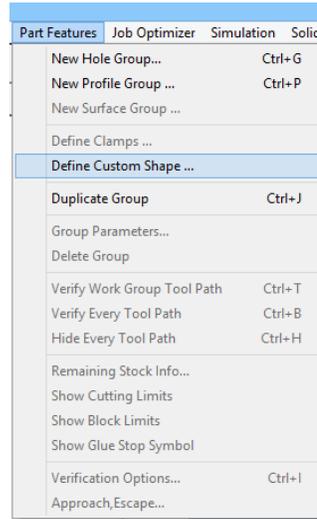
- Length (L):** Specifies the length along the X axis of the machining table
- Width (W):** Specifies the length along the Y axis of the machining table
- Height (H):** Specifies the height or thickness of the machining table
- Import From Machine File** Clicking this button allows you to import the parameters of a previously created machining table from an \*.MCH file which saves you from having to define a new machining table for every PartMaker JOB file
- Export to Machine File** Clicking this button allows you to export the parameters of a specific machining table as an \*.MCH file for use with future jobs. Exporting these parameters to a machine file allows you to setup a number of different types of machining tables
- Table Transparency** Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent
- Table Color** Allows you to choose the table color of the machining table from the available color pallet

## Stock

When the Table is set to **Standard**, you can choose from two types of stock, **Block** or **Cylinder**. The Block selection assumes you are working with prismatic stock, while the cylinder selection assumes you are working with cylindrical stock being held vertically.

## Define Custom Shape

Use the **Define Custom Shape** dialog to define a custom shape in PartMaker.



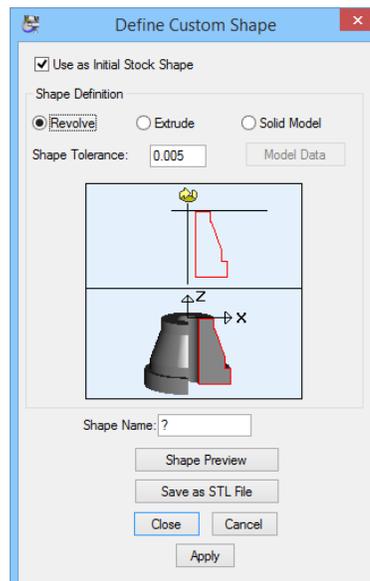
To display the **Define Custom Shape** dialog, select **Part Features > Define Custom Shape**.

The following settings are available:

**Use as Initial Stock Shape** — Select this option if you want to use the custom shape as the initial stock shape, as shown in simulation.

**Shape Definition** — Select the method for defining the custom shape from the geometry:

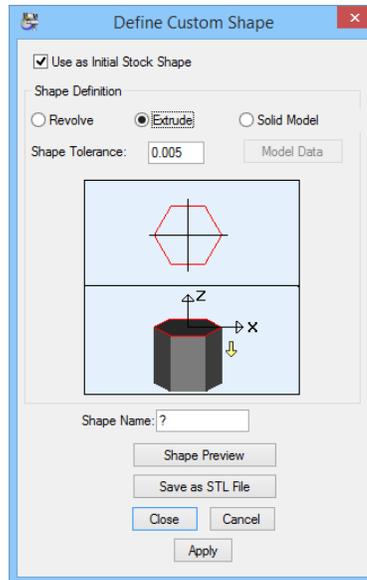
**Revolve:** If you are using 2D profile geometry to define the custom shape, selecting this option creates the shape by revolving the 2D profile around a vertical axis.



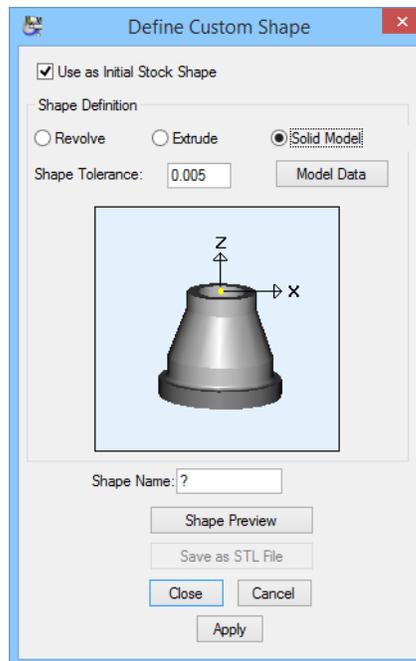


You must draw the profile in the fourth quadrant. Also ensure that 2D profile of the custom shape is **closed**, so the start and end points coincide.

**Extrude:** If you are using 2D profile geometry to define the custom shape, selecting this option creates the shape by extrusion. Specify the extrusion vector (the stock length) on the Setup dialog.



**Solid Model:** Select to create the shape using a solid model file. Click the <Model Data> button to display the [Model Data dialog](#), where you can select the solid model file you want to use.



**Shape Tolerance** — Enter a tolerance value for the custom shape. PartMaker uses this value when saving the custom shape to an .stl file and when calculating surface machining toolpaths whose machining limits are defined using this custom shape.

**Shape Name** — Enter a name for the custom shape.

**Shape Preview** — Click to display the **Stock Shape Preview** window, where you can preview a 3D representation of the custom shape.

**Save as STL File** — Click to save the custom shape to an **.stl** file. PartMaker displays the **Export STL File** dialog, where you can specify the folder location and filename of the file.

**Close** — Click to save any changes you have made and close the dialog.

**Cancel** — Click to discard any changes you have made and close the dialog.

**Apply** — Click to save any changes you have made to the dialog.

## Part ID

Specifies which part the current Face Window applies to on a machining table. Different face windows can be made for separate sides of the same part on a Standard Table, where each Face Window represents a different stock (see page 2-12 for an explanation Independent and Dependent stocks).

When multiple face windows representing different stocks have the same **Part ID**, the whole completed part can be shown in 3D simulation as PartMaker will know the relation to one another of the multiple stocks being machined as defined in the **Face Options** dialog (see page 2-15).

When multiple face windows representing different stocks have the different **Part IDs**, PartMaker will know different parts are being machined in each Face Window and prompt you for which completed part you wish to see in 3D simulation when choosing the **Show Finished Part** command.

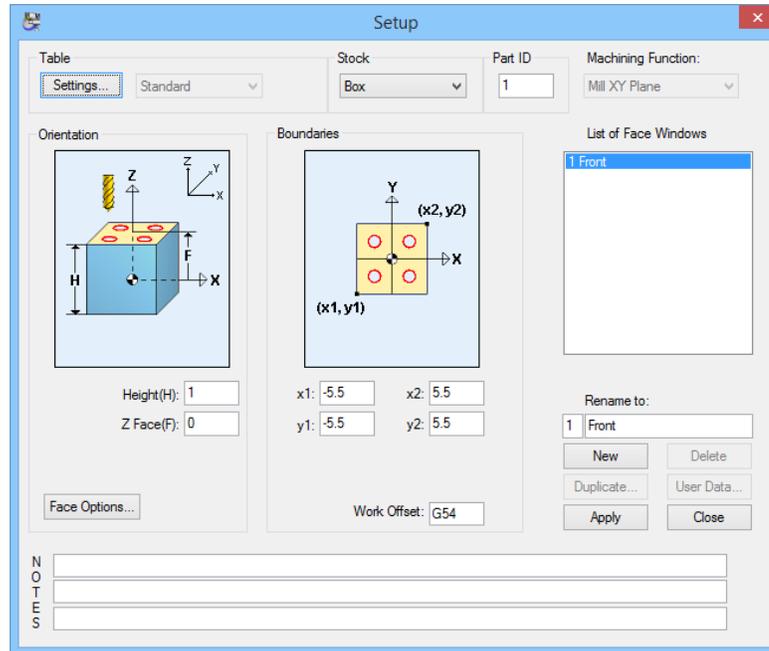
## Machining Functions for use with Table Type: Standard

The Machining Function drop down menu allows you to choose the type of machining being performed in a given Face Window.

For Standard Table type, the available Machining Function is **Mill XY Plane**.

## Mill XY Plane

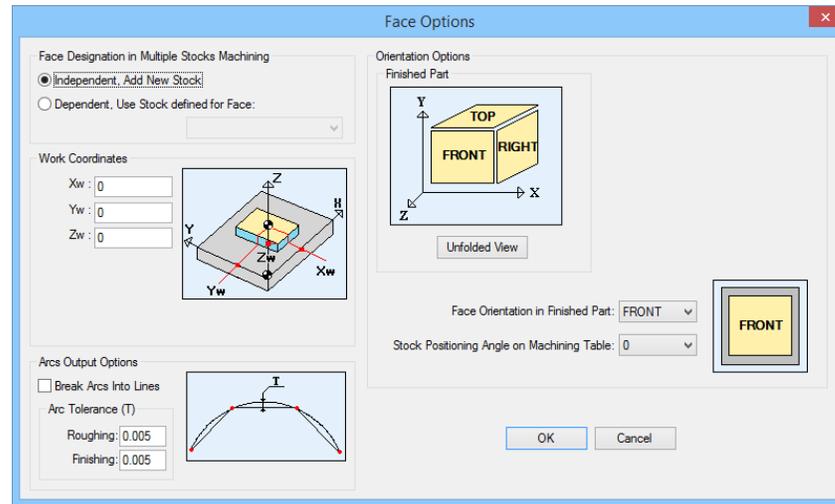
The Mill XY Plane machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Height (H):** Height of the initial stock being machined
- Z Face (F):** The location of the top of the stock relative to the Z = 0 Plane
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Face Options

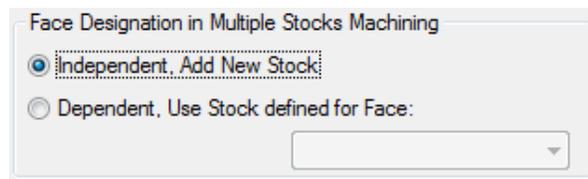
When the Table is set to Standard using the **Mill XY Plane** window, the **Face Options** dialog can be accessed by clicking the **Face Options** button. When doing so, the **Face Options** dialog will appear as shown below:



Each section of the **Face Options** dialog is explained below:

### Face Designation in Multiple Stocks Machining

This section of the **Face Options** dialog allows you determine whether the machining operations programmed in the current Face Window are being done on a separate stock from those being used in other Face Windows or on the same stock as another face window.



Checking the **Independent, Add New Stock** radio button indicates that tool paths created in the current Face Window are being machined on a separate stock from other face windows. Independent Stocks will be shown separately on the Standard Machining Table in PartMaker's 3D simulation.

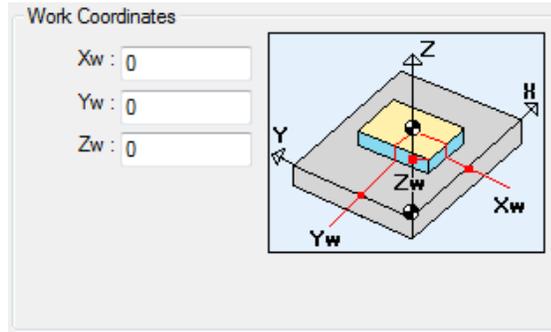


**Important!** Face windows containing **Independent Stocks** should each have a different **Work Offset** value assigned in the **Setup** dialog.

Checking the **Dependent, Use Stock defined for Face** radio button indicates that tool paths created in the current Face Window are being machined on the same stock defined in the Face Window designated. In the **Setup** dialog the **Part ID** will be disabled.

## Work Coordinates

This **Work Coordinates** section of the **Face Options** dialog allows you to determine the location of an **Independent Stock** on the machining table.



**Xw:** Allows you to define the X location of the center of the Independent Stock relative of the lower left hand corner of the machining table.

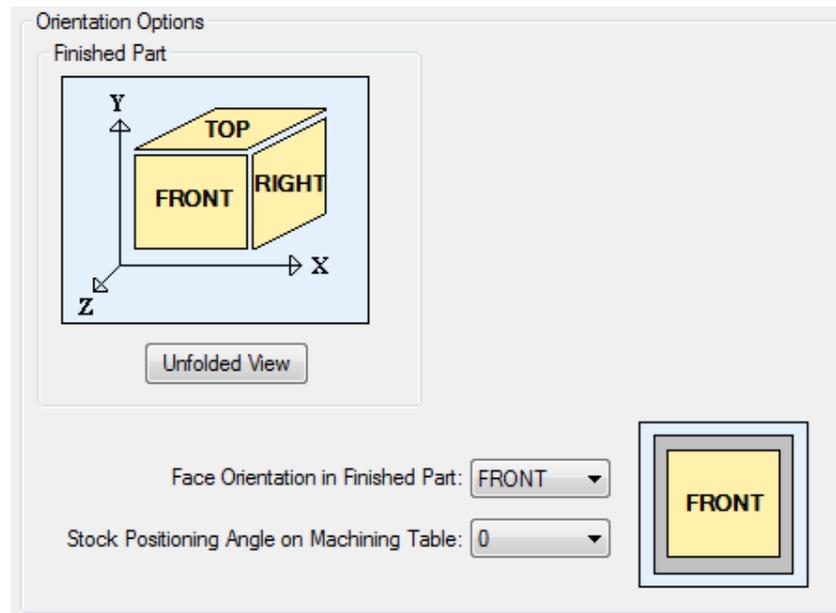
**Yw:** Allows you to define the Y location of the center of the Independent Stock relative of the lower left hand corner of the machining table.

**Zw:** Allows you to define the Z location of the top of the Independent Stock relative of the top of the machining table.

The values entered in these fields can be manipulated in PartMaker's 3D simulation window by double clicking on the stock using the **Change Stock Position** dialog.

## Orientation Options

This **Orientation Options** section of the **Face Options** dialog allows you to determine which face of a multi-sided part, i.e. the **FRONT, BACK, TOP, BOTTOM, RIGHT** or **LEFT** is being machined in the current Face Window.



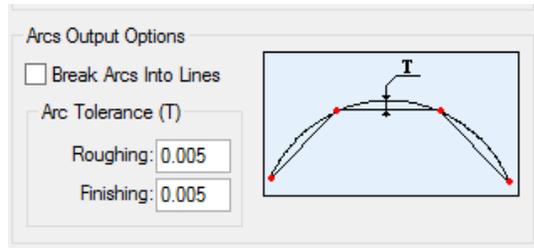
When using the **Table Type: Standard** PartMaker provides you with a very unique capability for programming a mutli-sided part on a vertical machining center where each side of the part is being machined in a separate vice and assigned a different work offset.

The relation of these faces is selected in the **Face Orientation in Finished Part** drop down selection and its orientation on the machining table is selected in the drop down dialog for **Stock Positioning Angle on Machining Table**.



*Note: When machining a multi-sided part using the Orientation Options section of the Face Options dialog with Independent Stocks (as defined in the Face Designation in Multiple Stocks Machining section of the Face Options dialog), you should make sure all the Face Windows corresponding to the same finished part should have the same Part ID in the Setup dialog.*

## Arc Output Options



**Break Arc Into Lines** — Select this option if you want PartMaker to break all toolpath arcs into small lines in the NC program when programming for machine controls that do not support circular interpolation of arcs. You can also use this option to break helical ramp entry into lines when the **Advanced Milling Toolpath** option is enabled.

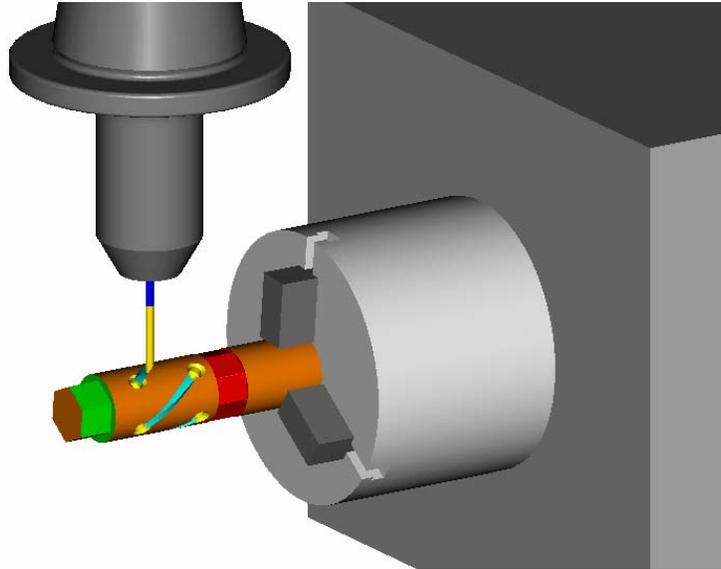
**Arc Tolerance** — When **Break Arc Into Lines** is selected, you can also specify:

**Roughing** — Enter the chord tolerance that PartMaker uses to break arcs into lines during roughing operation.

**Finishing** — Enter the chord tolerance that PartMaker uses to break arcs into lines during finishing, chamfering, and corner rounding operations.

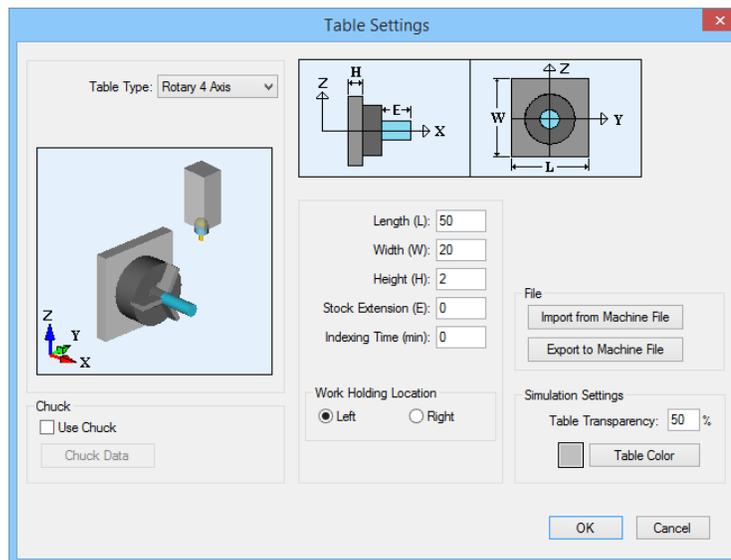
## Table Type: Rotary 4 Axis

Setting the Table Type to **Rotary 4 Axis** assumes you are working a vertical machining center with a Rotary Machining Table. (4th-axis-Rotary.JOB shown here)



### Settings

The **Settings** dialog is accessed for the **Table Type: 4 Axis Rotary** by clicking the **<Settings>** button. The **Settings** dialog is used to define the dimension and appearance of the machining table in PartMaker's 3D machining simulation for both block and cylinder stock.



- |                             |  |
|-----------------------------|--|
| <b>Length (L):</b>          | Specifies the table length.                              |
| <b>Width (W):</b>           | Specifies the table width.                               |
| <b>Height (H):</b>          | Specifies the table height.                              |
| <b>Indexing Time (min):</b> | Specifies the average 4-Axis Rotary table indexing time. |

|                                 |   |
|---------------------------------|---|
| <b>Work Holding Location</b>    | Select the location of the work holding ( <b>Left</b> or <b>Right</b> ). Specifies which side of the part will be in the holder.  |
| <b>Use Chuck</b>                | Specifies that Chuck will be used on 4-Axis Rotary Table  |
| <b>Chuck Data</b>               | Allows you to set the chuck parameters.   |
| <b>Import From Machine File</b> | Clicking this button allows you to import the parameters of a previously created machining table from an *.MCH file which saves you from having to define a new machining table for every PartMaker JOB file  |
| <b>Export to Machine File</b>   | Clicking this button allows you to export the parameters of the specific machining table as an *.MCH file for use with future jobs. Exporting these parameters to a machine file allows you to setup a number of different types of machining tables. |
| <b>Table Transparency</b>       | Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent   |
| <b>Table Color</b>              | Allows you to choose the color of the machining table from the available color pallet.  |

## Stock

When the Table is set to **Rotary 4 Axis**, you can choose from two types of stock, **Block** or **Cylinder**. The Block selection assumes you are working with prismatic stock, while the cylinder selection assumes you are working with cylindrical stock being held horizontally across the machining table and perpendicular to the tool spindle.

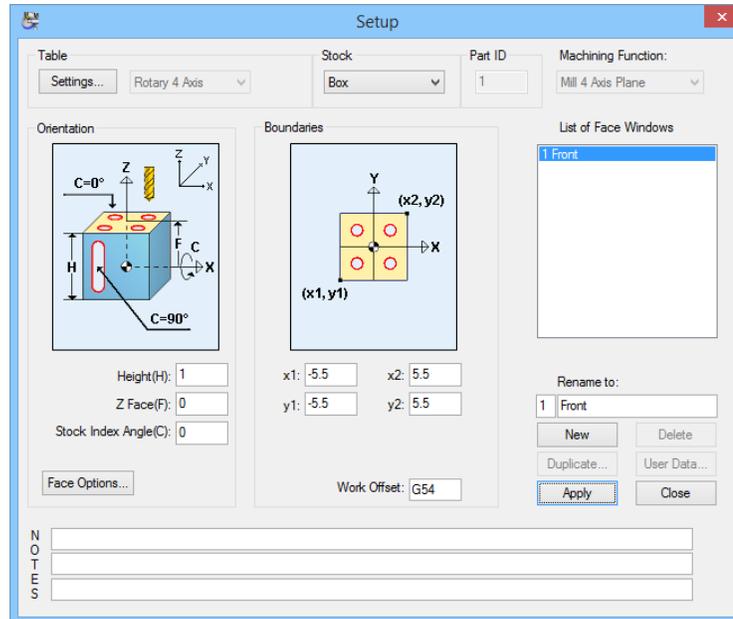
See page 2-5 of this manual for information on using custom stocks in PartMaker Mill.

## Part ID

Disabled when using the Rotary 4 Axis table, as you can only program one complete part per JOB file when using this table type.

## Machining Functions for Block Stock: Mill 4 Axis Plane

The Mill 4 Axis Plane machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 1/2 axis milled features such as pockets and contours and features requiring 3D surface machining.



- Height (H):** Height of the initial stock being machined.
- Z Face (F):** The Z coordinate of the surface of the part, measured from the rotary axis.
- Stock Index Angle (C):** Orientation angle of the stock to achieve the desired face position.
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

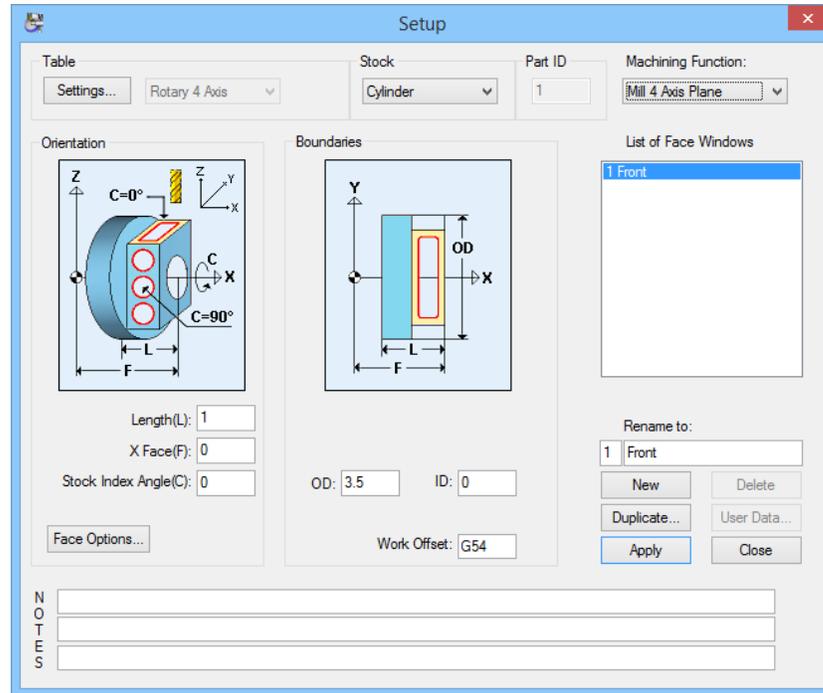
When the Table is set to Rotary 4 Axis, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Only **Arc Output Options** area of this dialog apply to the **Mill 4 Axis Plane** Machining Function when with **Table Type: Rotary 4 Axis** setting.

The **Arc Output Options** area of this dialog is explained on page 2-18.

## Machining Functions for Cylinder Stock: Mill 4 Axis Plane

The **Mill 4 Axis Plane** machining function supports 3-axis planar machining operations on a work piece that is cylindrical in nature. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



**Length (L):** Length of the initial stock being machined

**X Face (F):** The X coordinate of the initial front face

**Stock Index Angle (C):** Orientation angle of the stock to achieve the desired face position

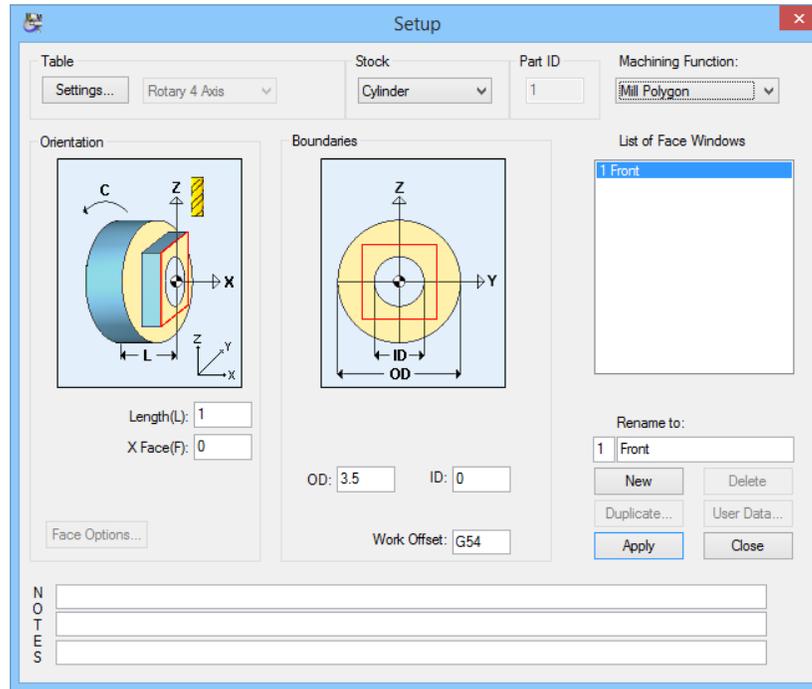
**OD:** The Outside diameter of the stock

**ID:** The Inside diameter of the stock

**Work Offset:** The work coordinate system of the stock in the current Face Window

## Machining Functions for Cylinder Stock: Mill Polygon

The **Mill Polygon** machining function allows you to create multiple flat surfaces on the OD of a part using the rotary-axis capability of your machine.



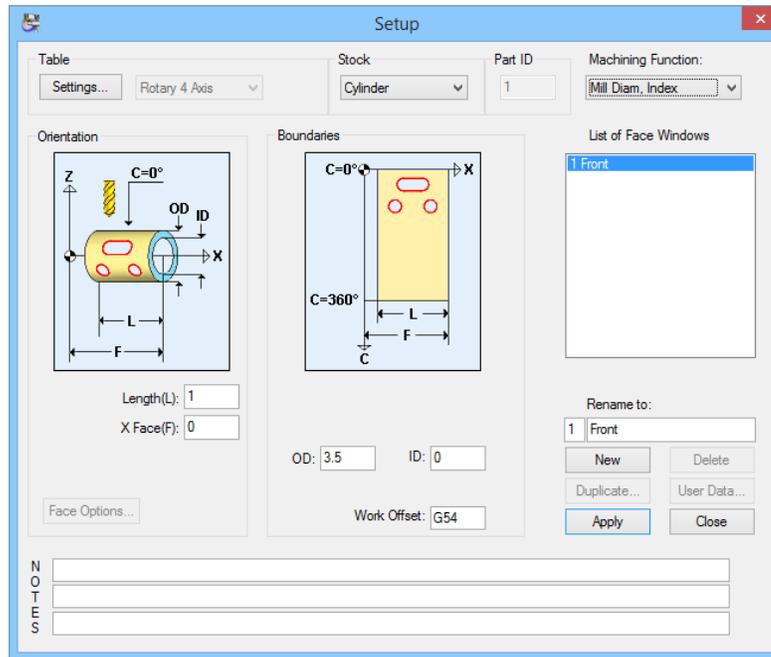
- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Machining Functions for Cylinder Stock: Mill Diam, Index

The **Mill Diam, Index** machining function allows you to create “cross holes” or horizontal slots on the OD of the part. In order to support this machining function, your mill must be able to support Rotary-axis indexing.

This machining function supports the following features cut on the outside diameter of the cylinder.

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



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

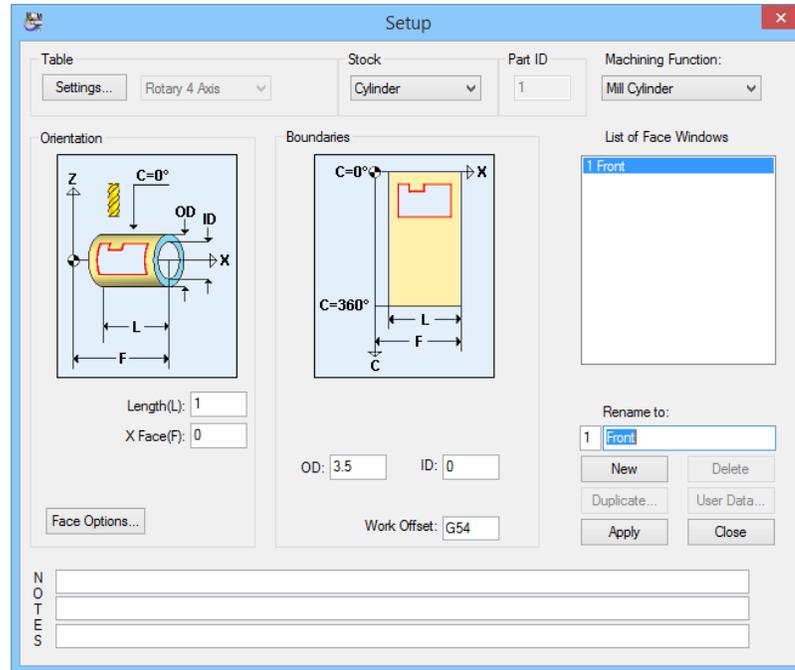
## Machining Functions for Cylinder Stock: Mill Cylinder

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

Using motions of this type, combined with a tool motion, profiles can be created on the outside diameter of the stock.

Tool positioning is programmed using X, Y, and C axis coordinates. The C position is programmed in terms of degrees of rotation relative to machine C=0.

Programming of this machining function requires that your Mill has a programmable Rotary-axis.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

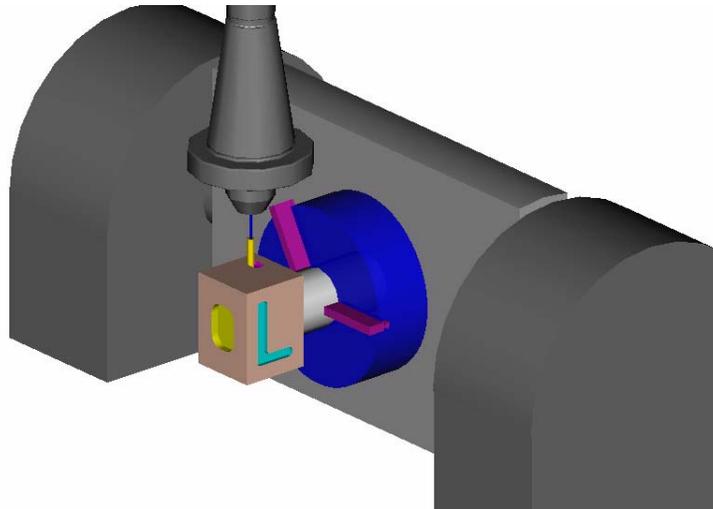
When the Table is set to Rotary 4 Axis, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Only **Arc Output Options** section of this dialog applies to the **Mill 4 Axis Plane** Machining Function when with **Table Type: Rotary 4 Axis** setting.

The **Arc Output Options** area of this dialog is explained on page 2-18.

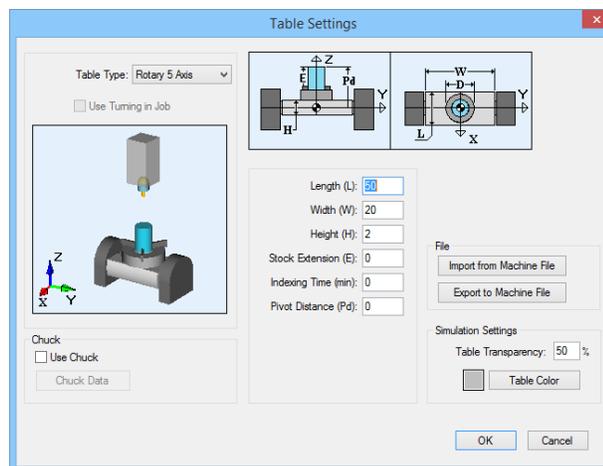
## Table Type: Rotary 5 Axis

Setting the Table Type to **Rotary 5 Axis** assumes you are working on a vertical machining center with a Rotary Table capable of rotating about two axes: X-axis and Z-axis, i.e. a 5 axis rotary table. (Multi-sided-Milling\6-sided-on-5-axis-table.JOB shown here)



### Settings

The **5-Axis Rotary Table Settings** dialog is accessed by clicking the **Settings** button in the **Setup** dialog. The **5-Axis Rotary Table Settings** dialog is used to define the dimension and appearance of the machining table in PartMaker's 3D machining simulation for both block and cylinder stock.



- |                             |  |
|-----------------------------|--|
| <b>Length (L):</b>          | Specifies the table length.  |
| <b>Width (W):</b>           | Specifies the table width.   |
| <b>Height (H):</b>          | Specifies the table height.  |
| <b>Stock Extension (E):</b> | Specifies the distance from the part face to the Chuck.                |
| <b>Pivot Distance (Pd):</b> | Specifies distance from the table pivot point to the top of the stock. |

|                                 |   |
|---------------------------------|---|
| <b>Use Turning in Job</b>       | Is only available with Cylinder Stock and allows the use of conventional turning tools moving in the X and Z-axes.  |
| <b>Indexing Time (min):</b>     | Specifies the average 5-Axis Rotary table index time.   |
| <b>Use Chuck</b>                | Specifies that Chuck will be used on 5-Axis Rotary Table  |
| <b>Chuck Data</b>               | Allows you to set the chuck parameters.   |
| <b>Import From Machine File</b> | Clicking this button allows you to import the parameters of a previously created machining table from an *.MCH file which saves you from having to define a new machining table for every PartMaker JOB file  |
| <b>Export to Machine File</b>   | Clicking this button allows you to export the parameters of the specific machining table as an *.MCH file for use with future jobs. Exporting these parameters to a machine file allows you to setup a number of different types of machining tables. |
| <b>Table Transparency</b>       | Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent   |
| <b>Table Color</b>              | Allows you to choose the table color of the machining table from the available color pallet.  |

## Stock

When the Table is set to **Rotary 5 Axis**, you can choose from two types of stock, **Block** or **Cylinder**. The Block selection assumes you are working with prismatic stock, while the cylinder selection assumes you are working with cylindrical stock initially being held vertically.

See page 2-10 of this manual for information on using custom stocks in PartMaker Mill.

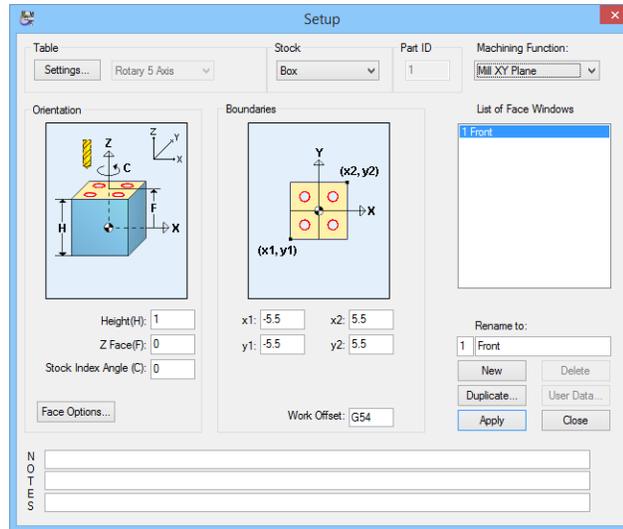
## Part ID

Disabled when using the Rotary 5 Axis table, as you can only program one complete part per JOB file when using this table type.

## Machining Functions for Block Stock: Mill XY Plane

The Machining Function drop down menu allows you to choose the type of machining being performed in a given Face Window. When the Table is set to Rotary 5 Axis and the stock type is set to Block, the Mill XY Plane, Mill 4 Axis Plane, and Mill 5 Axis Plane, are available for machining functions. Each of these functions is explained below.

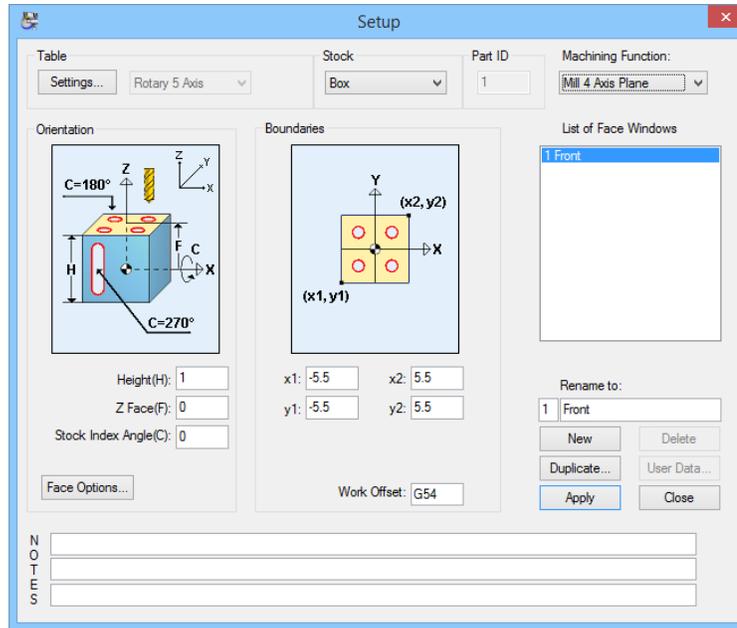
The **Mill XY Plane** machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Height (H)** Height of the initial stock being machined
- Z Face (F):** The location of the top of the stock relative to Z = 0 Plane
- Stock Index Angle (C):** Orientation angle of the stock to achieve the desired face position.
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Machining Functions for Block Stock: Mill 4 Axis Plane

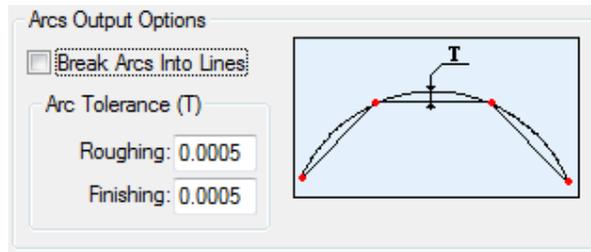
The **Mill 4 Axis Plane** machining function Face Window supports 4-axis planar machining operations to rotate the part about the machine's X axis. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining. You will use this machining function to machine the side faces of your part.



- Height (H):** Height of the initial stock being machined
- Z Face (F):** The location of the top of the stock relative to Z = 0 Plane
- Stock index Angle (C):** Orientation angle of the stock about the X-axis to achieve the desired face position
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Face Options

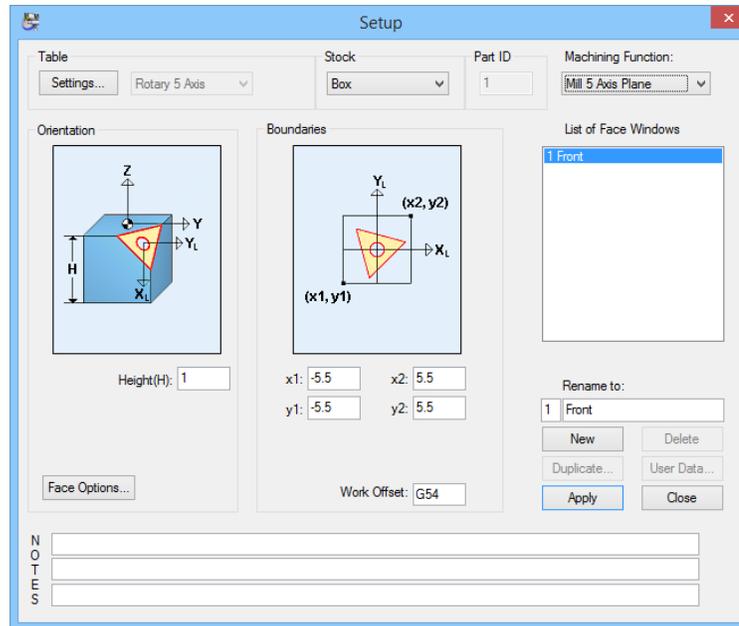
When the Table is set to Rotary 5 Axis using the **Mill XY Plane** or **Mill 4 Axis Plane** window, the **Face Options** dialog can be accessed by clicking the **Face Options** button. The only section of the **Face Options** dialog that applies to this table type is the **Arc Output Options** area shown here:



This should only be used when programming for machine controls that do not support circular interpolation of arcs (i.e. G2s and G3s). Checking the **Break Arcs Into Lines** box indicates to PartMaker to break all arc output into small lines in the NC program. The **Roughing** and **Finishing** field under **Arc Tolerance** specify the chord tolerance used when breaking arcs into lines for roughing and finishing operations, respectively.

## Machining Functions for Block Stock: Mill 5 Axis Plane

The **Mill 5 Axis Plane** machining function Face Window supports 3-axis planar machining operations within an inclined plane. This inclined plane can be achieved by using a tilting a 5-axis rotary table. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining that must be machined on an angle.



- Height (H):** Height of the initial stock being machined
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

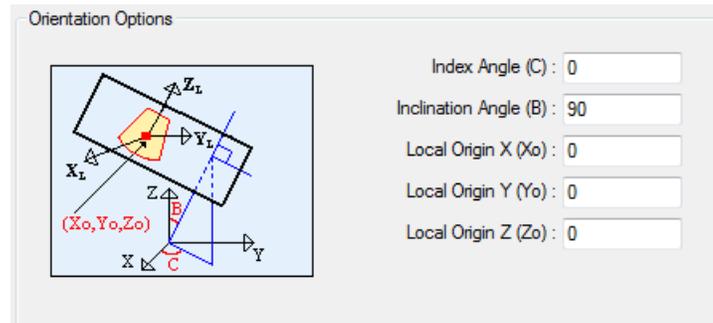
### Face Options

When the Table is set to Rotary 5 Axis, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Three sections of the **Face Options** dialog apply to the **Mill 5 Axis Plane** Machining Function when with Table Type: Rotary 5 Axis setting.

## Orientation Options

The **Orientation Options** section of the **Face Options** dialog allows you to set the location of local coordinate system being used on the inclined plane being machined.



**Index Angle (C):** Specifies the angle of the part rotation about the Z-axis to which the stock is indexed

**Inclination Angle (B):** Sets the inclination angle to which the table is tilted

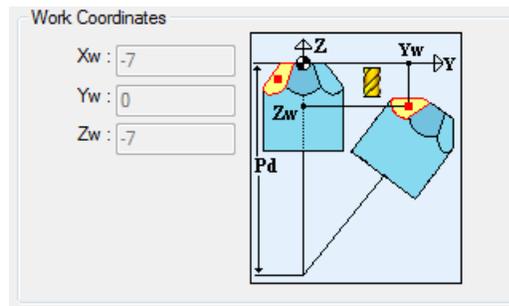
**Local X (X<sub>o</sub>):** These fields represent the co-ordinates for the local origin

**Local Y (Y<sub>o</sub>):** created for the planar face being machined. These fields represent the coordinates with respect to the work pieces'

**Local Z (Z<sub>o</sub>):** global coordinate system (part coordinate system of the solid model) origin after the work piece is indexed or rotated by the Index Angle (C).

## Work Coordinates

The **Work Coordinates** area of the **Face Options** dialog is shown here:



The **Work Coordinates** section of the **Face Options** dialog will display work coordinates based on calculations performed on the values entered in the Orientation Options fields and the Pivot Distance entered in the Table Setting Dialog. These work coordinates represent the distance from the origin of the Global Coordinate system (solid model coordinate system) to the origin of the local coordinate system for the 5-axis Face Window.

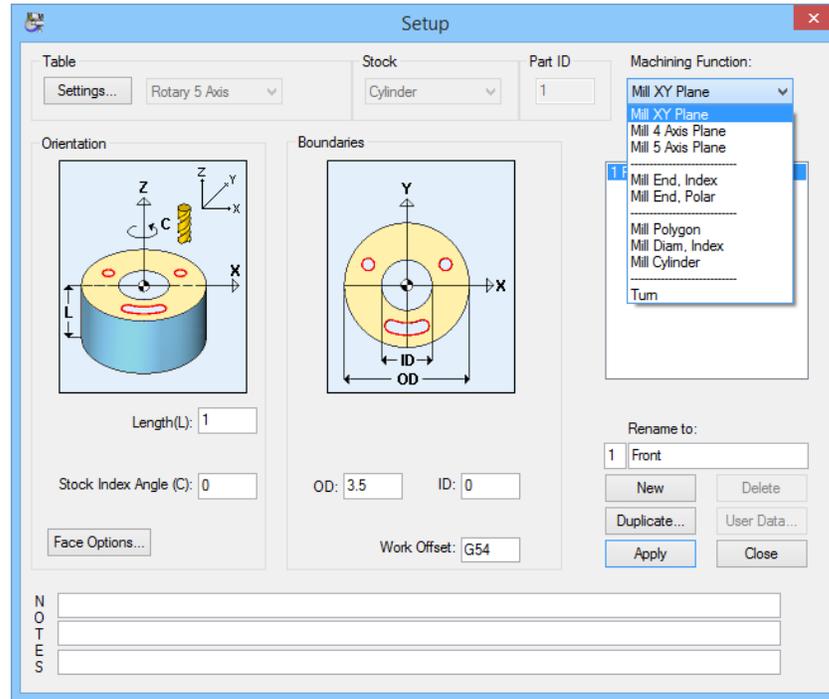
## Arc Output Options

See page 2-18 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

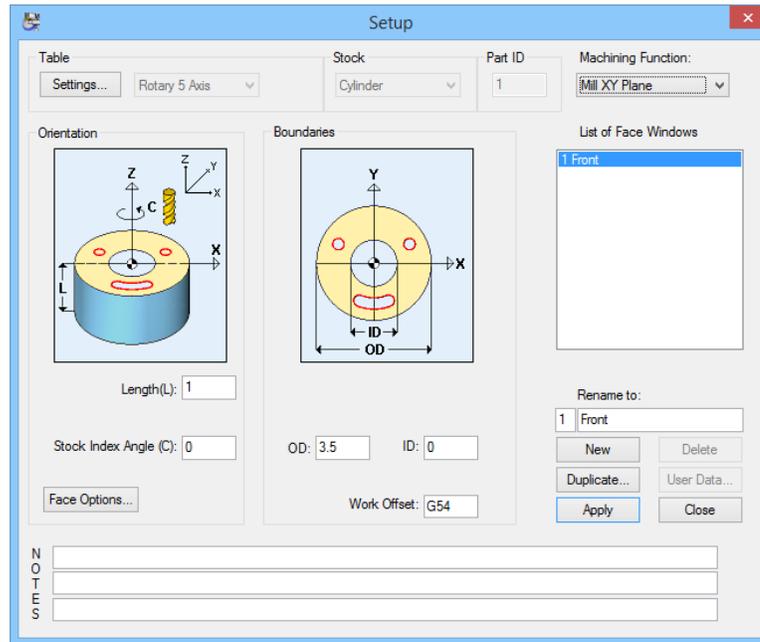
## Machining Functions for Cylinder Stock: Mill XY Plane

The Machining Function drop down menu allows you to choose the type of machining being performed in a given Face Window.

When the Table is set to Rotary 5 Axis and you are using Cylinder stock, there are nine types of Machining Functions, Mill XY Plane, Mill 4 Axis Plane, Mill 5 Axis Plane, Mill Polygon, Mill End Index, Mill End Polar, Mill Diameter Index, Mill Cylinder, and Turn. Each of these are explained below:



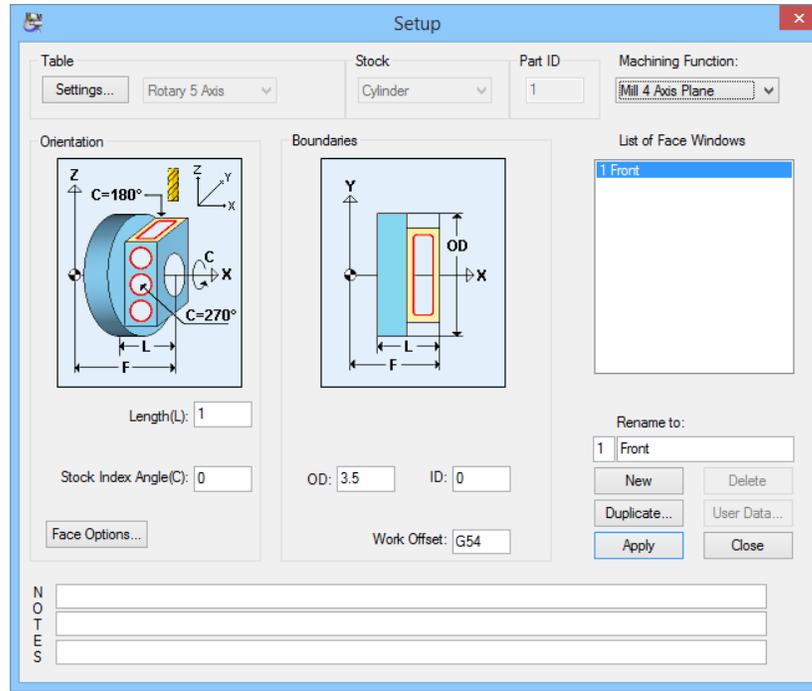
The **Mill XY Plane** machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the initial front face.
- Stock Angle Index** Orientation angle of the stock about the Z-axis to achieve the desired face position.
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Machining Functions for Cylinder Stock: Mill 4 Axis Plane

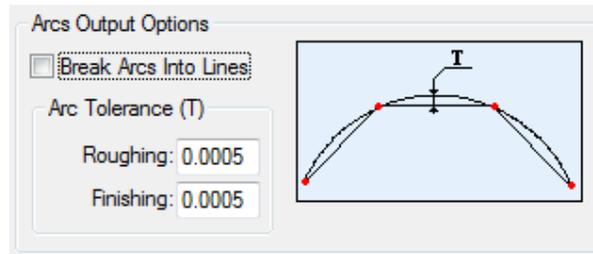
The **Mill 4 Axis Plane** machining function Face Window supports 4-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the initial front face
- Stock Index Angle (C):** Orientation angle of the stock about the X-axis to achieve the desired face position
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

## Face Options

When the Table is set to Rotary 5 Axis using the **Mill XY Plane** or **Mill 4 Axis Plane** window using **Cylinder Stock**, the **Face Options** dialog can be accessed by clicking the **Face Options** button. The only section of the **Face Options** dialog that applies to this table type is the **Arc Output Options** area shown here:

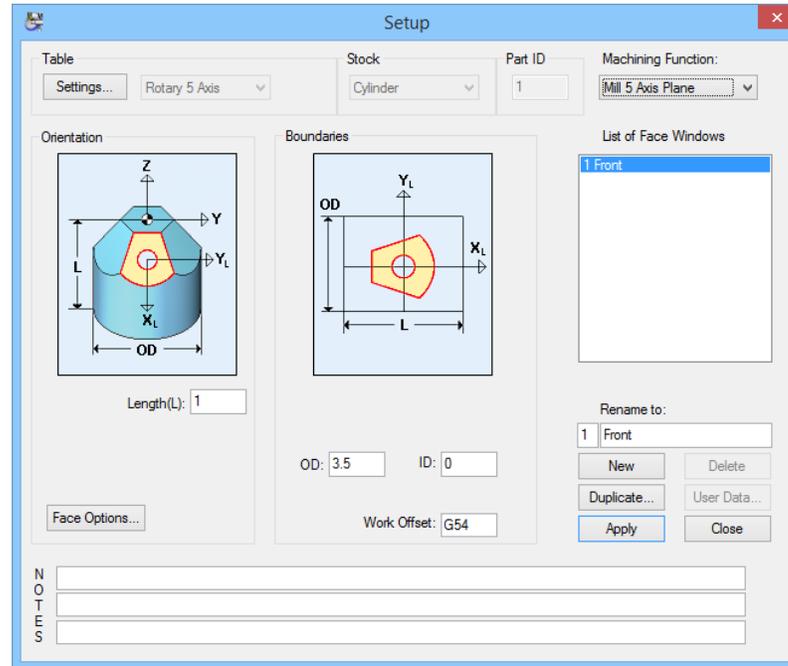


This should only be used when programming for machine controls that do not support circular interpolation of arcs (i.e. G2s and G3s). Checking the **Break Arcs Into Lines** box indicates to PartMaker to break all arc output into small lines in the NC program.

The **Roughing** and **Finishing** field under **Arc Tolerance** specify the chord tolerance used when breaking arcs into lines for roughing and finishing operations, respectively.

## Machining Functions for Cylinder Stock: Mill 5 Axis Plane

The **Mill 5 Axis Plane** machining function Face Window supports 3-axis planar machining operations within an inclined plane. This inclined plane can be achieved by tilting the 5-axis rotary table in an arbitrary 5-axis plane. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining that must be machined on an angle.



- Length (L):** Length of the initial stock being machined
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

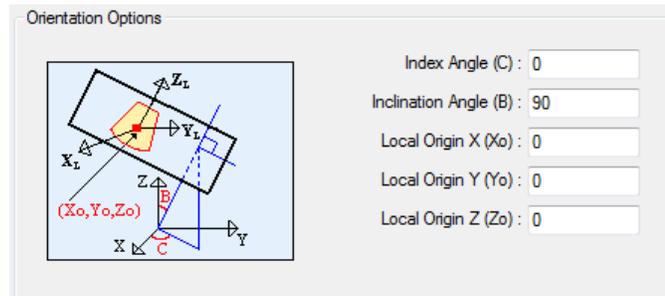
### Face Options

When the Table is set to Rotary 5 Axis, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Three sections of the **Face Options** dialog apply to the **Mill 5 Axis Plane** Machining Function when with Table Type: Rotary 5 Axis setting.

## Orientation Options

The **Orientation Options** section of the **Face Options** dialog allows you to set the location of local coordinate system being used on the inclined plane being machined.



**Index Angle (C):** Specifies the angle of the part rotation about the Z-axis to which the stock is indexed

**Inclination Angle (B):** Sets the inclination angle to which the table is tilted

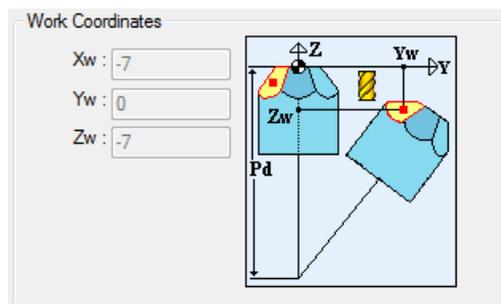
**Local X (X<sub>o</sub>):** These fields represent the co-ordinates for the local origin created for the planar face being machined. These fields represent the coordinates with respect to the work pieces' global coordinate system (part coordinate system of the solid model) origin after the work piece is indexed or rotated by the Index Angle (C).

**Local Y (Y<sub>o</sub>):**

**Local Z (Z<sub>o</sub>):**

## Work Coordinates

The **Work Coordinates** area of the **Face Options** dialog is shown here:



The **Work Coordinates** section of the **Face Options** dialog will display work coordinates based on calculations performed on the values entered in the Orientation Options fields and the Pivot Distance entered in the Table Setting Dialog. These work coordinates represent the distance from the origin of the Global Coordinate system (solid model coordinate system) to the origin of the local coordinate system for the 5-axis Face Window.

## Arc Output Options

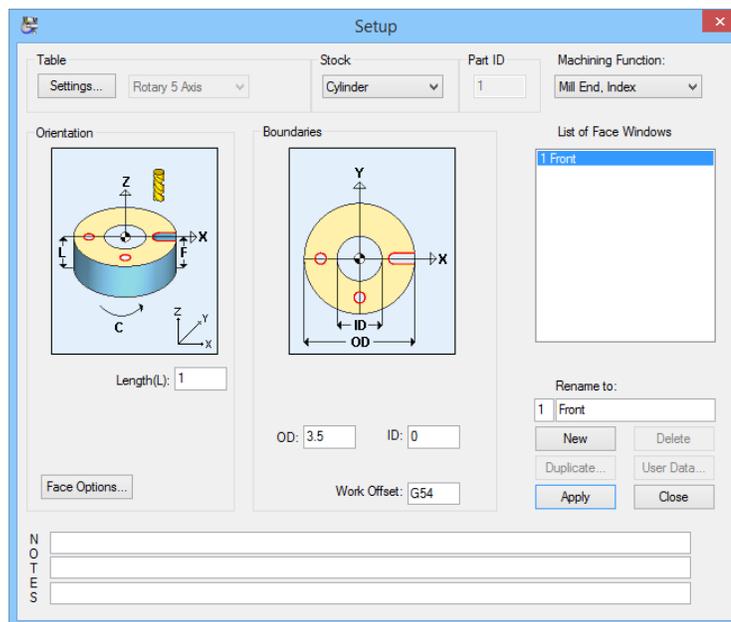
See page 2-16 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

## Machining Functions for Cylinder Stock: Mill End Index

The **Mill End, Index** machining function 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. This motion type supports off-center drilling on the end of the part, using C-axis indexing and X-axis or Y-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 or Y-axis motion is used to cut the part.



- 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

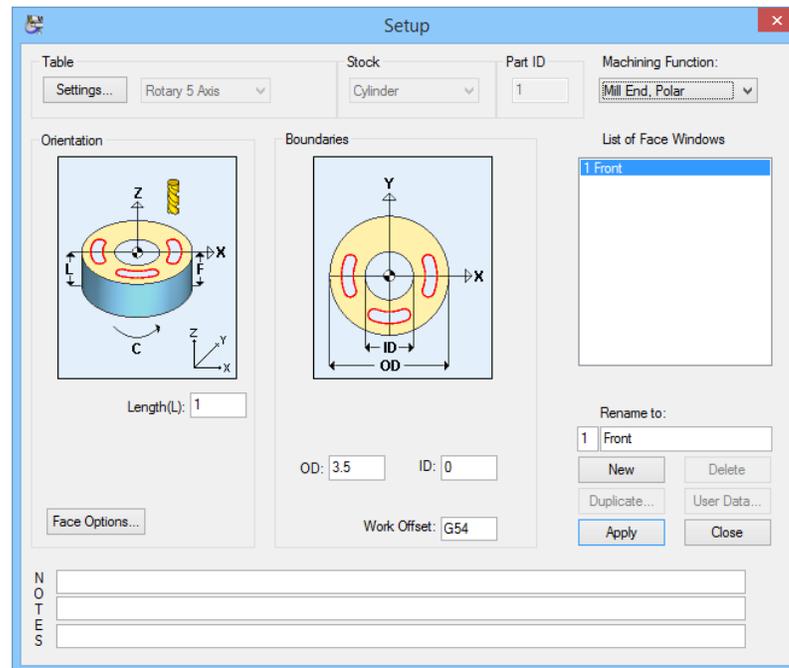
## Arc Output Options

See page 2-18 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

## Machining Functions for Cylinder Stock: 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 Mill Center has the ability to perform simultaneous feed motions with the X, Z and C-axes.

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



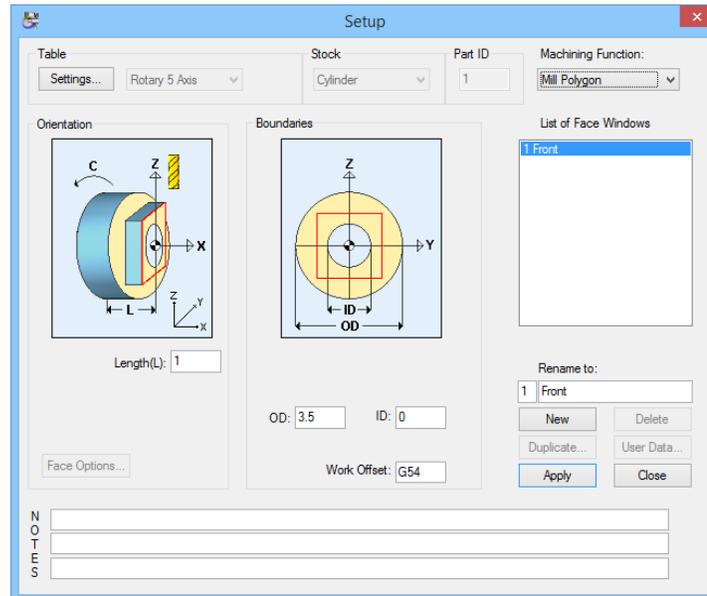
- Length (L)** Length of the finished part
- OD** Starting outside diameter of bar stock before machining
- ID** Starting inside diameter of bar stock before machining
- Work Offset** Specifies the work offset for the current face window

## Arc Output Options

See page 2-18 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

## Machining Functions for Cylinder Stock: Mill Polygon

The **Mill Polygon** machining function allows you to create multiple flat surfaces on the OD of a part using the rotary-axis capability of your machine.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

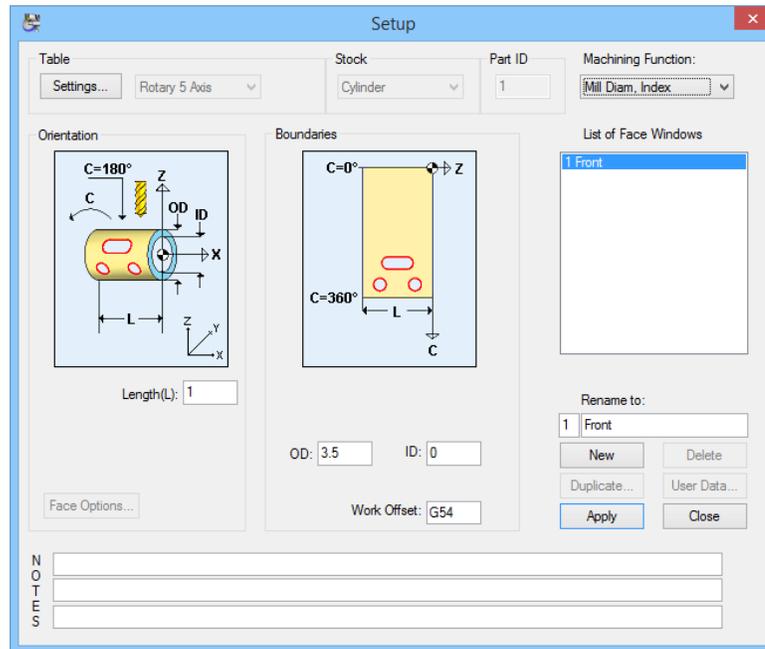
The Face Options dialog is disabled for the Mill Polygon Machining Function

## Machining Functions for Cylinder Stock: Mill Diam, Index

The **Mill Diam, Index** machining function allows you to create “cross holes” or horizontal slots on the OD of the part. In order to support this machining function, your mill must be able to support Rotary-axis indexing.

This machining function supports the following features cut on the outside diameter of the cylinder.

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



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

The **Face Options** dialog is disabled for the Mill Diameter Index Machining Function.

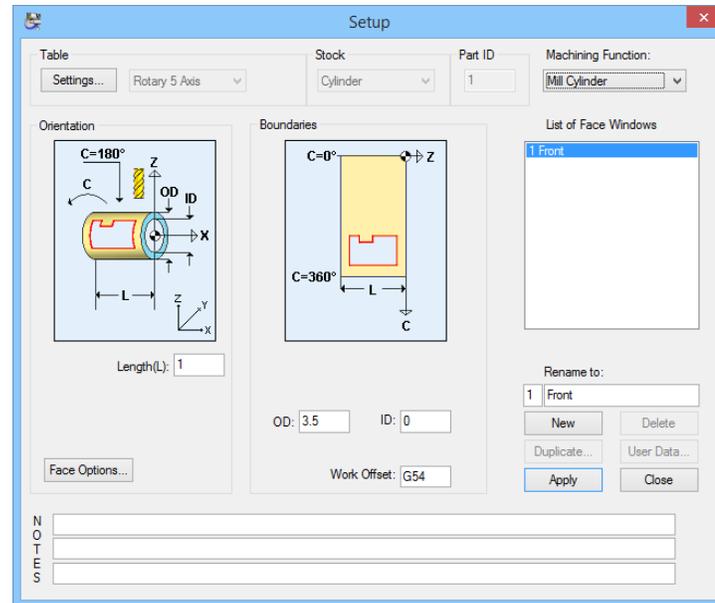
## Machining Functions for Cylinder Stock: Mill Cylinder

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

Using motions of this type, combined with a tool motion, profiles can be created on the outside diameter of the stock.

Tool positioning is programmed using X, Y, and C axis coordinates. The C position is programmed in terms of degrees of rotation relative to machine C=0.

Programming of this machining function requires that your Mill has a programmable Rotary-axis.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

When the Table is set to Rotary 4 Axis, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

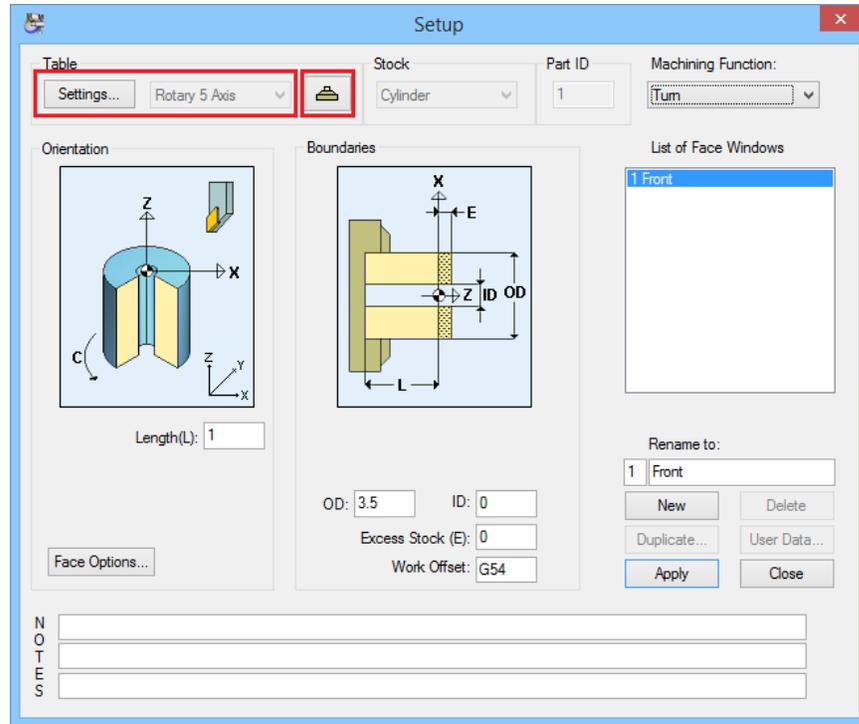
Only **Arc Output Options** section of this dialog applies to the **Mill Cylinder** Machining Function when with **Table Type: Rotary 4 Axis** setting.

The **Arc Output Options** area of this dialog is explained on page 2-18.

## Machining Functions for Cylinder Stock: Turning

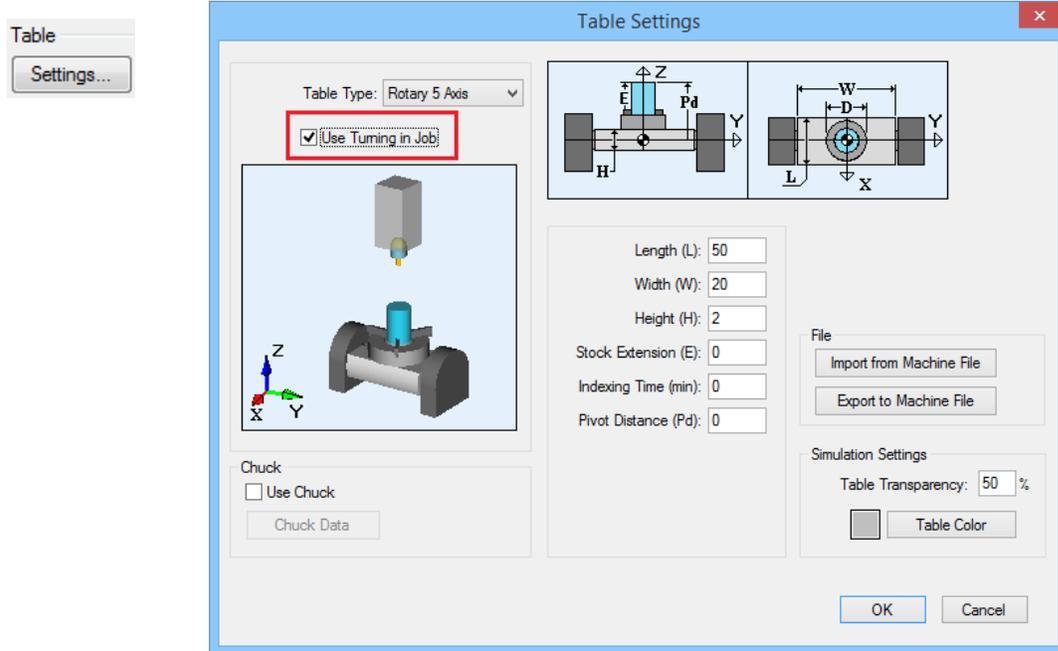
The picture below shows the appearance of the **Setup** dialog when performing Turning on the Rotary 5 Axis table.

The Turn machining function is only available when the Stock selection is set to Cylinder and the **Use Turning in Job** check box is selected in the **Settings** Dialog.



|                         |   |
|-------------------------|---|
| <b>Length (L)</b>       | Length of the finished part                             |
| <b>OD</b>               | Starting outside diameter of bar stock before machining |
| <b>ID</b>               | Starting inside diameter of bar stock before machining  |
| <b>Excess Stock (E)</b> | Rough stock left for facing on the front of the part    |
| <b>Work Offset</b>      | Specifies the work offset for the current face window   |

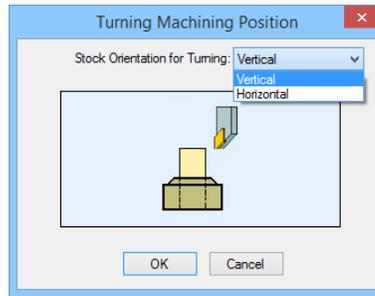
In the Set up Dialog click on the **Settings** button and select **Use Turning in Job**. This has to be selected while in any of the Mill Machining Functions.



- Length (L):** Specifies the table length.
- Width (W):** Specifies the table width.
- Height (H):** Specifies the table height.
- Stock Extension (E):** Specifies the distance from the part face to the Chuck.
- Indexing Time (min):** Specifies the average 4-Axis Rotary table indexing time.
- Work Holding Location** Select the location of the work holding (**Left** or **Right**). Specifies which side of the part will be in the holder.
- Use Chuck** Specifies that Chuck will be used on 4-Axis Rotary Table
- Chuck Data** Allows you to set the chuck parameters.
- Import From Machine File** Clicking this button allows you to import the parameters of a previously created machining table from an \*.MCH file which saves you from having to define a new machining table for every PartMaker JOB file
- Export to Machine File** Clicking this button allows you to export the parameters of the specific machining table as an \*.MCH file for use with future jobs. Exporting these parameters to a machine file allows you to setup a number of different types of machining tables.
- Table Transparency** Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent
- Table Color** Allows you to choose the color of the machining table from the available color pallet.



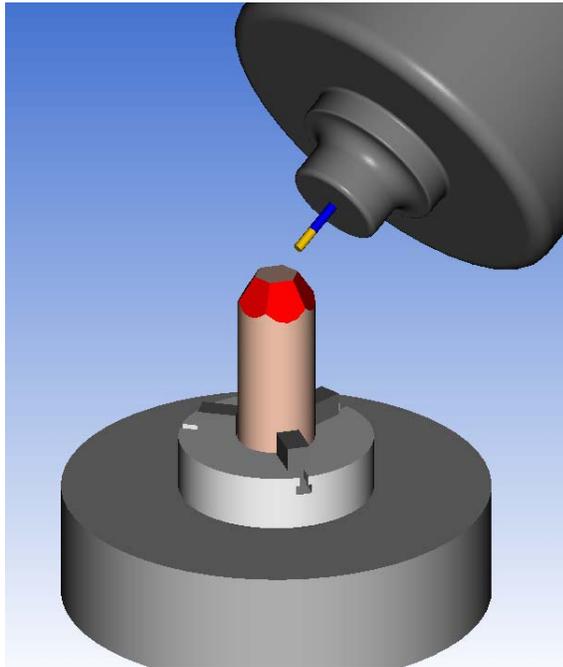
In PartMaker/Mill, you can use the **Turning Machining Position** dialog to change the orientation of the stock from Vertical to Horizontal for turning operations when you are using a **Rotary 5 Axis table**.



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.

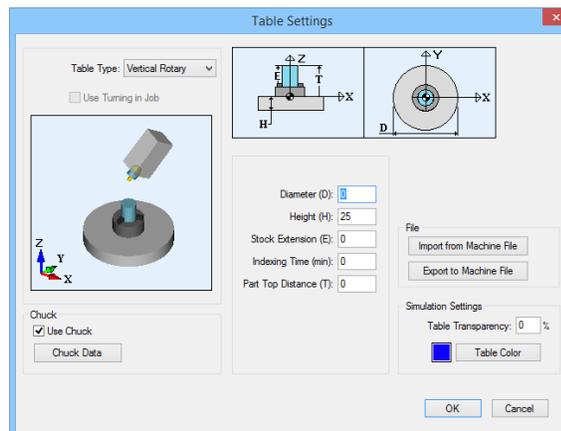
## Table Type: Vertical Rotary

Setting the Table Type to **Vertical Rotary** assumes you are working on a vertical machining center with a Rotary Table.



### Settings

The **Vertical Rotary Table Settings** dialog is accessed by clicking the **Settings** button in the **Setup** dialog. The **Vertical Rotary Table Settings** dialog is used to define the dimension and appearance of the machining table in PartMaker's 3D machining simulation for both block and cylinder stock.



- Length (L):** Specifies the table length.
- Diameter (D):** Specifies the diameter of the rotary table.
- Height (H):** Specifies the table height.
- Stock Extension (E):** Specifies the distance from the part face to the Chuck.

|                                 |   |
|---------------------------------|---|
| <b>Part Top Distance (T):</b>   | Specifies distance from the table top surface to the top of the stock.  |
| <b>Use Turning in Job</b>       | Is only available with Cylinder Stock and allows the use conventional turning tools moving in the X and Z-axes.   |
| <b>Indexing Time (min):</b>     | Specifies the average Rotary table index time.  |
| <b>Use Chuck</b>                | Specifies that Chuck will be used on Rotary Table   |
| <b>Chuck Data</b>               | Allows you to set the chuck parameters.   |
| <b>Import From Machine File</b> | Clicking this button allows you to import the parameters of a previously created machining table from an *.MCH file which saves you from having to define a new machining table for every PartMaker JOB file  |
| <b>Export to Machine File</b>   | Clicking this button allows you to export the parameters of the specific machining table as an *.MCH file for use with future jobs. Exporting a these parameters to a machine file allows you to setup a number of different types of machining tables. |
| <b>Table Transparency</b>       | Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent   |
| <b>Table Color</b>              | Allows you to choose the table color of the machining table from the available color pallet.  |

## Stock

When the Table is set to **Vertical Rotary**, you can choose from two types of stock, **Block** or **Cylinder**. The Block selection assumes you are working with prismatic stock, while the cylinder selection assumes you are working with cylindrical stock initially being held vertically.

See page 2-10 of this manual for information on using custom stocks in PartMaker Mill.

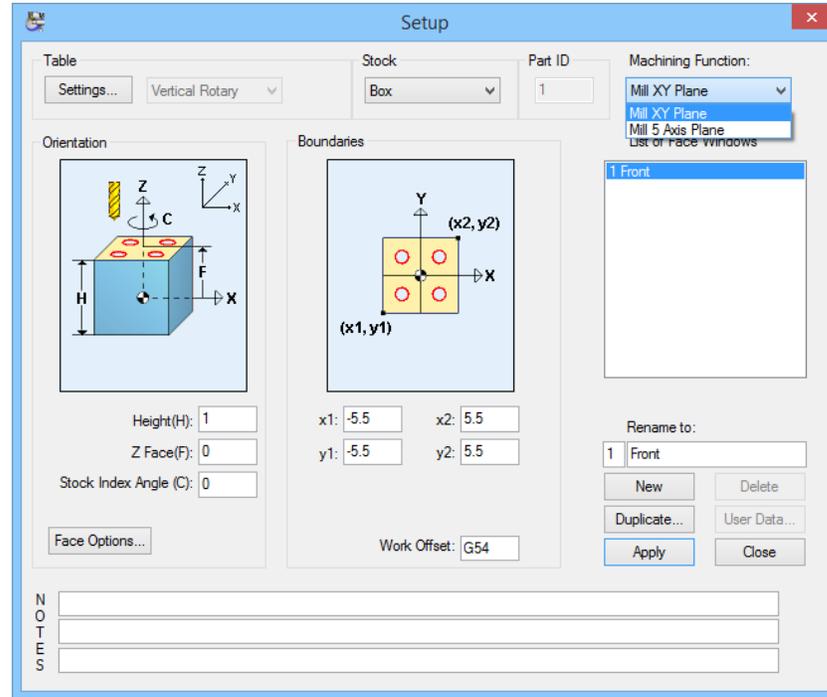
## Part ID

Disabled when using the Vertical Rotary table, as you can only program one complete part per JOB file when using this table type.

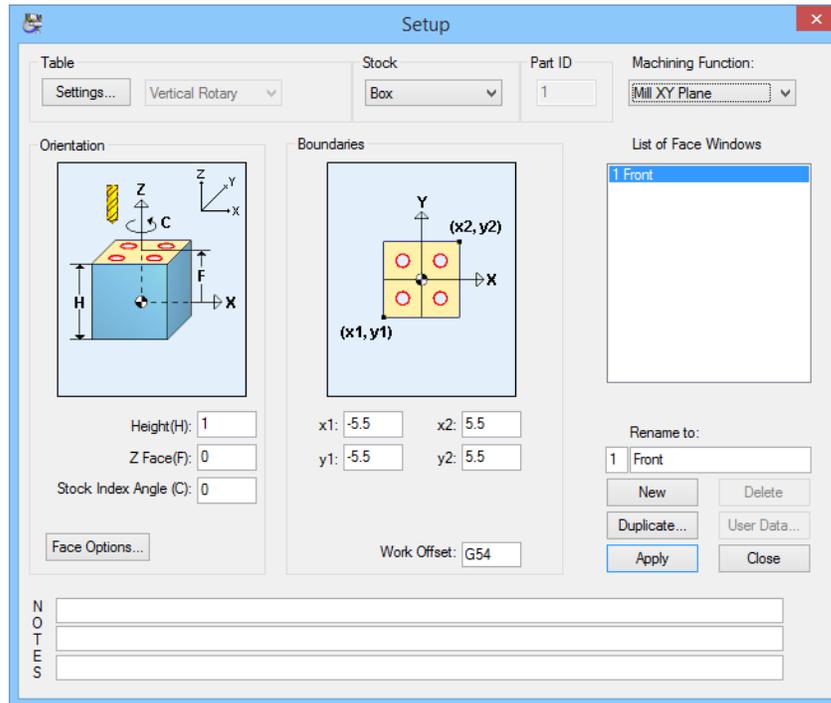
## Machining Functions for Block Stock: Mill XY Plane

The Machining Function drop down menu allows you to choose the type of machining being performed in a given Face Window.

When the Table is set to Vertical Rotary and the stock type is set to Block, the **Mill XY Plane**, and **Mill 5 Axis Plane**, are available for machining functions. Each of these functions is explained below.



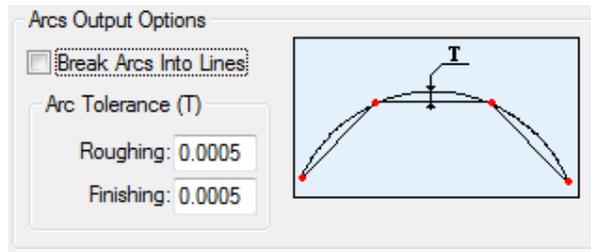
The **Mill XY Plane** machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Height (H)**      Height of the initial stock being machined
- Z Face (F):**      The location of the top of the stock relative to Z = 0 Plane
- Stock Index Angle (C):**      Orientation angle of the stock to achieve the desired face position.
- x1:**      X coordinate of the lower left hand corner of the rectangular machining area
- y1:**      Y coordinate of the lower left hand corner of the rectangular machining area
- x2:**      X coordinate of the upper right hand corner of the rectangular machining area
- y2:**      Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:**      The work coordinate system of the stock in the current Face Window

## Face Options

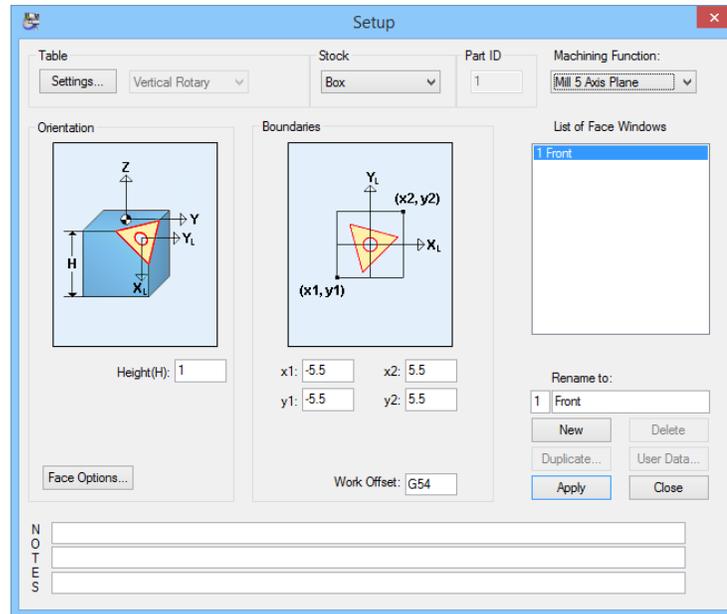
When the Table is set to Vertical Rotary using the **Mill XY Plane** window, the **Face Options** dialog can be accessed by clicking the **Face Options** button. The only section of the **Face Options** dialog that applies to this table type is the **Arc Output Options** area shown here:



This should only be used when programming for machine controls that do not support circular interpolation of arcs (i.e. G2s and G3s). Checking the **Break Arcs Into Lines** box indicates to PartMaker to break all arc output into small lines in the NC program. The **Roughing** and **Finishing** field under **Arc Tolerance** specify the chord tolerance used when breaking arcs into lines for roughing and finishing operations, respectively.

## Machining Functions for Block Stock: Mill 5 Axis Plane

The **Mill 5 Axis Plane** machining function Face Window supports 3-axis planar machining operations within an inclined plane. This inclined plane can be achieved by using a tilting a 5-axis rotary table. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining that must be machined on an angle.



- Height (H):** Height of the initial stock being machined
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

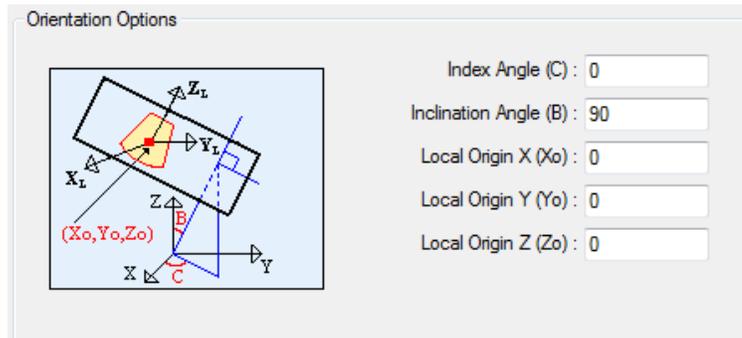
### Face Options

When the Table is set to Vertical Rotary, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Two sections of the **Face Options** dialog apply to the **Mill 5 Axis Plane** Machining Function with this Table Type:

## Orientation Options

The **Orientation Options** section of the **Face Options** dialog allows you to set the location of local coordinate system being used on the inclined plane being machined.



**Index Angle (C):** Specifies the angle of the part rotation about the Z-axis to which the stock is indexed

**Inclination Angle (B):** Sets the inclination angle to which the table is tilted

**Local X (X<sub>o</sub>):** These fields represent the co-ordinates for the local origin created for the planar face being machined. These fields represent the coordinates with respect to the work pieces' global coordinate system (part coordinate system of the solid model) origin after the work piece is indexed or rotated by the Index Angle (C).

**Local Y (Y<sub>o</sub>):**

**Local Z (Z<sub>o</sub>):**

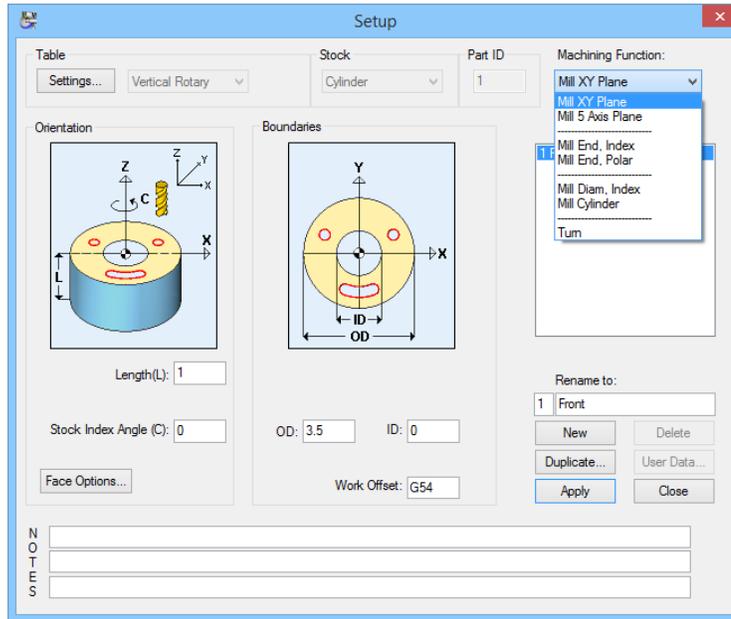
## Arc Output Options

See page 2-18 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

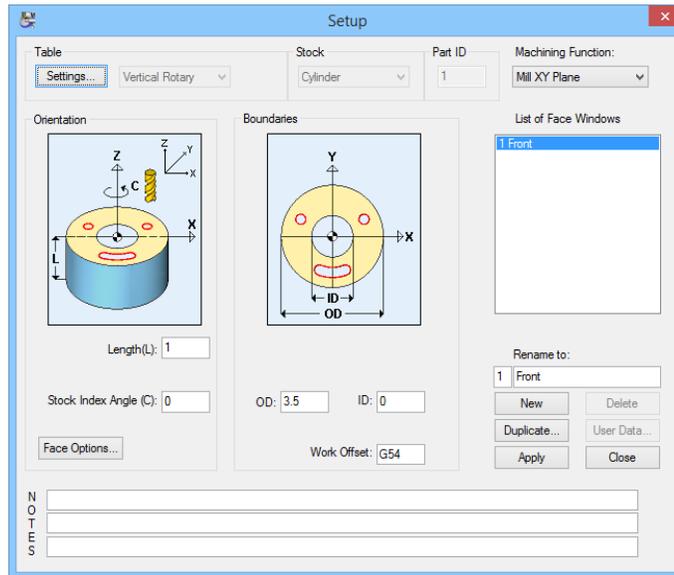
## Machining Functions for Cylinder Stock: Mill XY Plane

The Machining Function drop down menu allows you to choose the type of machining being performed in a given Face Window.

When the Table is set to Vertical Rotary and you are using Cylinder stock, there are seven types of Machining Functions, Mill XY Plane, Mill 5 Axis Plane, Mill End Index, Mill End Polar, Mill Diameter Index, Mill Cylinder, and Turn. Each of these is explained below:



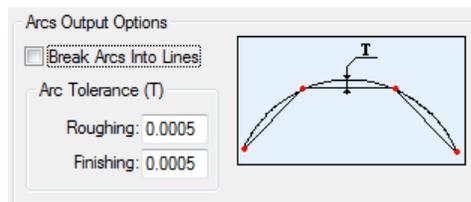
The **Mill XY Plane** machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Length (L):** Length of the initial stock being machined
- Stock Angle Index** Orientation angle of the stock about the Z-axis to achieve the desired face position.
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

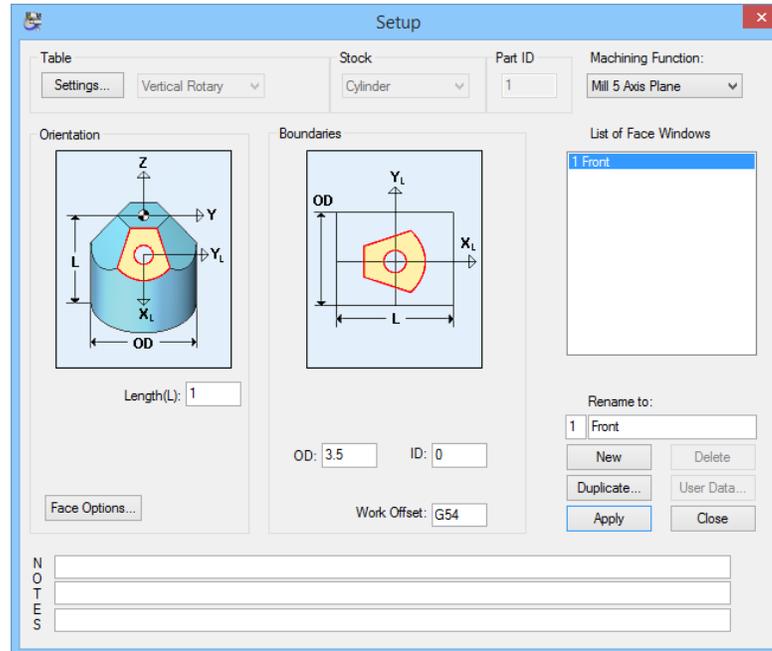
When the Table is set to Vertical Rotary using the **Mill XY Plane, Mill End Index Mill End Polar** window using **Cylinder Stock**, the **Face Options** dialog can be accessed by clicking the **Face Options** button. The only section of the **Face Options** dialog that applies to this table type is the **Arc Output Options** area shown here:



This should only be used when programming for machine controls that do not support circular interpolation of arcs (i.e. G2s and G3s). Checking the **Break Arcs Into Lines** box indicates to PartMaker to break all arc output into small lines in the NC program. The **Roughing** and **Finishing** field under **Arc Tolerance** specify the chord tolerance used when breaking arcs into lines for roughing and finishing operations, respectively.

## Machining Functions for Cylinder Stock: Mill 5 Axis Plane

The **Mill 5 Axis Plane** machining function Face Window supports 3-axis planar machining operations within an inclined plane. This inclined plane can be achieved by tilting the 5-axis rotary table in an arbitrary 5-axis plane. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining that must be machined on an angle.



**Length (L):** Length of the initial stock being machined

**OD:** The Outside diameter of the stock

**ID:** The Inside diameter of the stock

**Work Offset:** The work coordinate system of the stock in the current Face Window

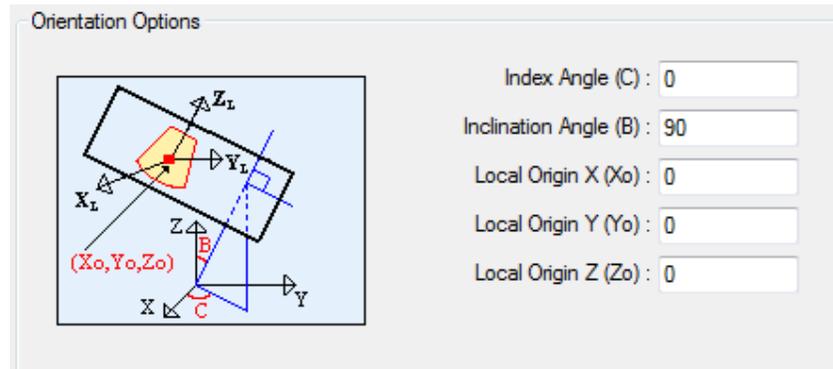
### Face Options

When the Table is set to Vertical Rotary, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

Two sections of the **Face Options** dialog apply to the **Mill 5 Axis Plane** Machining Function.

## Orientation Options

The **Orientation Options** section of the **Face Options** dialog allows you to set the location of local coordinate system being used on the inclined plane being machined.



- Index Angle (C):** Specifies the angle of the part rotation about the Z-axis to which the stock is indexed
- Inclination Angle (B):** Sets the inclination angle to which the table is tilted
- Local X (X<sub>o</sub>),  
Local Y (Y<sub>o</sub>)  
Local Z (Z<sub>o</sub>):** These fields represent the co-ordinates for the local origin created for the planar face being machined. These fields represent the coordinates with respect to the work pieces' global coordinate system (part coordinate system of the solid model) origin after the work piece is indexed or rotated by the Index Angle (C).

## Arc Output Options

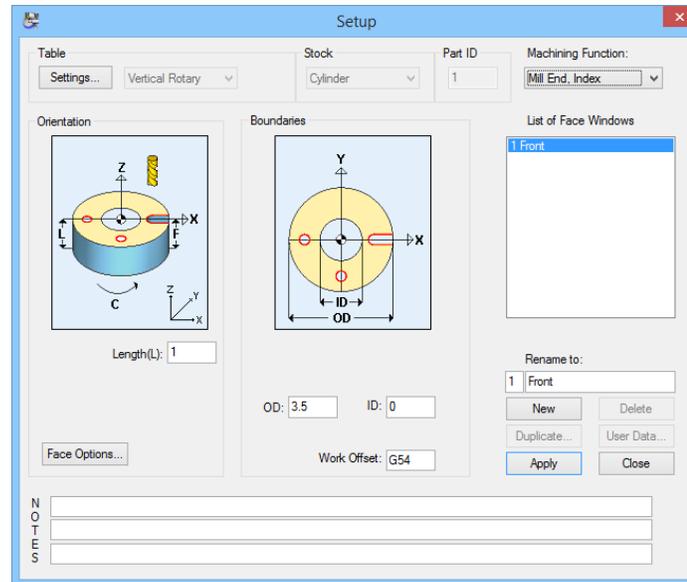
See page 2-18 of this guide for an explanation of the **Arc Output Options** section of **Face Options** dialog.

## Machining Functions for Cylinder Stock: Mill End Index

The Mill End, Index machining function 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. This motion type supports off-center drilling on the end of the part, using C-axis indexing and X-axis or Y-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 or Y-axis motion is used to cut the part.

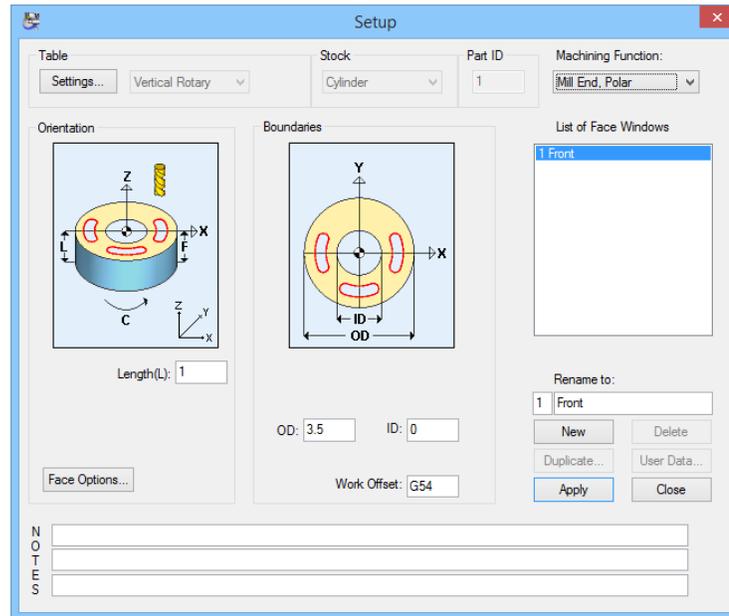


- Length (L)** Length of the finished part
- OD** Starting outside diameter of bar stock before machining
- ID** Starting inside diameter of bar stock before machining
- Work Offset** Specifies the work offset for the current face window

## Machining Functions for Cylinder Stock: 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 Mill Center has the ability to perform simultaneous feed motions with the X, Z and C-axes.

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



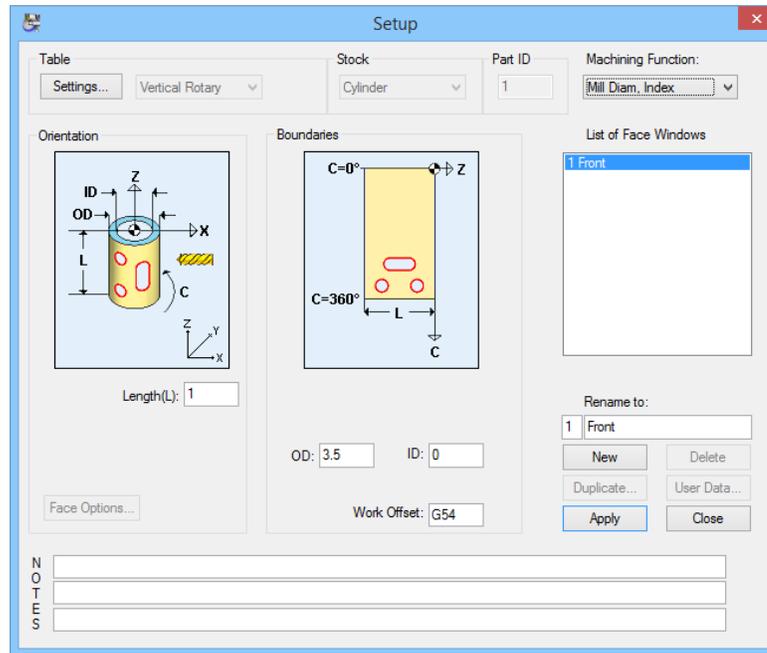
- Length (L)** Length of the finished part
- OD** Starting outside diameter of bar stock before machining
- ID** Starting inside diameter of bar stock before machining
- Work Offset** Specifies the work offset for the current face window

## Machining Functions for Cylinder Stock: Mill Diam, Index

The **Mill Diam, Index** machining function allows you to create “cross holes” or slots on the OD of the part. In order to support this machining function, your mill must be able to support Rotary-axis indexing.

This machining function supports the following features cut on the outside diameter of the cylinder.

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



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

The **Face Options** dialog is disabled for the Mill Diameter Index Machining Function.

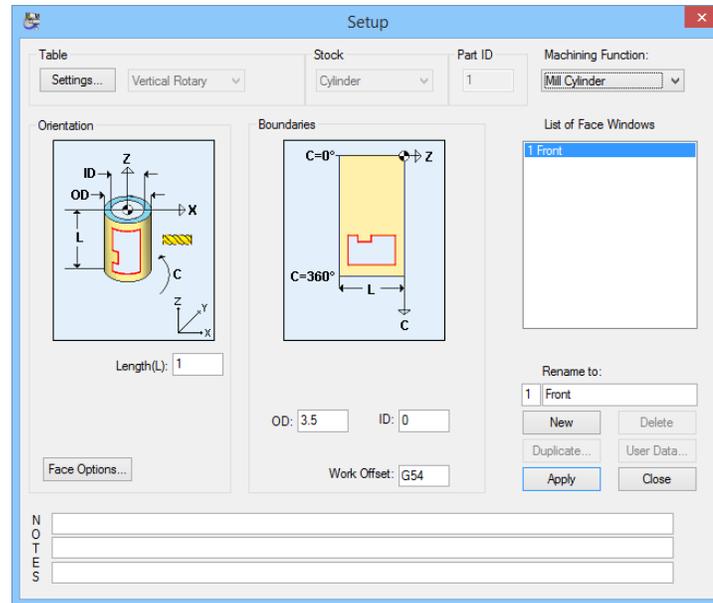
## Machining Functions for Cylinder Stock: Mill Cylinder

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

Using motions of this type, combined with a tool motion, profiles can be created on the outside diameter of the stock.

Tool positioning is programmed using X, Y, and C axis coordinates. The C position is programmed in terms of degrees of rotation relative to machine C=0.

Programming of this machining function requires that your Mill has a programmable Rotary-axis.



- Length (L):** Length of the initial stock being machined
- X Face (F):** The X coordinate of the front face of the initial stock
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

### Face Options

When the Table is set to Vertical Rotary, the **Face Options** dialog can be accessed by clicking the **Face Options** button.

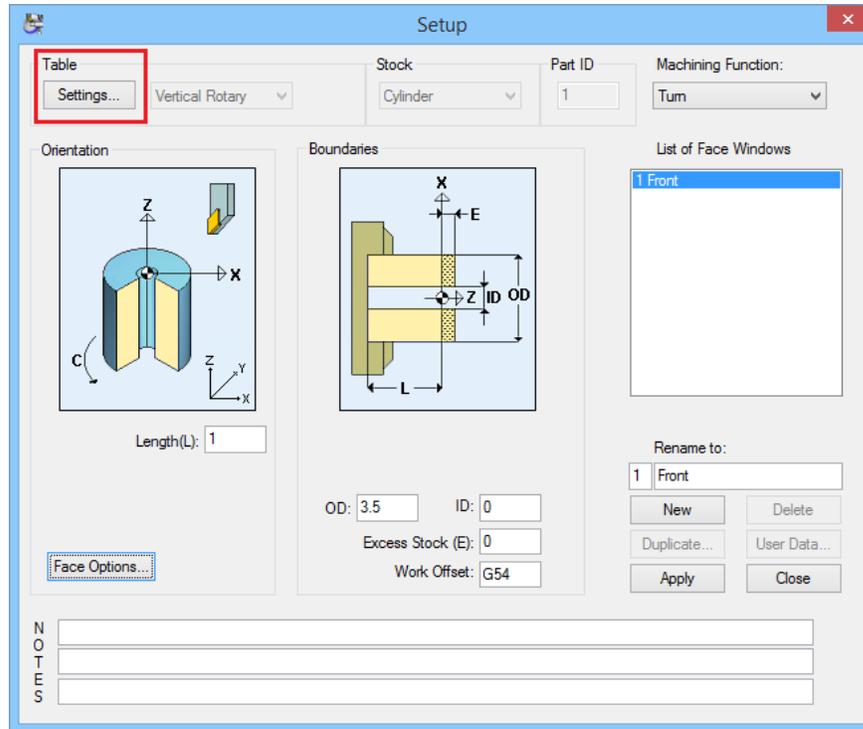
Only **Arc Output Options** section of this dialog applies to the **Mill Cylinder** Machining Function.

The **Arc Output Options** area of this dialog is explained on page 2-18.

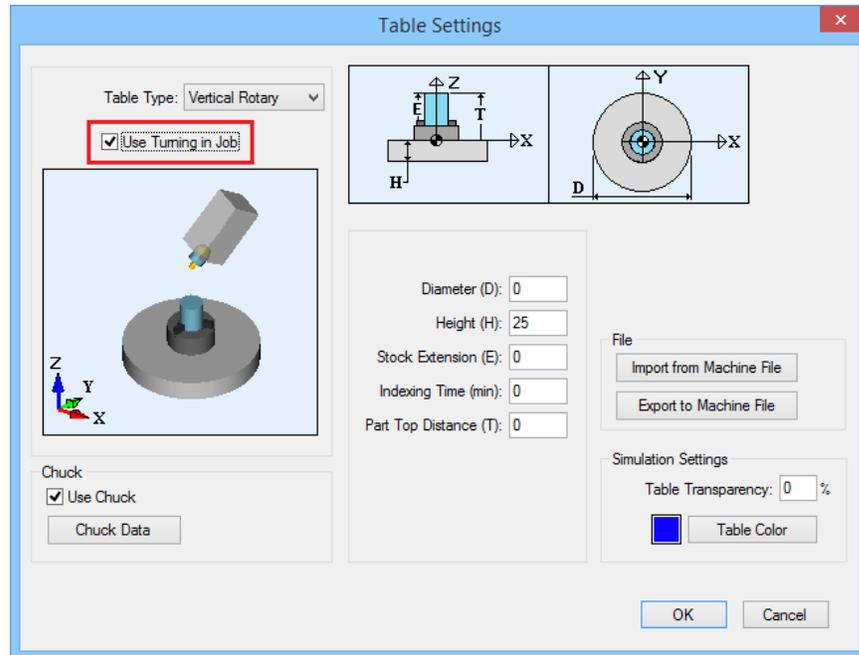
## Machining Functions for Cylinder Stock: Turning

The picture below shows the appearance of the **Setup** dialog when performing Turning on the Vertical Rotary table.

The Turn machining function is only available when the Stock selection is set to Cylinder and the **Use Turning in Job** check box is selected in the **Settings** Dialog.



In the Set up Dialog click on the **Settings** button and select **Use Turning in Job**. This has to be selected while in any of the Mill Machining Functions.

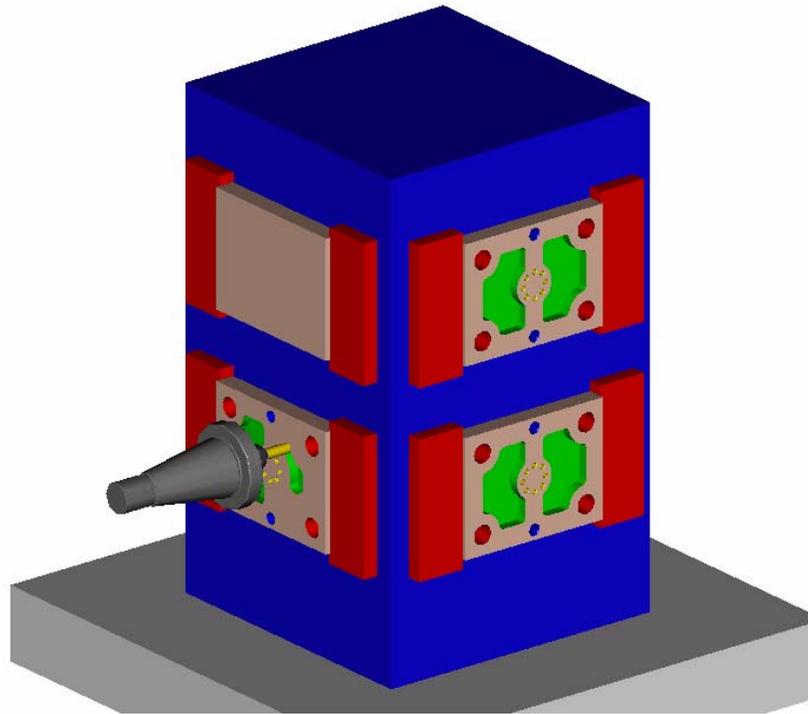


The Turn machining function uses conventional turning tools moving in the X and Z-axes. This machining function includes on centerline drilling where a drill is held stationary while the spindle holding the part rotates. All drilled holes must therefore be on the rotational centerline of the part.

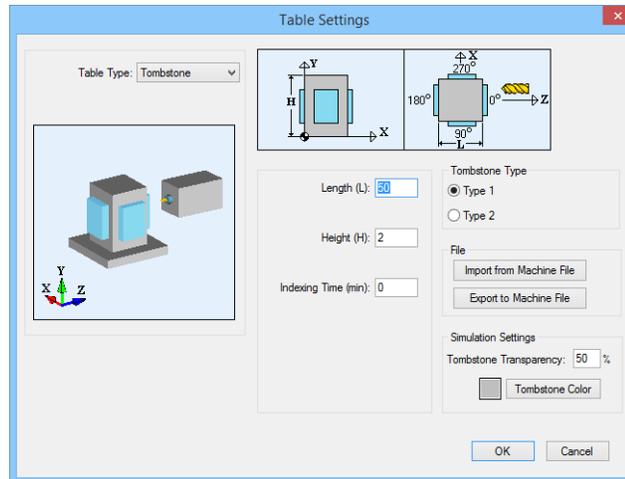
|                         |   |
|-------------------------|---|
| <b>Diameter (D)</b>     | Diameter of the Rotary Table                            |
| <b>OD</b>               | Starting outside diameter of bar stock before machining |
| <b>ID</b>               | Starting inside diameter of bar stock before machining  |
| <b>Excess Stock (E)</b> | Rough stock left for facing on the front of the part    |
| <b>Work Offset</b>      | Specifies the work offset for the current face window   |

## Table Type: Tombstone

Setting the Table Type to **Tombstone** assumes you are working on a horizontal machining center with a Tombstone machining table. This Table Type is especially helpful when machining multiple pieces in a single setup. With the Tombstone table setting, multiple stocks are allowed in simulation. The user can either choose an option to add a new stock for a given face or to "share" a stock defined on another face. The face where the stock is defined is called the "Independent" face; the face that uses a stock defined on another face is called the "dependent" face. Faces that "share" a stock may have different angles. Multiple parts can be machined by assigning a Part ID for every face. All of the faces with the same Part ID belong to a single part. This Part ID is used for "Show Finished Part" in Simulation to reconstruct a single part machined in multiple fixtures. (Tombstone-Many-Stocks\vice\_demo.JOB shown here)



## Settings



|                                 |   |
|---------------------------------|---|
| <b>Length (L):</b>              | Specifies the Tombstone length  |
| <b>Height (H):</b>              | Specifies the Tombstone height  |
| <b>Indexing Time (min):</b>     | Specifies the average Tombstone indexing time   |
| <b>Tombstone Type</b>           | Type 1: Specifies tombstone layout where angle corresponding to each side is incremented in Clockwise direction.<br>Type2: Specifies tombstone layout where angle corresponding to each side is incremented in Counter Clockwise direction. |
| <b>Import From Machine File</b> | Allows you to import the parameters of a previously created machining table from an *.MCH file which saves you from having to define a new machining table for every PartMaker JOB file   |
| <b>Export to Machine File</b>   | Allows you to export the parameters of the specific machining table as an *.MCH file for use with future jobs. Exporting a these parameters to a machine file allows you to setup a number of different types of machining tables.          |
| <b>Tombstone Transparency</b>   | Allows you to set the transparency of the table in PartMaker's 3D simulation, 0 being totally opaque, 100 being totally transparent   |
| <b>Tombstone Color</b>          | Allows you to choose the table color of the machining table from the available color pallet.  |

## Stock

When the Table is set to **Tombstone**, you can choose from two types of stock, **Block** or **Cylinder**. The Block selection assumes you are working with prismatic stock, while the cylinder selection assumes you are working with cylindrical stock being held horizontally, perpendicular to the tombstone surface.

## Part ID

Specifies which part the current Face Window applies to on a machining table. Different face windows can be made for separate sides of the same part on a Standard Table, where each Face Window represents a different stock (see page 2-15 below for an explanation Independent and Dependent stocks).

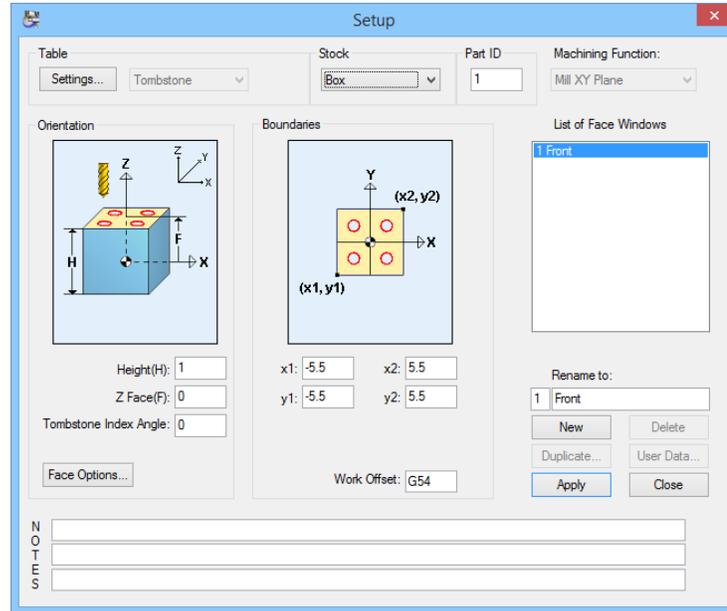
When multiple face windows representing different stocks have the same **Part ID**, the whole completed part can be shown in 3D simulation as PartMaker will know the relation to one another of the multiple stocks being machined as defined in the **Face Options** dialog (see page 2-15).

When multiple face windows representing different stocks have the different **Part IDs**, PartMaker will know different parts are being machined in each Face Window and prompt you for which completed part you wish to see in 3D simulation when choosing the **Show Finished Part** command.

## Machining Functions for Block Stock: Mill XY Plane

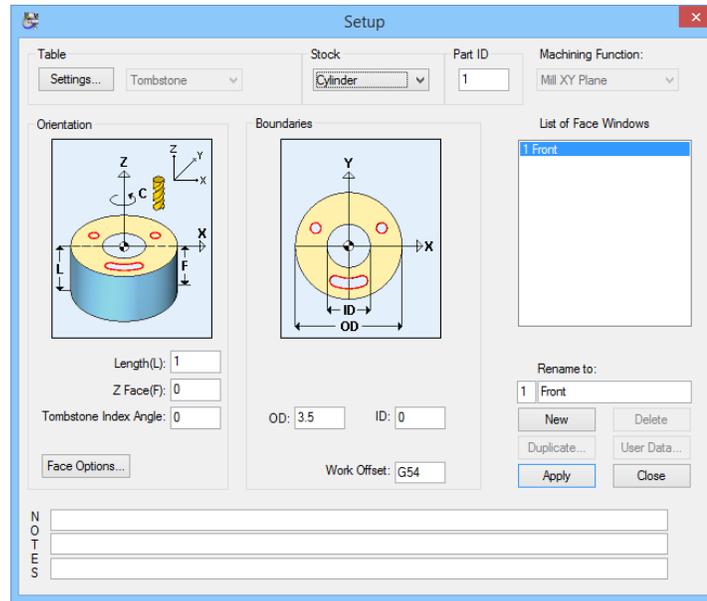
The Machining Function drop down menu is always set to **Mill XY Plane**.

The Mill XY Plane machining function supports 3-axis planar machining operations. Features that can be created by using this Face Window include holes, 2 ½ axis milled features such as pockets and contours and features requiring 3D surface machining.



- Height (H)** Height of the initial stock being machined
- Z Face (F):** The location of the top of the stock relative to Z = 0 Plane
- Tombstone Index Angle (C):** Orientation angle of the stock to achieve the desired face position.
- x1:** X coordinate of the lower left hand corner of the rectangular machining area
- y1:** Y coordinate of the lower left hand corner of the rectangular machining area
- x2:** X coordinate of the upper right hand corner of the rectangular machining area
- y2:** Y coordinate of the upper right hand corner of the rectangular machining area
- Work Offset:** The work coordinate system of the stock in the current Face Window

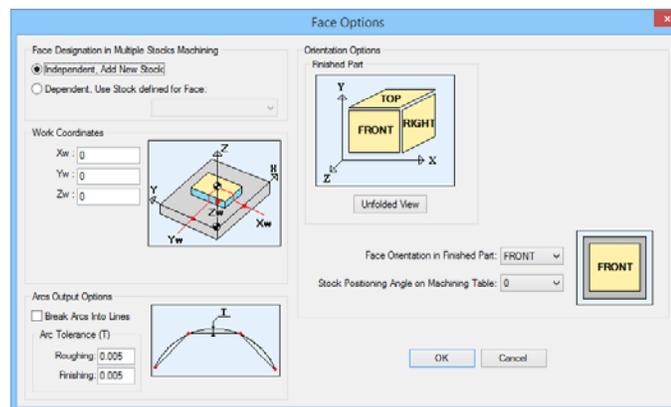
## Machining Functions for Cylinder Stock: Mill XY Plane



- Length (L):** Length of the initial stock
- Z Face (F):** The location of the top of the stock relative to Z = 0 Plane
- Tombstone Index Angle** Orientation angle of the tombstone to achieve the desired face position
- OD:** The Outside diameter of the stock
- ID:** The Inside diameter of the stock
- Work Offset:** The work coordinate system of the stock in the current Face Window

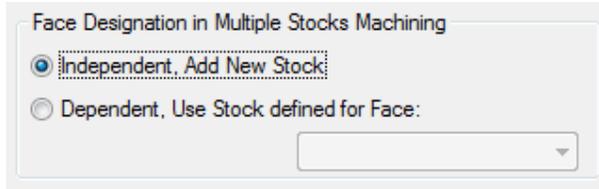
### Face Options

When the Table is set to **Tombstone** using **Box Stock**, the **Face Options** dialog can be accessed by clicking the **Face Options** button. When doing so, the **Face Options** dialog will appear as shown below:



Each section of the **Face Options** dialog is explained beginning on the next page.

## Face Designation in Multiple Stocks Machining



This section of the **Face Options** dialog allows you determine whether the machining operations programmed in the current Face Window are being done on a separate stock from those being used in other Face Windows or on the same stock as another face window.

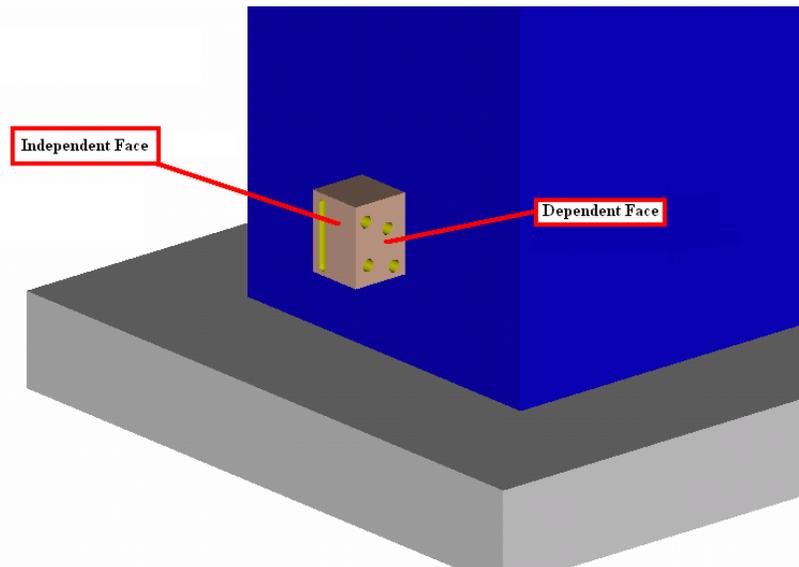
Checking the **Independent, Add New Stock** radio button indicates that tool paths created in the current Face Window are being machined on a separate stock from other face windows. Independent Stocks will be shown separately on the Tombstone in PartMaker's 3D simulation. This option will create a face in simulation that is parallel to the tombstone.



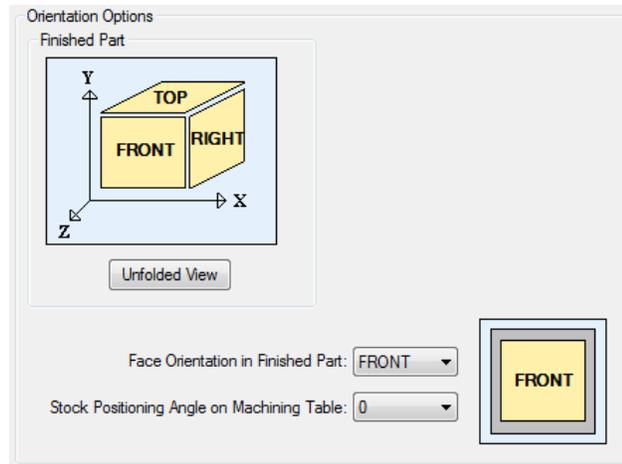
**Important!** Face windows containing *Independent Stocks* should each have a different **Work Offset** value assigned in the **Setup** dialog.

Checking the **Dependent, Use Stock defined for Face** radio button indicates that tool paths created in the current Face Window are being machined on the same stock defined in the face window designated. In the Setup dialog the **Part ID** will be disabled.

When creating dependent faces, the user must enter a value in **Tombstone Index Angle** field in the **Setup** dialog that corresponds to the index angle for the dependent face. For example, the FRONT Independent Face shown below can be specified at an index angle of 0 degrees. The RIGHT Dependent face would need an index angle of 270 degrees to have the face become aligned with the tool for machining. The FRONT and RIGHT designations used here are explained further below under **Orientation Options**.



## Orientation Options



This **Orientation Options** section of the **Face Options** dialog allows you determine which face of a multi-sided part, i.e. the FRONT, BACK, TOP, BOTTOM, RIGHT or LEFT is being machined in the current Face Window.

When using the **Tombstone** as the machining table PartMaker provides you with a very unique capability for programming a multi-sided part on a horizontal machining center where each side of the part is being machined in a separate vice and assigned a different work offset.

The following set of options is only available if **Block Stock** is selected in the **Setup** dialog. This feature allows you specify different Face Windows for different sides of the block, program them in the orientation needed to produce proper NC code (how it will be set up on the CNC machine table), and allow PartMaker simulation to connect the machining conducted on different stocks together to show the correct finished part.

The relation of these faces is selected in the **Face Orientation in Finished Part** drop down menu and its orientation on the tombstone is selected in the drop down menu for **Stock Positioning Angle on Machining Table**.

**Face Orientation in Finished Part:** This allows the user to select the side of the part which the face window will represent in the finished part during simulation. The **FRONT** face of the part is always parallel to the X-Y plane of the solid model and has the Z-axis pointing upward.



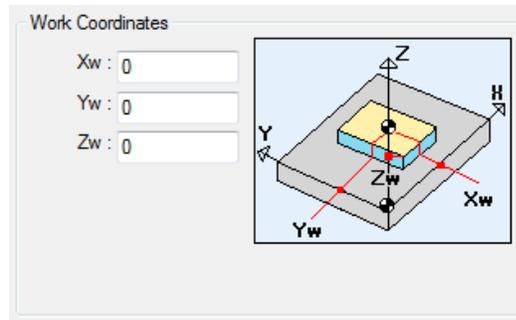
**Note:** If you work with a solid model the **Face Orientation in Finished Part** will be set automatically and the **Stock Positioning Angle** will be set to Zero.



**Note:** **FRONT** orientation MUST be used on every part otherwise an error message will display.

**Stock Position Angle on Machine Table:** Allows the user to specify how stock for a given face window will be oriented during machining when compared to finished part. The NC code will not be affected by this field but the tool path in PartMaker's 3D Simulation will be rotated according the value in this field. If you are rotating the part to 90 or 270 degrees, you will need to adjust the coordinates of the boundaries in the Setup Window.

## Work Coordinates



This **Work Coordinates** section of the **Face Options** dialog allows you determine the location of an **Independent Stock** on the machining table. The work coordinates are used to specify the coordinates of the stock on the tombstone in terms of the Tombstone Coordinate System with origin at bottom left corner of the tombstone.

**Xw:** Allows you to define the X location of the origin of the stock relative of the lower left hand corner of the Tombstone side where the stock is mounted.

**Yw:** Allows you to define the Y location of the origin of the stock relative of the lower left hand corner of the Tombstone side where the stock is mounted.

**Zw:** Allows you to define the Z location of the origin of the stock relative of the lower left hand corner of the Tombstone side where the stock is mounted.

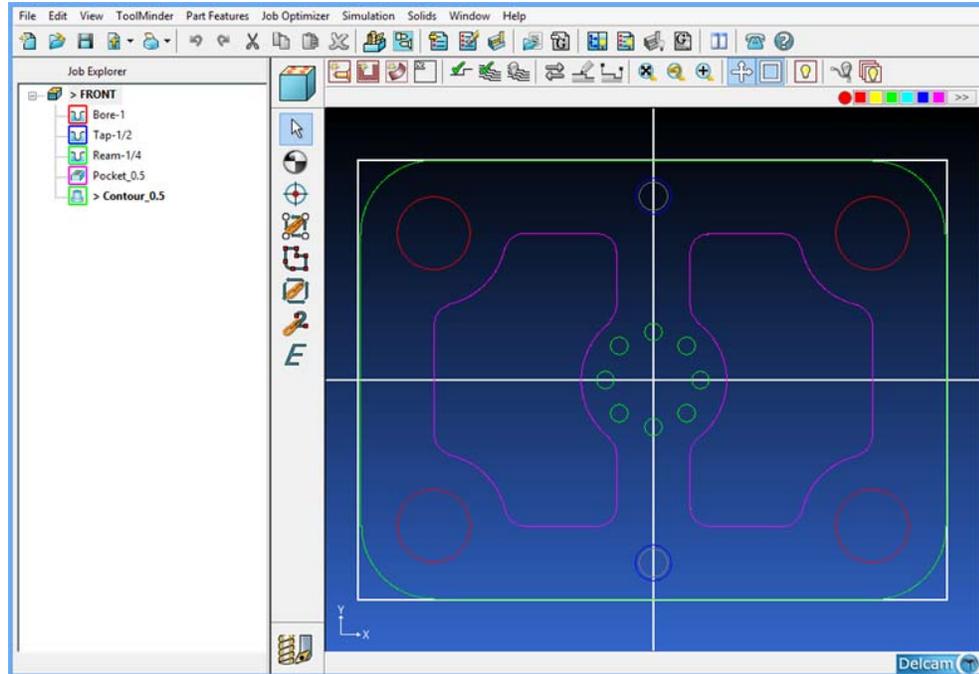
The values entered in these fields can be manipulated in PartMaker's 3D simulation window by double clicking on the stock using the **Change Stock Position** dialog.

## Arc Output Options

The **Arc Output Options** area of this dialog is explained on page 2-18.

## Working in a Face Window

A **Face Window** is the work area in which you define groups of holes and profiles that need to be machined. The main area of the window displays the part boundaries; the left side displays group icons, which have cycle name labels needed to machine the current part.



### Display the Axes



You can display the axes in a face window by choosing Show Axes from the View menu.

### Display the Grid



You can display the grid in a face window by choosing Show Grid from the View menu.

### Display the Boundaries



You can display the part boundaries in a face window by choosing Show Boundaries from the View menu.

## Using the CAD/CAM Switch



The left icon represents CAM mode. Clicking on it will switch to the CAD mode represented by the icon on the right.



## 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** (round color icon).



*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 (round icon) 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 (round icon) 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

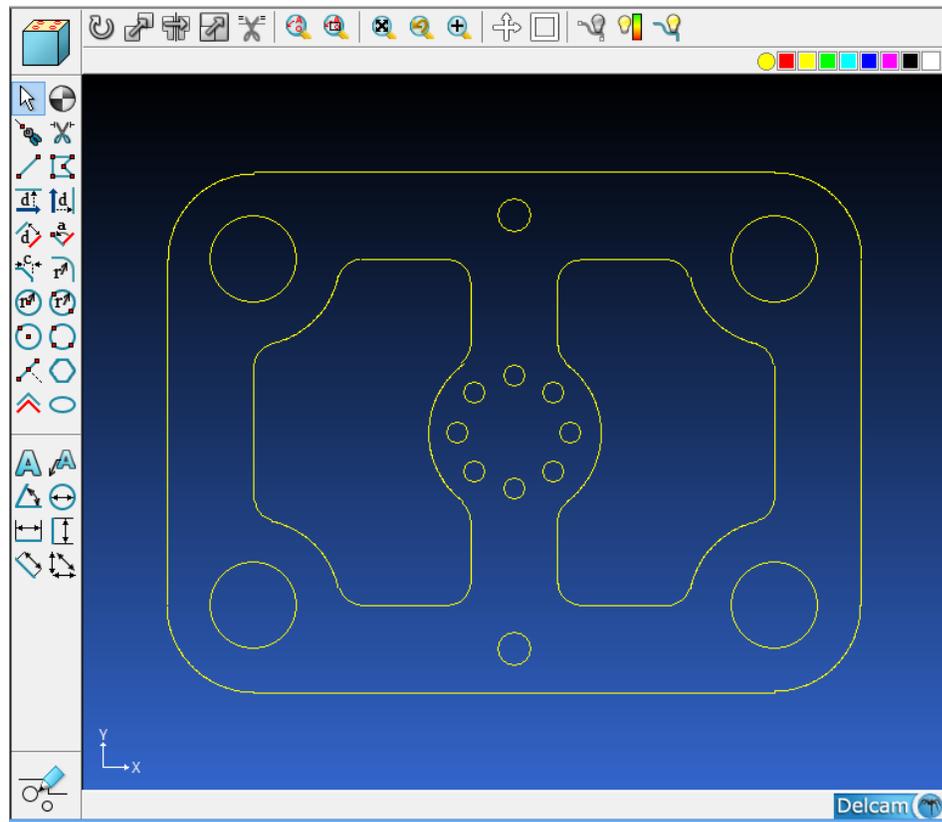
The **Face Window Toolbar** lets you perform many of the functions in PartMaker's menus with the click of a button. The **Face Window Toolbar** applies only to the active **Face Window** and may look differently depending on the **Machining Function** Face Window being used.





## Using the Graphics Icons in PartMaker CAD mode

You can create your part drawings in **PartMaker** in the CAD mode. Alternatively, you can 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 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.



-  **XY Coordinates:** Used for entering Cartesian coordinates.  
For **Face Windows** of various machining functions, this snap mode will appear differently.
  - XY** for Mill XY Plane, Mill Polygon, Mill End Index, MillEnd Polar and Mill Diam Polar
  - ZX** for Mill ZX Plane and Turn
  - ZC** for Mill Diam Index and Mill Cylinder
  - ZY** for Mill ZY Plane
  
-  **Polar Coordinates:** Used for entering Polar coordinates
  
-  **Point on a Circle:** Clicking a point on an existing arc or circle and entering an Angle
  
-  **Screen:** Clicking a cursor location anywhere within the face window
  
-  **Grid:** Clicking a grid location (grid can be turned on from the View Menu by choosing Show Grid. Grid size can be set in the Preferences dialog accessed from the View menu).
  
-  **Circle Center:** Clicking the center an existing circle
  
-  **Closest Intersection:** Clicking the closest intersection of geometric elements
  
-  **End of an Element:** Clicking the end of an existing line or arc
  
-  **Middle of an Element:** Clicking the middle of an existing line or arc
  
-  **Tangency:** Clicking near the tangency point that will be calculated by the system
  
-  **Horizontal Constraint** allows you to specify a point whose vertical coordinate remains constant, allowing changes in horizontal coordinate only.
  
-  **Vertical Constraint** allows you to specify a point whose horizontal coordinate remains constant, allowing changes in vertical coordinate only.

## Using the Graphics Icons in PartMaker CAM mode

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

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



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

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



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

### Graphics Icons: Machining Functions Mill XY Plane, Mill 4 Axis Plane, Mill 5 Axis, Mill Polygon, Mill Diam Index and Mill Cylinder

The Machining Functions Mill XY Plane, Mill ZY Plane, Mill Polygon, Mill End Polar, Mill Diam Polar and 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.



**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** 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.



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



**Hole Pattern Icon** is displayed in PartMaker/Mill as a dropdown menu:

## Graphics Icons: Machining Functions Mill End, Index and Mill Diam, Index

The Machining Functions Mill End, Index and 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.

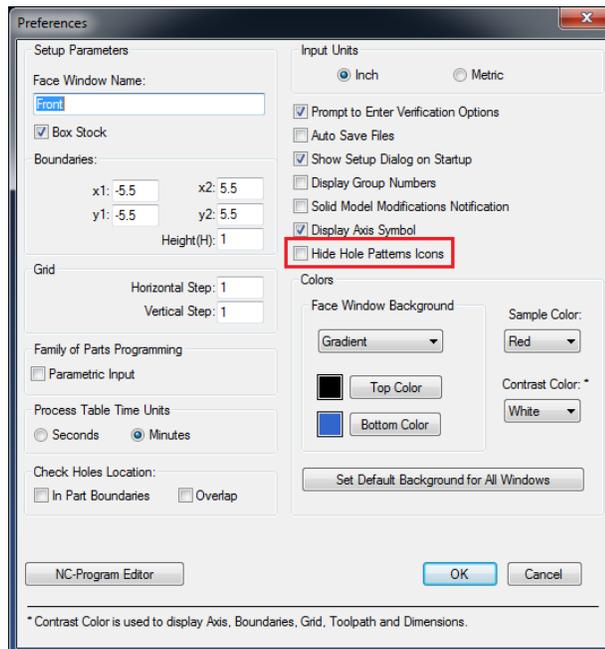


**Chain Icon** enables you to specify that the currently active group symbol is to be applied to a profile that is created when PartMaker automatically connects all the elements that form an unbroken chain with an initial line or arc clicked. When performing indexing operations, the Chain Icon may only join two points during its usage.

## Hole Pattern Icon – Machining Function Mill XY Plane, Mill 4 Axis Plane, Mill 5 Axis.

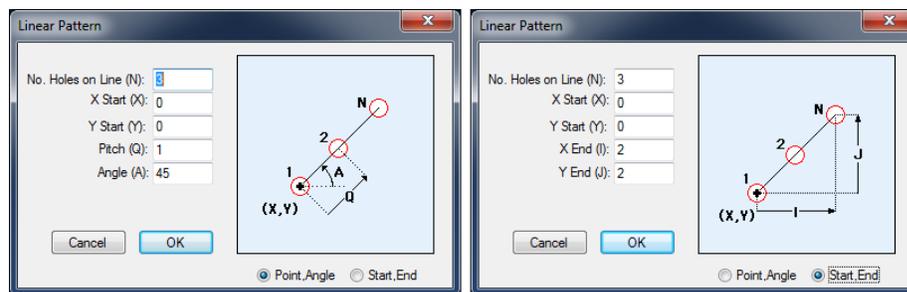
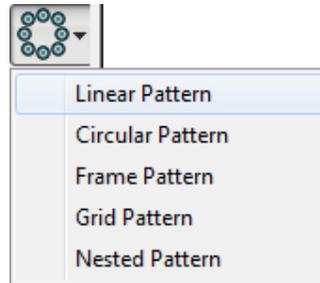


The Graphics icons on the left side of a face window allow you to define holes by entering individual point coordinates. If you prefer, you can define “patterns” of holes by using the **Hole Pattern Icon**; A dialog will be used to specify options such as the number of holes, X and Y starting coordinates, pitch, angle, radius, length, and/or offset. (To see complete list below, click on **View, Preferences, Uncheck Hide Hole Pattern Icon**, click ok.)



## Using the Linear Pattern Icon

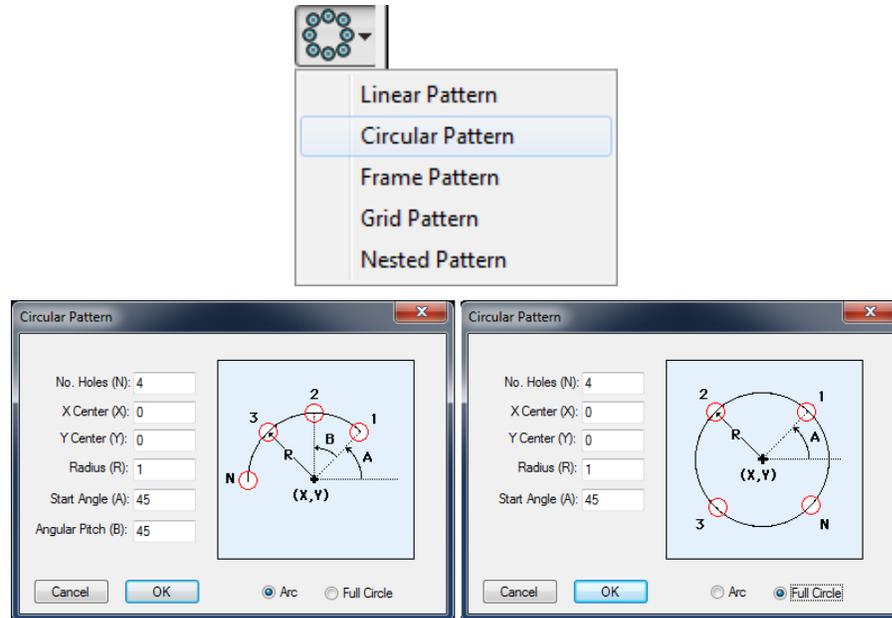
The Linear Pattern icon lets you specify a linear pattern of holes in a face window. When you select the Linear Pattern icon, the Linear Pattern dialog appears. After entering parameters in the dialog, a linear pattern of holes is displayed in the face window.



- |                              |   |
|------------------------------|---|
| <b>No. Holes on Line (N)</b> | Specifies the number of holes in the pattern  |
| <b>X Start (X)</b>           | Specifies the X starting coordinate for the first hole in the pattern                     |
| <b>Y Start (Y)</b>           | Specifies the Y starting coordinate for the first hole in the pattern                     |
| <b>Pitch (Q)</b>             | Specifies the pitch of holes in the pattern from the X, Y starting point                  |
| <b>Angle (A)</b>             | Specifies the angle of holes in the pattern from the X, Y starting point                  |
| <b>Point, Angle</b>          | Creates the hole pattern based on the starting location and angle entered in the dialog   |
| <b>Start, End</b>            | Creates the hole pattern based on the starting and ending locations entered in the dialog |
| <b>X End (I)</b>             | Specifies the X coordinate of the ending location for the pattern                         |
| <b>Y End (J)</b>             | Specifies the +Y coordinate of the ending location for the pattern                        |

## Using the Circular Pattern Icon

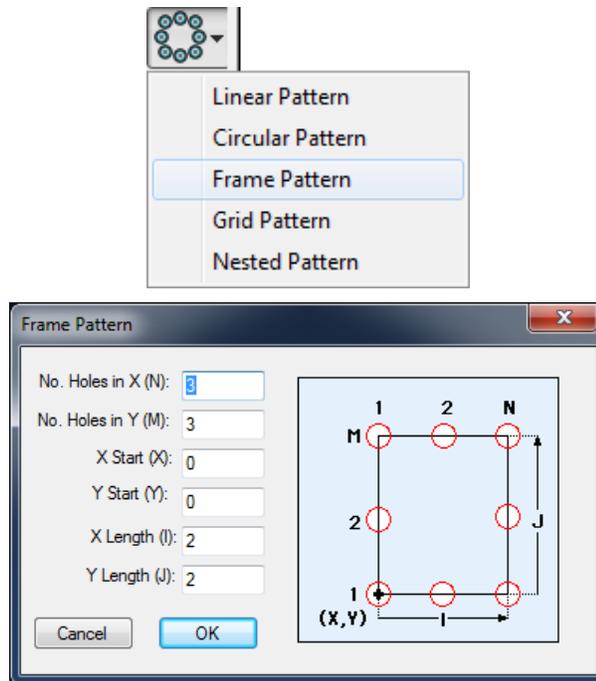
The Circular Pattern icon lets you specify a circular or arc-shaped pattern of holes in a face window. When you select the Circular Pattern icon, the Circular Pattern dialog appears. After entering parameters in the dialog, a circular pattern of holes is displayed in the face window.



|                          |   |
|--------------------------|---|
| <b>No. Holes (N)</b>     | Specifies the number of holes in the pattern  |
| <b>X Center (X)</b>      | Specifies the X starting coordinate for the pattern center                                      |
| <b>Y Center (Y)</b>      | Specifies the Y starting coordinate for the pattern center                                      |
| <b>Radius (R)</b>        | Specifies the radius of the pattern center  |
| <b>Start Angle (A)</b>   | Specifies the starting angle from which the pattern is generated                                |
| <b>Angular Pitch (B)</b> | Specifies the pitch for arc-shaped patterns (This option is not used for full circle patterns.) |
| <b>Arc</b>               | Creates an arc-shaped pattern of holes  |
| <b>Full Circle</b>       | Creates a circular pattern of holes   |

## Using the Frame Pattern Icon

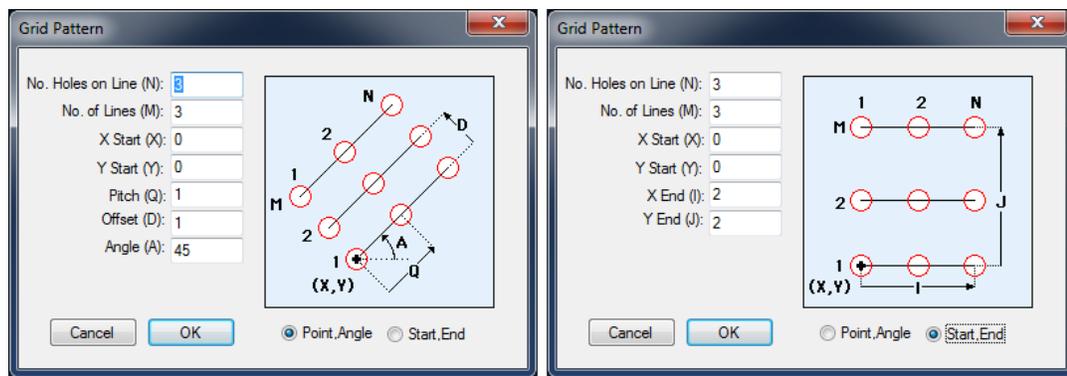
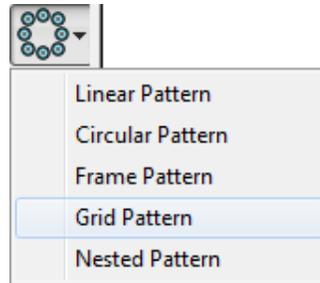
The Frame Pattern icon lets you specify a rectangular pattern of holes in a face window. When you select the Frame Pattern icon, the Frame Pattern dialog appears. After entering parameters in the dialog, a rectangular pattern of holes is displayed in the face window.



- No. Holes in X (N)** Specifies the number of holes in the pattern in the X (horizontal) direction
- No. Holes in Y (M)** Specifies the number of holes in the pattern in the Y (vertical) direction
- X Start (X)** Specifies the X starting coordinate for the first hole in the pattern
- Y Start (Y)** Specifies the Y starting coordinate for the first hole in the pattern
- X Length (I)** Specifies the total length of horizontal holes from the center of the first hole to the center of the last hole
- Y Length (J)** Specifies the total length of vertical holes from the center of the first hole to the center of the last hole

## Using the Grid Pattern Icon

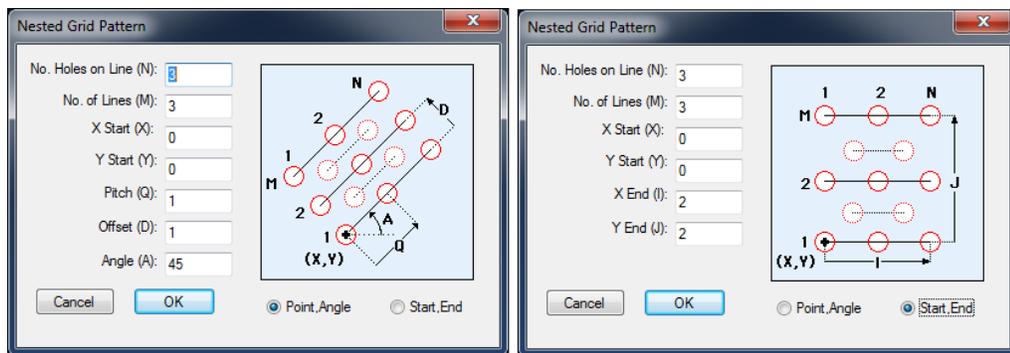
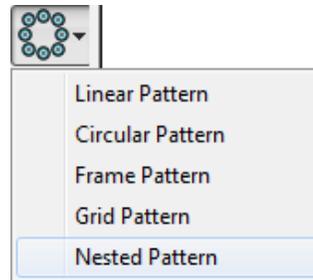
The Grid Pattern icon lets you specify a rectangular grid pattern of holes in a face window. When you select the Grid Pattern icon, the Grid Pattern dialog appears. After entering parameters in the dialog, a grid pattern of holes is displayed in the face window.



- No. Holes on Line (N)** Specifies the number of holes in the pattern
- X Start (X)** Specifies the X starting coordinate for the first hole in the pattern
- Y Start (Y)** Specifies the Y starting coordinate for the first hole in the pattern
- Pitch (Q)** Specifies the pitch of holes in the pattern from the X, Y starting point
- Offset (D)** Specifies the offset distance for each line of holes in the pattern
- Angle (A)** Specifies the angle of holes in the pattern from the X, Y starting point
- Point, Angle** Creates the hole pattern based on the starting location and angle entered in the dialog
- Start, End** Creates the hole pattern based on the starting and ending locations entered in the dialog
- X End (I)** Specifies the X coordinate of the ending location for the pattern
- Y End (J)** Specifies the +Y coordinate of the ending location for the pattern

## Using the Nested Grid Pattern Icon

The Nested Grid Pattern icon lets you specify a rectangular nested grid pattern of holes in a face window. When you select the Nested Grid Pattern icon, the Nested Grid Pattern dialog appears. After entering parameters in the dialog, a nested grid pattern of holes is displayed in the face window.



- No. Holes on Line (N)** Specifies the number of holes in the pattern
- X Start (X)** Specifies the X starting coordinate for the first hole in the pattern
- Y Start (Y)** Specifies the Y starting coordinate for the first hole in the pattern
- Pitch (Q)** Specifies the pitch of holes in the pattern from the X, Y starting point
- Offset (D)** Specifies the offset distance for each line of holes in the pattern
- Angle (A)** Specifies the angle of holes in the pattern from the X, Y starting point
- Point, Angle** Creates the hole pattern based on the starting location and angle entered in the dialog
- Start, End** Creates the hole pattern based on the starting and ending locations entered in the dialog
- X End (I)** Specifies the X coordinate of the ending location for the pattern
- Y End (J)** Specifies the +Y coordinate of the ending location for the pattern

---

# Chapter 3: Creating a Sample Part

## Introduction

The illustration on the following page represents a typical task you might perform. The name of this part is Milldemo. You'll create the Milldemo part in this chapter using **PartMaker**.

This tutorial is designed to help you learn some of the most common **PartMaker** commands and features you'll use every day. Before you begin, we recommend you read Chapter 2 if you haven't already done so to familiarize yourself with some basic **PartMaker** concepts and features.



**Important!** *The tutorial below is presented in both inch and metric units, depending on how you are working with PartMaker. The inch units of measure are given first, with the metric entries given in parenthesis ( ).*

For example you may see instruction that says: Enter an X value 1 (25), where 1 is 1 inch and 25 is 25 mm.

## How You Will Create Milldemo

Here are the major steps you'll follow to create Mill-demo:

- ✓ Start **PartMaker** and open a new **Face Window**
- ✓ Specify part boundaries
- ✓ View tool information for the part
- ✓ View cycle information for the part
- ✓ Select a material for the part
- ✓ Set up **PartMaker** defaults
- ✓ Create Geometry
- ✓ Define holes for a boring cycle
- ✓ Define holes for a tapping cycle
- ✓ Define holes for a reaming cycle
- ✓ Define profiles for Pocket Mill
- ✓ Define profiles for Contour Mill
- ✓ Select a post processor
- ✓ Generate a process table containing machining data
- ✓ Generate an NC program for the part
- ✓ Print the NC program for the part

Job File: Milldemo.job (Milldemo\_metric.job)

**Group 1: Holes 1" (24mm) Bore**

| Operation    | Inch Diameter | Metric Diameter |
|--------------|---------------|-----------------|
| Center Drill | 7/32          | 5               |
| Drill        | 31/32         | 23.5            |
| *Bore        | 1"            | 25              |
| Chamfer      | 1"            | 25              |

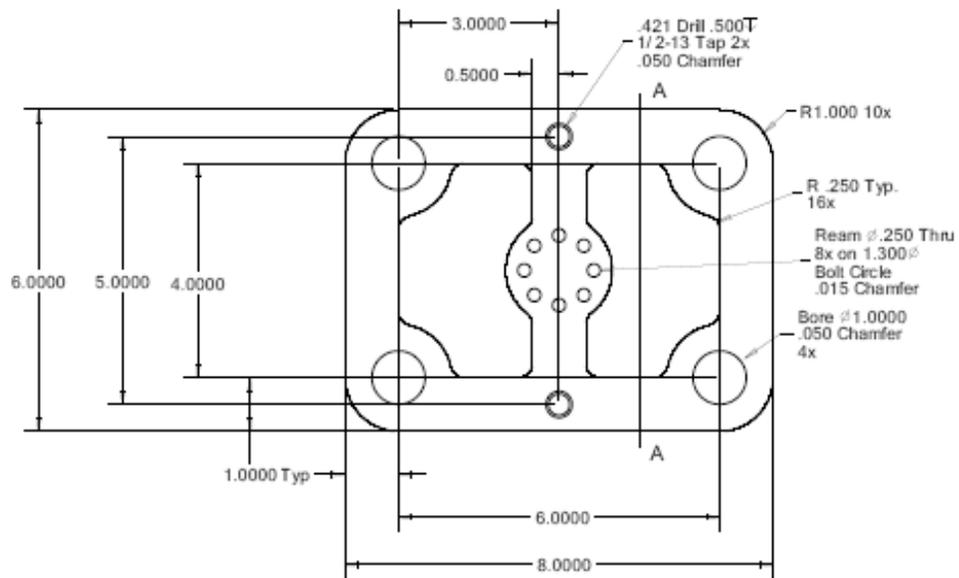
**Group 2: Holes 1/2" (12mm) Tap**

| Operation    | Inch Diameter | Metric Diameter |
|--------------|---------------|-----------------|
| Center Drill | 7/32          | 5               |
| Drill        | 27/64         | 10              |
| *Tap         | 1/2"          | 12.5            |
| Chamfer      | 1/2"          | 12.5            |

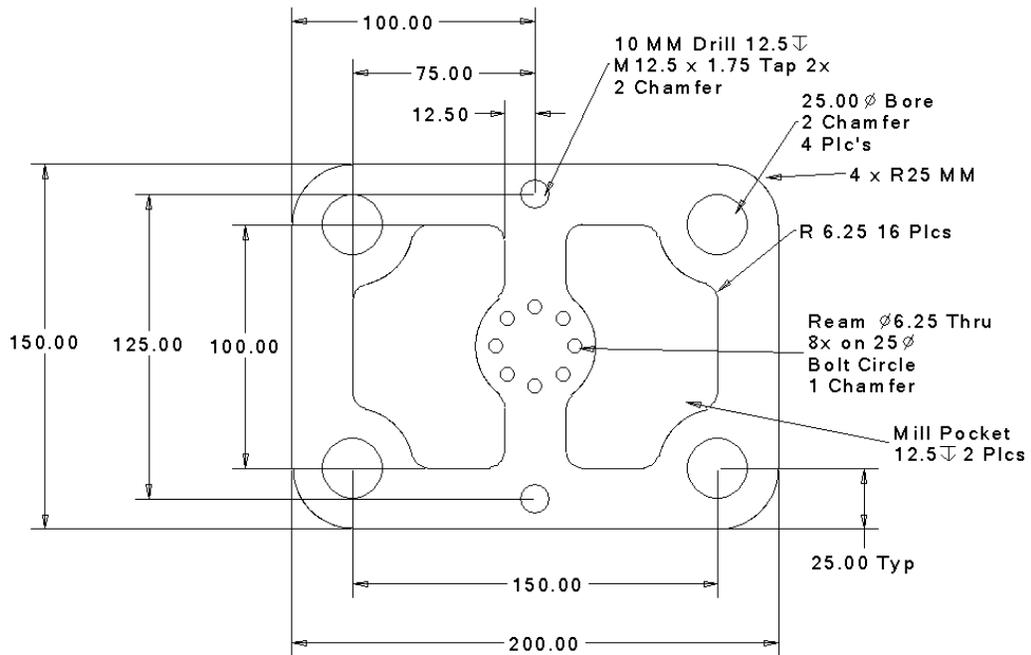
**Group 3: Holes 1/4" (6mm) Ream**

| Operation    | Inch Diameter | Metric Diameter |
|--------------|---------------|-----------------|
| Center Drill | 7/32          | 5               |
| Drill        | 15/64         | 5.5             |
| *Ream        | 1/4"          | 6.25            |
| Chamfer      | 1/4"          | 6.25            |

Inch Units



## Metric Units



The mill demo has three groups of holes. Each group has a cycle with a **major operation** indicated by an asterisk in the illustration on the previous page. Operations are shown for each group that is used for machining the part. The hole diameter information and hole locations for each group of holes are also shown in the illustration.

We have provided the boring and tapping cycle information and tool information for Milldemo; you'll create the reaming cycle needed to complete the part. (A complete list of what you'll be doing in the tutorial is shown in the next section.)

You may find it useful to refer to the illustration as you create Milldemo. We have provided the finished part. To view the part, do one of the following:

Choose **Open All Files** from the **PartMaker** File drop down menu, locate and select the file named Milldemo.job (Milldemo\_metric.job) in the **PartMaker** directory.

## Start PartMaker

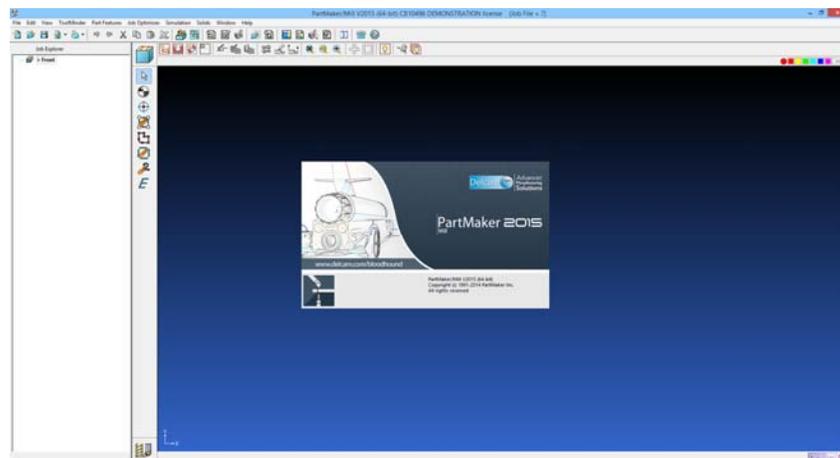
You will now look in detail at each of the steps necessary to program each of the components of the finished part. The components of the completed tutorial Mill part can be found in your PartMaker Mill directory under the following file names:

|                             |                                    |
|-----------------------------|------------------------------------|
| Completed Job File:         | C:\PartMaker\PM-Mill\Milldemo.job  |
| Completed Tools Data Base:  | C:\PartMaker\ PM-Mill\Milldemo.tdb |
| Completed Cycles Data Base: | C:\PartMaker\ PM-Mill\Milldemo.cdb |

In the Mill tutorial, the default **Tools Data Base** and **Cycles Data Base** will be the same as Milldemo.tdb and Milldemo.cdb found in the **PM-Mill** root directory.

- 1 Double Click on the PartMaker Mill icon on your Windows desktop, or in the Windows Start button, locate the **PartMaker** Group and double click the **PartMaker** Mill application icon.

A window containing the **PartMaker** logo appears.

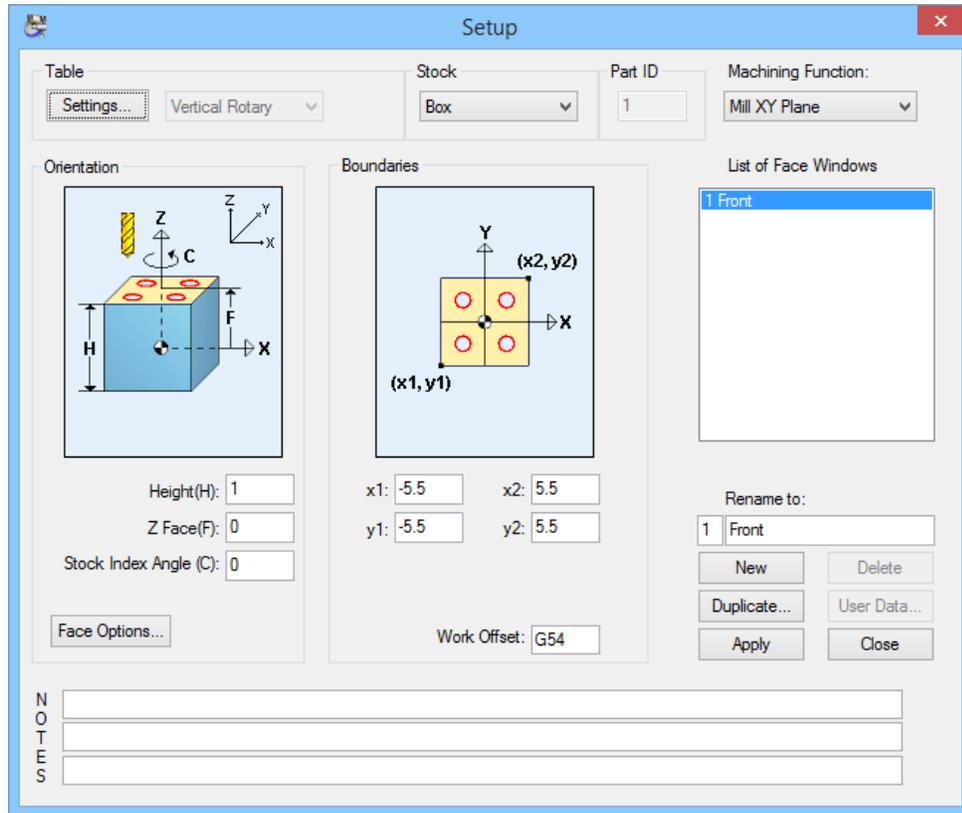


- 2 After a few seconds, you will see the Setup definition dialog box open. The **Setup** dialog appears on top of the **Face Window**.

## Setup Part Boundaries

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

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



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



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

- 2 From the **Table** drop down menu, choose **None**
- 3 Enter the following values next to these options in the dialog:

**x1**      -4.5    (-105)

**y1**      -3.5    (-80)

**x2**      4.5     (105)

**y2**      3.5     (80)

**Height** 1      (25)

- 4 Click <Apply>.
- 5 Click the close box in the lower-right corner of the dialog to close it and return to the **Face Window**.
- 6 Click anywhere in the **Face Window** to make sure it is active.

## View the Tool Information

The tool library for the Mill tutorial has been created.

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

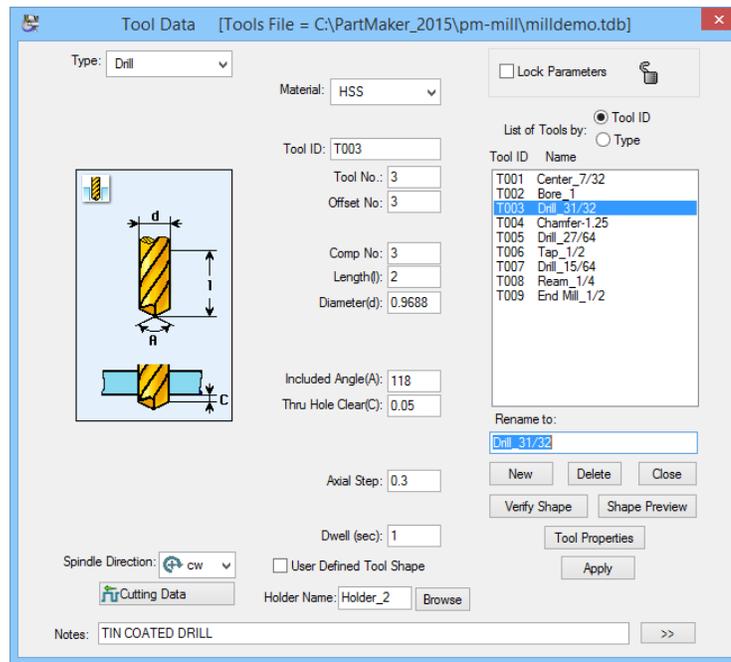


- 2 Open the **Milldemo.tdb** from the **C:\PartMaker\PM-Mill** directory as shown below by choosing **Open Tools File** from the **File** menu.

- 3 Choose **Tools** from the **ToolMinder** menu to display the Tool Data dialog.

- 4 Select the **Tool ID T003** from the list shown on the right side of the dialog to view a graphic display of the tool and its characteristics. Each tool you use in **PartMaker** has a tool type name, material, and a unique tool ID, which are used by the cycles you create for machining.

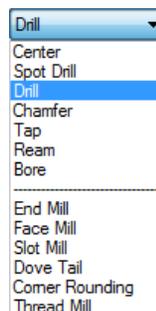
Notice that each tool picture corresponds to the selected tool type in the dialog.



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

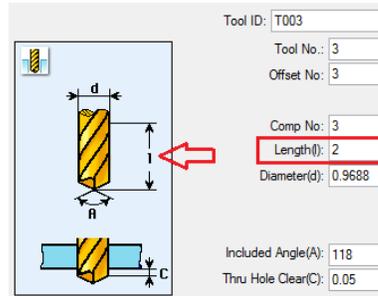
Each tool has many parameters that need to be fully defined so that **PartMaker** can accurately program your parts.

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



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

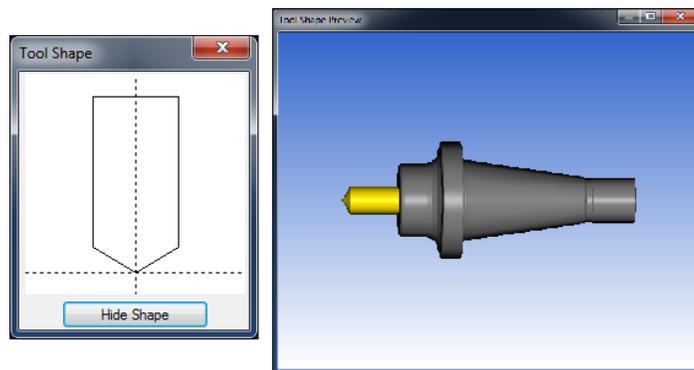
The color graphic to the left in the dialog box clearly describes most of the parameters as shown in the example below.



The following list explains the remaining parameters:

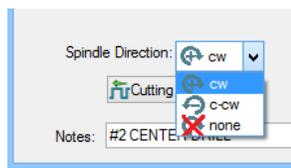
- **Material:** defines the cutting insert material, typically either **HSS** or **Carbide**.
- **Tool ID:** A unique ID number that is used by **PartMaker** to keep track of the tools you have defined.
- **Tool No.:** The number as it appears in the actual part program.
- **Comp No.:** The tool nose radius compensation number used for the tool.
- **Offset No.:** The offset register number used to define the length and diameter offset values for the defined tool.
- **Axial Step:** Specifies the peck depth for drills, the roughing depths for end mills. Specifies step in the axial cutting to be used by the currently-selected tool. The programmer may change this value at any time during the programming process as needed.

Below is a button labeled **Verify Shape** this button is used to display the 2 dimensional shape of the tool defined by the user parameters. The button labeled **Shape Preview** will display the 3 dimensional models of the tool and a holder if you have selected one. Press this button now to see the exact shape of the **31/32 (23.5) Drill**.



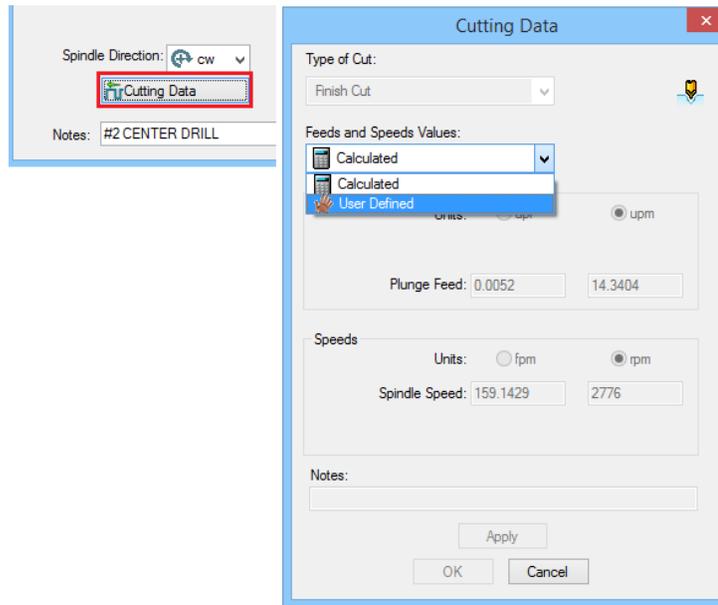
Click on the **Hide Shape** button to close the dialog.

**Spindle Direction** can be set to CW, CCW or None:



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

By Selecting the **Cutting Data** button you can choose to use Speeds and Feeds calculated from the Material file or set as User Defined values.



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



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

### View the Cycle Information

You can view information about the boring and tapping cycles used for Milldemo in the Cycles dialog.

**PartMaker** lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a cycle. The Cycle information you create is saved in a Cycles Database for easy retrieval; reusing cycle information is as easy as opening a cycles file from the cycle's database.



- 1 Choose **Cycles** from the **ToolMinder** menu to display the Cycles dialog.
- 2 Open the **Milldemo.cdb (Milldemo\_metric.cdb)** from the **C:\PartMaker\PM-Mill** directory as shown below by choosing **Open Cycles File** from the **File** menu.

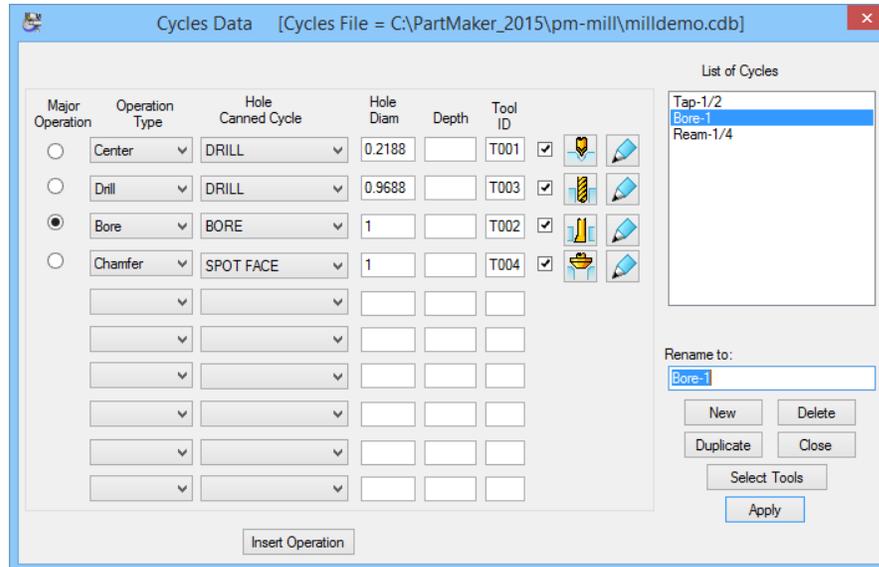
The cycle you are viewing is indicated by the selected cycle name under **List of Cycles**. The Major Operation type is displayed for each cycle, as well as the Canned Cycle, Hole Diameter, Depth, and Tool ID.

- 3 Select BORE-1 (BORE-24) on the right side of the dialog to view the information for this cycle, which will be used in Milldemo.job (Milldemo\_metric.job).



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

#### Cycle information for the BORE-1 cycle



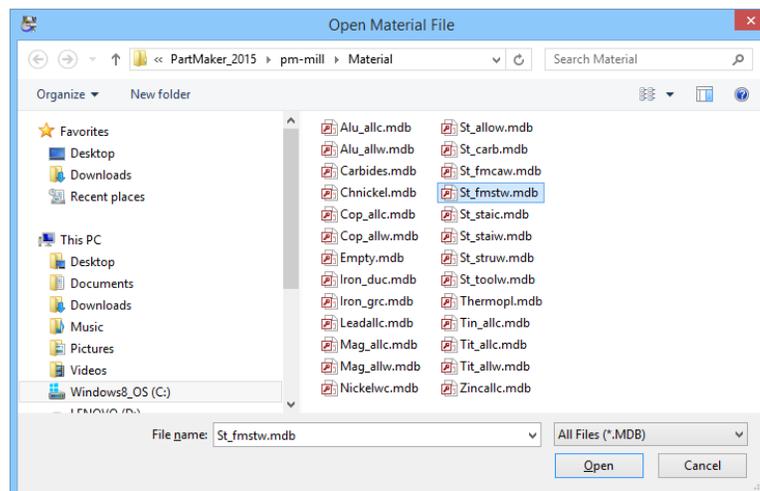
- 4 When you're finished viewing cycle information, click the **<Close>** button or double-click the close box in the upper-left corner of the dialog to return to the **Face Window**.

### Select a Material for the Part

Before you create the process table for Milldemo, you must select the type of material to be machined in the **Open Material File** dialog. Milldemo requires Free Machining Stainless Steel.

**PartMaker** has a materials database from which you can select a variety of material types with different characteristics. A material database allows you to store your shop's accumulated materials data. **PartMaker** uses this data for the automatic calculation of feedrates and spindle speeds.

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



- 2 Locate and select ST\_FMSTW (Free Machining Stainless Steel Wrought) from the list on the left side of the dialog, then click **<OK>**. (Refer to Appendix A for a complete list of materials.)

You are returned to the **Face Window**.



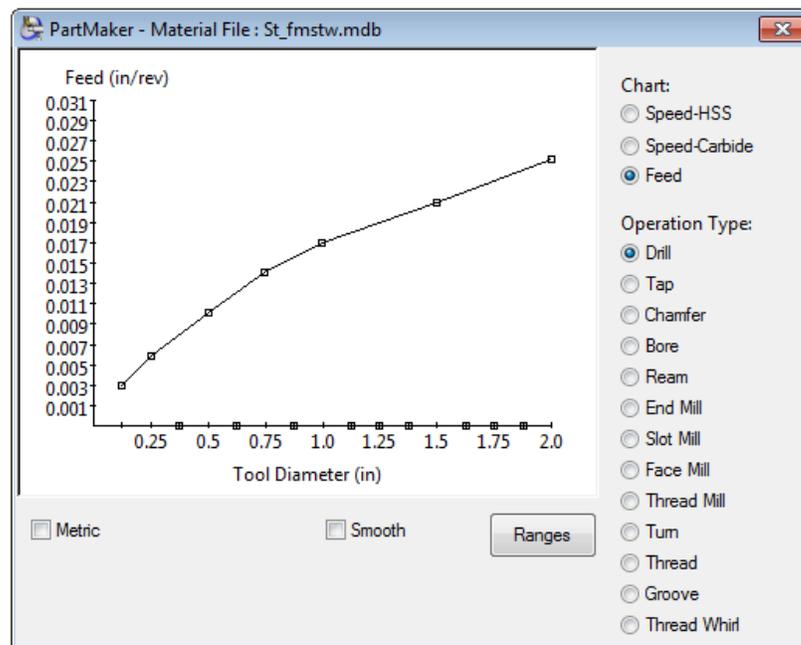
**Note:** Sometimes the **Face Window** loses focus and its title bar becomes gray. You may find a number of menu commands being disabled (gray) if a front window is not active (not in focus). To make sure the **Face Window** is the active window, click its title bar.

## View the Material Characteristics

In this section, you'll view the characteristics of the material you selected with respect to feedrates and/or speed, and the operation type.

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

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



- 2 Click the **<Feed>** button to see how the feed varies for the currently selected operation.
- 3 Click an **Operation Type** to see the feed for that operation.
- 4 Drag a point on the graph in the dialog to see how you can change the feed for the selected operation.
- 5 Double-click the close box in the upper-left corner of the dialog (on icon) to return to the **Face Window**.

## Setting Defaults

In this section, you'll use the Defaults dialog to specify a Z depth of one inch for the three groups of holes in Milldemo.

Since the tutorial is designed for a part using the inch mode, you must be sure that **PartMaker** is in the correct mode. To verify:

Choose **<Preferences>** from the **View** menu to display the Preferences dialog. If the **<Inch>** button under Input Units is not selected, click the radio button to the left. Click **<OK>** to close the dialog box.

Default settings are used in the program generated for a part. Hole group parameters are used for each new group (cycle) you create. You can change Process Parameters at any time in the Process Parameters dialog (**Process Parameters** command in the **Job Optimizer** menu); you can change Part Boundaries at any time in the Setup dialog (**Setup** command in the **View** menu).

- 1 Select **<Defaults>** from the **Job Optimizer** menu to display the Defaults dialog.

Defaults for Milling

Group Parameters

- Through Hole
- Diameter: 0.25
- Chamfer: 0
- Z\_Surf: 0
- Z\_Depth: 1
- Z\_Rapid: 0.05
- Z\_Clear: 0.05
- Width of Cut: %Tool Diameter
- Width of Cut Value: 35
- Surface Machining Module
  - SMW (Legacy)
  - ASM
- Surfacing Defaults

Process Parameters:

- Apply Comp in PartMaker
- Coolant: Standard
- Feed: 10
- Speed: 1000
- Tool Ch X: 0
- Tool Ch Y: 0
- Tool Ch Z: 5

Machining Data:

- Maximum Speed: 5000
- Maximum Feed: 200
- Tool Change Time (min): 0.25
- Rapid Feed: 1000

Leads

- Arc Radius: 75 %dia
- Line Length: 75 %dia
- Lead Angle: 45 deg

Process Table Display Option

- Feeds in Units per Revolution and Surface Speed

Arc Tolerance: 0.005

- Corner Rounding

All Feeds here are in Units Per Minute (upm)  
All Speeds here are in Revolutions Per Minutes (rpm)

OK Cancel

- 2 Enter **1 (25)** next to Z\_Depth.
- 3 Select Standard from the Coolant drop-down list in the dialog, and then click **<OK>** to return to the **Face Window**.

**Width of Cut:** Has two options; **"% Tool Diameter"** and **"Part Units"**. This facilitates direct input of Width of Cut in addition to **% Diameter** in Profile Group Parameters dialog for Milling.



**Tip:** The default button in all **PartMaker** dialogs is the heavily-outlined button (**<OK>** or **<Apply>**, for example). You can use the default button by clicking it or by pressing the **<Enter>** key.

## Create Geometry

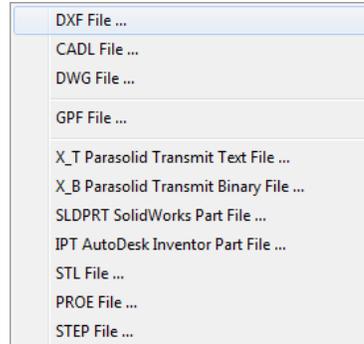
There are two methods of creating Geometry in **PartMaker**. You can either import Geometry from a CAD file (method 1) or define Geometry using **PartMaker**'s own CAD mode (method 2). If you want to use method 2 skip to the section called "**Set up Part Boundaries.**"

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

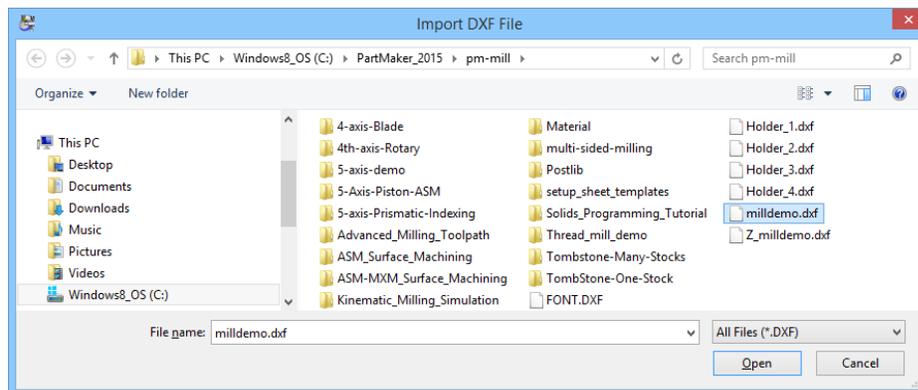
To import Geometry for Milldemo, do the following:



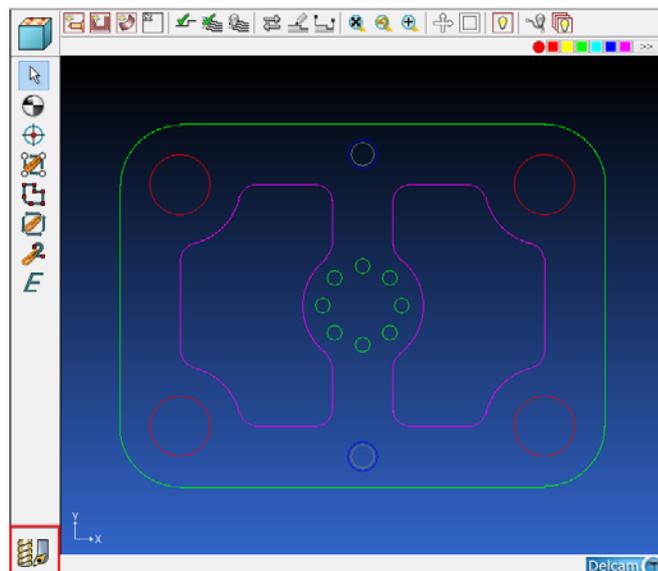
- 1 Choose **Import > DXF File** from the **File** menu to display Import DXF File dialog.



- 2 Locate and select **Milldemo.dxf** (milldemo\_metric.dxf), then click **<Open>**.



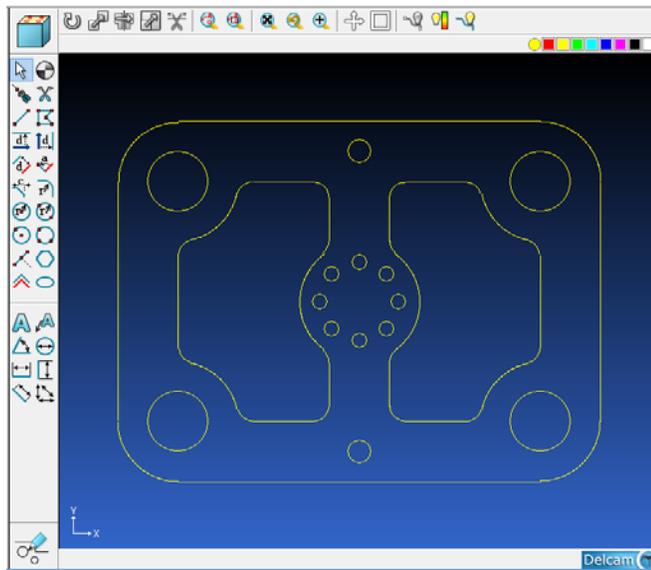
Your screen should look like the illustration below.



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

- 1 Click on the CAD/CAM Switch icon on the lower left hand side to switch to CAD mode (see picture above)

The screen should appear as shown below.



- 2 From the **View** menu, you have the following **Zoom** options available:



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



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



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

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

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

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



Select **Zoom to Fit** to create the smallest possible window that will contain all of the geometry, dimensions and notes in the current window



Select **Zoom All** to create the smallest possible window that will contain all of the geometry, dimensions, notes and the boundaries in the current window.



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

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



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

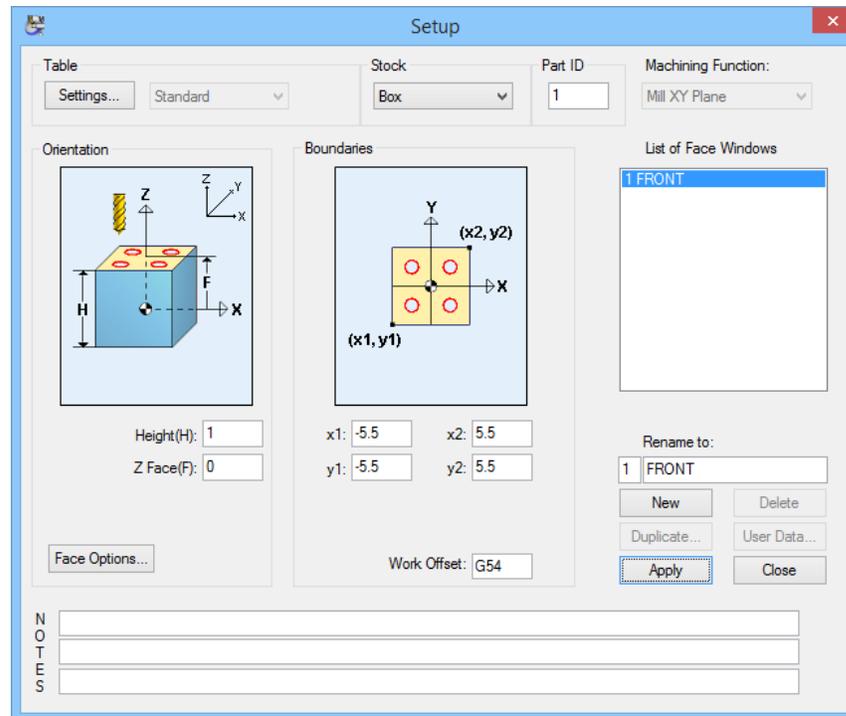
Depending on your needs, any or all of these display options may be activated at any time during the drawing (CAD) or programming (CAM) process.

## Method 2: Create Geometry in PartMaker CAD

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

The method that we will demonstrate in the following pages will demonstrate how geometry can quickly and easily be created in **PartMaker** CAD. It is recommended you try drawing the part to gain a fuller understanding of the capabilities of **PartMaker** CAD.

- 1 First, create a new job by selecting **New Job** from the **File** Menu.
- 2 Make sure you have setup your part defaults correctly as shown below. From the **View** menu, select **Setup**.



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

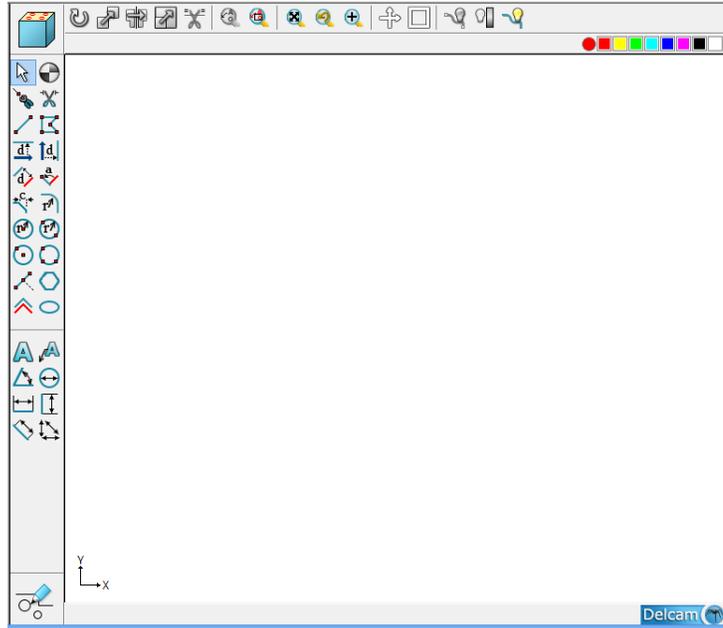


The button looks like this in **CAM mode**



The button looks like this in **CAD mode**

The window displayed should be empty and appear as below:



**Note:** To view a white background, go to the **View>Preferences** menu option and select **White** under **Face Window background** drop down menu

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

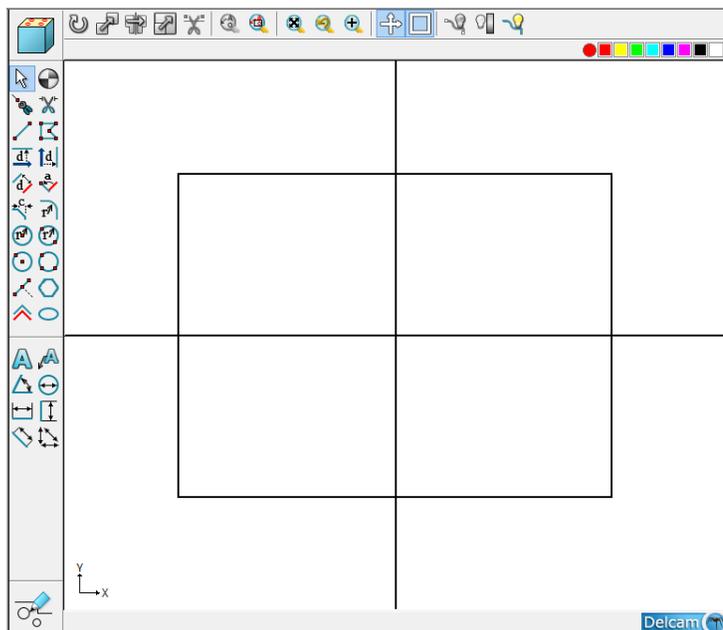


Show Boundaries



Show Axis.

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



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

## Create Five Circles with Radius = 1" (25 mm)

To do so:



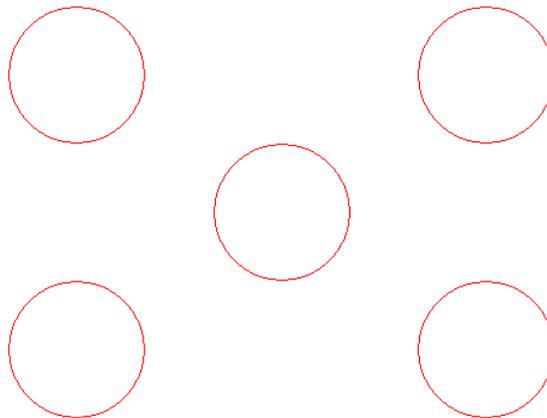
- 1 Select the Circle with a Known Radius and Center icon.
- 2 When prompted to enter 1 (25) as the "r" value on the bottom of the screen, then press the **<Enter>** key.
- 3 When snap mode icon bar appears, click the **XY Coordinates Snap Mode** icon.
- 4 Enter the following coordinates for the circles centers, accepting the same radius value in between each entry that you entered in the step above by pressing **<Enter>**:

| Z value, X value: Inch | Z value, X value: Metric |
|------------------------|--------------------------|
| 0, 0                   | 0, 0                     |
| 3, 2                   | 75, 50                   |
| 3, -2                  | 75, -50                  |
| -3, 2                  | -75, 50                  |
| -3, -2                 | -75, -50                 |



**Note:** In PartMaker, you do not need to re-enter a numerical value that has not changed from the previous entry.

The screen will look as shown below:



## Create Four Tangent Lines

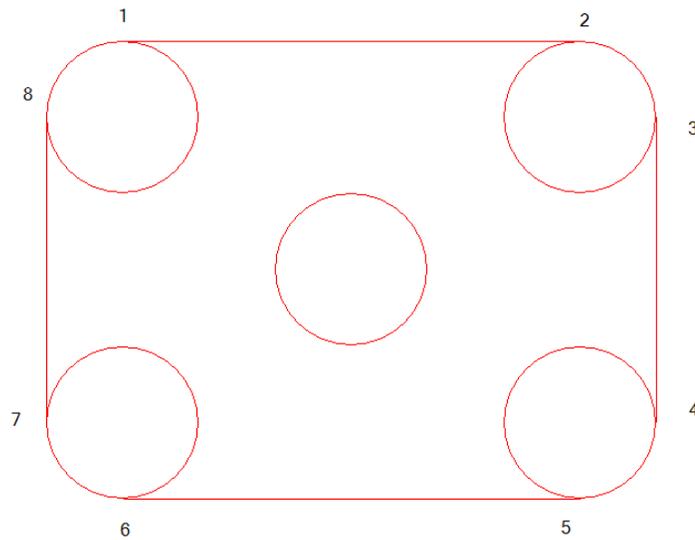


- 1 Select the Line Thru Two Points icon.
- 2 Select the Tangency Snap Mode icon.
- 3 Click the cursor at locations 1 and 2 to create first line.
- 4 Click the cursor at locations 3 and 4 to create a second line.
- 5 Click cursor at locations 5 and 6 to create third line.
- 6 Click cursor at locations 7 and 8 to create fourth line.



**Note:** The numbers on the illustration below indicate the location of the cursor clicks:

The screen will look as shown below:



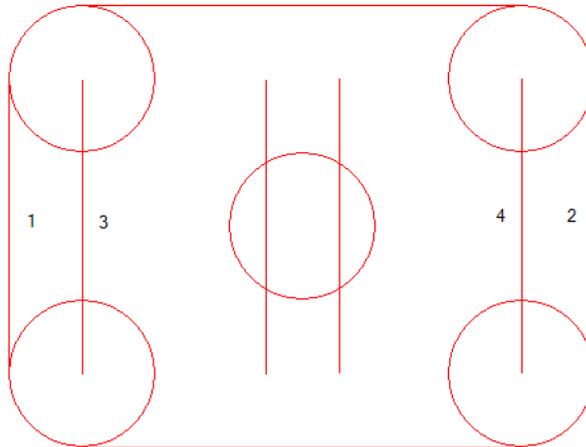
**Tip:** If you make a mistake, use the **Undo** command from the **Edit** menu or select the geometry you wish to remove and press the backspace button.

### Create Four Parallel Lines



- 1 Select the Parallel Element Icon.
- 2 When prompted enter 1 (25) as the "d" value on the bottom of the screen.
- 3 Click on location 1, then on location 2 as indicated below.
- 4 When prompted enter 2.5 (62.5) as the new "d" value at the bottom of the screen
- 5 Click on location 3, then on location 4.

The screen will look as shown below:

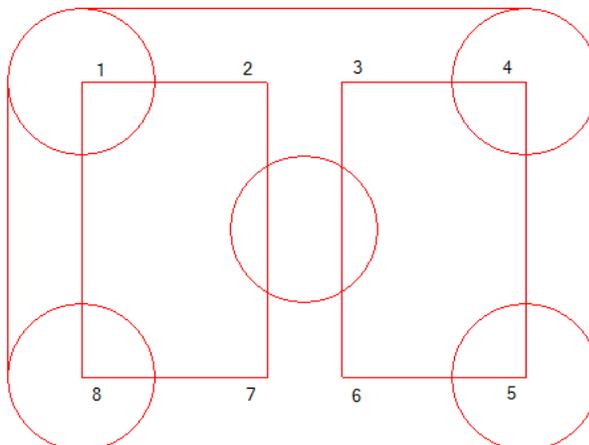


### Create Four Lines through Two Points



- 1 Select the Line Thru Two Points Icon.
- 2 Select the End of Element Snap Mode icon
- 3 Click on locations 1 and 2 to create the first line.
- 4 Click on locations 3 and 4 to create the next line.
- 5 Click on locations 5 and 6 to create the next line.
- 6 Click on locations 7 and 8 to create last line.

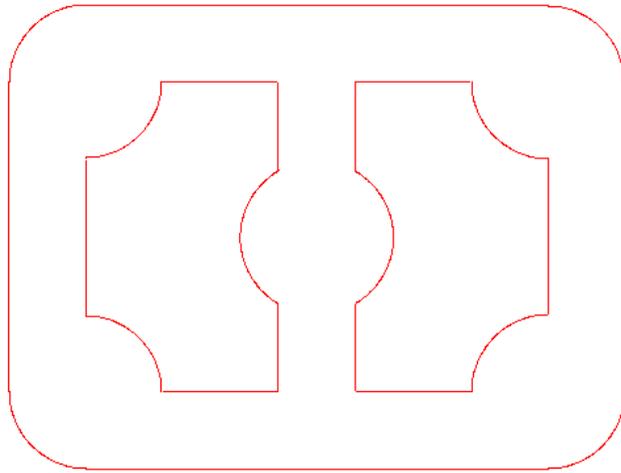
The screen will look as shown below:



## Remove Redundant Geometry



- 1 Select the **Remove** icon.
- 2 Click geometry to remove it so your image appears as below:

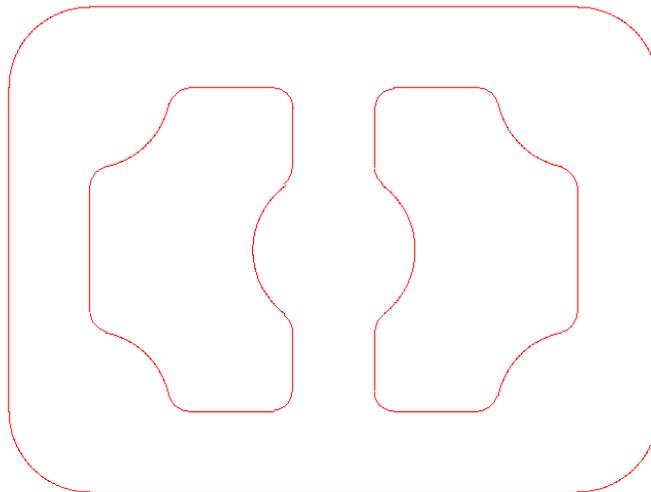


## Insert Fillets



- 1 Select the **Fillet** icon.
- 2 When prompted to enter 0.25 (6.25) as the "r" value on the bottom of the screen.
- 3 Click the cursor *inside* of every sharp corner to insert a fillet.

The screen will look as shown below:



## Create Seven Circles Representing Holes



1 Select the Circle with a Known Radius and Center Icon.

2 When prompted to enter 0.5 (12.5) as the "r" value on the bottom of the screen.



3 Select the Circle Center Snap Mode icon.

4 Click the cursor at the center of the four outside arcs until the four circles are created.

5 Next enter 0.25 (6.25) as the new "r" value.



6 Select the XY Coordinates Snap Mode icon.

7 Enter the following coordinates for the circle centers for X and Y respectively:

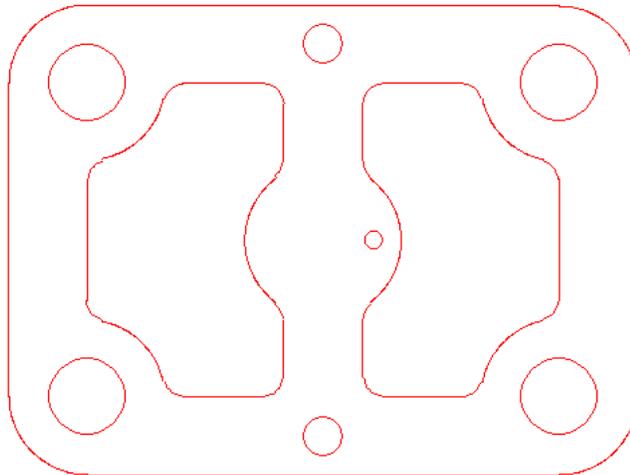
**Top circle:** 0, 2.5 ( 62.5)

**Bottom circle:** 0, -2.5 (-62.5).

8 Next enter 0.125 (3.125) as the new "r" value.

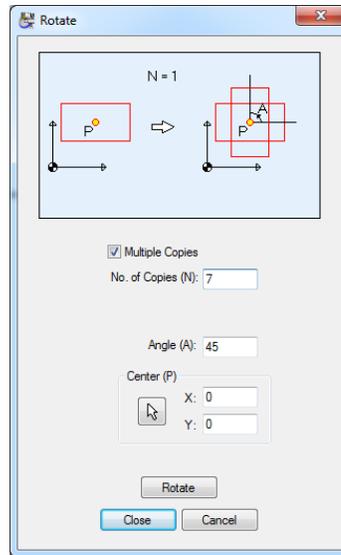
9 Enter 0.65, 0 (16.25, 0) for the center coordinates for the last circle and press **<Enter>**.

The screen will look as shown below:

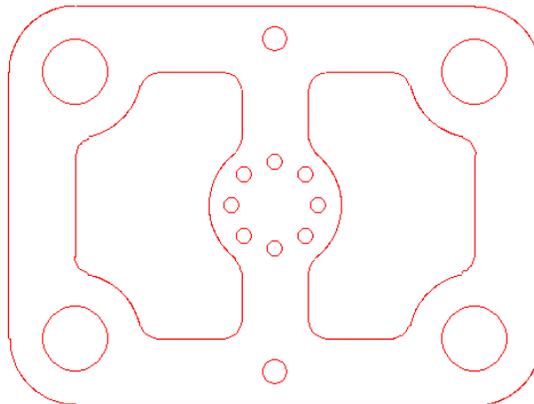


## Create Bolt Circle

-  1 Verify that the Selection icon is select and then click to select the small circle in the middle of the CAD drawing.
-  2 From the **Edit > Transform** menu, choose the **Rotate**. A dialog will display as shown below:



- 3 Click the Multiple Copies checkbox
  - 4 Enter 7 for **No. of Copies**. Click **<Rotate>** and then **<Close>**.
  - 5 Click anywhere in the **Face Window** to deselect circles.
- You should see a picture as shown below.



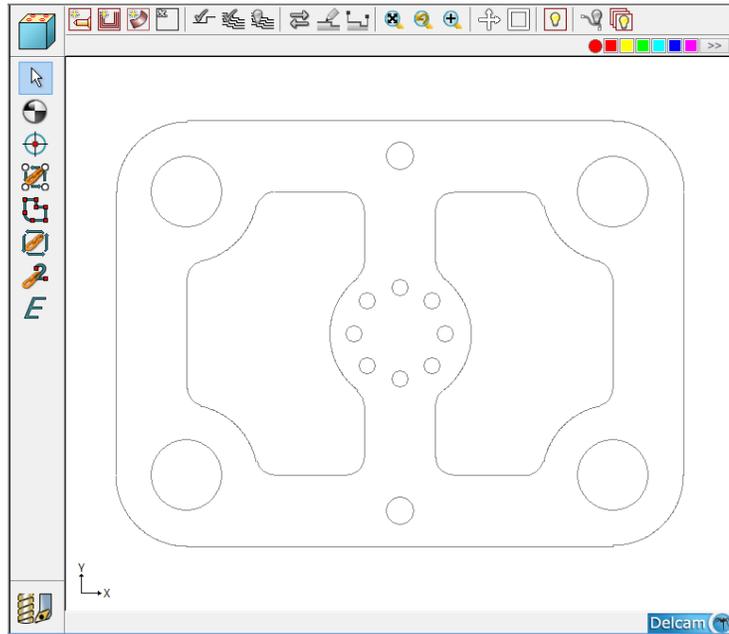
## Creating Milldemo in PartMaker CAM Mode

Now that you have created the geometry for Milldemo, you are ready to begin working in **PartMaker** CAM mode. You are in CAM mode when the CAD/CAM switch in the lower left-hand corner of the screen looks as follows:



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

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

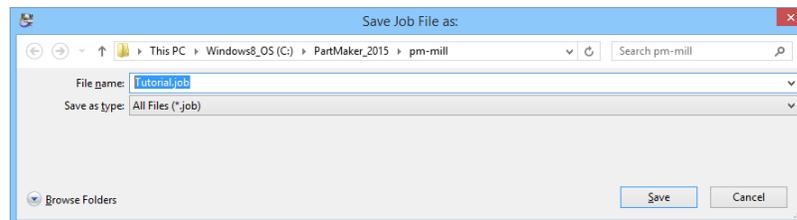


## Save Your Work

It is always advisable to save your work periodically as you program jobs in PartMaker. To do so:



- 1 Choose **Save Job File As...** from the **File** menu to display the Save Job File dialog.
- 2 Enter the name **TUTORIAL** in the space provided.

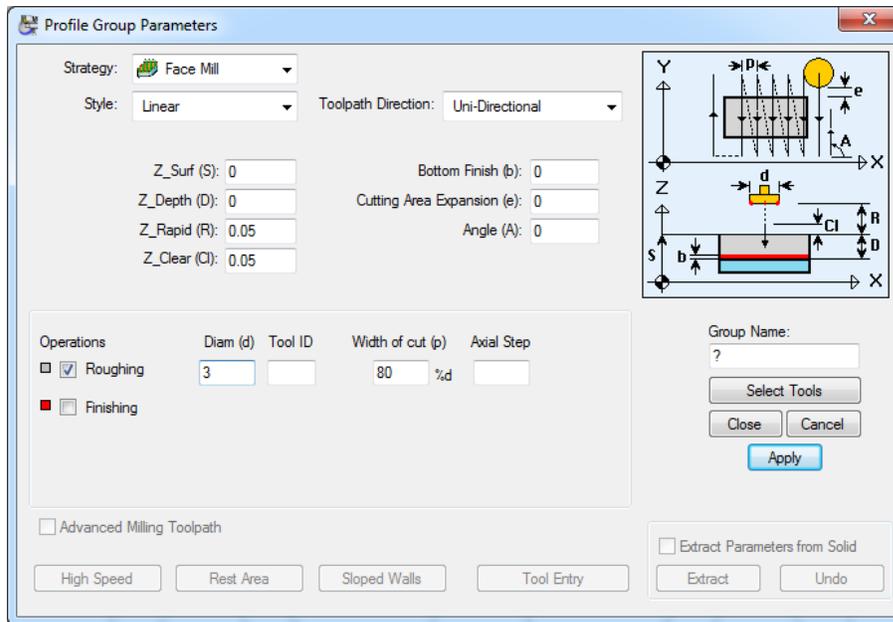


- 3 Locate and select the **PartMaker** directory, then click **<SAVE>**.

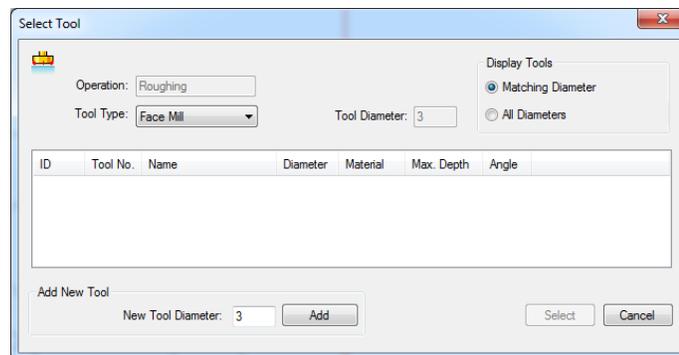
## Define Group 1 Profiles for Face Mill Cycle



- 1 On the color bar click Red. 
- 2 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
- 3 Choose **"Face Mill"** from the **Strategy** drop-down menu.
- 4 Enter **0** next in the **Z\_Depth (D)** field.
- 5 Enter 3.0 (76.0) under **Diam (d)**.
- 6 Choose Uni-Directional from the **Tool Path Direction** drop-down menu.

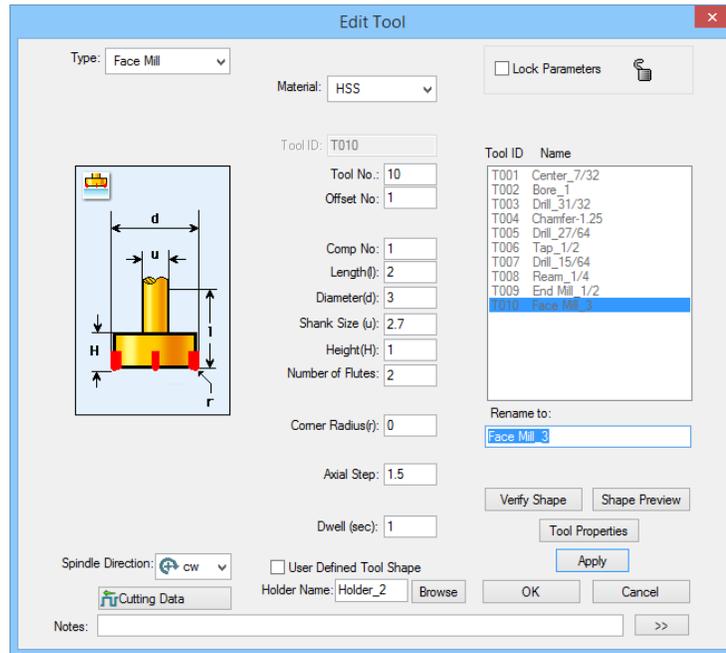


- 7 Click the **<Select Tools>** button. The Select Tool dialog will display as shown below:



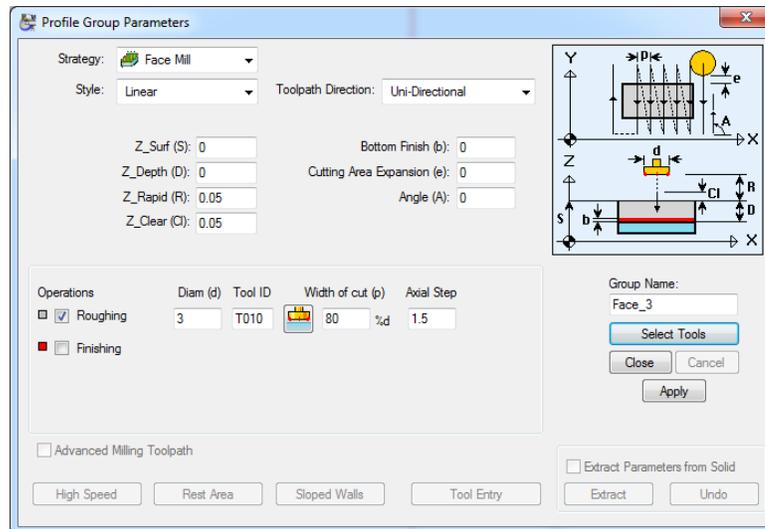
You will now create a new Face Mill Tool. To do so:

- 1 Click on the Add button to create the 3 inch (76 MM) face Mill.
- 2 Change the tool description to match the dialog below.



- 3 Select **<Apply>** and **<OK>**.

When you are finished the dialog box should appear as the one shown below.

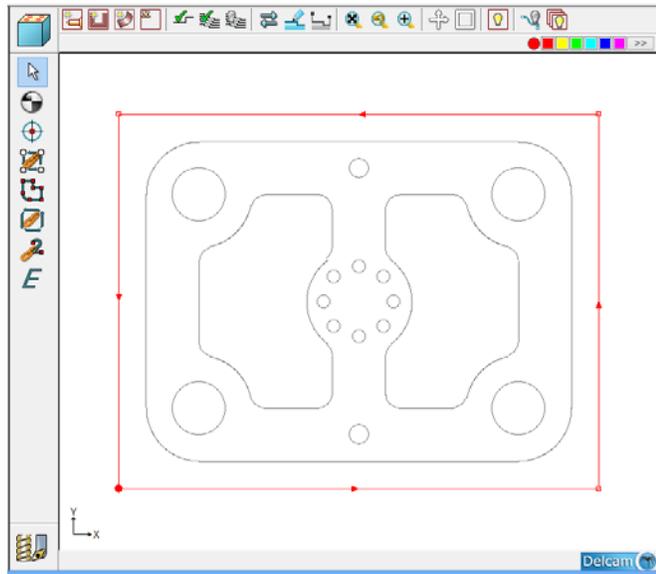


- 4 Enter Face\_3 (Face\_76) in the **Group Name** field. If you do not name a milling cycle, PartMaker will give it a default name when you close the dialog box.
- 5 A group icon is displayed in the upper left corner of the face window in the Job Explorer Tree as shown:



- 6 Click the **<Close>** button.

A tool path is automatically created for you. Your screen will appear as below:



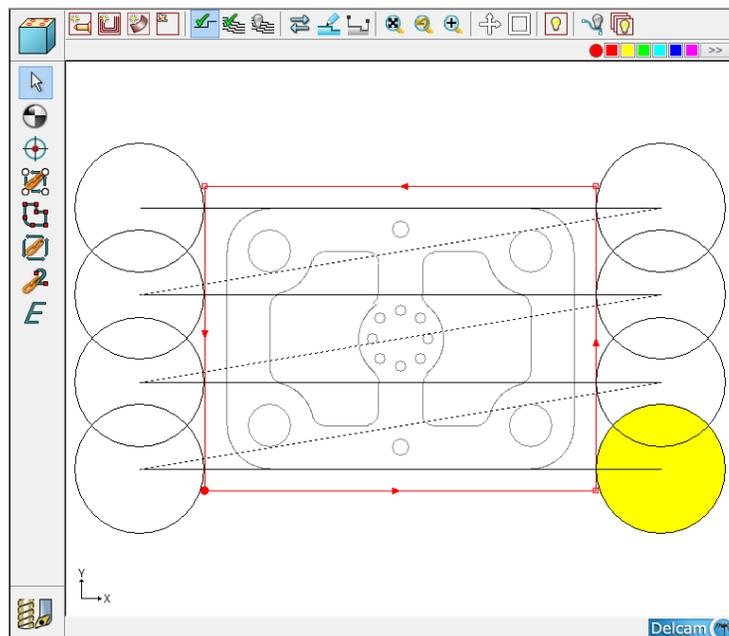
### Verify Face Tool Path



- 1 Select the **Verify Work Group Tool Path** command from the Part Features menu.
- 2 Set options to match the dialog shown below and click <OK>



You will see a picture as below:

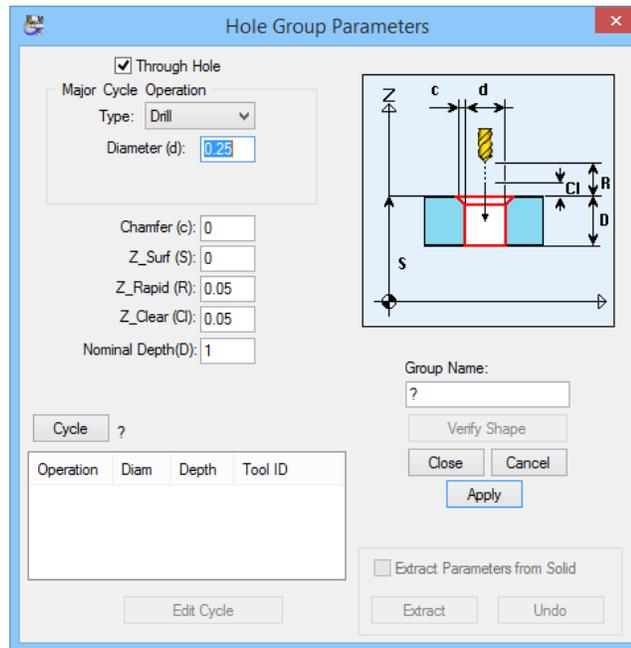


- 3 Select the **Hide Every Tool Path** command from the Part Features menu.

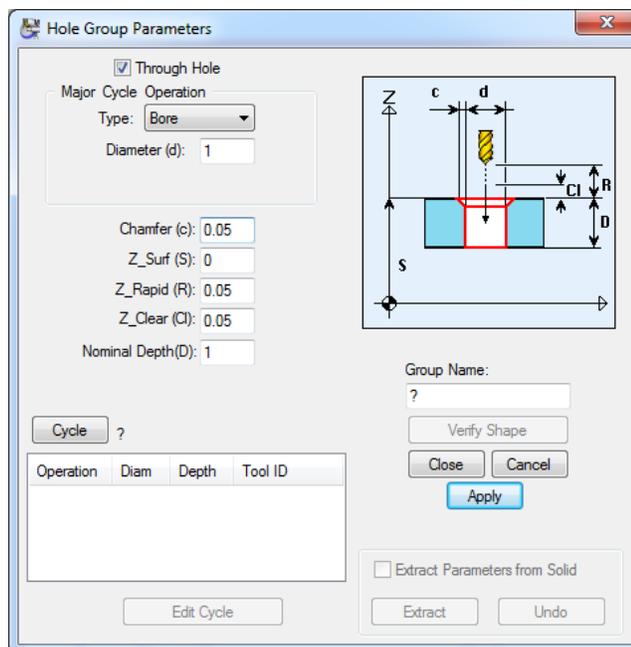
## Define Group 2 Holes for the Boring Cycle

The first operation in **Milldemo** is to Bore four 1.0" (25 mm) diameter holes.

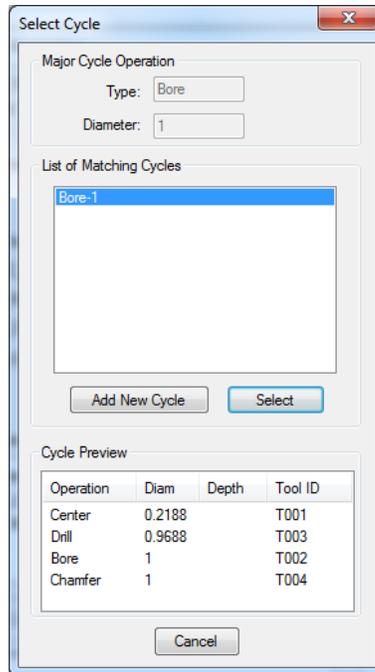
- 1 Click anywhere in the **Face Window** to make it active.
- 2 Click the blue square in the **Color Bar** 
- 3 Choose **New Hole Group** from the **Part Features** menu to display the Profile Group Parameters dialog.



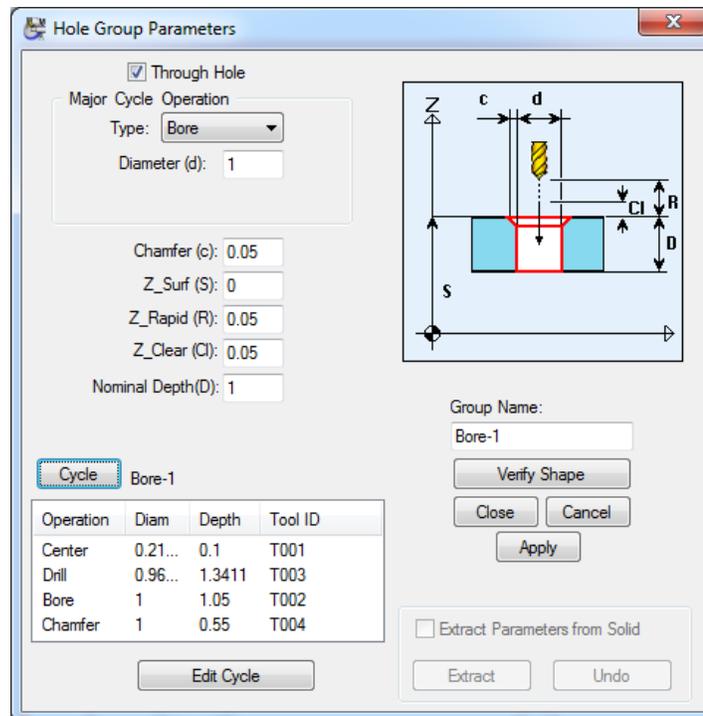
- 4 From the Major Cycle Operation menu, select Bore
- 5 Enter **1 (25)** in the **Diameter (d)** field for the major hole diameter.
- 6 Enter .05 (2) in the **Chamfer (c)** field



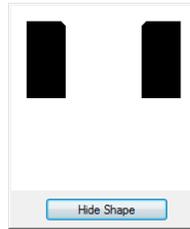
- Click the **<Cycle>** button and select the Bore-1 (Bore-25) cycle.



*PartMaker finds the appropriate cycle in the database Bore-1 (Bore-25). Tool information for the cycle is displayed at the bottom of the dialog when the appropriate cycle is found. If there is more than one cycle that matches the Major operation and Diameter you can check each one to see the tool information for that cycle. The dialog box should appear as shown below.*



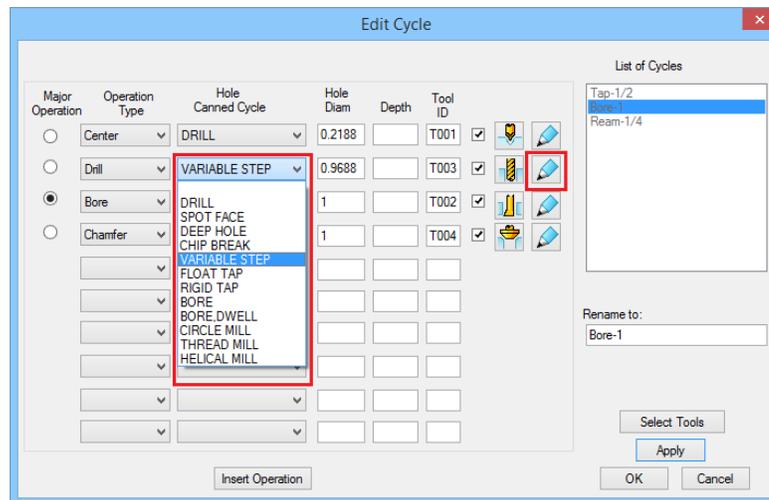
To verify the hole shape, click the **Verify Shape** button; click the **Hide Shape** button under the picture to hide the hole shape and return to the dialog.



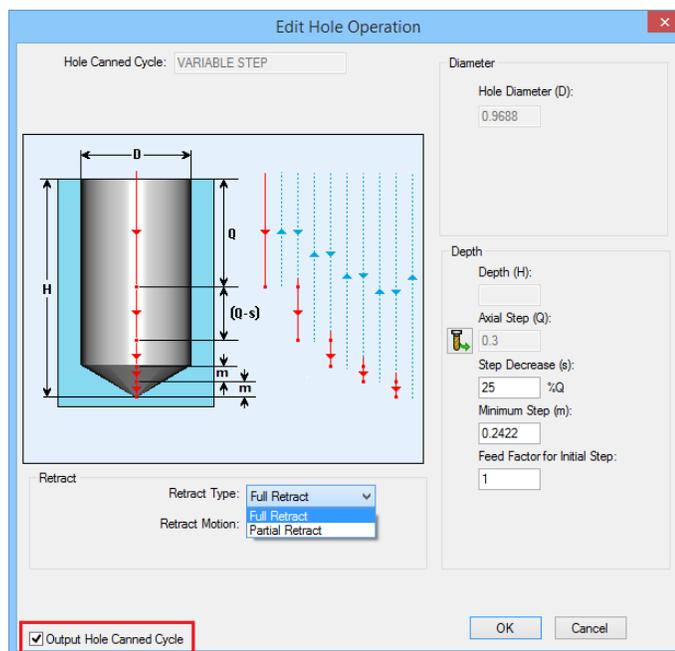
- Click on the Edit Cycle button in Hole Group Parameters dialog to access the selected cycle. Only the selected cycle will be available for modification, any changes that are made will automatically update the Hole Group Parameters.



- Select the drop down menu for the 31/32 drill and you will see options for the drilling cycle.



By selecting the **Edit Hole Operations** button to the right of the **Tool ID** field you can review and modify the parameters for this type of operation.

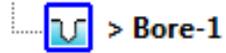




**Output Hole Canned Cycle** — This option is selected by default so the machining operation for the hole is output as a canned cycle in the NC Program file. If your postprocessor does not support all of the parameters for this canned cycle, deselect this option.

- 10 When you are satisfied that you have correctly defined the Group Parameters, **Close** the dialog.

A Blue square displaying the group name BORE-1(BORE-25) appears in the upper right corner of the **Face Window** as shown below. The square is a *group icon*. Each part you create in **PartMaker** will display one or more group icons in the **Face Window** to indicate the various machining cycles for a part.



## Hole Creation Method

The icons (called **Snap Modes**) at the top of the **Face Window** represent various ways of defining locations such as hole center. In the next several sections of the tutorial, you'll use some of these methods to define holes for TUTORIAL.

Using the **Single Hole** icon and a **Snap Mode**, you can define single holes by X,Y coordinates, polar coordinates, a point on an arc or circle, a screen location, a grid location, a circle center location, the closest intersection an endpoint of a line, or a midpoint of a line.

### Single Hole Icon Snap Modes



XY RA Define Location of Hole Center



**Note:** The Snap Modes are used with the Single Hole icon only.

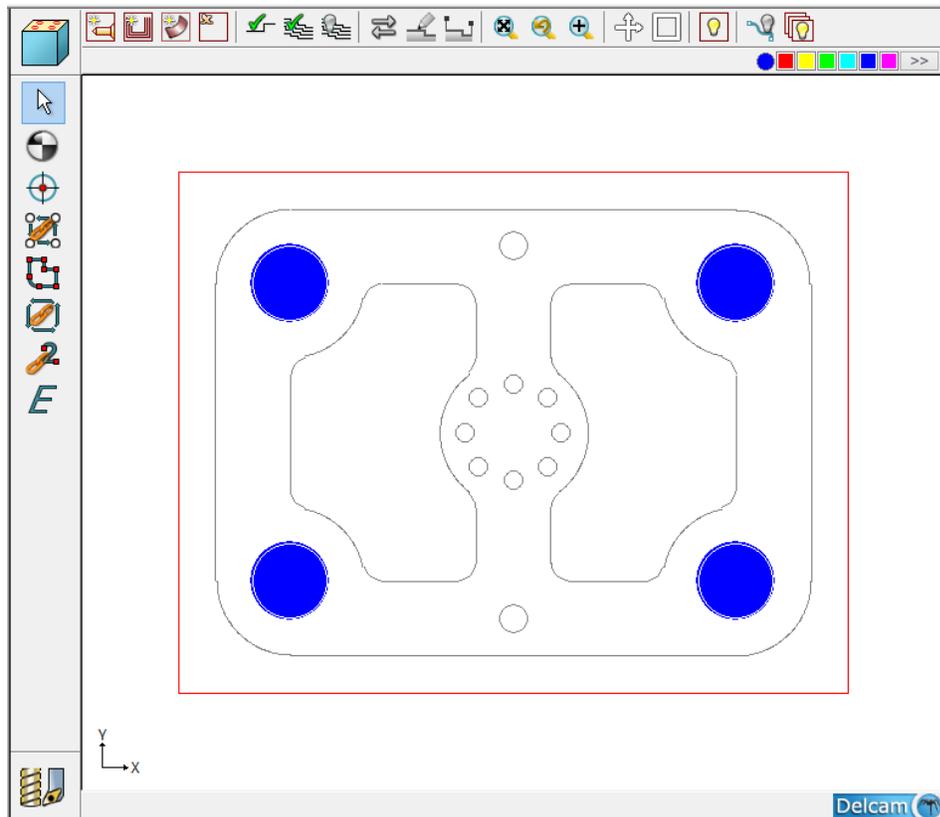


You can define multiple holes quickly using the **Chain Circle** icon.

### Specifying Holes Using the Chain Circle Icon



- 1 Select the **Chain Circles Geometry** icon on the left side of the CAM **Face Window**.
- 2 Click on any of the four 1 inch (25 mm) diameter bored holes. Your Face Window should appear as shown below:



Since all of the holes were defined at the same time, they all remain selected in the **Face Window**. You can click anywhere in the **Face Window** to deselect holes.

### Save Your Work

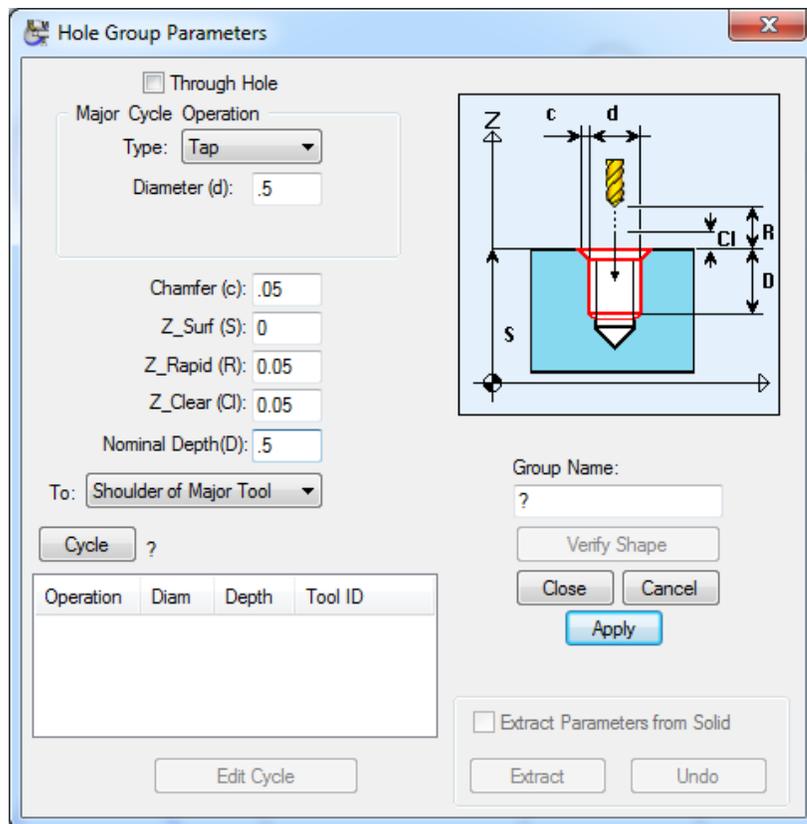


Choose **Save** from the **File** menu or press **<CTRL + S>**.

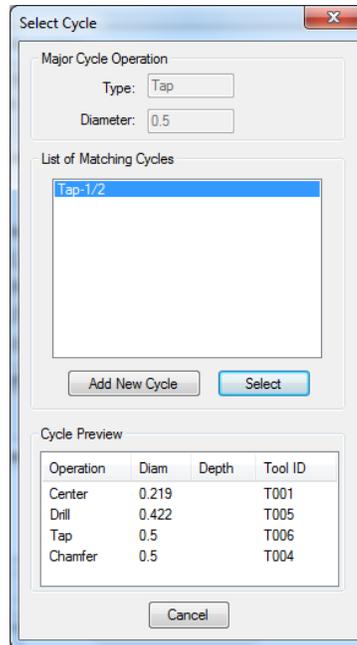
## Define Group 3 Holes for the Tapping Cycle

In this section, you'll specify the holes for Group 2, the tapping cycle. Notice that **PartMaker** accepts fractions or decimal values in any dialog.

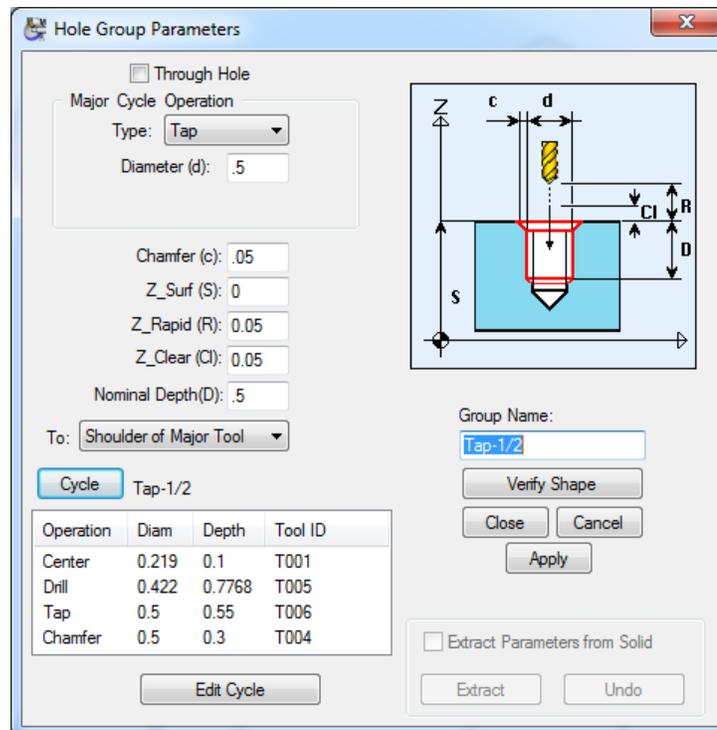
- 1 Click the Red color square in the color bar at the top of the **Face Window** to specify the color for the new group. 
- 2 Choose **New Hole Group** from the **Part Features** menu to display the Hole Group Parameters dialog.
- 3 Choose **Tap** from the Type pop-up menu.
- 4 Turn off the **Through Hole** check box.
- 5 Enter **1/2 (12.5 mm)** next to the Diameter option.
- 6 Enter **0.05 (2mm)** for the Chamfer value.
- 7 Enter **0.5 (12.5 mm)** for the Nominal Depth value.
- 8 Select **Shoulder of Major Tool** for the depth.



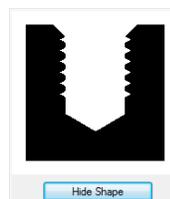
- Click the **<Cycle>** button and select the Tap ½ (Tap-12.5) cycle.



When you are done your screen should appear as below.



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



A group icon for Group 3 displays in the upper-right corner of the **Face Window** under the Bore-1 group icon. The group name Tap-1/2 (Tap-12.5) appears to the right of the group symbol.

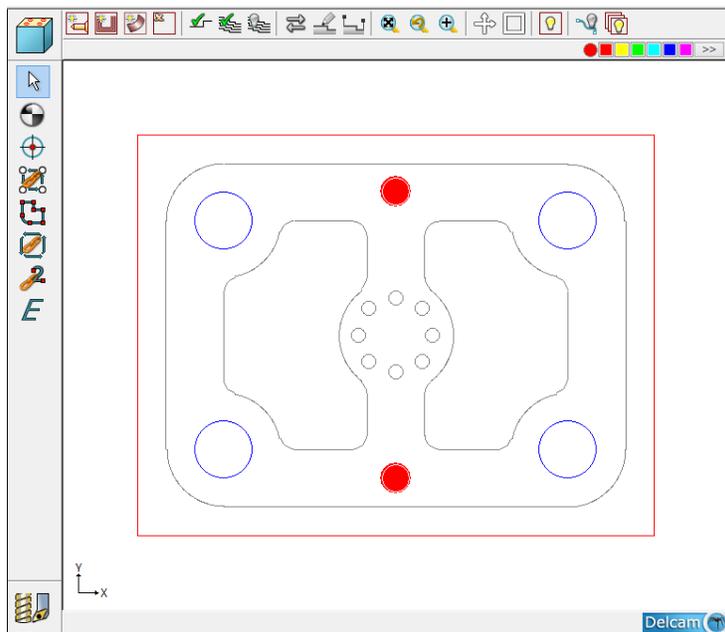


- 11 Click the **<Close>** button to exit the dialog and return to the **Face Window**.

### Specify Holes Using a Chain Circles Icon



- 1 Select the **Chain Circles** icon (fourth from the bottom) on the left side of the Face Window.
- 2 Click the cursor at the location shown below. Holes are automatically created in the centers of the 2 circles.



### Save Your Work

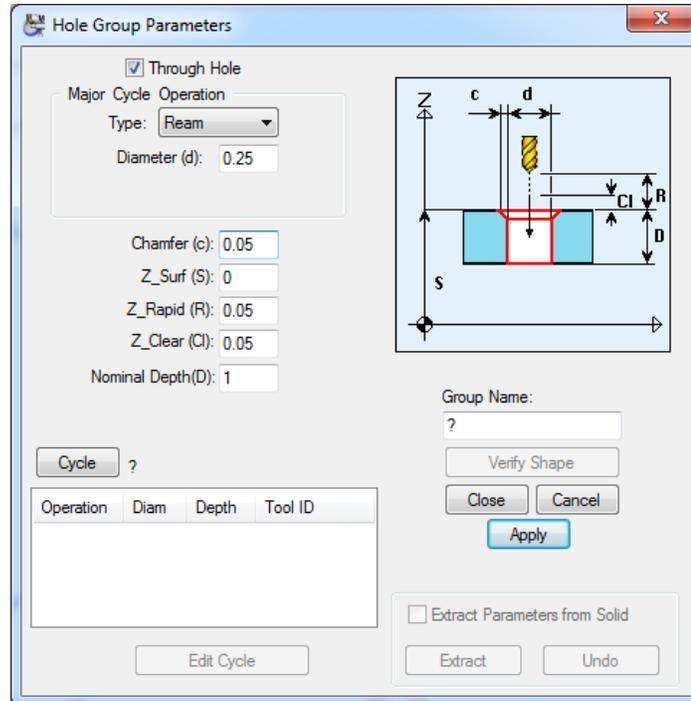


- Choose **<Save>** from the **File** menu.

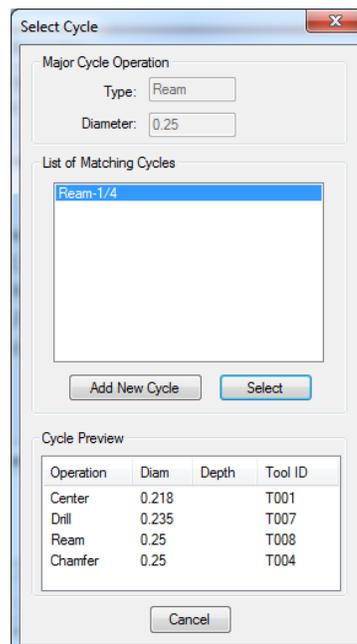
## Define Group 4 Holes for the Reaming Cycle

In this section, you'll specify the holes for Group 3, the reaming cycle.

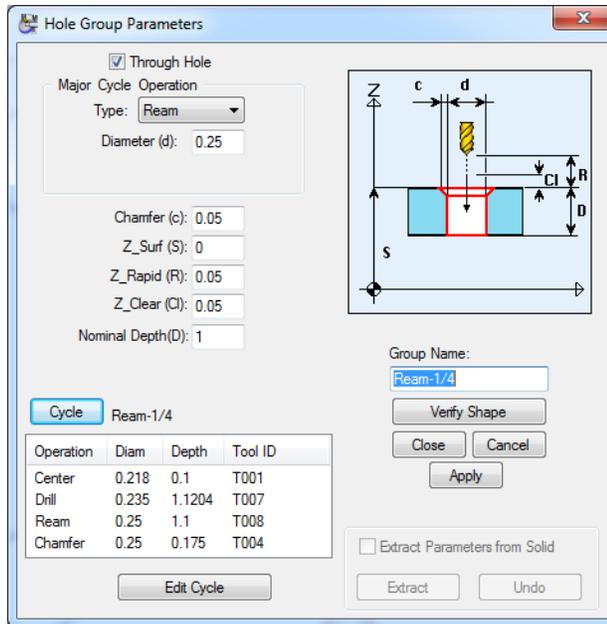
- 1 Click the Yellow color square in the color bar at the top of the **Face Window** to specify the color for the new group. 
- 2 Choose **New Hole Group** from the **Part Features** menu to display the Hole Group Parameters dialog.
- 3 Choose **Ream** from the Type drop-down menu.
- 4 Enter **.25 (6.25mm)** next to the Diameter option.
- 5 Enter **0.05 (1mm)** for the Chamfer option.



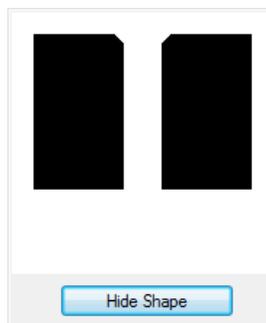
- 6 Click the **<Cycle>** button and select the Ream-1/4 (Ream-6.25) cycle.



- 7 When you are done your screen should appear as below.



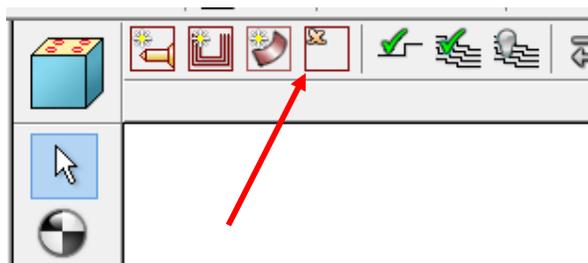
- 8 To verify the shape of the hole, click the **<Verify Shape>** button; click the **<Hide Shape>** button under the picture to return to the dialog.



- 9 Click the **<Close>** button to exit the dialog and return to the **Face Window**.
- 10 The group icon for the Ream-1/4 (Ream-6.25) cycle appears on the right side of the **Face Window**.



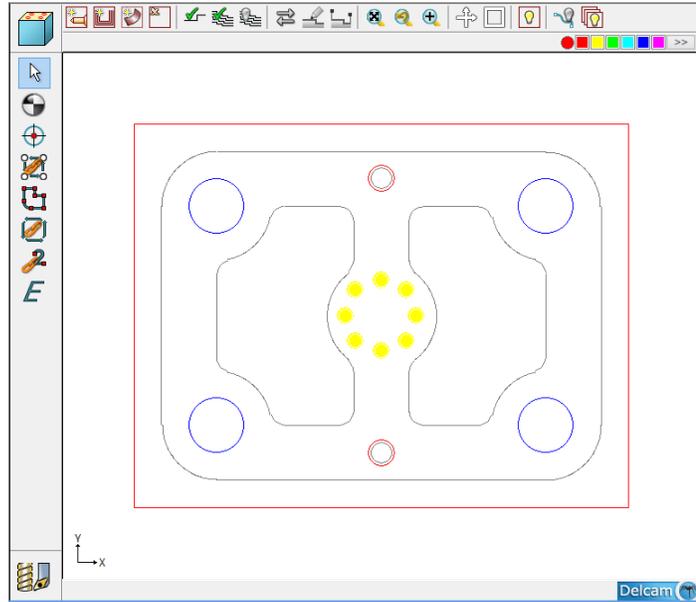
**Note:** If a group icon appears with a question mark (?) instead of a name, it means that you didn't specify the correct operation or diameter in the Hole Group Parameters dialog. If this happens, you can choose the Group Parameters command (**Part Features** menu) to check the diameter or use the Cycle button in the dialog to select a new cycle. If this group is not wanted you can right mouse click on it and select **Delete Group** or select the **Delete Group** on the CAM mode tool bar.



## Specify Holes Using a Chain Circles Icon



- 1 Select the Chain Circles Icon (fourth from the top) on the left side of the Face Window.
- 2 Click the cursor at the location shown below. Holes are automatically created in the centers of the 8 circles.



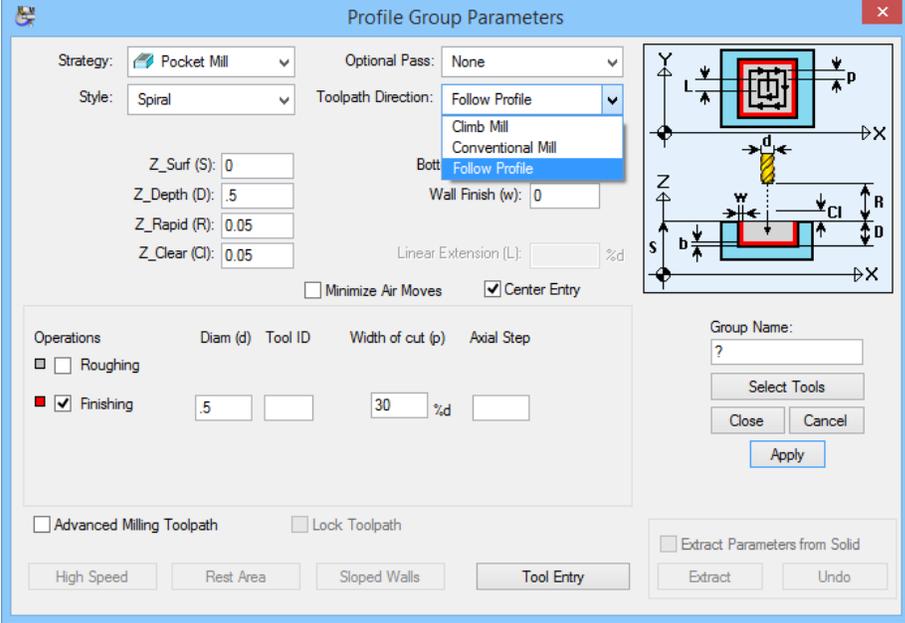
## Save Your Work

Choose **<Save>** from the **File** menu.

## Define Group 5 Profiles for Pocket Mill Cycle



- 1 On the color bar click magenta. 
- 2 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
- 3 Choose **"Pocket Mill"** from the **Strategy** drop-down menu.
- 4 Enter **0.5 (12.5)** next in the **Z\_Depth** field.
- 5 Enter **0.5 (12.5)** under Diam (d).
- 6 Enter **30** in the **Width of Cut** field.
- 7 Select the Toolpath Direction you prefer from the drop down menu.



Profile Group Parameters

Strategy:  Optional Pass:

Style:  Toolpath Direction:

Z\_Surf (S):  Bot:

Z\_Depth (D):  Wall Finish (w):

Z\_Rapid (R):  Linear Extension (L):  %d

Z\_Clear (C):

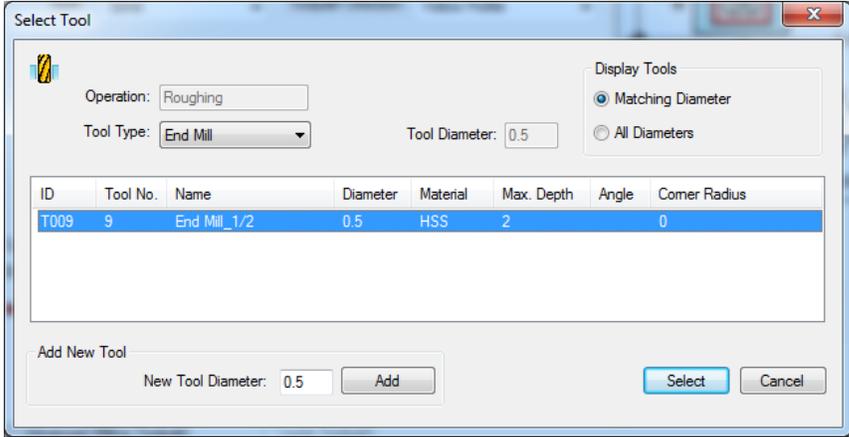
Minimize Air Moves  Center Entry

| Operations                                    | Diam (d)                         | Tool ID                       | Width of cut (p)                   | Axial Step                    |
|---|----------------------------------|-------------------------------|------------------------------------|-------------------------------|
| <input type="checkbox"/> Roughing             |                                  |                               |                                    |                               |
| <input checked="" type="checkbox"/> Finishing | <input type="text" value="0.5"/> | <input type="text" value=""/> | <input type="text" value="30"/> %d | <input type="text" value=""/> |

Advanced Milling Toolpath  Lock Toolpath

Group Name:

- 8 Click the **<Select Tools>** button. Select tool T009 in the database.



Select Tool

Operation:  Display Tools:  Matching Diameter  All Diameters

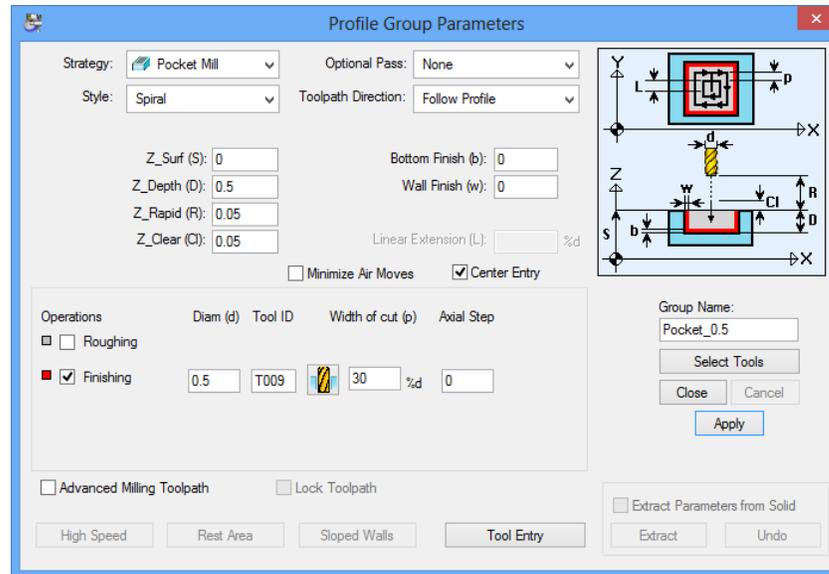
Tool Type:  Tool Diameter:

| ID   | Tool No. | Name         | Diameter | Material | Max. Depth | Angle | Corner Radius |
|------|----------|--------------|----------|----------|------------|-------|---------------|
| T009 | 9        | End Mill_1/2 | 0.5      | HSS      | 2          | 0     |               |

Add New Tool

New Tool Diameter:

- 9 When you are finished the dialog box should appear as the one shown below.



- 10 You can enter a name in the **Group Name** field. If you do not name a milling cycle PartMaker will give it a default name when you close the dialog box, as shown below.
- 11 A group icon is displayed in the upper right corner of the face window under the Ream-1/4 (Ream-6.25) group symbol icon. The default group name Pocket\_0.5 (Pocket\_12.5) appears to the right of the group symbol.



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

## Profile Creation Methods

There are two methods to creating profiles: **Define Profile** and **Chain Geometry**.



Using the **Define Profile** icon and a Snap Mode, you can define a profile by connecting points defined by X,Y coordinates, polar coordinates, a point on an arc or circle, a screen location, a grid location, a circle center location, the closest intersection and endpoint of a line, or a midpoint of a line.



Using the **Chain Geometry** icon you can define a profile by selecting an endpoint of a line or an arc. **PartMaker** automatically finds all lines and arcs that are connected together.

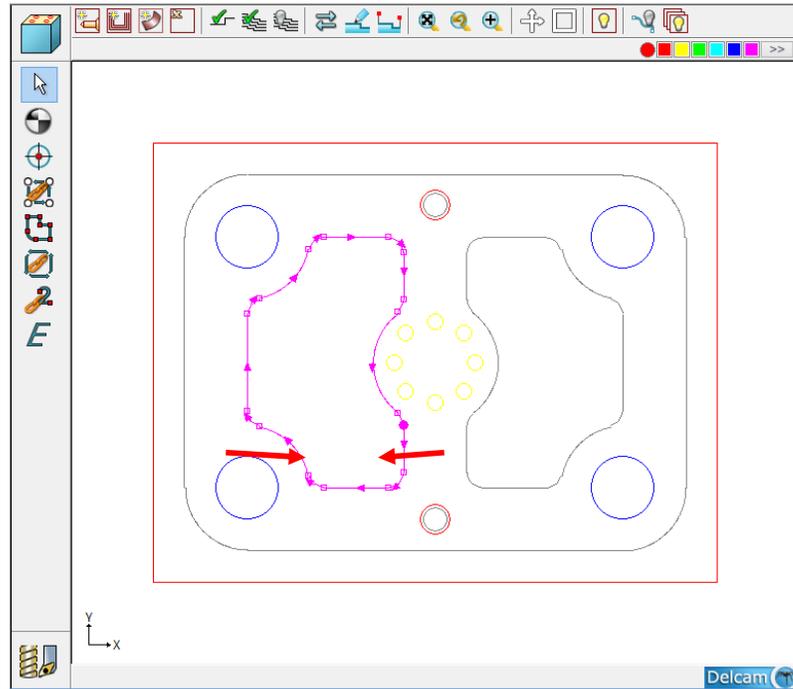


This command allows you to enter coordinates for either approach or escape point for a selected profile.

## Create Profiles Using Chain Geometry Icon



- 1 Select the **Chain Geometry** icon on the left side of the **Face Window**. Click the cursor on the location shown on the screen below. A profile is automatically created.

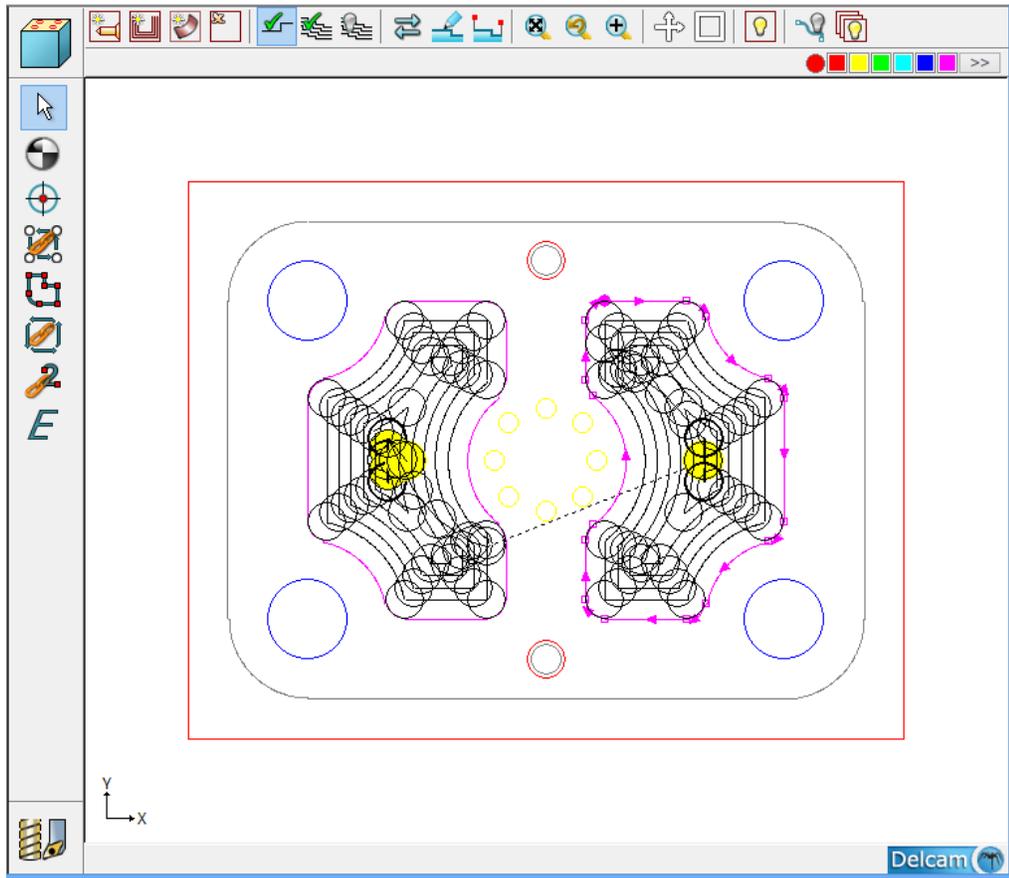


- 2 Select the **Chain Geometry** icon on the left side of the **Face Window** again. Click the cursor on the screen corresponding to the pocket on the right side of the illustration. Another profile is automatically created.

## Verify Pocket Tool Path



- 1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as below.

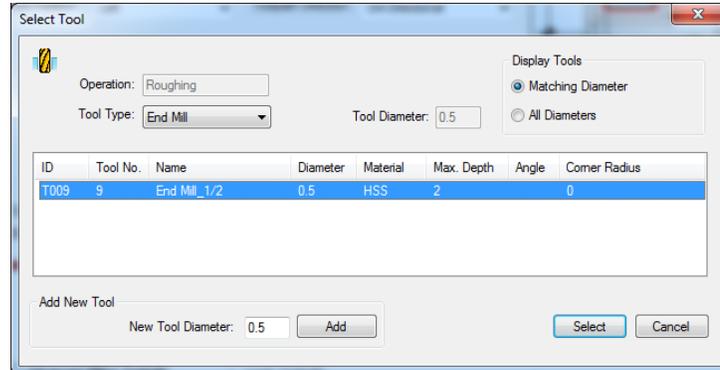


- 2 Select the **Hide Every Tool Path** command from the **Part Features** menu.

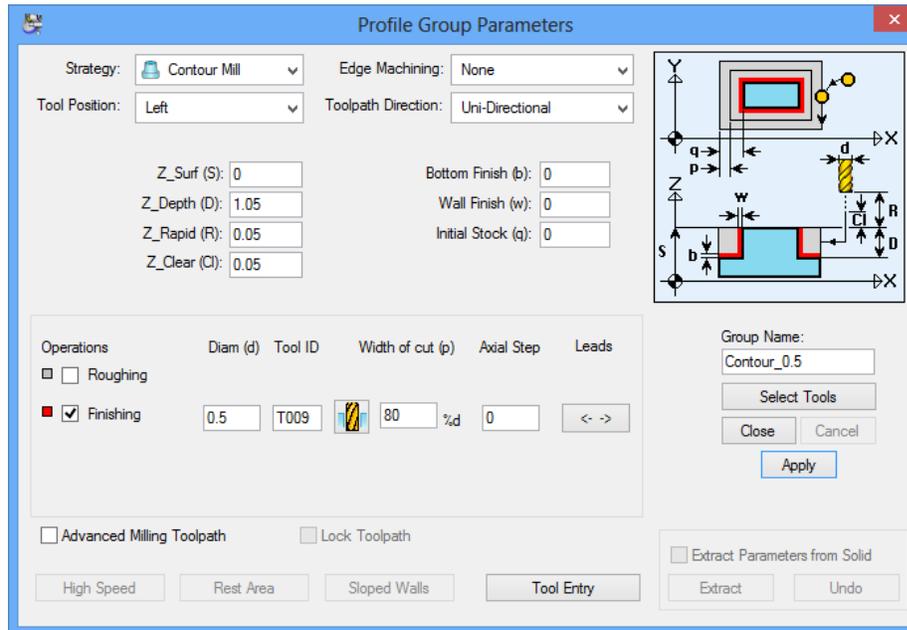
## Define Group 6 Profiles for Contour Mill Cycle



- 1 On the color bar click green. 
- 2 Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
- 3 Choose **Contour Mill** from the **Strategy** drop-up menu.
- 4 Enter **1.05 (25)** in the **Z\_Depth** field.
- 5 Enter **0.5 (12.5)** in the **Diam(d)** field.
- 6 Under Operations group, uncheck **Roughing** and check **Finishing**
- 7 Click the **<Select Tools>** button. Select tool T009.



When you are finished your dialog box should appear like the one below:



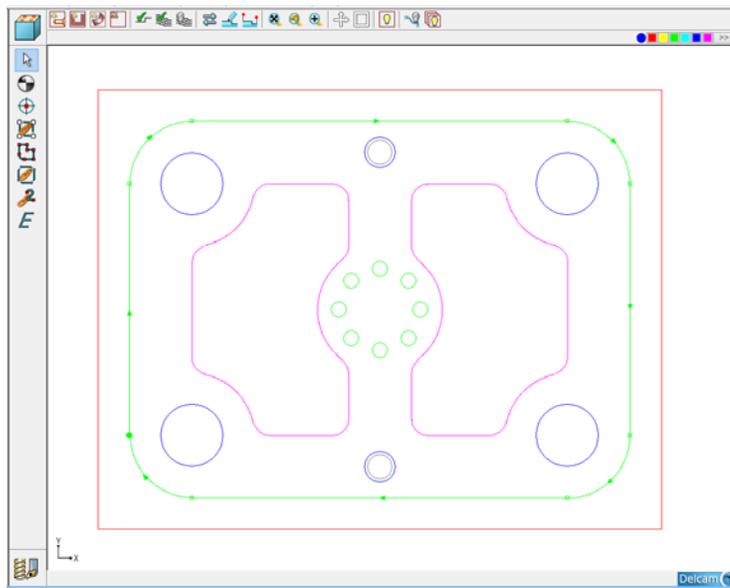
- 8 You can enter a name in the **Group Name** field containing "?". If you do not name a milling cycle **PartMaker** will give it a default name as shown below.
- 9 A group icon for Group 6 is displayed in the upper right corner of the **Face Window** under the Pocket\_0.5 (Pocket 12.5) group icon. The default group name Contour\_0.5 (Contour\_12.5) appears to the right of the group symbol.



- 10 Click the **<Close>** button.

## Create Profiles Using Chain Geometry Icon

- 1 Select the Chain Geometry icon on the left side of the **Face Window**.
- 2 Click the cursor on the outside profile of the following illustration. A profile is automatically created.

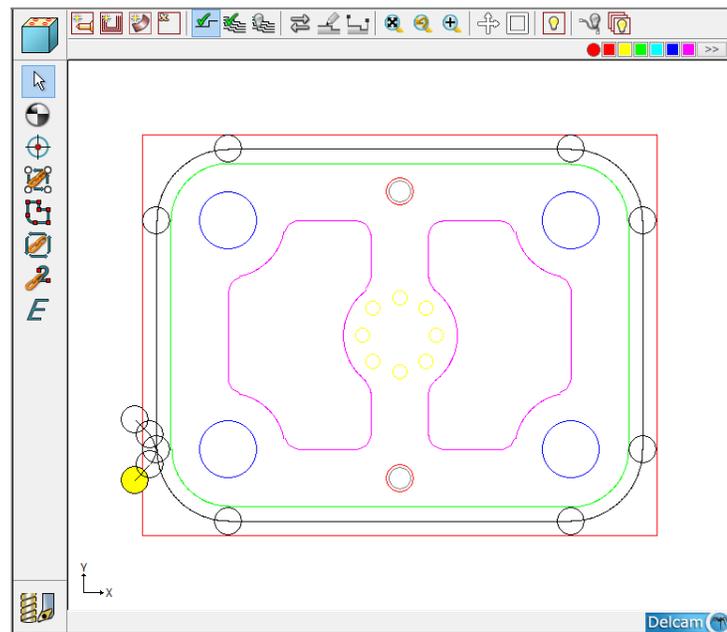


**Note:** The profile shows direction arrows. This means it is selected and can be reversed, translated or rotated.

## Verify Contour Tool Paths



- 1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as shown below.



- 2 Select the Hide Work Group Tool Path command from the Part Features menu.



**Note:** Group symbols can be dragged and dropped to change the order of machining.

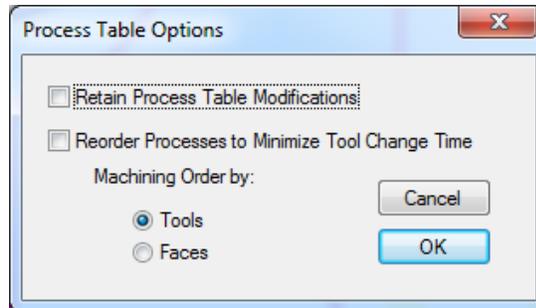
## Generate a Process Table

A **process table** shows you all of the processes for a part. Before generating a process table, you can choose to reorder processes automatically by tool diameter (smallest to largest) and/or reorder them to minimize tool change time and optimize machining.

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



- 1 Choose **Generate Process Table** from the **Job Optimizer** menu to display the Process Table Options dialog. The dialog box should look like the picture below.



- 2 Click **<OK>** to display the process table window.

**PartMaker** generates and displays the Process Table as shown below:

| Proc ID | Tool ID | Tool No. | Tool Name    | Proc Task      | Group       | Face  | Feed    | Speed   | Time  | Status |
|---------|---------|----------|--------------|----------------|-------------|-------|---------|---------|-------|--------|
| P01     | T010    | 12       | Face Mill_3  | Surface Rough  | Face_3      | Front | 38.9upm | 795rpm  | 1.42  | ✓      |
| P02     | T001    | 1        | Center_7/32  | DRILL          | Bore-1      | Front | 14.3upm | 2776rpm | 0.07  | ✓      |
| P03     | T003    | 3        | Drill_31/32  | DRILL          | Bore-1      | Front | 10.4upm | 626rpm  | 0.61  | ✓      |
| P04     | T002    | 2        | Bore_1       | BORE           | Bore-1      | Front | 2.6upm  | 756rpm  | 3.79  | ✓      |
| P05     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Bore-1      | Front | 1.4upm  | 392rpm  | 2.01  | ✓      |
| P06     | T001    | 1        | Center_7/32  | DRILL          | Tap-1/2     | Front | 14.3upm | 2776rpm | 0.03  | ✓      |
| P07     | T005    | 5        | Drill_27/64  | DRILL          | Tap-1/2     | Front | 12.7upm | 1439rpm | 0.15  | ✓      |
| P08     | T006    | 6        | Tap_1/2      | FLOAT TAP      | Tap-1/2     | Front | 19.7upm | 255rpm  | 0.14  | ✓      |
| P09     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Tap-1/2     | Front | 1.4upm  | 392rpm  | 0.60  | ✓      |
| P10     | T001    | 1        | Center_7/32  | SPOT FACE      | Ream-1/4    | Front | 14.3upm | 2776rpm | 0.24  | ✓      |
| P11     | T007    | 7        | Drill_15/64  | DRILL          | Ream-1/4    | Front | 14.3upm | 2591rpm | 0.73  | ✓      |
| P12     | T008    | 8        | Ream_1/4     | BORE,DWELL     | Ream-1/4    | Front | 3.7upm  | 911rpm  | 5.59  | ✓      |
| P13     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Ream-1/4    | Front | 1.4upm  | 392rpm  | 1.59  | ✓      |
| P14     | T009    | 9        | End Mill_1/2 | Pocket Rough   | Pocket_0.5  | Front | 3.7upm  | 946rpm  | 29.87 | ✓      |
| P15     | T009    | 9        | End Mill_1/2 | Contour Finish | Contour_0.5 | Front | 3.7upm  | 946rpm  | 9.21  | ✓      |

Material File: ST\_FMSTW.MDB Total Time: 59.66 min.

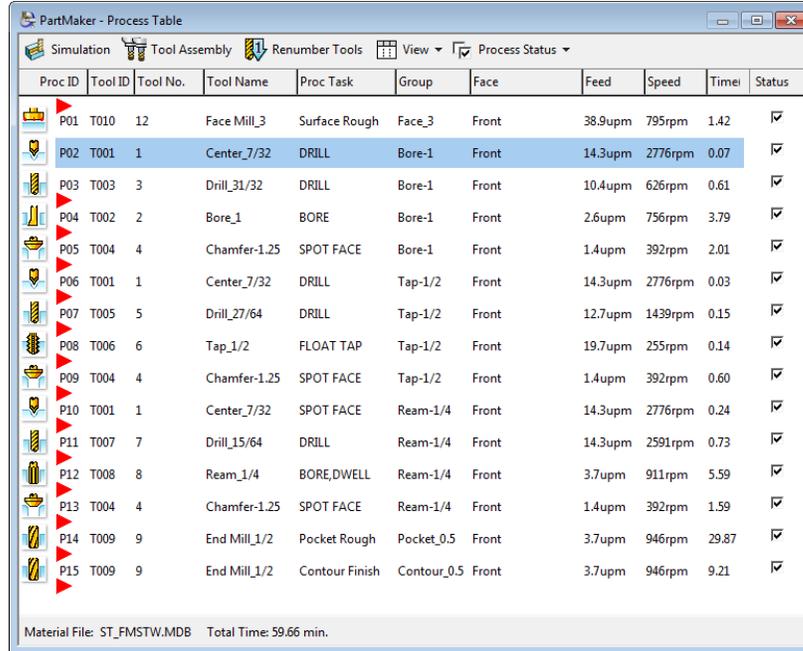


*Tip: If tool numbers are not in the proper order, you can choose **Renumber Tools** from the **Job Optimizer** menu to automatically renumber the tools before generating an NC part program.*

## Modify the Spindle Speed

Process 1 is used on all holes in TUTORIAL.JOB . In this section, you'll change the spindle speed (RPM) used for this process.

- 1 Click anywhere on the second process line in the process table window. The line is highlighted in blue as shown below.



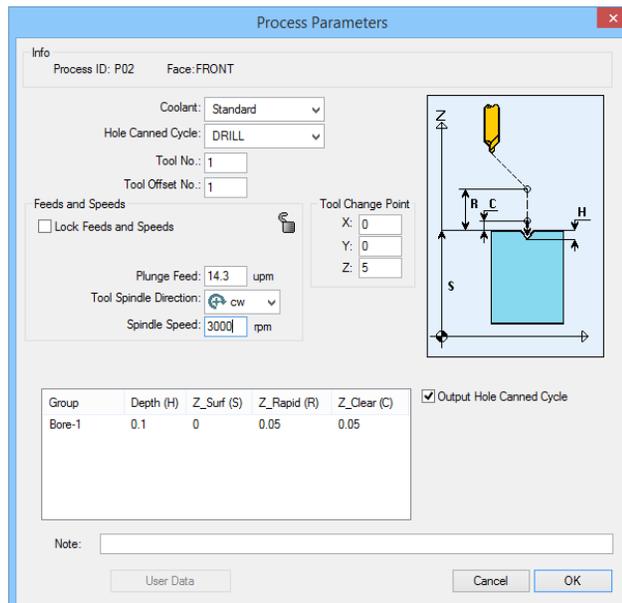
| Proc ID | Tool ID | Tool No. | Tool Name    | Proc Task      | Group       | Face  | Feed    | Speed   | Time  | Status |
|---------|---------|----------|--------------|----------------|-------------|-------|---------|---------|-------|--------|
| P01     | T010    | 12       | Face Mill_3  | Surface Rough  | Face_3      | Front | 38.9upm | 795rpm  | 1.42  | ✓      |
| P02     | T001    | 1        | Center_7/32  | DRILL          | Bore-1      | Front | 14.3upm | 2776rpm | 0.07  | ✓      |
| P03     | T003    | 3        | Drill_31/32  | DRILL          | Bore-1      | Front | 10.4upm | 626rpm  | 0.61  | ✓      |
| P04     | T002    | 2        | Bore_1       | BORE           | Bore-1      | Front | 2.6upm  | 756rpm  | 3.79  | ✓      |
| P05     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Bore-1      | Front | 1.4upm  | 392rpm  | 2.01  | ✓      |
| P06     | T001    | 1        | Center_7/32  | DRILL          | Tap-1/2     | Front | 14.3upm | 2776rpm | 0.03  | ✓      |
| P07     | T005    | 5        | Drill_27/64  | DRILL          | Tap-1/2     | Front | 12.7upm | 1439rpm | 0.15  | ✓      |
| P08     | T006    | 6        | Tap_1/2      | FLOAT TAP      | Tap-1/2     | Front | 19.7upm | 255rpm  | 0.14  | ✓      |
| P09     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Tap-1/2     | Front | 1.4upm  | 392rpm  | 0.60  | ✓      |
| P10     | T001    | 1        | Center_7/32  | SPOT FACE      | Ream-1/4    | Front | 14.3upm | 2776rpm | 0.24  | ✓      |
| P11     | T007    | 7        | Drill_15/64  | DRILL          | Ream-1/4    | Front | 14.3upm | 2591rpm | 0.73  | ✓      |
| P12     | T008    | 8        | Ream_1/4     | BORE,DWELL     | Ream-1/4    | Front | 3.7upm  | 911rpm  | 5.59  | ✓      |
| P13     | T004    | 4        | Chamfer-1.25 | SPOT FACE      | Ream-1/4    | Front | 1.4upm  | 392rpm  | 1.59  | ✓      |
| P14     | T009    | 9        | End Mill_1/2 | Pocket Rough   | Pocket_0.5  | Front | 3.7upm  | 946rpm  | 29.87 | ✓      |
| P15     | T009    | 9        | End Mill_1/2 | Contour Finish | Contour_0.5 | Front | 3.7upm  | 946rpm  | 9.21  | ✓      |

Material File: ST\_FMSTW.MDB Total Time: 59.66 min.



**Note:** Red triangles appear on the left side of the process table window. To move a selected process in the table, click a triangle—it will become the new location for the selected process. If you make a mistake when moving a process in the table, choose **Undo** from the **Edit** menu to move it back to its original position.

- 2 Double click on this process or choose **Process Parameters** from the **Job Optimizer** menu to display the **Process Parameters** dialog.
- 3 Enter **3000** next to Spindle RPM as shown below.



Process Parameters

Info  
Process ID: P02 Face: FRONT

Coolant: Standard  
Hole Canned Cycle: DRILL  
Tool No.: 1  
Tool Offset No.: 1

Feeds and Speeds  
 Lock Feeds and Speeds  
Plunge Feed: 14.3 upm  
Tool Spindle Direction: cw  
Spindle Speed: 3000 rpm

Tool Change Point  
X: 0  
Y: 0  
Z: 5

Group Depth (H) Z\_Surf (S) Z\_Rapid (R) Z\_Clear (C)  
Bore-1 0.1 0 0.05 0.05

Output Hole Canned Cycle

Note:  
User Data

Cancel OK

- 4 Check the box for Lock Feeds and Speeds.
- 5 Click **<OK>** to return to the Process Table.

Compare the second line of the process table to the previous illustration to see the change.

| Proc    | Tool ID | Tool No.     | Tool Name      | Proc Task   | Group | Face    | Feed    | Speed | Time(min)                           | Status |
|---------|---------|--------------|----------------|-------------|-------|---------|---------|-------|-------------------------------------|--------|
| PC T010 | 10      | Face Mill_3  | Surface Rough  | Face_3      | FRONT | 5.3upm  | 325rpm  | 10.05 | <input checked="" type="checkbox"/> |        |
| PC T001 | 1       | Center_7/32  | DRILL          | Bore-1      | FRONT | 14.3up  | 3000rpm | 0.07  | <input checked="" type="checkbox"/> |        |
| PC T003 | 3       | Drill_31/32  | DRILL          | Bore-1      | FRONT | 10.4upm | 626rpm  | 0.61  | <input checked="" type="checkbox"/> |        |
| PC T002 | 2       | Bore_1       | BORE           | Bore-1      | FRONT | 2.6upm  | 756rpm  | 3.79  | <input checked="" type="checkbox"/> |        |
| PC T004 | 4       | Chamfer-1.25 | SPOT FACE      | Bore-1      | FRONT | 1.4upm  | 392rpm  | 2.01  | <input checked="" type="checkbox"/> |        |
| PC T001 | 1       | Center_7/32  | DRILL          | Tap-1/2     | FRONT | 14.3upm | 2776rpm | 0.03  | <input checked="" type="checkbox"/> |        |
| PC T005 | 5       | Drill_27/64  | DRILL          | Tap-1/2     | FRONT | 12.7upm | 1439rpm | 0.13  | <input checked="" type="checkbox"/> |        |
| PC T006 | 6       | Tap_1/2      | FLOAT TAP      | Tap-1/2     | FRONT | 19.7upm | 255rpm  | 0.11  | <input checked="" type="checkbox"/> |        |
| PC T004 | 4       | Chamfer-1.25 | SPOT FACE      | Tap-1/2     | FRONT | 1.4upm  | 392rpm  | 0.60  | <input checked="" type="checkbox"/> |        |
| P1 T001 | 1       | Center_7/32  | SPOT FACE      | Ream-1/4    | FRONT | 14.3upm | 2776rpm | 0.24  | <input checked="" type="checkbox"/> |        |
| P1 T007 | 7       | Drill_15/64  | DRILL          | Ream-1/4    | FRONT | 14.3upm | 2591rpm | 0.73  | <input checked="" type="checkbox"/> |        |
| P1 T008 | 8       | Ream_1/4     | BORE,DWELL     | Ream-1/4    | FRONT | 3.7upm  | 911rpm  | 5.59  | <input checked="" type="checkbox"/> |        |
| P1 T004 | 4       | Chamfer-1.25 | SPOT FACE      | Ream-1/4    | FRONT | 1.4upm  | 392rpm  | 1.36  | <input checked="" type="checkbox"/> |        |
| P1 T009 | 9       | End Mill_1/2 | Pocket Finish  | Pocket_0.5  | FRONT | 3.7upm  | 946rpm  | 25.18 | <input checked="" type="checkbox"/> |        |
| P1 T009 | 9       | End Mill_1/2 | Contour Finish | Contour_0.5 | FRONT | 3.7upm  | 946rpm  | 9.24  | <input checked="" type="checkbox"/> |        |

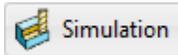
Material File: St\_fmstwm.db Total Time: 62.64 min.



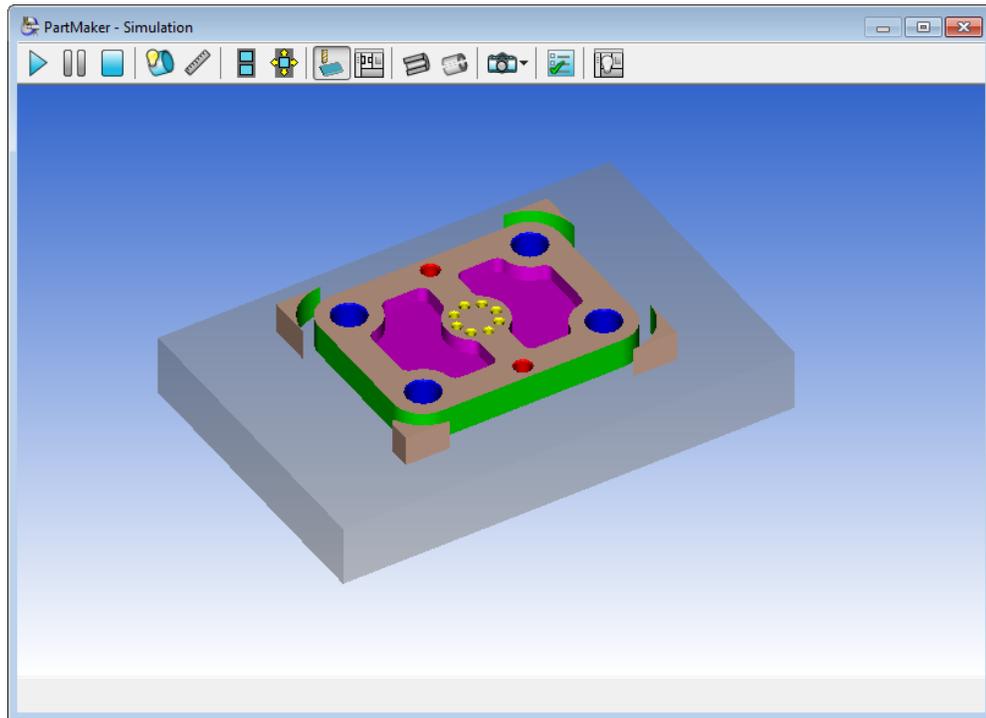
**Tip:** Practice selecting and modifying other processes. If you want to remove a process from simulation and simply un-check the box at the right of the process. This un-checked process will not simulate or be used to generate NC code. If you want to delete a selected process, choose **Delete** from the **Edit** menu or press the **<Delete>** key. You can undo a command by choosing **Undo** from the **Edit** menu immediately.

## Simulate the Process

Once you are satisfied with the appearance of your Process Table you can simulate cutting.

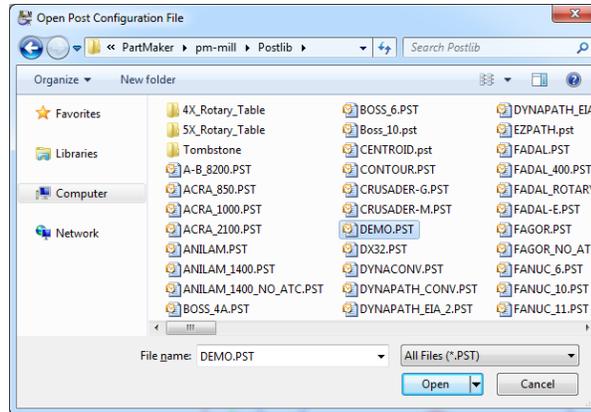


- 1 From the Process Table, click the **Simulate** in the lower left hand corner of the **Process Table** or just press the **<Space Bar>**.
- 2 When the simulation Window appears, press the **<Space Bar>** or choose **Simulation** from the **Simulate** menu to initiate 3D simulation. Your completed part should appear as shown below:



## Generate an NC Program

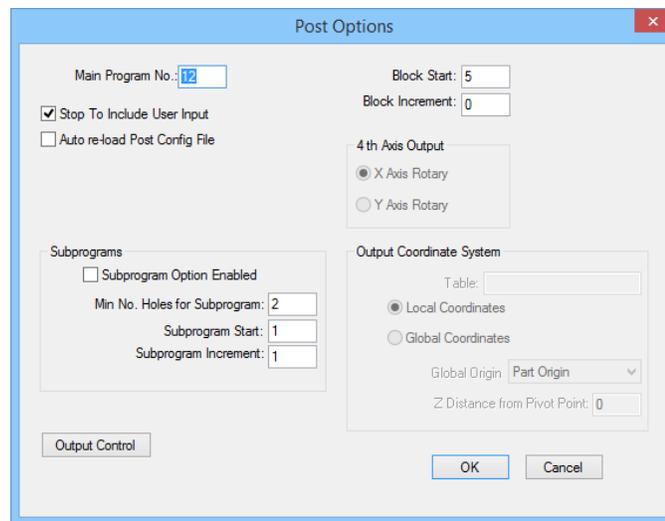
- 1 Choose **Post Config File= ?** From the **Job Optimizer** menu to display the Open Post Configuration File dialog.
- 2 Locate and double-click POSTLIB to open the Post Configuration file.
- 3 Select the DEMO.PST postprocessor, and then click **<Open>**.



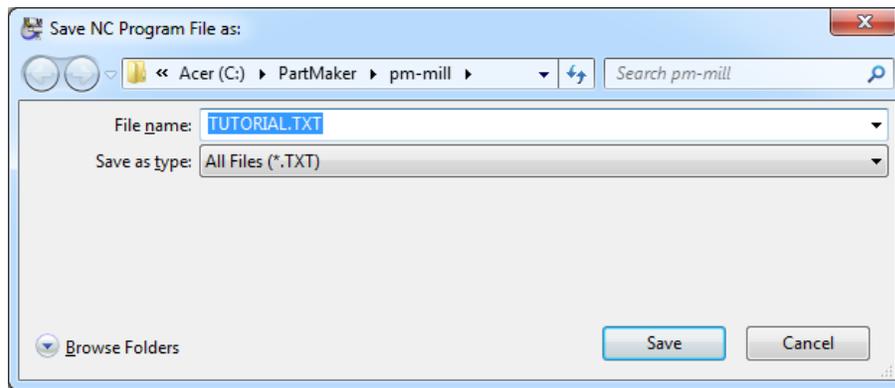
*Tip: If you use the same .PST file all the time, move it from POSTLIB (the post processor library) to your working directory/folder where it will be loaded automatically when you generate an NC program.*



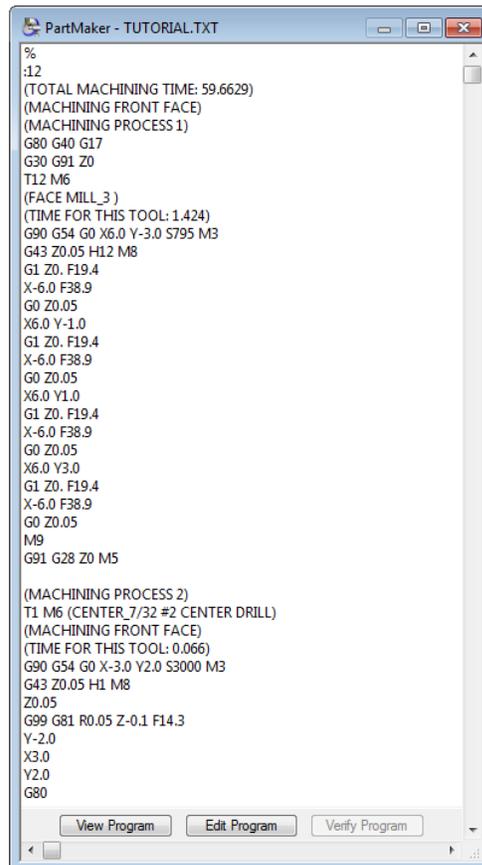
- 4 Choose Generate NC Program from the **Job Optimizer** menu to display the Save NC Program dialog. When you generate an NC Program for the first time during a programming session the Post Options dialog is displayed before the Save NC Program dialog.



- 5 Choose the Post Options you prefer. If you leave Block Increment at zero there will be no sequence numbers in your part program. Click **<OK>** to proceed to the Save NC Program dialog:



- 6 Enter the name TUTORIAL.TXT for the NC program, then click **<Save>** to generate the NC Program.



---

# Chapter 4: PartMaker® Simulation

## Introduction

PartMaker Versions 5.0 and higher come standard with an integrated 3D solid modeling process simulation and verification module. The purpose of simulation is to let you visualize all the processes as ordered on PartMaker's **Process Table**. You should run Simulation after the Process Table is arranged to your liking.

PartMaker's integrated process simulation module allows you to:

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

## Using PartMaker Simulation

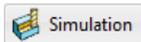
PartMaker Simulation can be viewed at any point in the programming process. You need not complete programming a part to see the results of your work.



**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. It is not necessary to complete synchronization of simultaneous operations before launching PartMaker Simulation.

## Launching Simulation

To launch PartMaker Simulation:



- 1 Press the **Simulation** button in on the Main tool bar.

or



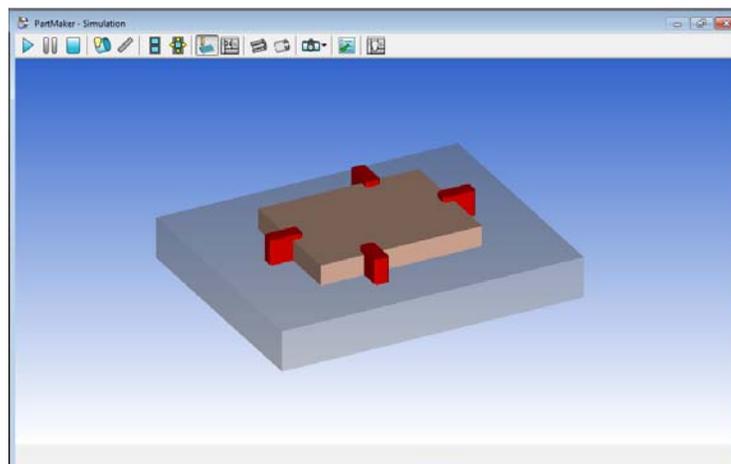
Press the Simulation button in the lower left hand corner of the Process Table

or



Choose Basic Simulation or Full Machine Simulation from the Simulation menu

Upon launching simulation, you will see an uncut work piece and the machining table that has been defined. A newly launched simulation window is shown below:



## Running Simulation

Once you have launched PartMaker Simulation, to start machining:



2 To start Simulation press the **Start Simulation** button.

or

Choose Start Simulation from the Simulation menu

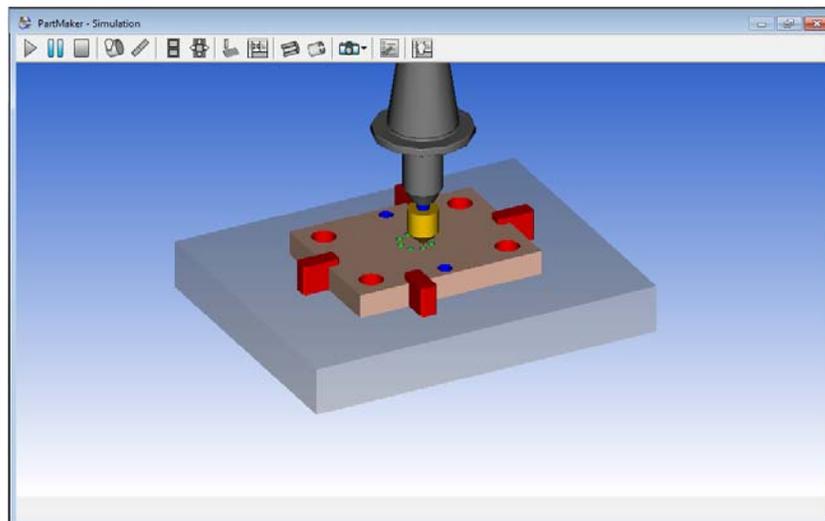
or

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

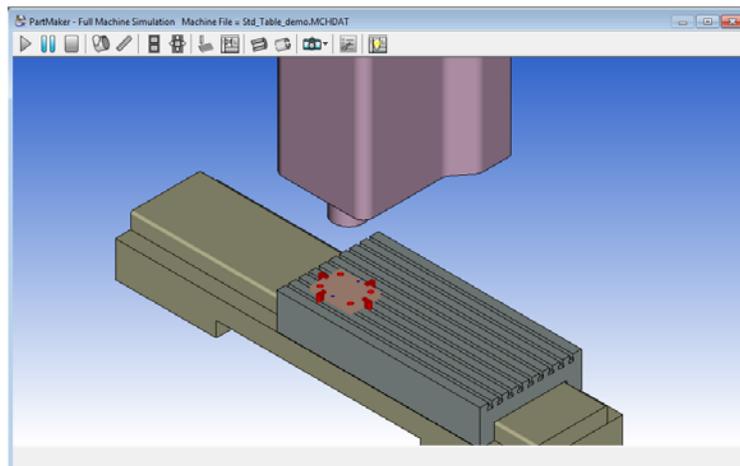


Once simulation has started, you can suspend cutting by pressing the **Pause** or the **<Esc>** button on your keyboard.

Below, is an example of suspending Basic simulation in progress:



Below, is an example of suspending Full Machine simulation in progress:



## 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.
-  Automatically arranges the milling Stock to fit on the table
-  Allows the user to change stock position interactively
-  Allows basic Simulation
-  Allows Full Machine Simulation
-  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

## 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: some of these features are not available in PartMaker Mill.

### Reset



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

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

### Show Travel Limits

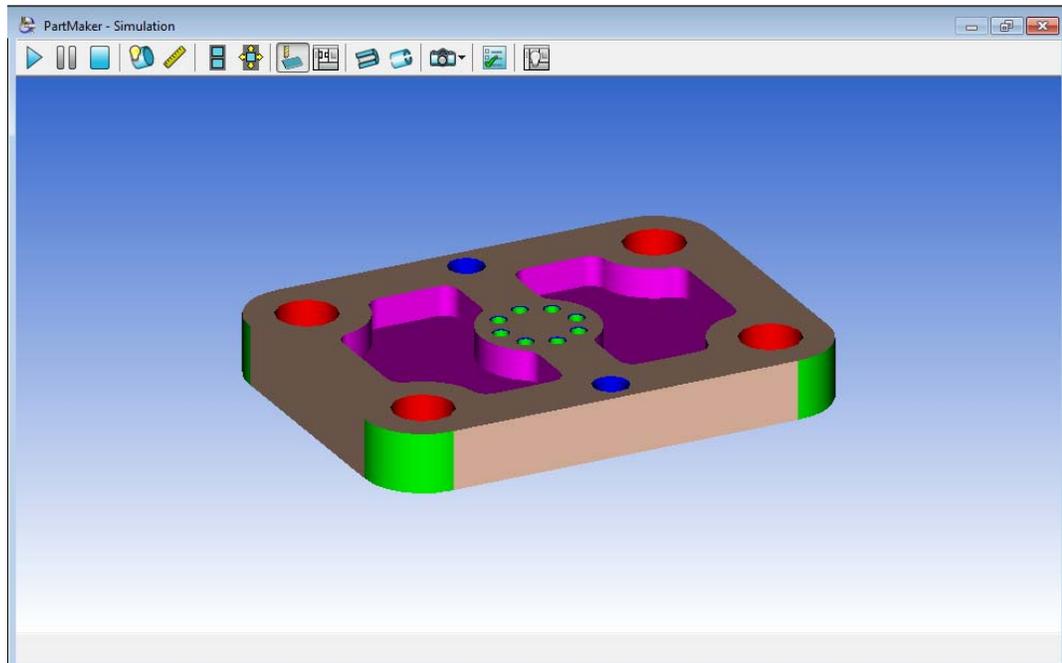
Displays the travel limits for machine being simulated for each machine component depending on how the machine is equipped. Travel limits for each machine components including main spindle, sub spindle and tool posts are set in the **Machine Data File** dialog accessed from the **ToolMinder** menu.

### Show Finished Part



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

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

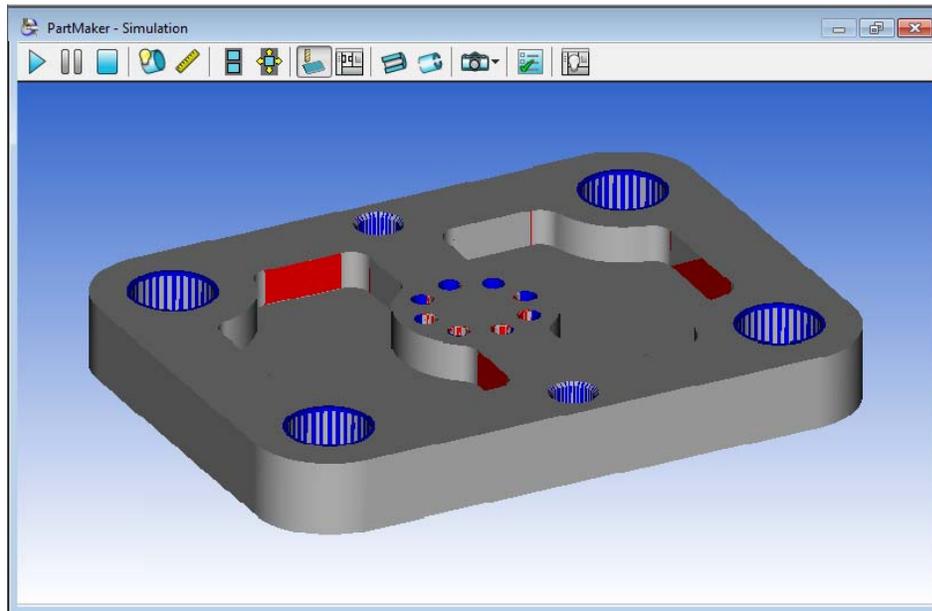


### Save Finished Part As STL file

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

## Compare Finished Part to Solid Model

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



## Comparison Options

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

## View Setting Features

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

### Default

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

### Best Fit

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

### XY View

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

### YZ View

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

## ZX View

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

## Isometric View

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

## Show Coordinate Axes

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

## Show Finished part Origin

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

## Show Coordinate axis

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

## Background Color

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

## Datum Color

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

## View Manipulation Features: Rotate, Move and Zoom

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

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

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

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

### Rotate

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

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

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

### Move

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

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

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

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

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

### Zoom

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

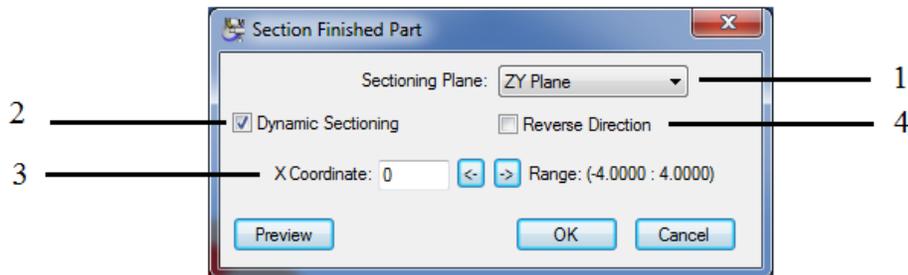
When **Zoom** is active, you can scale the process simulation or finished part by clicking the left hand mouse button and dragging it upward to increase the size of the part and dragging downward to decrease the size of the part.

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

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

## Sectioning Features

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



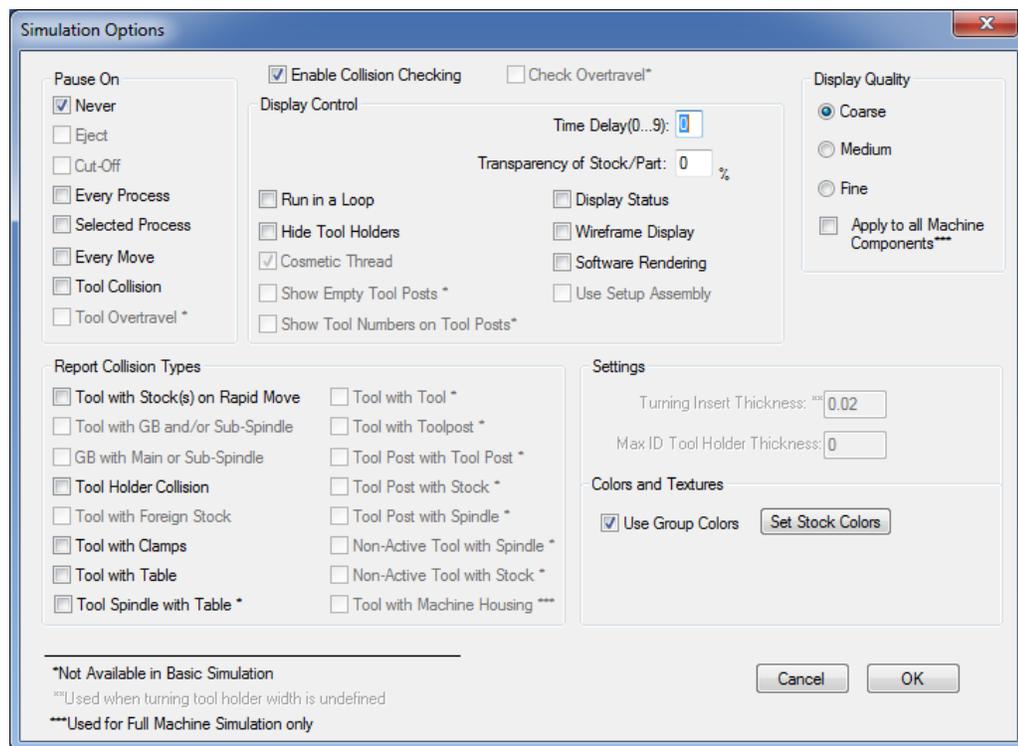
- 1 Sectioning Plane** – The Sectioning Plane section of this dialog allows you to determine the plane across which sectioning will occur normal to. The user can section a finished part normal to the XY, ZY and ZX plane. The axes along which the part is defined are constantly active in the lower left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

## Simulation Options

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

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

The **Options** dialog as it appears in Basic Simulation for PartMaker Turn-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** box allows you to detect the following collision types in Simulation:

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

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

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

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

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

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

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

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

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

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

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

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

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

## Display Control

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

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

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

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

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

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

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

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

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

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

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

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

## Display Quality

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

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

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

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

## Colors and Textures

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

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

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

---

# Appendix A:Materials

## PartMaker Material Library

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

| <b>File</b>  | <b>Description</b>              |
|--------------|---------------------------------|
| 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          |

| <b>File</b>  | <b>Description</b>                      |
|--------------|---|
| 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: Included Post Configuration Files

## PartMaker Post Library

**PartMaker** software package includes a library called POSTLIB that contains sample Post Configuration Files (called Config files below) for a variety of popular CNC machines.

You can use the ConfigPost application to customize and create additional Config files for CNC machines in your shop. **PartMaker** uses a Config file as a template to convert the Process Table into a part program file according to CNC machine guidelines. This conversion step is called "Post Processing".

Part programs produced when using these Config files can be fully verified with N-See verification software unless otherwise stated below.

All Config files are distributed with the Macro (Subroutines) option disabled in the General Info section. Customization of ConfigPost's Macro Related Formats may be required because of a great variety of macro calling conventions and user's preferences.

If it is stated below that Milling and Drilling Macros (subroutines) are supported, they can be enabled in ConfigPost's General Info Dialog.

## Inch Post Configuration Files

### 1 **AB8200.PST**

- Sample Config file for Allen Bradley 8200 control.
- Cutter diameter compensation is supported.
- Multiple fixture numbers H1,H2... are supported.
- Enter the fixture number in PartMaker Faces Dialog in the Work Offset field.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

### 2 **ACRA850.PST**

- Sample Config file for Vickers Acramatic 850 control as found on the Arrow mill (and others).
- Arc centers are absolute
- Multiple fixture offsets are supported
- Cutter diameter compensation is supported

### 3 **ACRA1000.PST**

- Sample Config file for Acramatic controls 10H, 10HC-1000/1500/2500 ("Big Blue")
- Single quadrant circular interpolation
- Vector type (P,Q) cutter diameter compensation is supported
- Consult PartMaker Inc. regarding availability of a translator to fully verify NC programs generated for this control.

#### **4 ACRA2100.PST**

- Sample Config file for Vickers Acramatic 2100 control
- Arc centers are absolute
- Multiple fixture offsets are supported
- Cutter diameter compensation is supported

#### **5 ANILAM14.PST**

- Sample Config file for Anilam 1400 control on a mill with an automatic tool changer
- Arc centers are always absolute
- Multiple work offsets are supported
- Uses M6 for tool change command

#### **6 1400-1.PST**

- Sample Config file for Anilam 1400 control on a mill with a manual tool changer
- Arc centers are always incremental
- Multiple work offsets are supported
- Uses M6 for tool change command

#### **7 BOSS4A.PST**

- Sample Config file for Bridgeport BOSS4,5 controls.
- Uses single quadrant circular interpolation with unsigned I,J values.
- Cutter diameter compensation is not supported.

#### **8 BOSS6.PST**

- Sample Config file for Bridgeport BOSS5,6,7 controls.
- Cutter diameter compensation is supported.
- 16 Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog

#### **9 BOSS10.PST**

- Sample Config file for Bridgeport BOSS8,9,10 and new BOSS10 compatible controls.
- Cutter diameter compensation is supported.
- 99 Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.
- M3 / M4 and M5 are not output for spindle control. If these codes are required, then use the DX32.PST

#### **10 CRUSAD-G.PST**

- Sample Config file for Anilam Crusader-G mill
- Cutter diameter compensation is supported

#### **11 CRUSAD-M.PST**

- Sample Config file for Anilam Crusader-M mill
- Cutter diameter compensation is supported

#### **12 DEMO.PST**

- Functionally the same as Fanuc.pst. Additional comments included to further illustrate how ConfigPost can be used to improve NC program format.

#### **13 DX32.PST**

- Sample Config file for Bridgeport DX32 and new BOSS10 compatible controls that require M3 / M4 and M5 to activate the spindle.
- Cutter diameter compensation is supported.
- 99 Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

#### **14 DYNACONV.PST**

- Sample Dynapath 10/20/30 Config file in CONVERSATIONAL mode.
- Each line must begin with an event (1) and end with a \$
- Drilling Macros are not supported
- Milling Macros are not supported
- Multiple Work Offsets are supported
- Cutter diameter compensation is supported

#### **15 DYNAEIA2.PST**

- Sample Dynapath 20/30 Config file for EIA (G-code) programming with 2 digit tool numbers.
- Cutter diameter compensation is supported
- Multiple Work Offsets are supported
- Drilling macros are not supported
- Milling macros are not supported

#### **16 DYNAEIA4.PST**

- Sample Dynapath 20/30 Config file for EIA (G-code) programming with 4 digit tool numbers.
- Cutter diameter compensation is supported
- Multiple Work Offsets are supported
- Drilling macros are not supported
- Milling macros are not supported

#### **17 FADAL.PST**

- Sample Config file for FADAL controls.
- Cutter diameter compensation is supported.
- Multiple work offset coordinate systems are supported

## 18 FADAL-A.PST

- Sample Config file for FADAL controls with programmable A rotary axis.
- Cutter diameter compensation is supported.
- Fixture offsets E1 – E99 supported
- In order to verify part programs produced when using this Config file you must locate a file FADAL.MCF in N-SEE\MCF directory and copy it to a file FADAL\_A.MCF.

## 19 FADAL-E.PST

- Sample Config file for FADAL controls.
- Cutter diameter compensation is supported.
- Multiple fixture numbers E1,E2... are supported.
- You should enter the fixture number in PartMaker Faces Dialog in the Work Offset field.
- Fanuc style Milling and Drilling Macros (subroutines) are supported.
- Milling and Drilling Macros can be enabled in ConfigPost's General Info Dialog.
- In order to verify part programs produced when using this Config file you must locate a file FADAL.MCF in N-SEE\MCF directory and copy it to a file FADAL\_E.MCF.

## 20 FAGOR.PST

- Sample Config file for Fagor 80XX series control
- Tool changes are assumed to be with an automatic changer, using the Taa.bb format where aa is the tool number and bb is the offset number.
- Multiple Work Offsets are supported
- Drilling macros are supported
- Milling macros are supported
- Cutter diameter compensation is supported

## 21 FAGOR-M.PST

- Sample Config file for Fagor 80XX series control
- Tool changes are assumed to be with a manual changer, using the T.aa format where aa is the tool offset to be applied.
- Multiple Work Offsets are supported
- Drilling macros are supported
- Milling macros are supported
- Cutter diameter compensation is supported

## 22 FANUC.PST

- Sample Config file for Fanuc type controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Multiple work offsets (G54,G55,..) are supported.
- Milling and Drilling Macros (subroutines) are supported.
- Milling and Drilling Macros can be enabled in ConfigPost's General Info Dialog.

## 23 FANUC-1.PST

- Sample Config file for Fanuc type controls without M6.
- Cutter diameter compensation is supported.
- Multiple work offsets (G54,G55,..) are supported.
- Milling and Drilling Macros (subroutines) are supported.
- Milling and Drilling Macros can be enabled in ConfigPost's General Info Dialog.

## 24 FANUC-A.PST

- Sample Config file for Fanuc control with rotary A axis
- Drilling macros are supported
- Milling macros are supported
- Multiple Work Offsets are supported
- Cutter Diameter compensation is supported

## 25 FANUCO.PST

- Sample Config file for the Fanuc O series control
- Drilling macros are supported
- Milling macros are supported
- Multiple Work Offsets are supported
- Cutter Diameter compensation is supported

## 26 GE1050IJ.PST

- Sample Config file for GE 1050 controls.
- Uses unsigned I,J values for circular interpolation.

## 27 HAAS.PST

- Sample Config file for HAAS controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Multiple work offsets (G54,G55,..) are supported.
- Fanuc style Milling and Drilling Macros (subroutines) are supported.
- Milling and Drilling Macros can be enabled in ConfigPost's General Info Dialog.

## 28 HG\_VMC.PST

- Sample Config file for Hardinge VMC and VMC II (600, 800, 1000)with Fanuc OT control.
- Drilling macros are supported
- Milling macros are supported
- Multiple Work Offsets are supported
- Cutter Diameter compensation is supported

## 29 HHCON.PST

- Sample Config file for Heidenhain conversational control.
- Cutter diameter compensation is supported.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

## 30 HURCOII.PST

- Sample Hurco Ultimax II or Ultimax III Config file
- Cutter Diameter compensation is supported
- Drilling Macros are not supported
- Milling Macros are not supported

## 31 MAZAK.PST

- Sample Config file for MAZAK controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Tools preloading is not supported.
- Multiple work offsets (G54,G55,..) are supported.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

## 32 MAZAK-1.PST

- Sample Config file for MAZAK controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Tools preloading is supported.
- Multiple work offsets (G54,G55,..) are supported.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

### **33 MELDAS.PST**

- Sample Config file for Mitsubishi MELDAS control
- Cutter compensation is supported
- Multiple work offsets are supported
- Tools are changed with a M6 command
- Drilling Macros are supported
- Milling Macros are supported

### **34 MELDAS-1.PST**

- Sample Config file for Mitsubishi MELDAS control for use with machines without automatic tool changer. Tool changes are assumed to be by hand.
- T-Codes are not used. Appropriate tool is determined by the H-code used in the G43 line associated with the tool changing process.
- Cutter compensation is supported
- Multiple work offsets are supported
- Drilling Macros are supported

### **35 OSP5020M.PST**

- Sample Config file for OKUMA 5000 series controls.
- Cutter diameter compensation is supported.
- Single fixture number G15H1 is supported.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.

### **36 OSP\_H.PST**

- Sample Config file for OKUMA 5000 series controls.
- Cutter diameter compensation is supported.
- Multiple fixture numbers G15H1, G15H2 are supported.
- Enter the fixture number in PartMaker Faces Dialog in the Work Offset field.
- Milling Macros (subroutines) are supported.
- Drilling Macros are not supported.
- Milling Macros can be enabled in ConfigPost's General Info Dialog.
- In order to verify part programs produced when using this Config file you must locate a file OSP5020M.PST in N-SEE\MCF directory and copy it to a file OSP\_H.PST.

### **37 PROT\_MX3.PST**

- Sample Config file for ProtoTrack-MX3 controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Fully Fanuc compatible.

### 38 **PROT\_PLU.PST**

- Sample Config file for ProtoTrack-PLUS controls.
- Cutter diameter compensation is supported.
- Not fully Fanuc compatible because it uses ";" at the end of each line.
- Milling Macros are supported
- Consult PartMaker Inc. regarding availability of a translator to fully verify NC programs generated for this control.

### 39 **QUICK.PST**

- Sample Config file for QUICKMILL (Canadian).
- Cutter diameter compensation is supported
- Multiple work offsets are supported
- Drilling Macros are supported
- Milling Macros are supported

### 40 **TECHNO.PST**

- Sample Config file for TECHNO mill, which is basically a subset of standard FANUC control language.
- Cutter compensation is not supported
- Milling macros are not supported
- Drilling macros are not supported

### 41 **YASNAC.PST**

- Sample Config file for YASNAC controls.
- Cutter diameter compensation is supported.
- Uses M6 for tool change.
- Multiple work offsets (G54, G55...) are supported.
- Fanuc style Milling and Drilling Macros (subroutines) are supported.
- Milling and Drilling Macros can be enabled in ConfigPost's General Info Dialog.

## Metric Post Configuration Files

### 1 **M\_FANUC.PST**

- Sample Config file for Fanuc type controls with metric output.

---

# Appendix C: GPF Files

## Why GPF Files

In addition to the interactive input, **PartMaker** version 2.3 and higher is capable of reading a text file in the format shown in Fig.1 below. The format of such a file is very similar to a part program file where drilling canned cycles were replaced by **PartMaker** compound cycles. It is not a complete part program yet since essential machining parameters, namely feedrates, spindle speeds, tool numbers, etc. are missing. The name **G**eneric **P**rogram **F**ormat or, in short, **GPF** was chosen for such a format. A GPF file can be created via a text editor or generated by **PartMaker** or another CAD/CAM program.

GPF files can be used for various purposes. The three most frequently used cases are:

- Repetitive patterns of most commonly used Part Features
- Font files for Engraving
- Interfacing to other CAM/NC systems

## GPF Files Import/Export

An example of GPF file is shown below:

```
$Drill(Tap-1/2) Chamfer(0.0500) Surface(0.0000) Depth(0.5000) Rapid(0.0500) Blind
X0.000000Y2.500000
X0.000000Y-2.500000

$Mill(Contour_0.5) Surface(0.0000) Depth(1.0000) Rapid(0.0500) Tool(0.5000) On
G0 X-3.000000 Y-3.000000
G1 X3.000000 Y-3.000000
G3 X4.000000 Y-2.000000 I3.000000 J-2.000000 R1.000000
G1 X4.000000 Y2.000000
G3 X3.000000 Y3.000000 I3.000000 J2.000000 R1.000000
G1 X-3.000000 Y3.000000
G3 X-3.999957Y1.999955 I-3.000000 J2.000000 R1.000000
G1 X-4.000000 Y-2.000000
G2 X-3.000000 Y-3.000000 I-3.000000 J-2.000000 R1.000000
$$End
```

**Figure 1**

The import of the foregoing GPF file into **PartMaker** results in two tapped blind holes at locations (0,2.5) and (0,-2.5) and a rectangular profile 8"x6" with 1" radius corner fillets.

Export GPF File command in **PartMaker**'s File menu creates a GPF file that contains all Groups of Part Features in the active (front most) **Face Window**.

Import GPF File command in **PartMaker**'s File menu merges (attaches) all Groups of Part Features from a GPF file to the list of Groups in the active (front most) **Face Window**.

## Details of GPF File Format

The GPF file consists of statements and data blocks. There are 3 types of statements: **\$\$Mill**, **\$\$Drill**, and **\$\$End**. The statement type must be followed by parameters as specified below:

### 1 **\$\$Drill Statement Structure**

- `$$Drill(Tap-1/2)Chamfer(0.0500)Surface(0.0000)Depth(0.5000)Rapid(0.0500)Blind`
- The word **\$\$Drill**
- The name of the cycle Tap-1/2 in PartMaker database.
- Word Chamfer or letter "C" followed by the hole chamfer value.
- Word Surface or letter "S" followed by the machining surface Z coordinate
- Word Depth or letter "D" followed by the nominal depth
- Word Rapid or letter "R" followed by the rapid traverse level. (The latter is defined as unsigned distance from the part surface)
- Letter "T" for the through hole or "B" for blind hole

### 2 **\$\$Drill** statement is followed by hole location blocks

- Each block contains X, Y coordinates that must be in decimal format without leading zeros. Both coordinates must be in the first block.
- Sequence numbers are allowed.

Example1: X0.000000 Y2.500000

Example2: N020 X0.000000 Y2.500000

### 3 **\$\$Mill** statement structure

- `$$Mill (Contour_0.5)Surface(0.0000)Depth(1.0000)Rapid(0.0500)Iool(0.5000) On`
- The word **\$\$Mill**
- The name of a milling group Contour\_0.5. Any combination of letters and digits can be used.
- Word Surface or letter "S" followed by the machining surface Z coordinate
- Word Depth or letter "D" followed by the cutting depth
- Word Rapid or letter "R" followed by the rapid traverse level. (The latter is defined as unsigned distance from the part surface)
- Word Iool or letter "T" followed by the tool diameter.
- Word "On" or letter "O" for the center line cutting, or Word "Left" or letter "L" for offset Left, or Word "Right" or letter "R" for offset Right, or Word "**Pocket**" or letter "P" for a pocket.

### 4 **\$\$Mill** statement is followed by one or several profiles that represent tool path.

- Profiles consist of straight lines and circular arcs. Profiles can be closed or open. The profiles are defined as:

```
G0 X-3.000000 Y-3.000000 //Start point: (-3,-3)
G1 X3.000000 Y-3.000000 //Line to (3,-3)
G3 X4.000000 Y-2.000000 I3.000000 J-2.000000 R1.000000 //Counter-Clockwise arc to (4,-2).
//Circle center at (3,-2). Radius=1
```

**G2** modifier is used for clockwise arc.

- Each block must contain both X and Y coordinates in decimal format without leading zeros. Sequence numbers are allowed.

### 5 **\$\$End** statement

- This statement indicates the end of valid data. Comments can be placed after the **\$\$End** statement.

## Usage of GPF Files as Font Files for Engraving

When you select engraving icon in PartMaker you are prompted to enter text, text height, and spacing between letters.

The software loads the default font from a file "FONT.GPF" located in the working directory. If such file is not present PartMaker displays a standard file open dialog that allows the user to designate another "\*\*\*.GPF" file as a font file.

- When you create your own font file, you must place each character, namely: ["A...Z", "0-9", ".", ",", "-", ";", "/"] in a separate group named after this character. The size of a character and its origin can be arbitrary.

*Page Intentionally Left Blank*

---

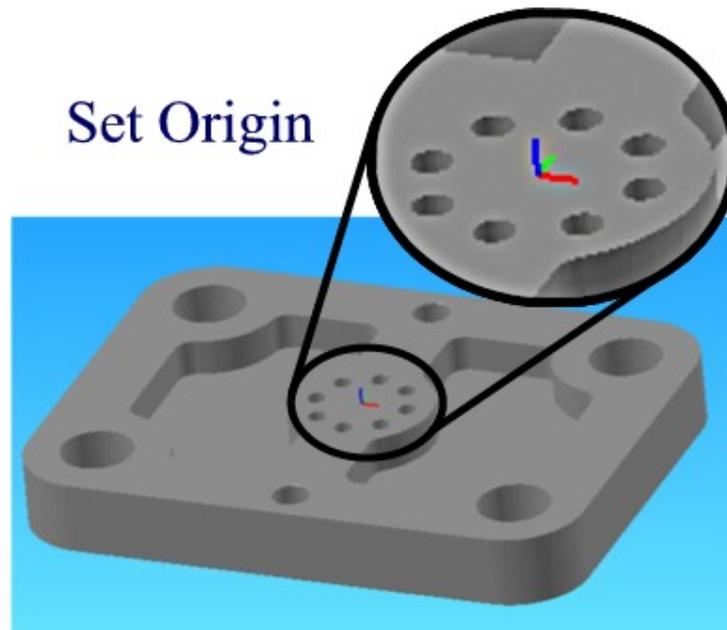
# Appendix D: Programming Directly on a Solid Model

## Introduction

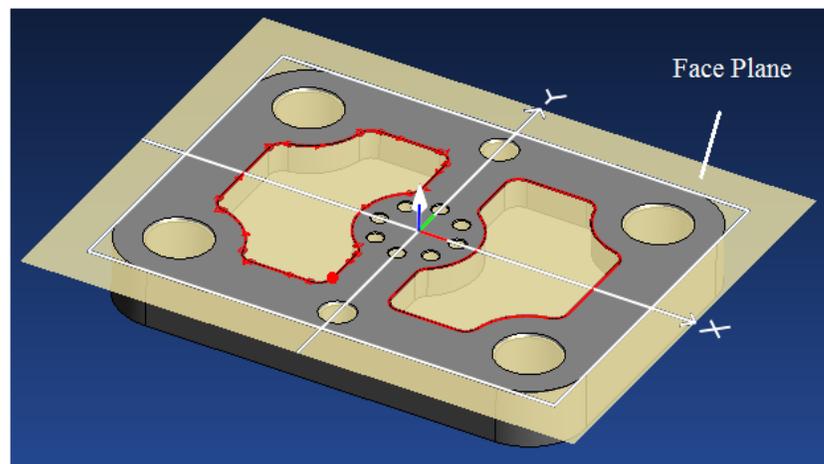
This document describes the process of programming directly on a Solid Model in PartMaker/Mill version 9 and higher.

## Key Definitions

**Part Coordinate System (PCS):** The Origin of a PCS is on a Solid Model. After being set, the PCS should not change during the programming session.

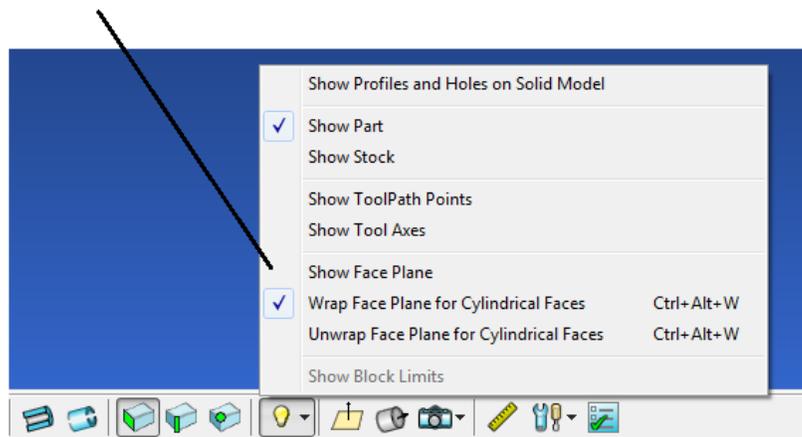


**Face Coordinate System (FCS):** This is a coordinate system with two axes in the Face Plane and the third axis perpendicular to the Face Plane. A Face Plane is the plane in which a PartMaker Face Window is programmed. Each Face Window can have its own Face Plane. The Face Plane must be set in the correct place to properly program on the Solid Model.



## Face Plane Icons for Solid Model:

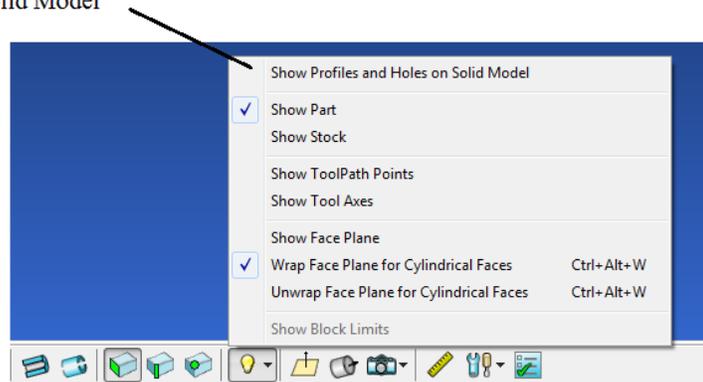
Show Face Plane



Menu

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

Show Profiles and Holes on Solid Model



Menu

## Solids Programming Tutorial

This tutorial is designed to help you learn the steps to go through in using **PartMaker** Mill to program on a Solid Model.

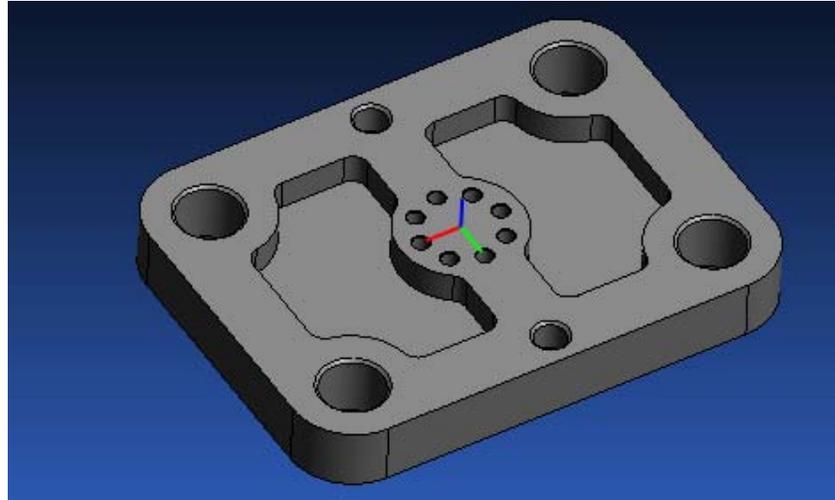
The Complete Finished Job Files can be found in the C:\PartMaker\PM-Mill\Solids\_Programming\_Tutorial Directory.

### How you will create the Sample part

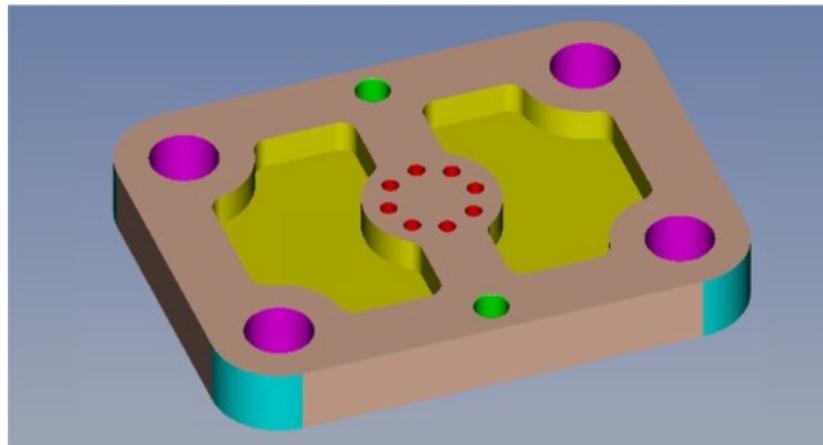
Here are the major steps you will follow to create the Swiss tutorial part:

- √ 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:

### Mill XY:

Use an end mill to create Pockets and Profiles.

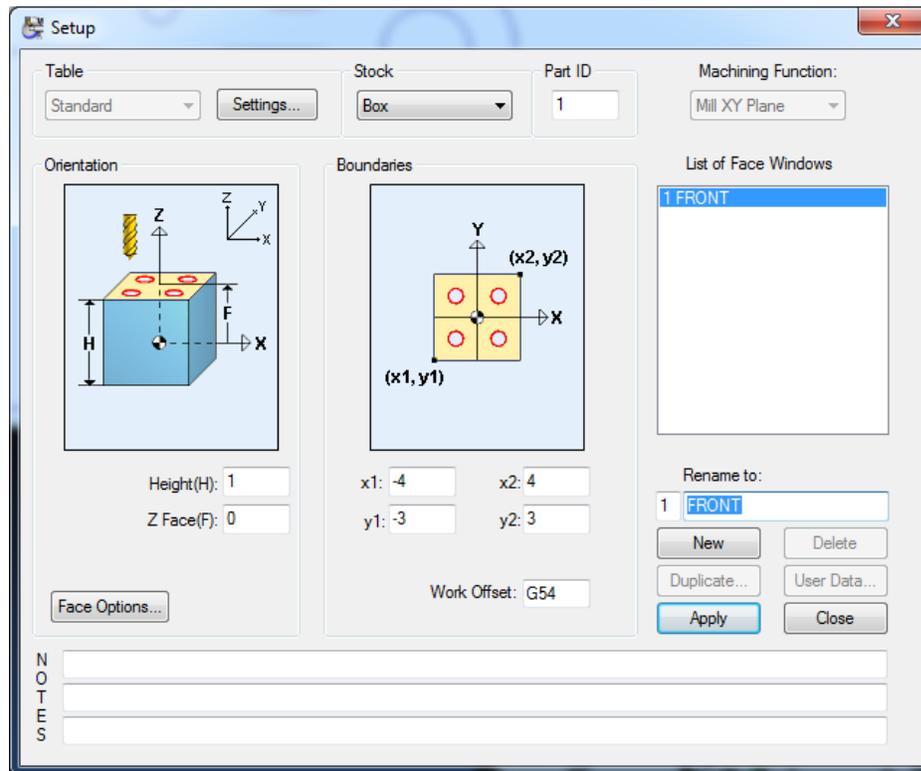
Use Hole Groups and Cycles.

Programming on the Solid Model can accomplish almost every Machining Function needed. In Most Cases you will not need to work in the CAD window at all.

Now you will look in detail at the steps necessary to program each of the components of the finished part.

## Starting PartMaker

- 1 Double click on the PartMaker Mill icon.
- 2 A picture 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.
- 3 This dialog will be explained in greater depth later in this document.



## View the Finished Sample

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

**C:\PartMaker\PM-Mill\ Solids\_Programming\_Tutorial**

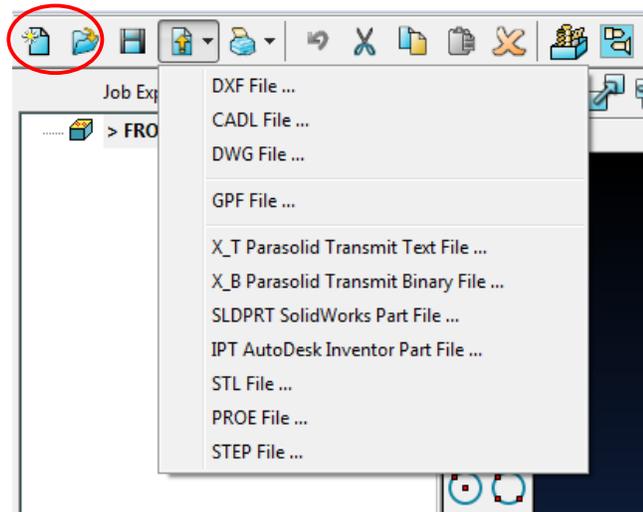


*NOTE: This will also open the Sample Tools and Cycles files for you to use when you create your own Part.*

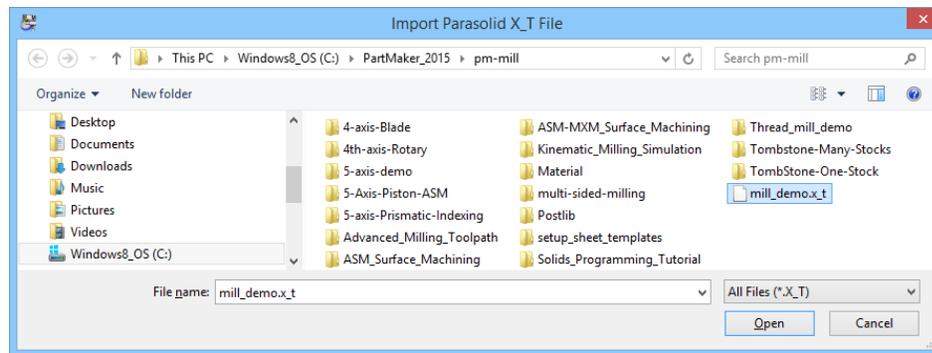
- 3 Look through the Sample Job file and familize yourself with the Profiles and Hole Groups.
- 4 When Ready, Choose **New Job** from the **File** Menu.

## Importing a Solid Model

- 1 Select the **Import** icon and select the X\_T Solid Model.



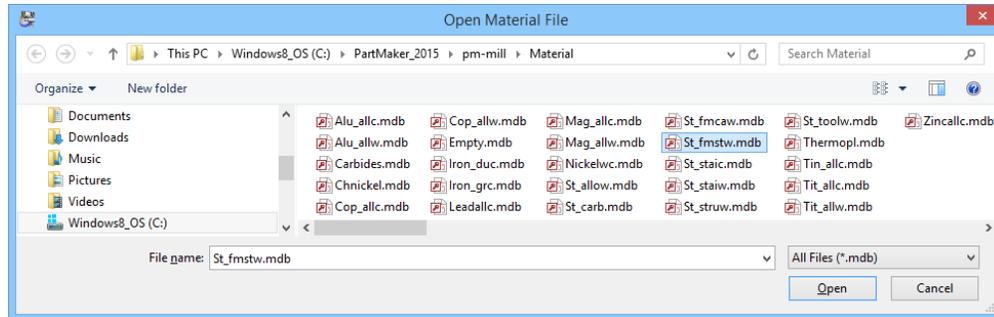
- 2 Choose the file, **SampleSolidProgramming.X\_T**, and click **<Open>**.



## 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\_fmstwmdb** as shown below and click **<Open>**.



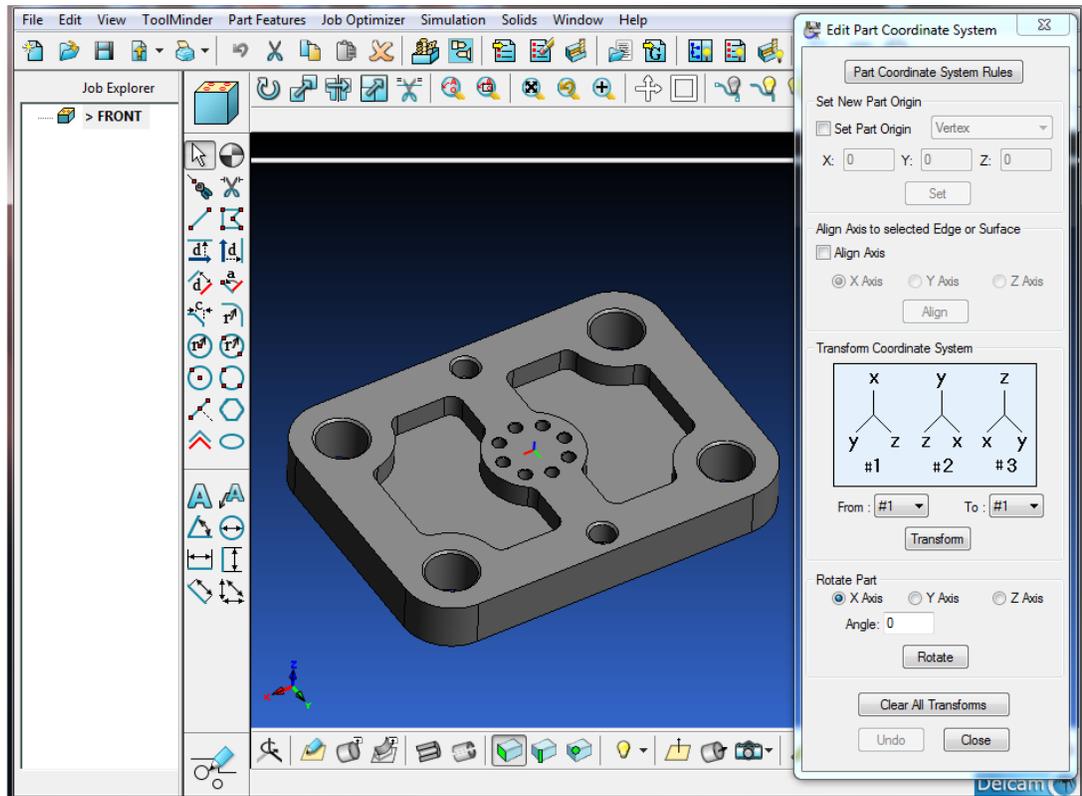
You have now loaded the tools, cycle, and material files that were previously developed for the Solid Model Programming Sample.

## Editing the Coordinate System

In this section, you will edit the Coordinate System on the Solid Model.



- 1 On the Solids Tool Bar, Choose the **Edit Part Coordinate System**.
- 2 Edit the Coordinate System on the Solid Model so the Origin is at the Center of the Part and the Z positive is pointing up away from the Part.

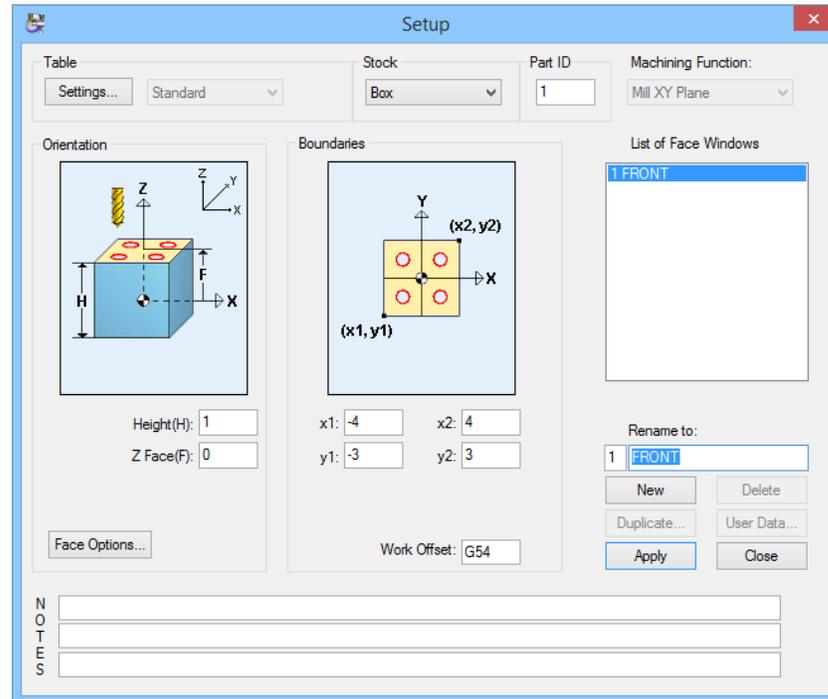


## Setting up the Face Window

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

- 1 You will need to make certain changes to the Set-up Dialog before moving to the Solid Model.

|                           |               |
|---------------------------|---------------|
| <b>Table</b>              | Standard      |
| <b>Stock</b>              | Block         |
| <b>Machining Function</b> | Mill XY Plane |

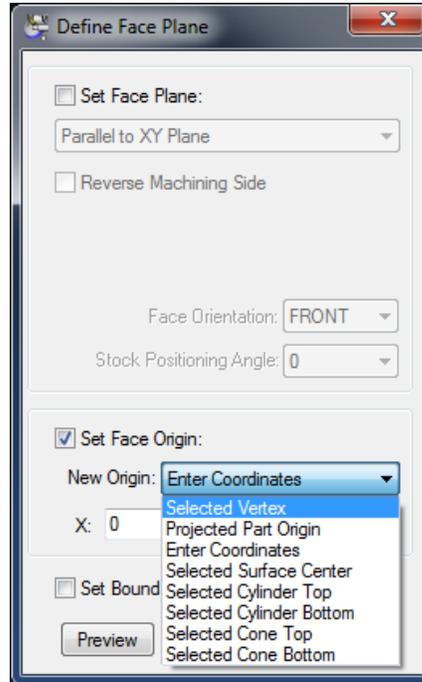


- 2 When Programming with a Solid Model. Many items will be set automatically from the information obtained from the Solid Model. To import these settings into the Set-up Window, you will need to define a face plane.
- 3 Select the **Define Face Plane** Icon in the Solids Tool Bar.
- 4 For Block Mill parts, you can define your Face Plane **Parallel to XY Plane**. Check the **Set Face Plane** box. Choose **Parallel to XY Plane** in the Drop Down Menu, and check the **Set Boundaries** Box.

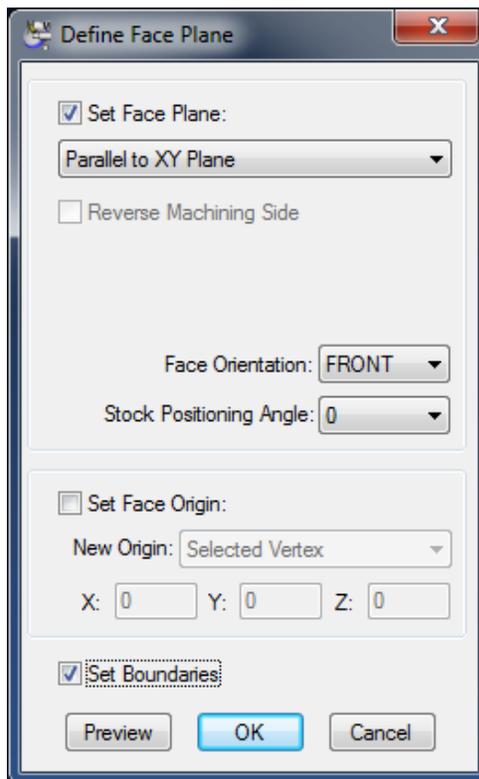


*Note: There are many choices to select the Origin for the Face Plane.*

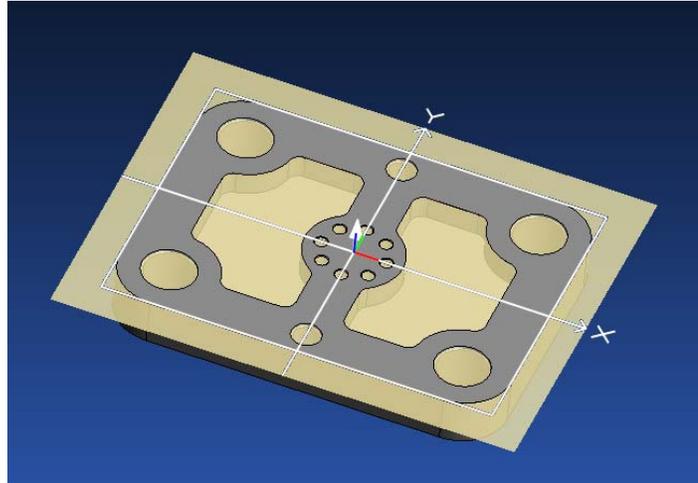
- 5 Click the New Origin drop down menu to view the options available as shown below:



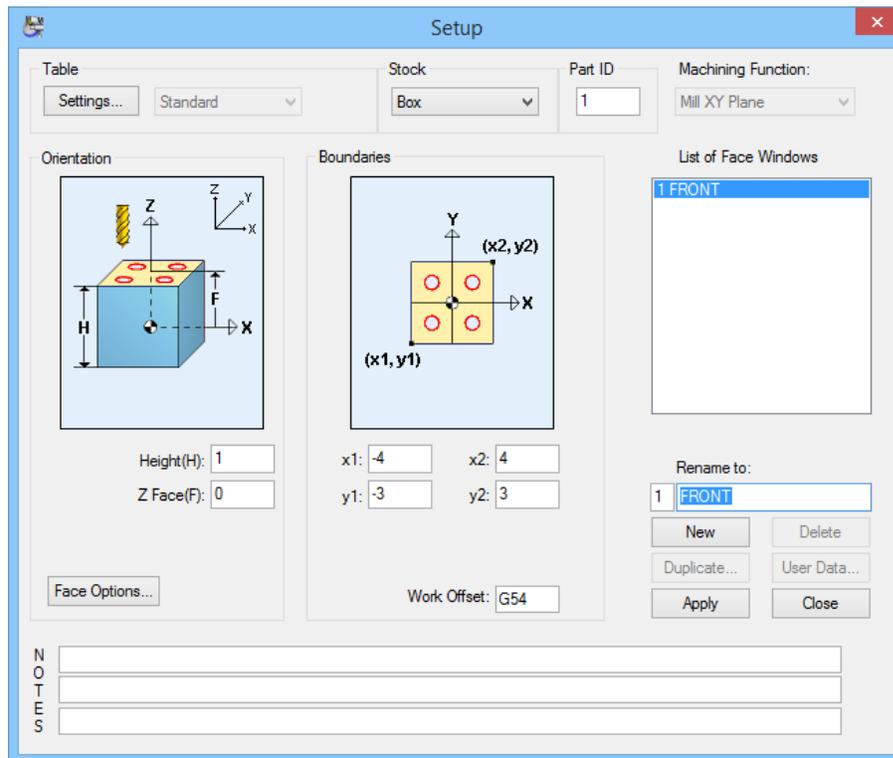
- 6 Select Parallel to XY Plane at this time.



- 7 Click the **<Preview>** Button to see the Face Plane on the Solid Model update as shown here:



- 8 Click **<OK>** and open the Set-up Window. Note the Part height, X1, X2, Y1, Y2 were imported from the Solid Model. You could now make any other changes required for your face window.



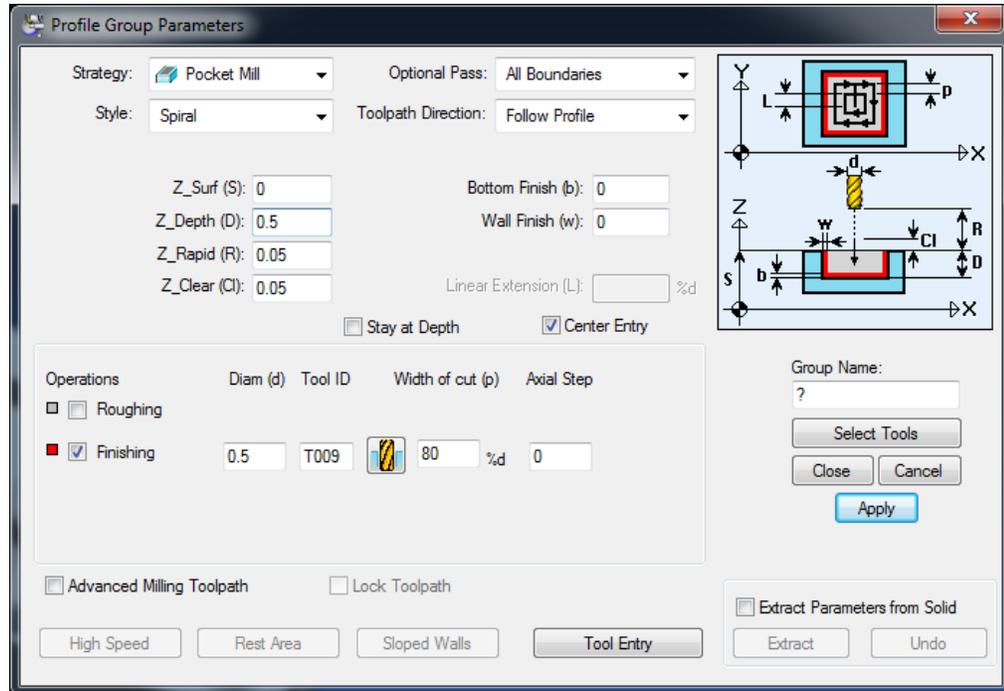
- 9 Click **<Apply>** and **<Close>**.

## Programming a Pocket Operation

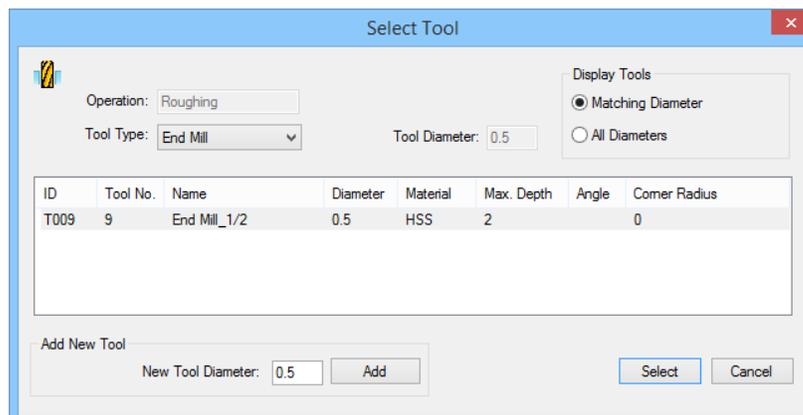


- 1 Select the **New Profile Group** icon.
- 2 Change the following settings on the Profile Group Parameters dialog:

|  |             |
|--|-------------|
| <b>Strategy</b>                                | Pocket Mill |
| <b>Remove Check</b>                            | Roughing    |
| <b>Check</b> (checked when Roughing unchecked) | Finishing   |
| <b>Diameter</b>                                | 0.5         |

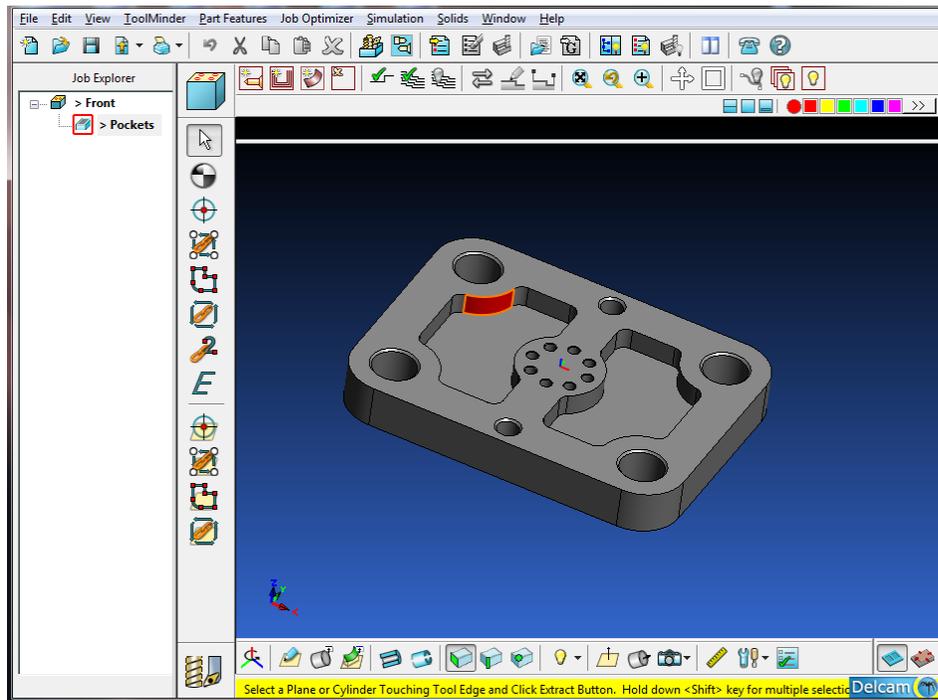


- 3 Click **<Select Tools>** and choose the 1/2 End Mill.

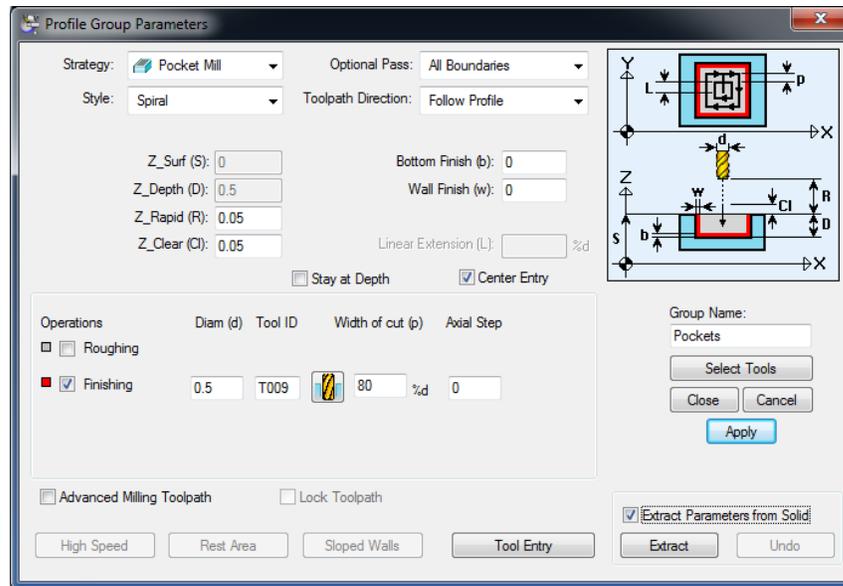


- 4 Name the Group, **Pockets**.

- 5 Click the **Extract Parameters from Solid** check box.
- 6 Select the Wall of one of the Pockets.



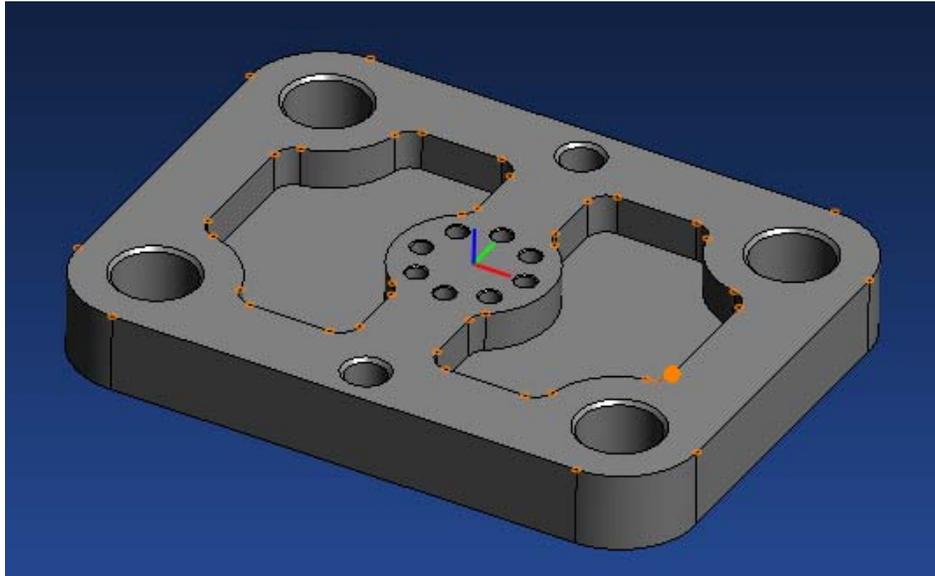
- 7 Click the **<Extract>** Button.
- 8 Your dialog should look like the following.



- 9 Click **<Apply>** and **<Close>**.



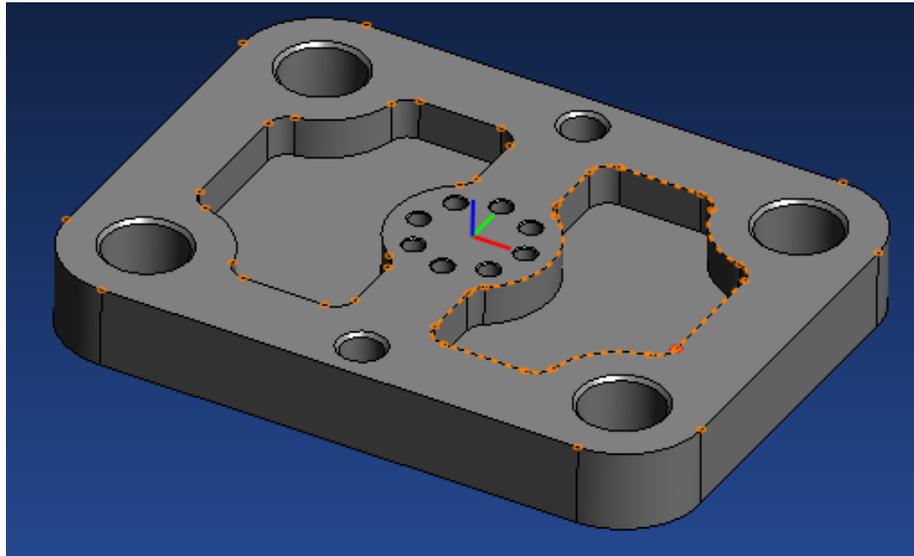
- 10 Select the **Chain Geometry on Solid Model** icon.
- 11 Select a Start Point of the 1<sup>st</sup> Pocket. When choosing start point, click on the side of the point that you want the Chain to follow. This will determine the direction the chain will follow.



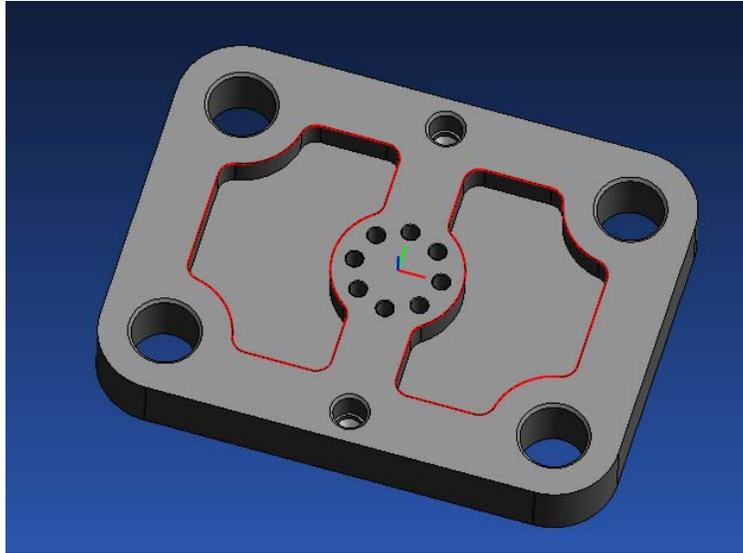
- 12 Select the End Point of the 1<sup>st</sup> Pocket.



**Note:** The Dotted line indicates the Profile Preview.



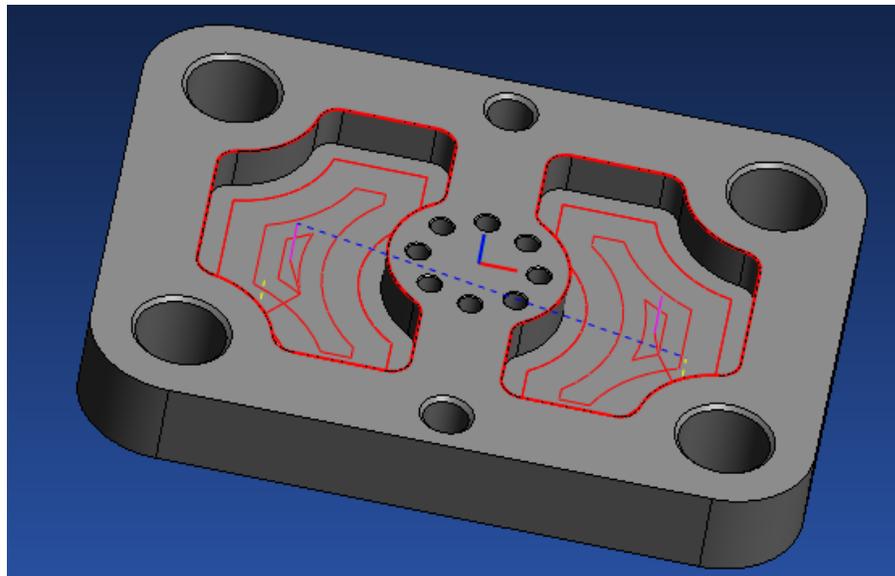
13 Repeat these steps for the Second Pocket.



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

15 The **Tool Path Verification Options** dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**.

Your screen should appear as below:



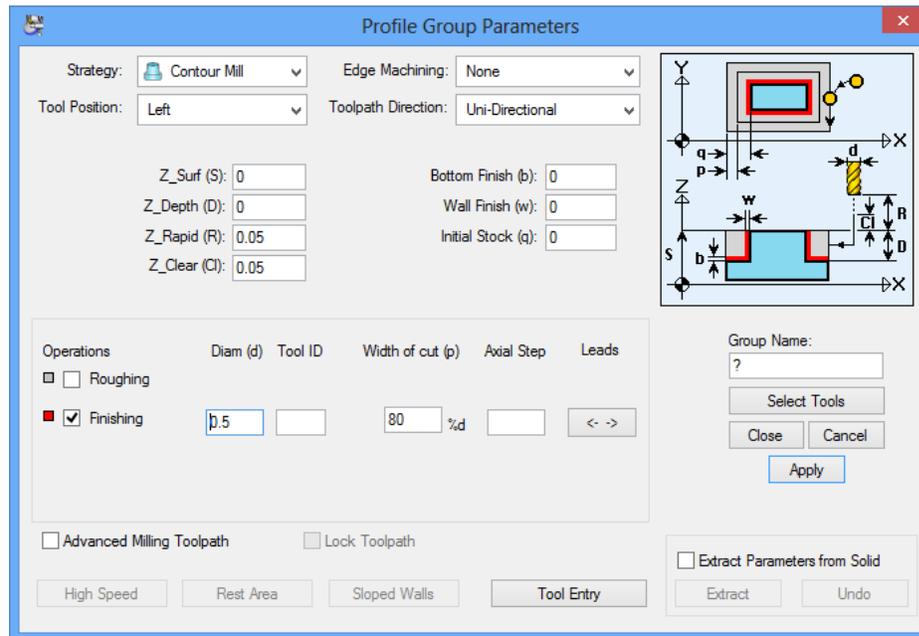
16 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

## Programming a Profile Operation

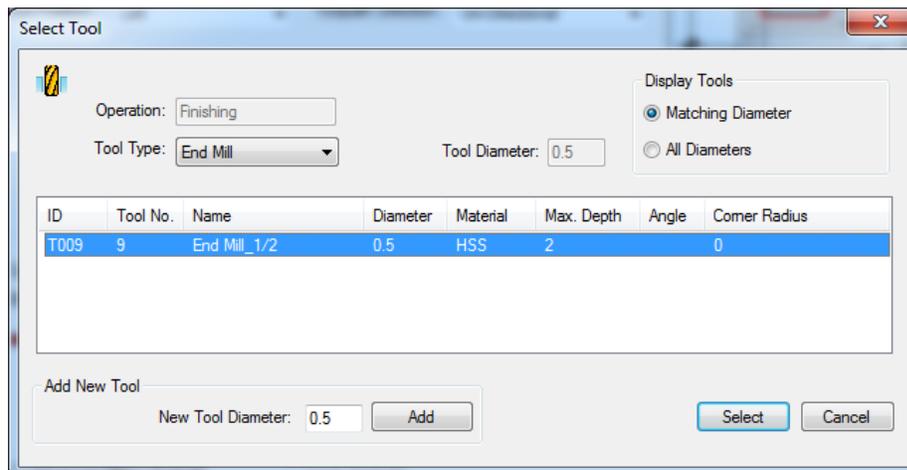


- 1 Select the **New Profile Group** icon.
- 2 Change the Following settings in the Profile Group.

|  |              |
|--|--------------|
| <b>Strategy</b>                                | Contour Mill |
| <b>Tool Position</b>                           | Left         |
| <b>Remove Check</b>                            | Roughing     |
| <b>Check</b> (checked when Roughing unchecked) | Finishing    |
| <b>Diameter</b>                                | 0.5          |

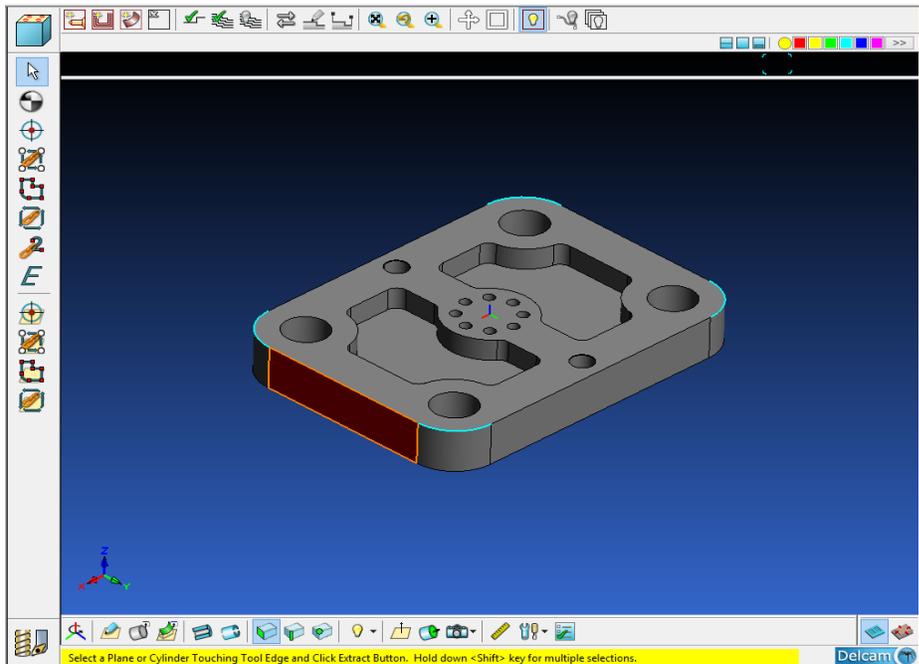


- 3 Click **<Select Tools>** and choose the 1/2 End Mill.

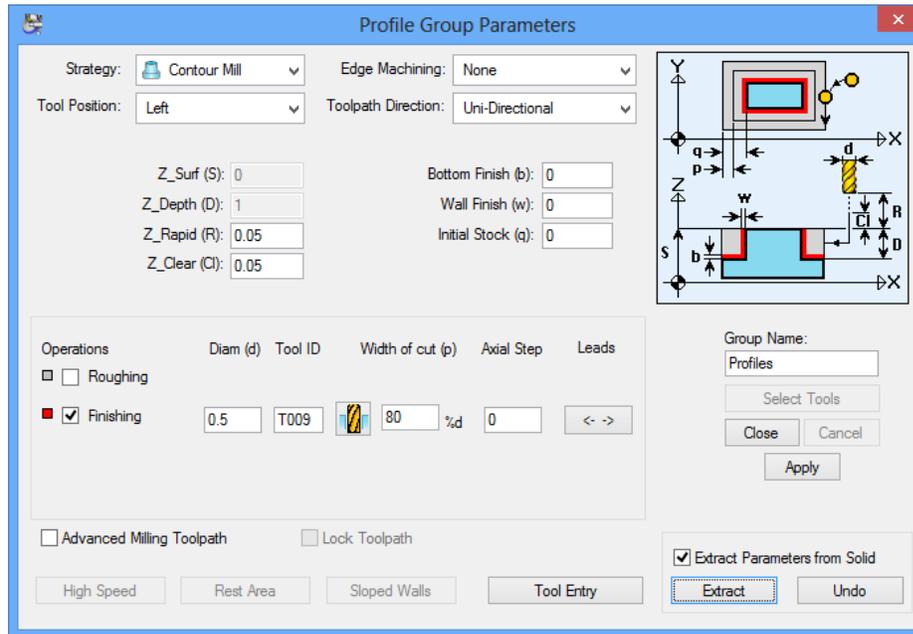


- 4 Name the Group Profiles.

- 5 Click the **Extract Parameters from Solid** check box.
- 6 Select the Wall of one of the Outside Radii.



- 7 Click the **<Extract>** Button.
- 8 Your dialog should look like the following.

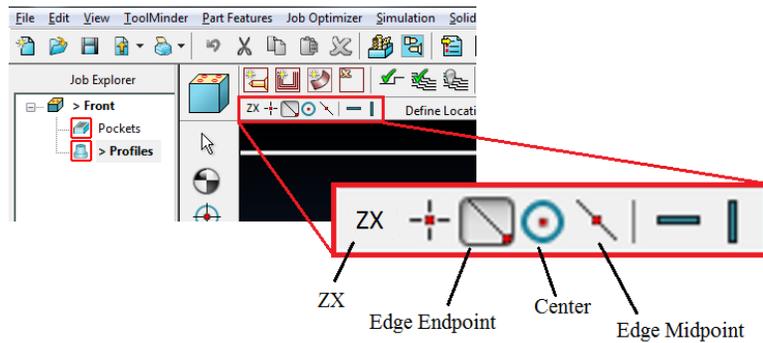


- 9 Click **<Apply>** and **<Close>**.

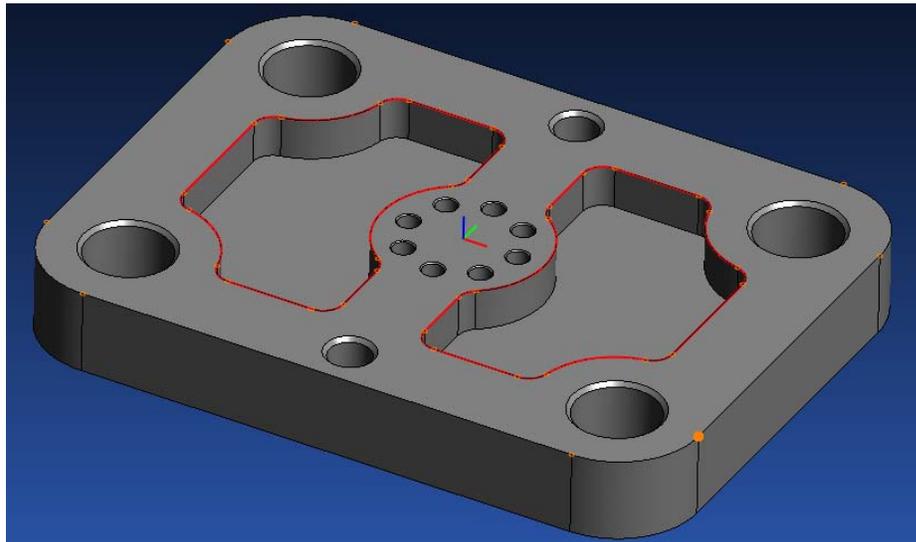


10 Select **Define Profile on Solid Model** icon.

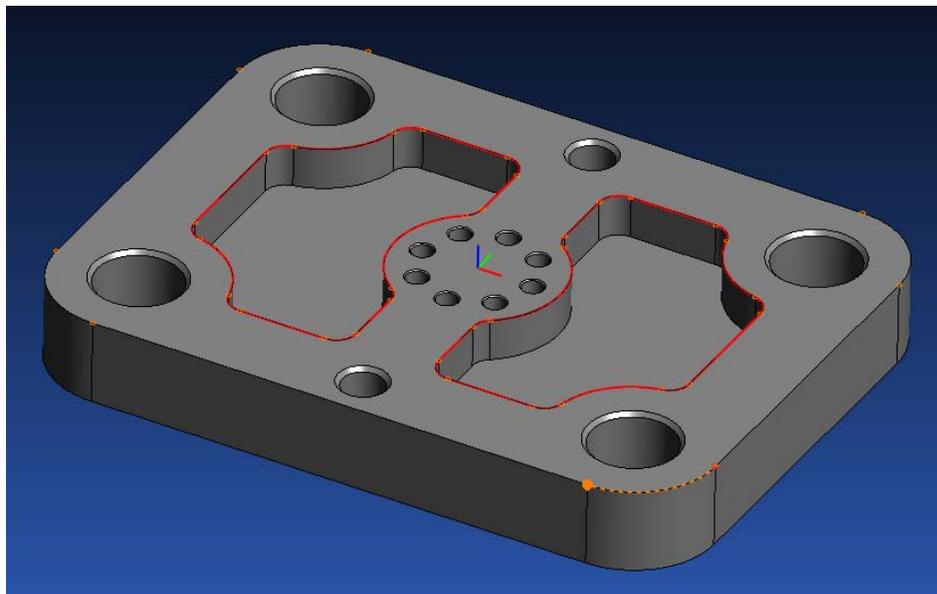
11 Select **Edge Endpoint Snap Mode** icon.



12 Select a Start Point of the 1<sup>st</sup> Corner.

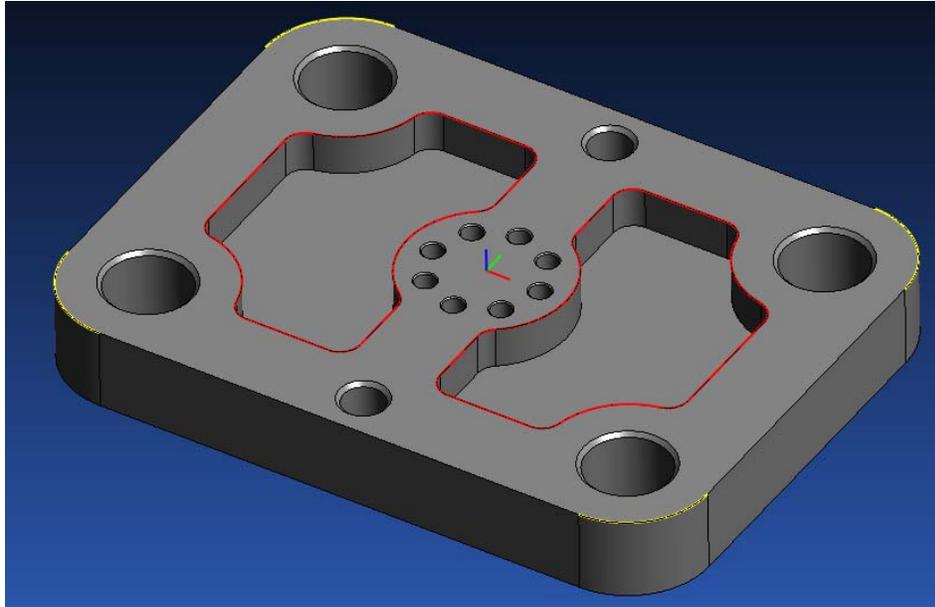


13 Select the End Point of the 1<sup>st</sup> Corner. Note: The Dotted line indicates the Profile Preview.



14 Press the **Selection** icon.

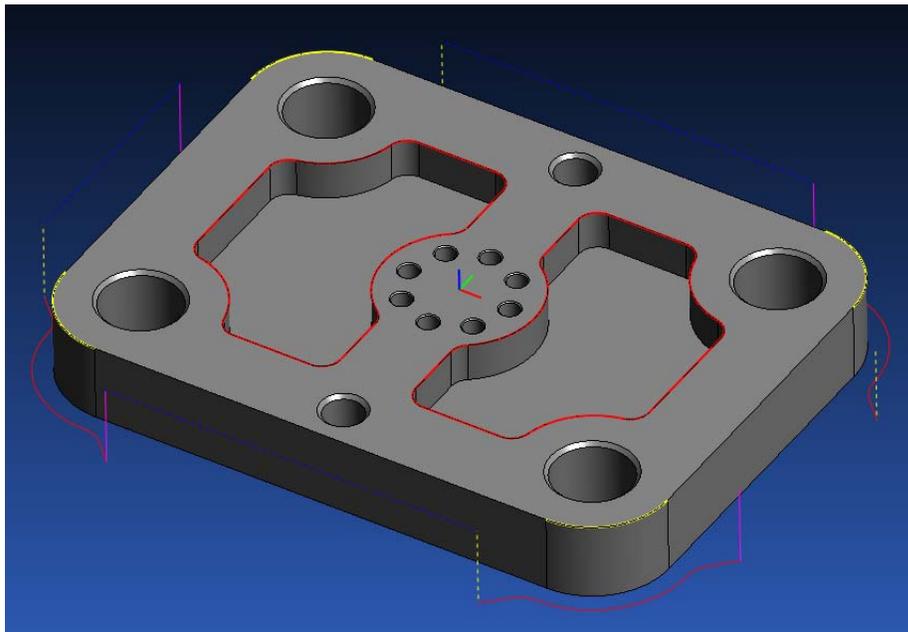
15 Repeat this process for 3 the other Corners.



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

17 The **Tool Path Verification Options** will now Display. Make sure to check the Option for **Enable Verification on Solid Model**.

Your screen should appear as below:

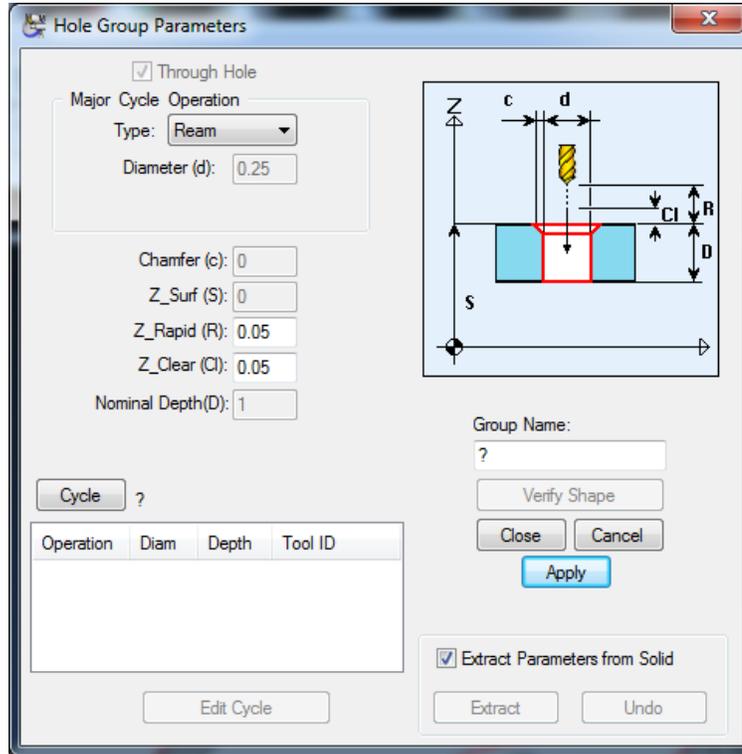


18 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

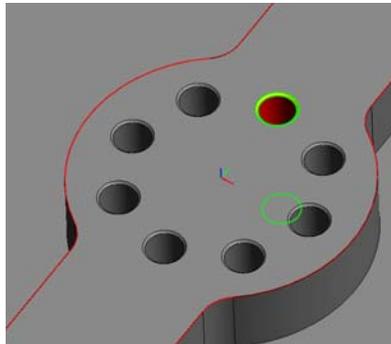
## Programming a Reamed Hole Group



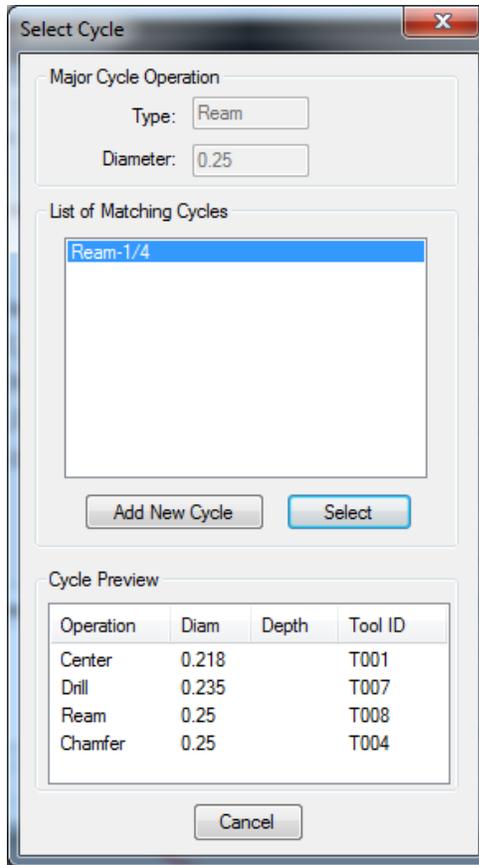
- 1 Select the **New Hole Group** icon.
- 2 Change the **Major Cycle Operation** type to Ream.
- 3 Click the **Extract Parameters from Solid** check box.
- 4 The **Hole Group Parameters** dialog should appear as follow.



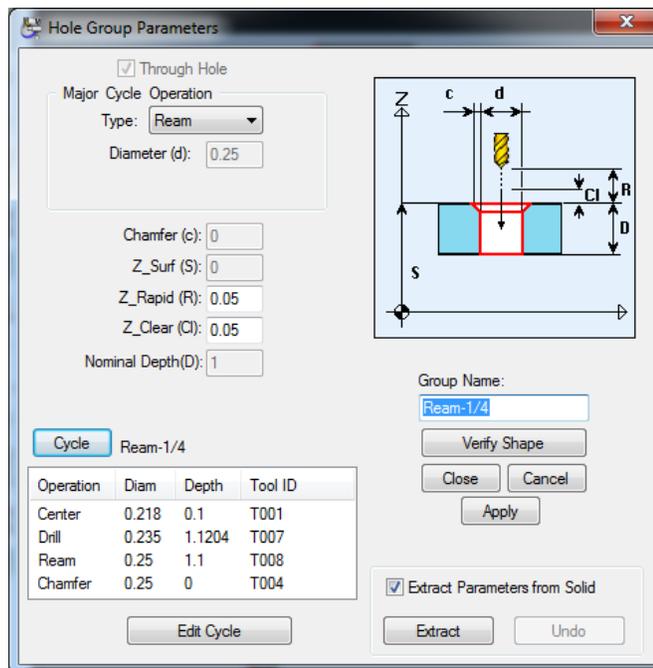
- 5 Select one of the Holes around the Center of the Part.
- 6 Click the **<Extract>** Button.



- 7 The **Select Cycle** dialog will appear; select the **Ream 1/4** Cycle.



- 8 The **Hole Group Parameters** dialog should appear as follow.

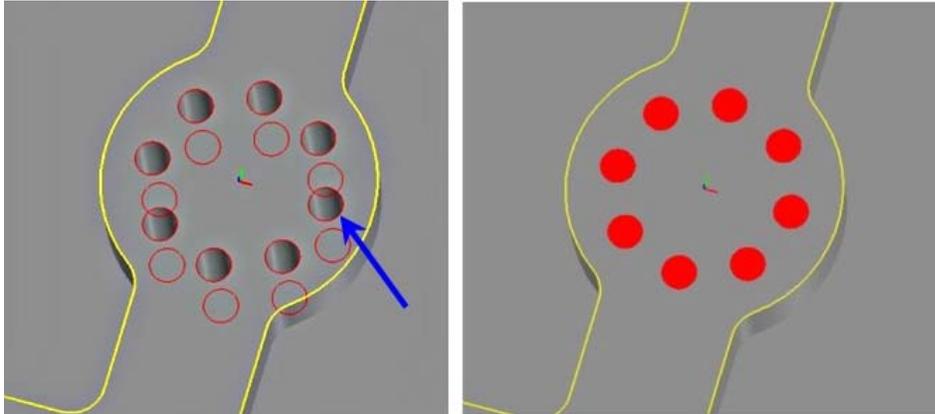


- 9 Click **<Apply>** and **<Close>**.



10 Select **Chain Holes on Solid Model** icon.

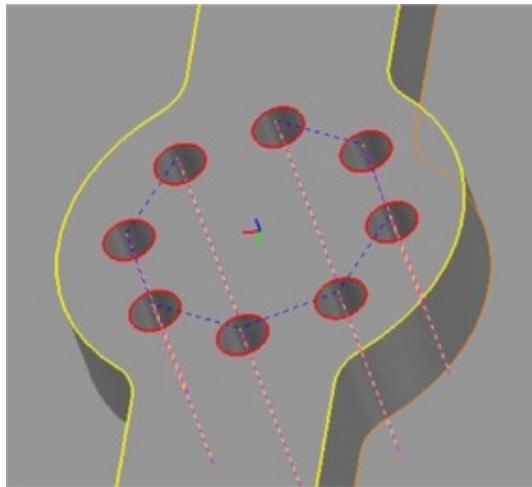
11 Select one of the holes around the Center of the Part. Note: Other holes with exact same feature are also highlighted.



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

13 The **Tool Path Verification Options** dialog will now Display. Make sure to check the Option for **Enable Verification on Solid Model**.

Your screen should appear as below:



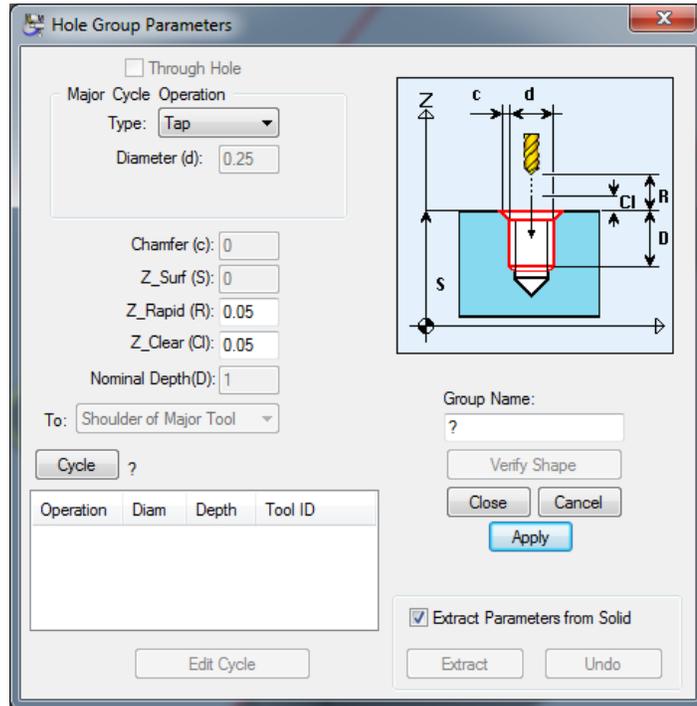
14 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

## Programming a Tapped Hole Group

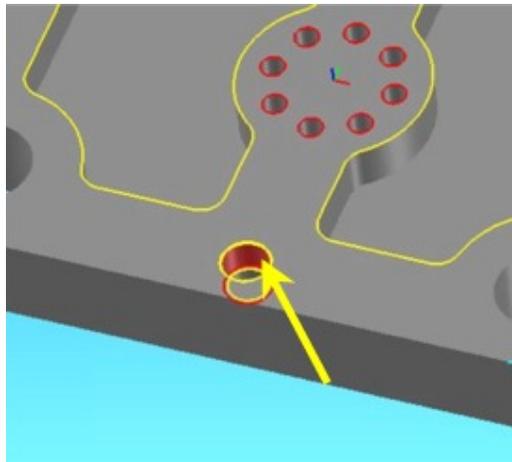


- 1 Select the **New Hole Group** icon.
- 2 Uncheck **Through Hole**
- 3 Change the **Major Cycle Operation** type to Tap.
- 4 Click the **Extract Parameters from Solid** check box.

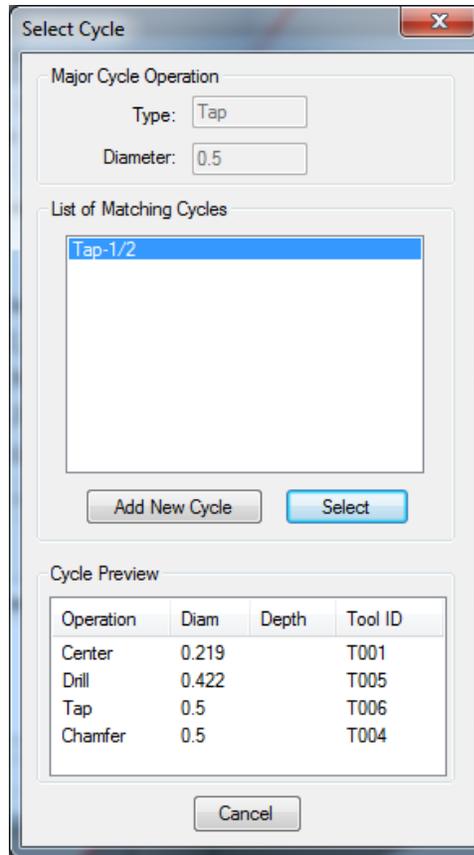
Your dialog should look like the following image:



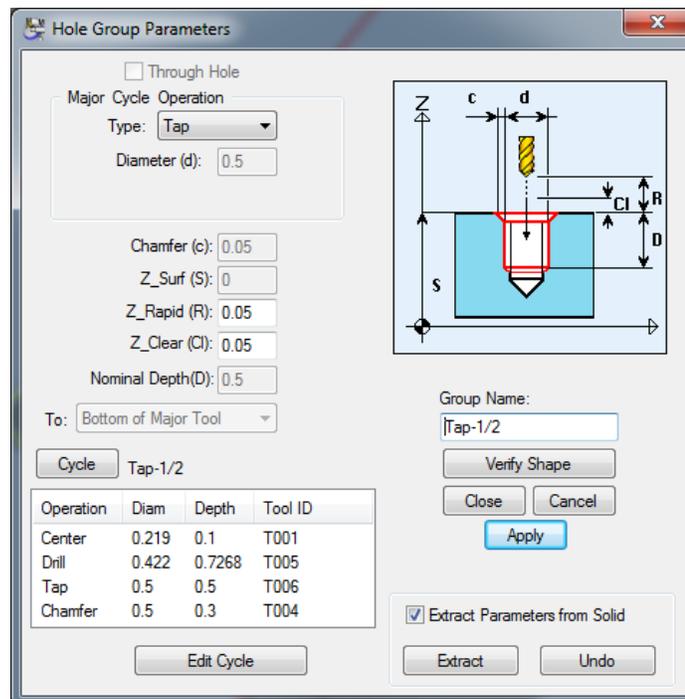
- 5 Select one of the two blind holes on the Part.
- 6 Click the **<Extract>** Button.



- 7 The **Select Cycle** dialog will appear; select the **Tap 1/2** Cycle.



- 8 The Hole Group Parameters Dialog should appear as follows:

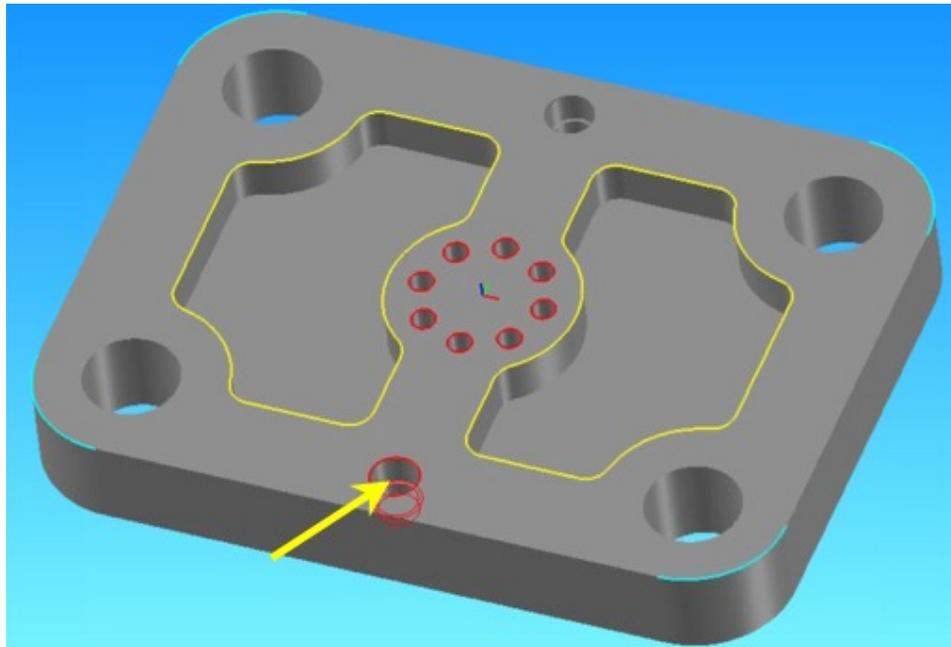


- 9 Select **<Apply>** and **<Close>**.

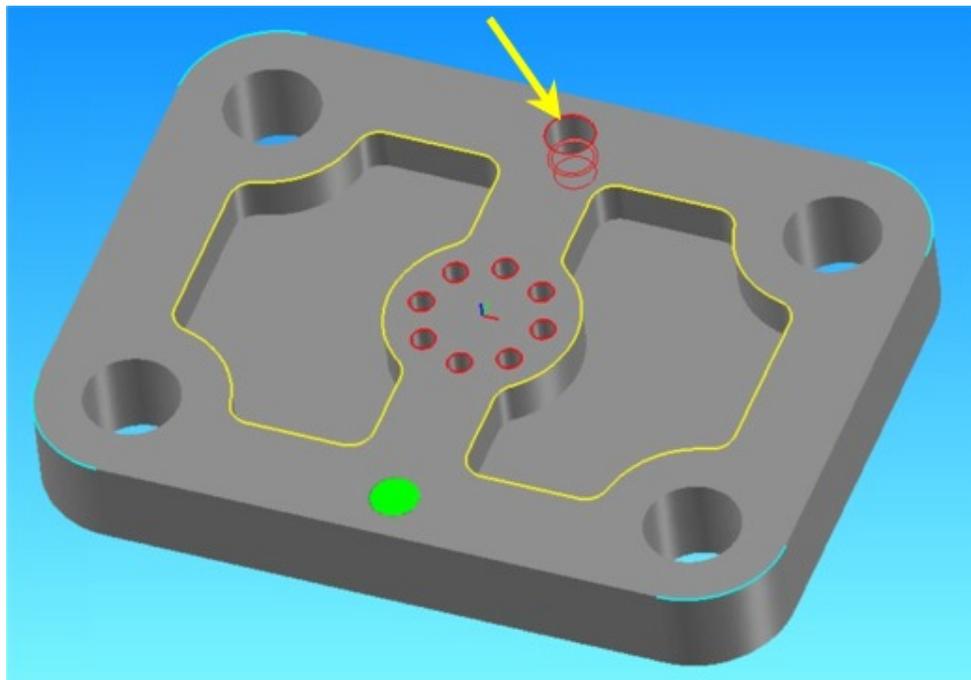


10 Select **Single Hole on Solid Model** icon.

11 Select The First Hole at the Bottom of the Part.



12 Then Select the Second Hole at the Top of the Part.

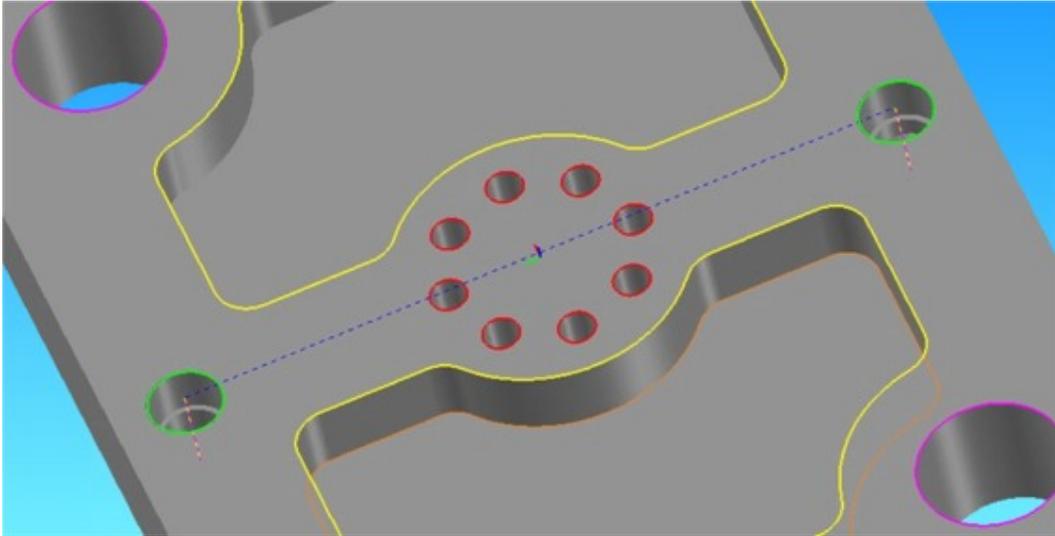


13 To verify the Profile Group, Select the **Verify Work Group Toolpath** icon.



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

Your screen should appear as below:



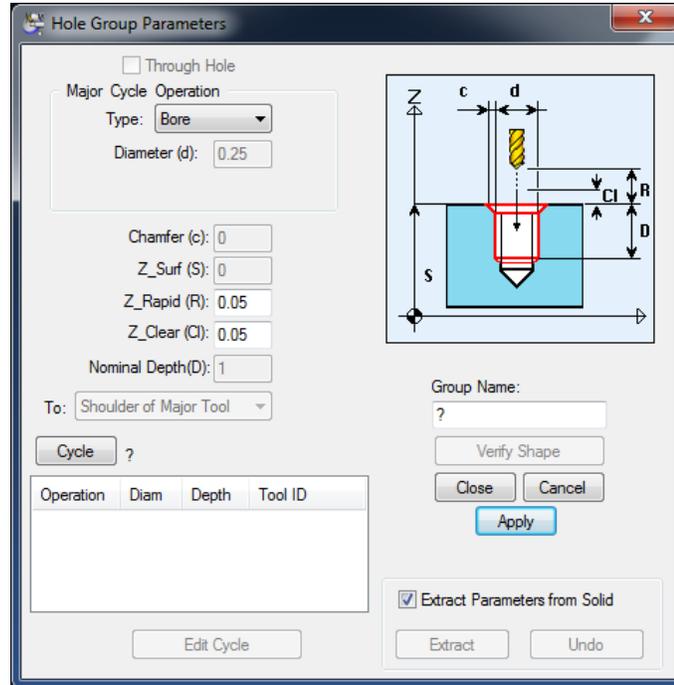
- 15 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

## Programming a Bore Hole Group

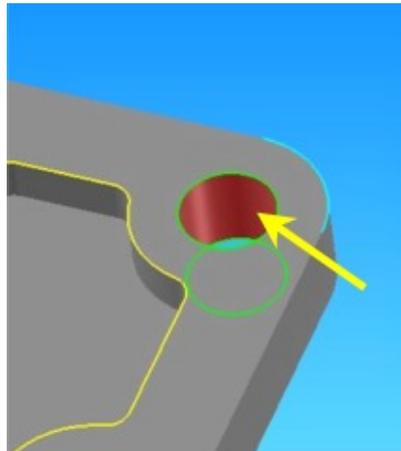


- 1 Select the **New Hole Group** icon.
- 2 Change the **Major Cycle Operation** type to Bore.
- 3 Click the **Extract Parameters from Solid** check box.

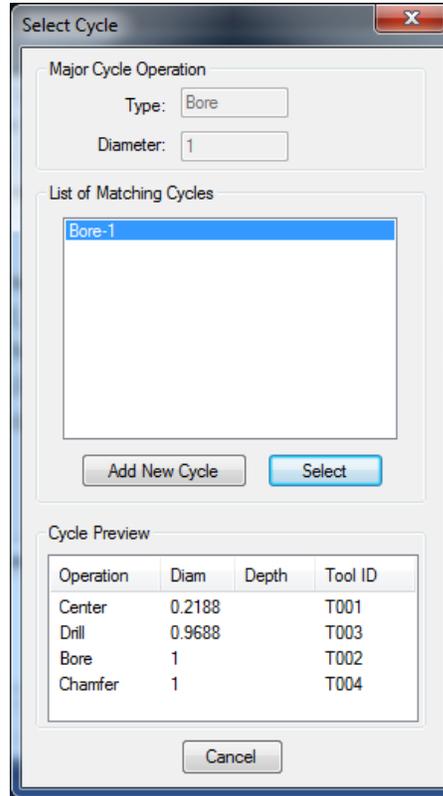
Your dialog should look like the following.



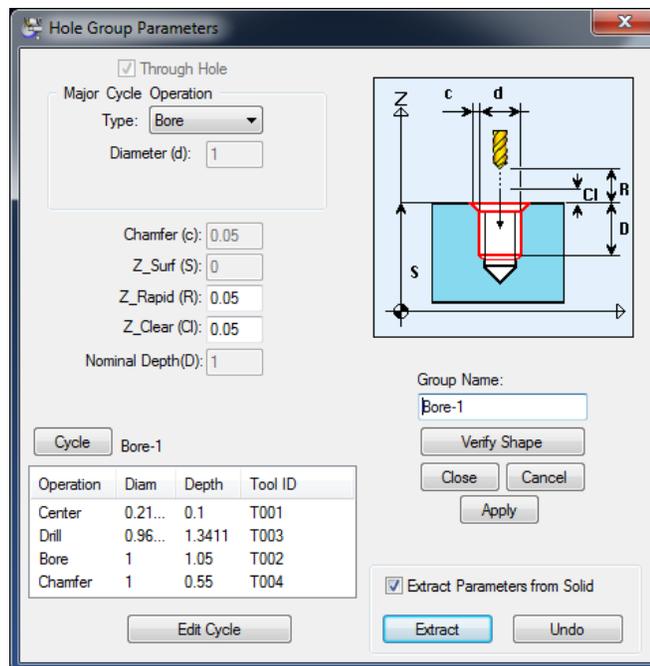
- 4 Select one of the four outside corners of the part.
- 5 Click the **<Extract>** Button.



- 6 When the **Select Cycle** dialog appears, select the **Bore-1** Cycle as shown below:



The Hole Group Parameters Dialog should appear as follows.



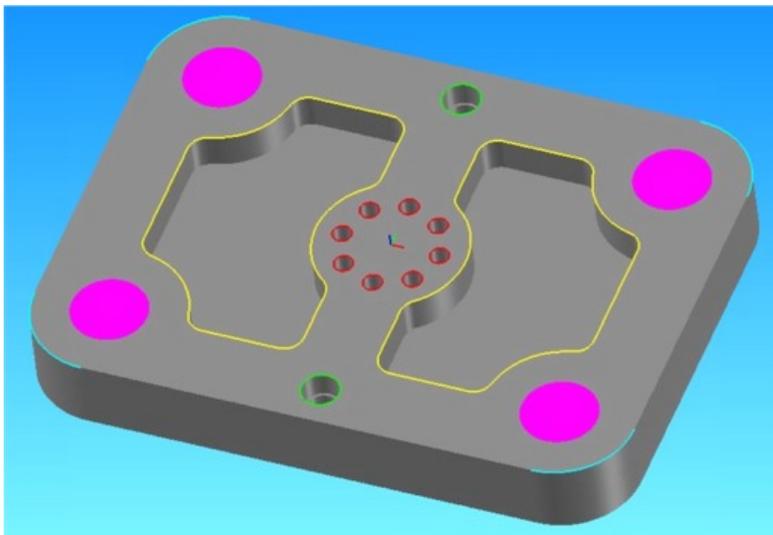
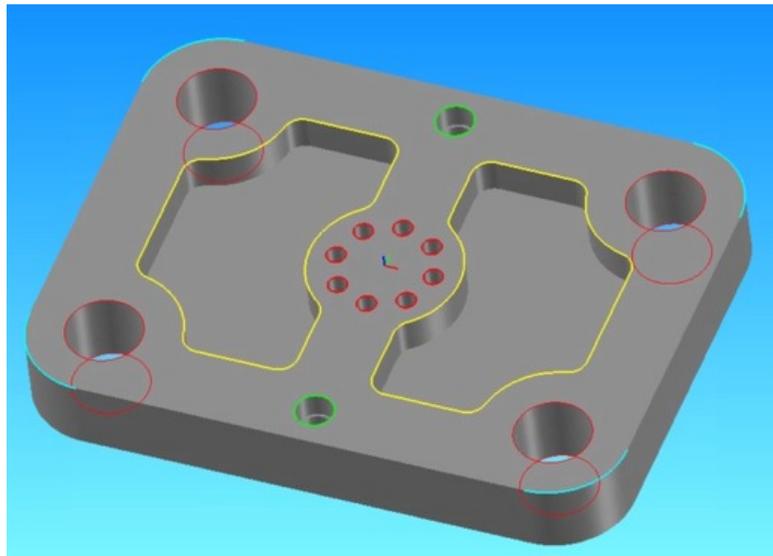
- 7 Select **<Apply>** and **<Close>**.



- 8 Select **Chain Holes on Solid Model** icon.
- 9 Select one of the 4 holes at the outside corners of the Part.

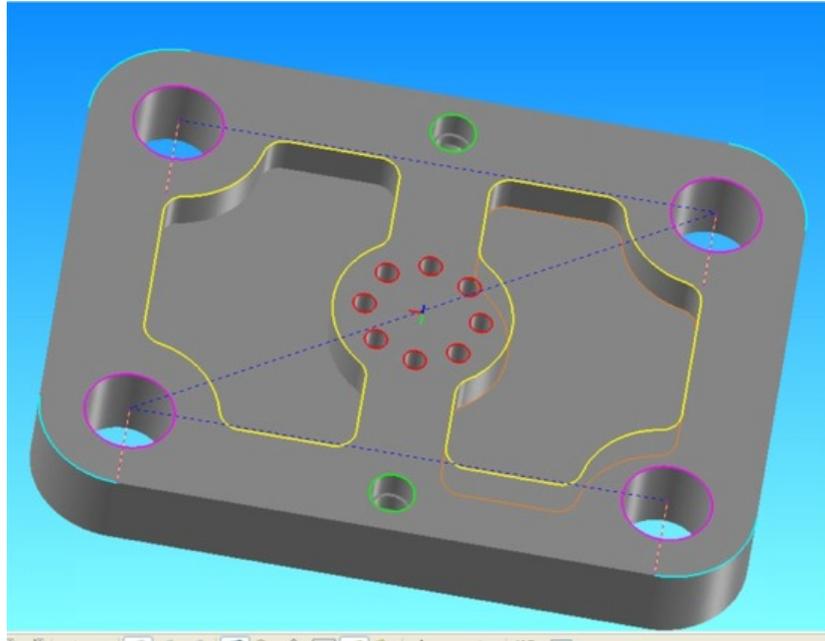


*Note: Other holes with exact same feature are also highlighted.*



- 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 for **Enable Verification on Solid Model**.

Your screen should appear as below:

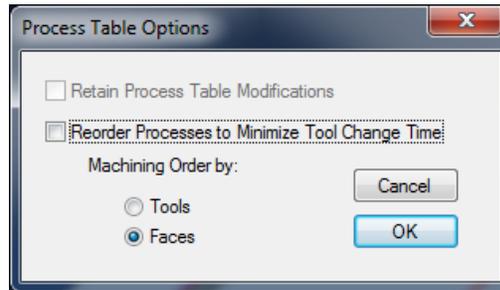


12 After Verifying the Tool Path, Click the **Hide Every Tool Path** icon to hide all verifications.

## Generating the Process Table



- 1 Choose the **Generate Process Table** icon.
- 2 Click **<OK>** for the Process Table Options.

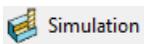


- 3 Your Process Table should look like the picture below.

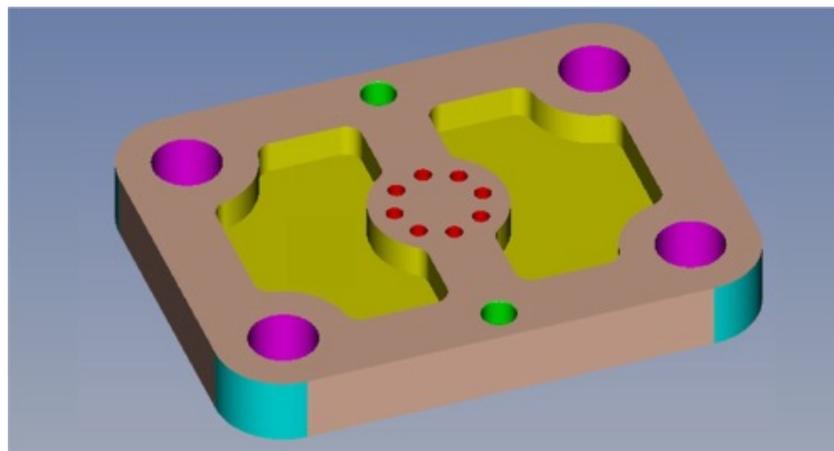
| Proc ID | Tool ID | Tool No. | Tool Name    | Proc Task     | Group  | Face  | Feed    | Speed   | Time(min) | Status |
|---------|---------|----------|--------------|---------------|--------|-------|---------|---------|-----------|--------|
| P01     | T009    | 9        | End Mill_1/2 | Pocket Finish | Pocke  | Front | 3.7upm  | 946rpm  | 19.00     | ✓      |
| P02     | T009    | 9        | End Mill_1/2 | Contour Finis | Profil | Front | 3.7upm  | 946rpm  | 6.40      | ✓      |
| P03     | T001    | 1        | Center_7/32  | SPOT FACE     | Ream   | Front | 14.3upm | 2776rpm | 0.24      | ✓      |
| P04     | T007    | 7        | Drill_15/64  | DRILL         | Ream   | Front | 14.3upm | 2591rpm | 0.73      | ✓      |
| P05     | T008    | 8        | Ream_1/4     | BORE,DWELL    | Ream   | Front | 3.7upm  | 911rpm  | 5.59      | ✓      |
| P06     | T001    | 1        | Center_7/32  | DRILL         | Tap-1  | Front | 14.3upm | 2776rpm | 0.03      | ✓      |
| P07     | T005    | 5        | Drill_27/64  | DRILL         | Tap-1  | Front | 12.7upm | 1439rpm | 0.14      | ✓      |
| P08     | T006    | 6        | Tap_1/2      | FLOAT TAP     | Tap-1  | Front | 19.7upm | 255rpm  | 0.13      | ✓      |
| P09     | T004    | 4        | Chamfer-1.25 | SPOT FACE     | Tap-1  | Front | 1.4upm  | 392rpm  | 0.60      | ✓      |
| P10     | T001    | 1        | Center_7/32  | DRILL         | Bore:- | Front | 14.3upm | 2776rpm | 0.07      | ✓      |
| P11     | T003    | 3        | Drill_31/32  | DRILL         | Bore:- | Front | 10.4upm | 626rpm  | 0.61      | ✓      |
| P12     | T002    | 2        | Bore_1       | BORE          | Bore:- | Front | 2.6upm  | 756rpm  | 3.79      | ✓      |
| P13     | T004    | 4        | Chamfer-1.25 | SPOT FACE     | Bore:- | Front | 1.4upm  | 392rpm  | 2.01      | ✓      |

## 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:



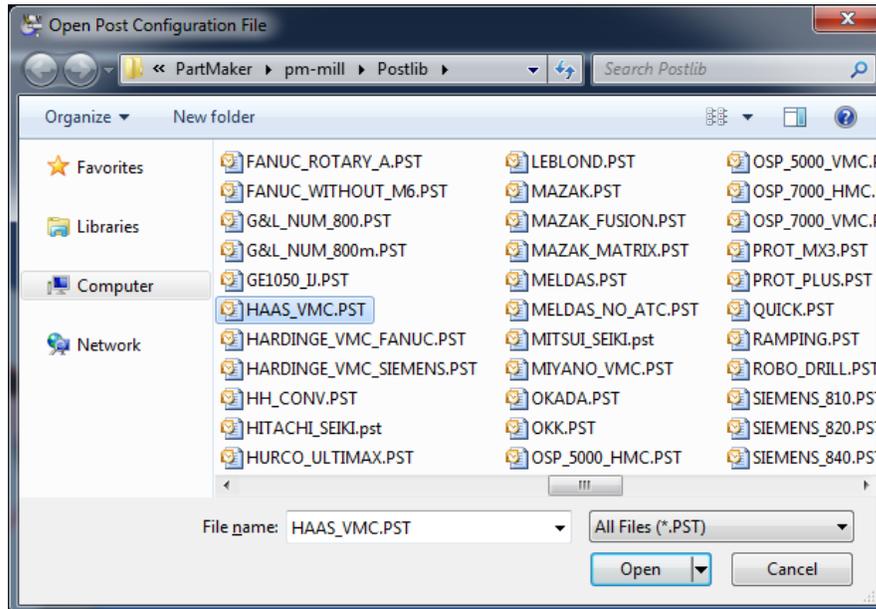
- 1 Choose the **Simulation** icon to watch process run and see the finished part.
- 2 Click the **Show Finished Part** icon to view the completed part:



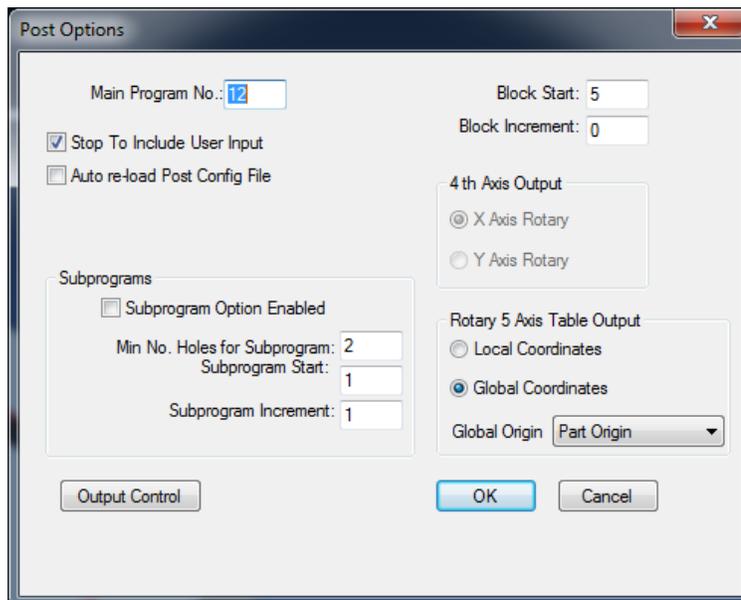
## Generating the NC Program



- 1 Choose the **Open Config Post File** icon.
- 2 Open the Post for the Appropriate Machine. (Haas Post Shown)

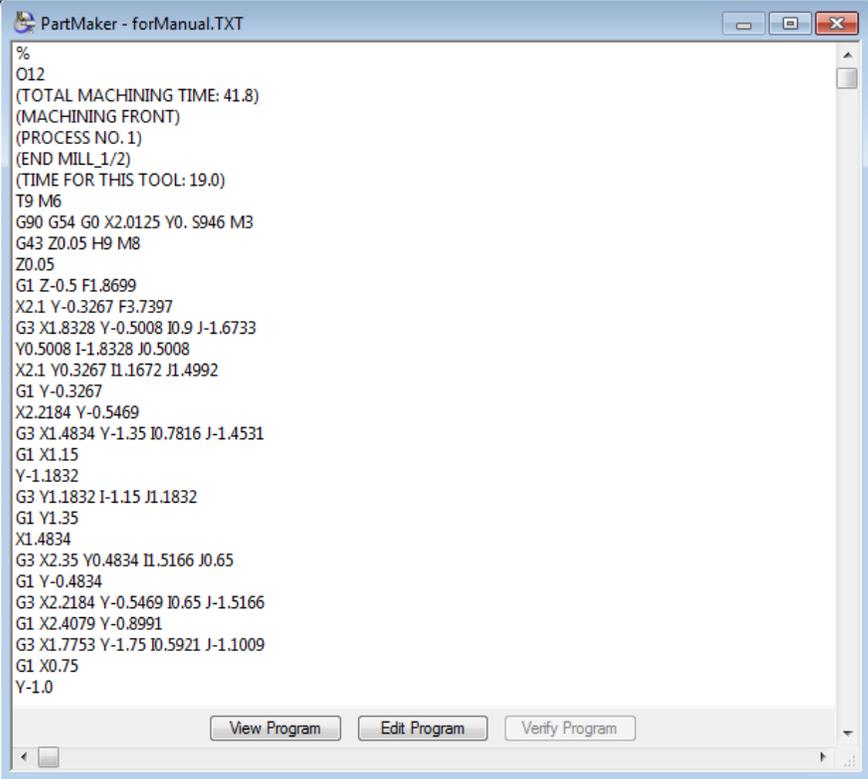


- 3 Choose the **Generate NC Program** icon. The Post Options dialog will display.



- 4 Review Post Options and make changes as needed.
- 5 Click **<OK>** to display the Save Program File dialog.

- 6 Enter a name for your file and click **<Save>** to generate the NC program.  
You will see your program as shown below:

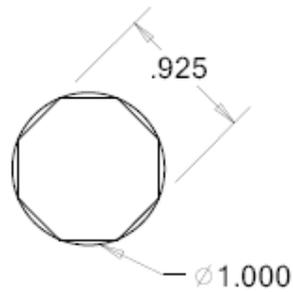
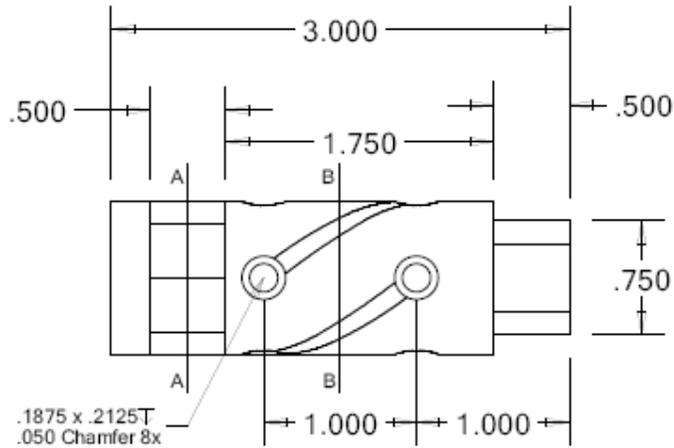


```
%  
O12  
(TOTAL MACHINING TIME: 41.8)  
(MACHINING FRONT)  
(PROCESS NO. 1)  
(END MILL_1/2)  
(TIME FOR THIS TOOL: 19.0)  
T9 M6  
G90 G54 G0 X2.0125 Y0.5946 M3  
G43 Z0.05 H9 M8  
Z0.05  
G1 Z-0.5 F1.8699  
X2.1 Y-0.3267 F3.7397  
G3 X1.8328 Y-0.5008 I0.9 J-1.6733  
Y0.5008 I-1.8328 J0.5008  
X2.1 Y0.3267 I1.1672 J1.4992  
G1 Y-0.3267  
X2.2184 Y-0.5469  
G3 X1.4834 Y-1.35 I0.7816 J-1.4531  
G1 X1.15  
Y-1.1832  
G3 Y1.1832 I-1.15 J1.1832  
G1 Y1.35  
X1.4834  
G3 X2.35 Y0.4834 I1.5166 J0.65  
G1 Y-0.4834  
G3 X2.2184 Y-0.5469 I0.65 J-1.5166  
G1 X2.4079 Y-0.8991  
G3 X1.7753 Y-1.75 I0.5921 J-1.1009  
G1 X0.75  
Y-1.0
```

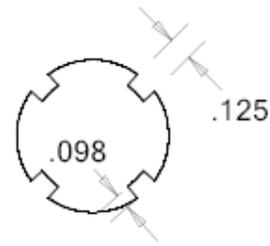
# Appendix E: The 4th-Axis Tutorial Part

The drawing below shows the part to be programmed.

**Inch Units**

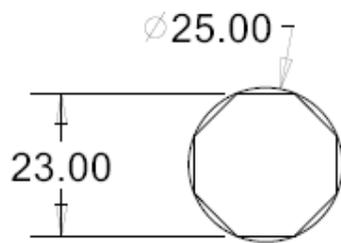
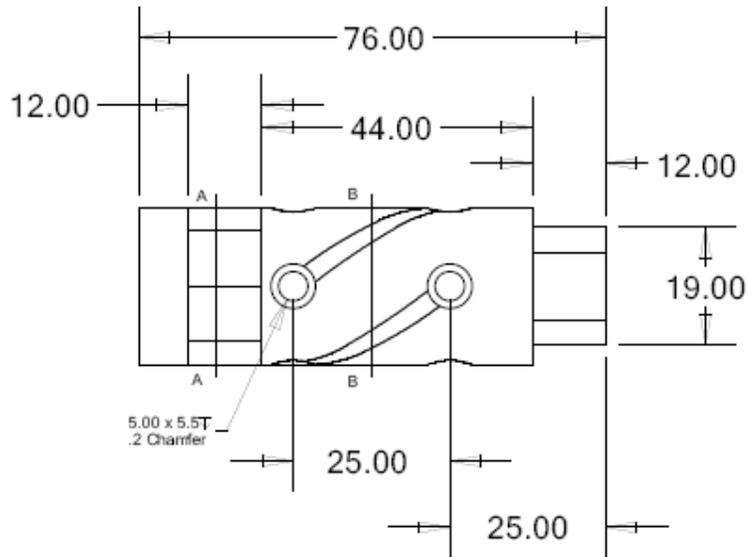


Section AA

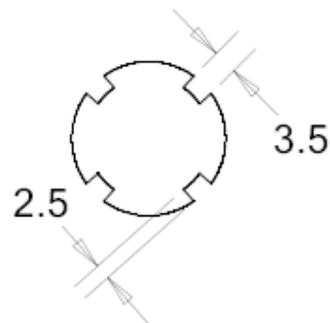


Section BB

**Metric Units**



Section AA

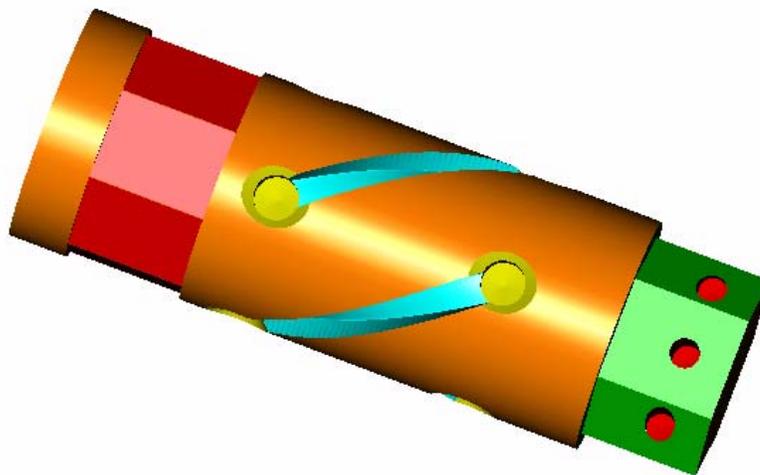


Section BB



**Important!** To complete this tutorial, you must license the PartMaker Production Milling Module (PMM).

When you complete this tutorial, you will have a part that looks like the picture on the page to follow.



## Introduction

The first part of any programming exercise with PartMaker's 4<sup>th</sup> Axis milling module is to decide which machining functions will be used.



**Important!** *The tutorial below is presented in both inch and metric units, depending on how you are working with PartMaker. The inch units of measure are given first, with the metric entries given in parenthesis ( ).*

For example you may see instruction that says: Enter an X value 1 (25), where 1 is 1 inch and 25 is 25 mm.

This tutorial is for use with PartMaker Version 7.8 and higher.

The machining functions used in this tutorial are as follows:

### Mill Polygon:

Use an end mill to create hexagonal flats on the end of the part.

### Mill Polygon:

Use an end mill to create a set of octagonal flats that begin 2.25" (56 MM) from the end of the part.

### Mill Diameter - Index:

Use a spot drill and a drill to create the eight holes arranged around the diameter of the part.

### Mill Cylinder:

Use an end mill to create four channels around the cylinder of the workpiece. Programming of these channels will use the simultaneous X and A axis feed capabilities of the machines' rotary axis.

### Mill 4 Axis Plane:

Use a spot drill and a drill to create the three .125" (3 MM) diameter holes .200" (5 MM) from the face of the part and arranged .300" (7.5MM) apart.

## Getting Started

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 tutorial for 4<sup>th</sup> Axis part can be found in the following directories:

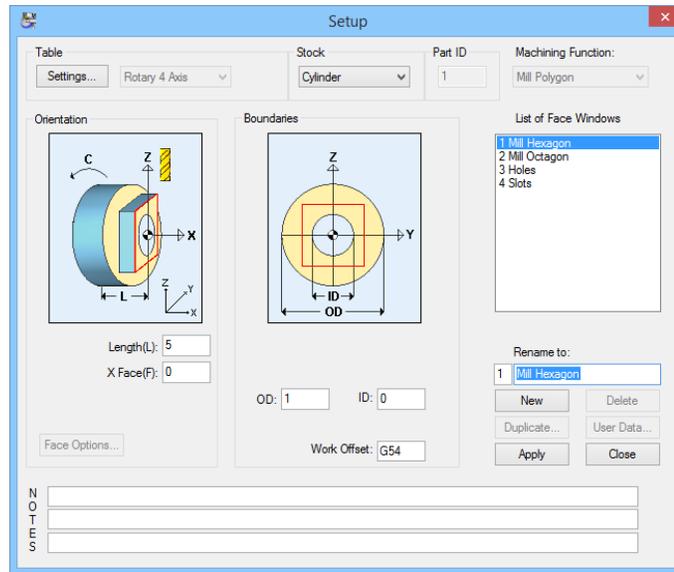
**Completed Job File:** C:\PartMaker\PM-Mill\4th-axis-Rotary.job

**Completed Tools File:** C:\PartMaker\PM-Mill\4th-axis-Rotary.tdb

**Completed Cycles File:** C:\PartMaker\PM-Mill\4th-axis-Rotary.cdb

## Starting PartMaker Mill

Start the PartMaker Milling application. 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. This dialog will be explained in greater detail starting on page E-9.

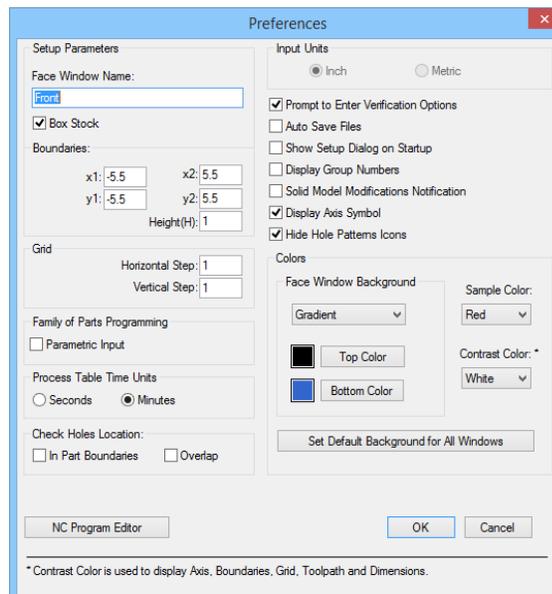


## Setting Preferences for the 4th-Axis Exercise

In this section, you will setup the necessary preferences for this exercise.

- 1 Choose **Preferences** from the **View** menu.
- 2 Check the **Display Axis Symbol** box. It may already be checked. Checking this box will display the axes you are working with in the lower left hand corner of each Face Window.
- 3 Make sure to **uncheck** the **In Part Boundaries** box. This will allow you to create part features on the edge of a cylinder.
- 4 Click **<OK>** to Exit the Preferences dialog.

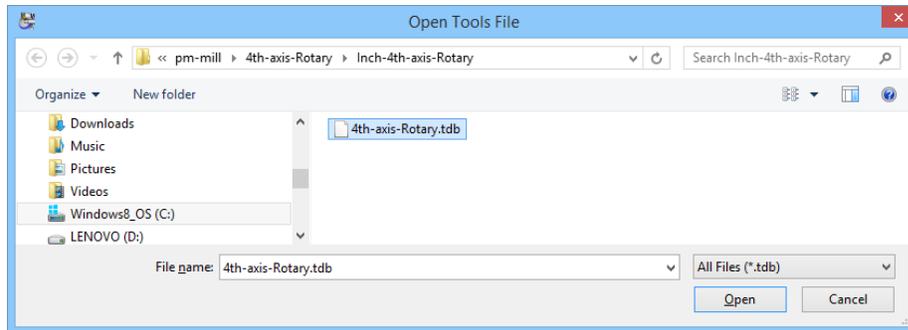
Your completed preferences dialog should appear as the one shown below:



## Opening the Tools file

The tool library for the 4th-Axis Programming Exercise has already been created.

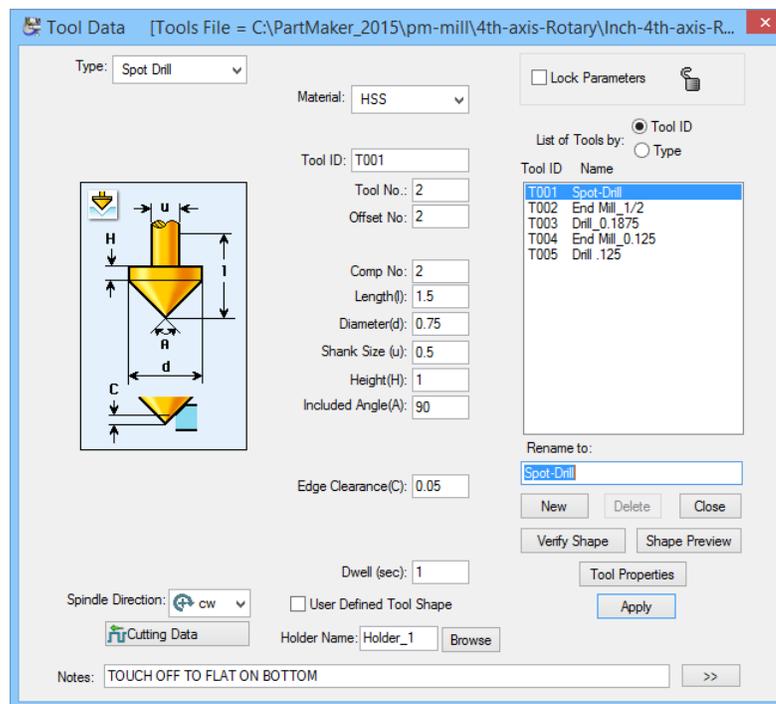
- 1 Choose **Open Tools File** from the **File** menu
- 2 Open the **4th-axis-Rotary.tdb (4th-axis-Rotary\_Metric.tdb)** from the C:\PartMaker\PM-Mill\4th-axis-Rotary directory as shown below:



- 3 Select the file to be opened and click the **<Open>** button to load the tools file.

Once you have loaded the tools database, you can view the tools that you will be using for this programming exercise. To do so

- 4 Choose **Tools** from the **ToolMinder** menu. This tools file contains five tools. The first of these appears below:

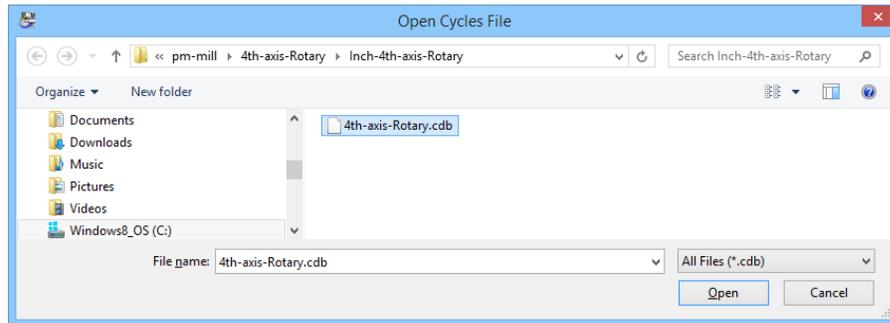


- 5 Click the **<Close>** button to close the **Tools** dialog.

## Opening the Cycles file

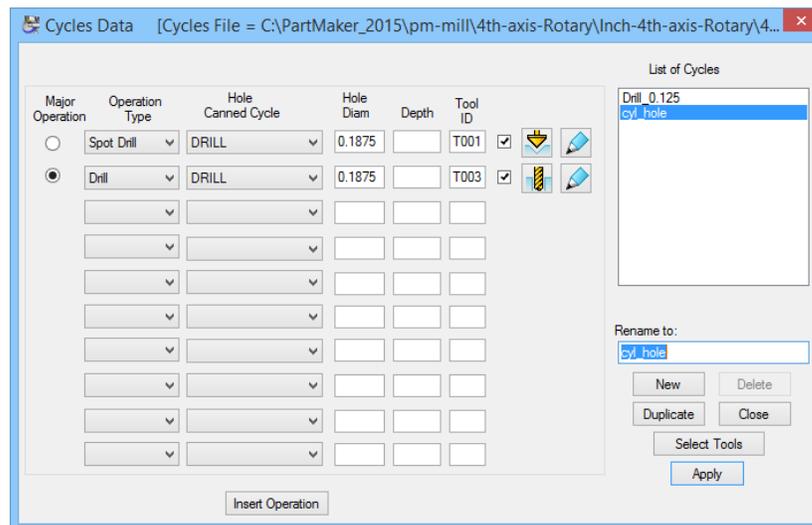
At this point, you should be familiar with the concept of Cycles in PartMaker. Cycles are sequences of repetitive operations that go into machining a hole. The cycles library for the 4th-Axis Programming Exercise has already been created.

- 1 Choose **Open Cycles File** from the **File** menu.
- 2 Open the **4th-axis-Rotary.cdb (4th-axis-Rotary\_Metric.cdb)** from the C:\PartMaker\PM-Mill\4th-axis-Rotary directory as shown below:



You can view the cycle that you will be using for this programming exercise. To do so:

- 1 Select the file to be opened and click the **<Open>** button to load the tools file.
- 2 Choose **Cycles** from the **ToolMinder** menu.

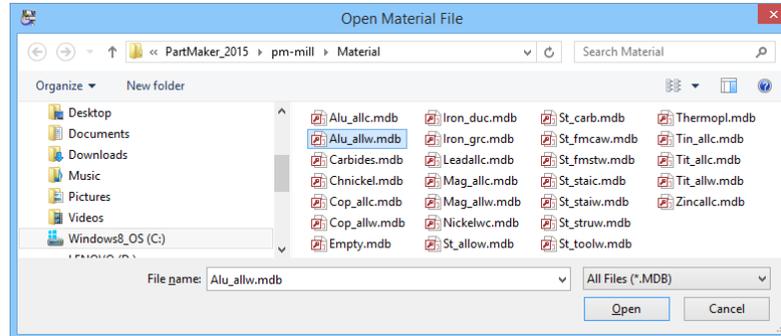


- 3 Click the **<Close>** button to close the **Cycles** dialog.

## Select a Material for the Part

The work piece in this exercise will be machined from **aluminum alloy wrought**. To load this material:

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



You have now loaded the tools, cycle, and material files that were previously developed for the 4th Axis 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.

## Creating the Flats Using Mill Polygon

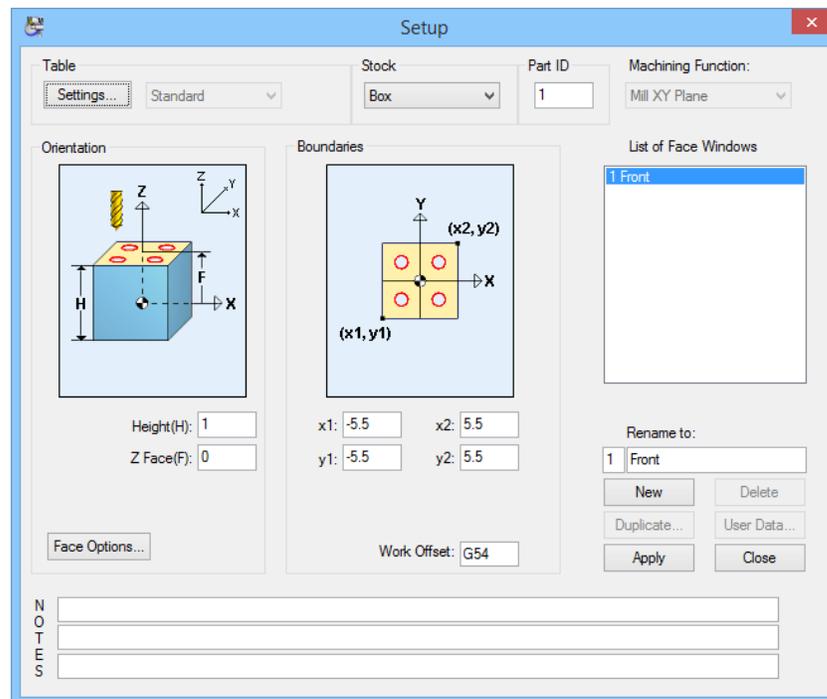
This section of the tutorial will instruct you in the steps necessary to create the hexagonal flats at the end of the part.

### Setting Up the Face Window

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

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

The default setup screen will appear as shown below.



- 2 From the **Table** drop down menu select **Rotary 4 Axis**.
- 3 From the **Stock** drop down menu select **Cylinder**.
- 4 From the **Machining Function** drop down menu select **Mill Polygon**.
- 5 Rename the current face window by clicking on **Front** in the **List of Face Windows:** field and type **Mill Hexagon** in the **Rename To:** field and click the **<Apply>** button.
- 6 Now enter the parameters listed below in the appropriate fields:

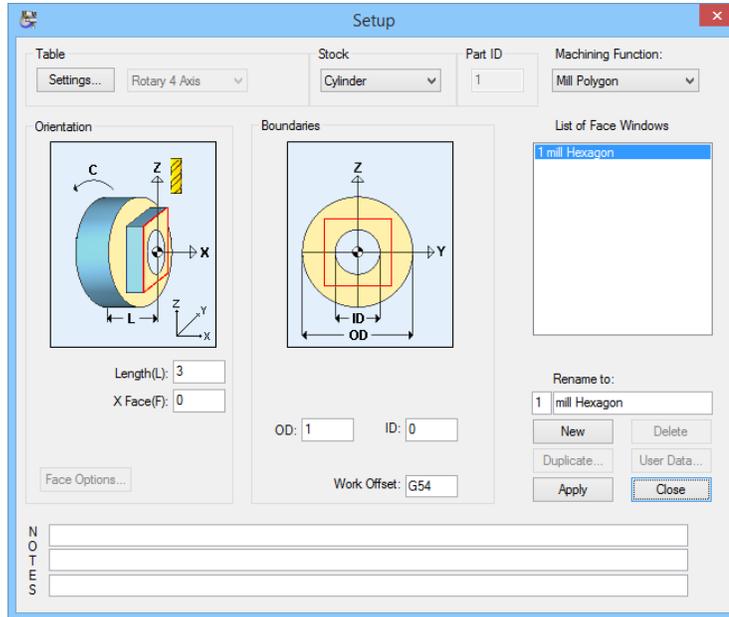
**Length (L):** 3.0 (76.0)

**X Face (F):** 0 (0)

**OD:** 1.0 (25.0)

**ID:** 0 (0)

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



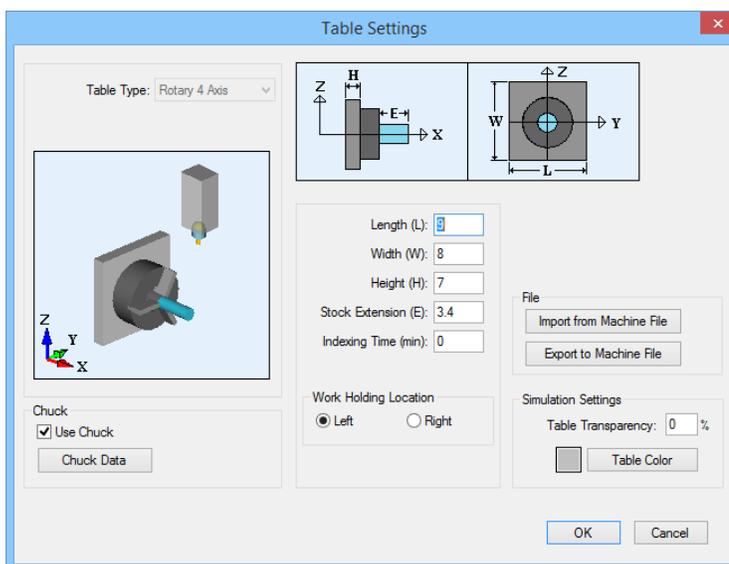
- 7 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current face window.

### Creating the Settings for the Rotary 4 Axis Table

Before exiting the setup dialog, you will define the settings of the rotary 4th Axis table being used on this part.

The **Settings** dialog is accessed for the **Table Type: 4 Axis Rotary** by clicking the **<Settings>** button in the Setup dialog. The **Settings** dialog is used to define the dimension and appearance of the machining table in PartMaker's 3D machining simulation for both block and cylinder stock.

A description of all of the fields for the **4 Axis Rotary Table Settings** is located in Chapter 2 of the Part Mill User Manual.



- 1 Click the **Settings** button in the **Setup** dialog to enter the **4-axis Rotary Table Settings** dialog.

- Enter the values into the following values into **4 Axis Rotary Table Settings** dialog:

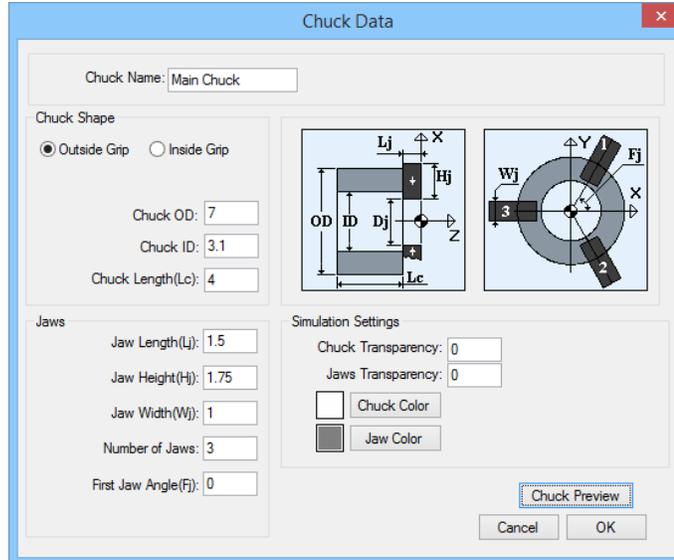
**Length (L):** 9 (230)  
**Width (W):** 8 (200)  
**Height (H):** 7 (175)  
**Stock Extension (E):** 8.25 (175)  
**Indexing time:** 0.1

Once you have entered the parameters as shown above, you will then create the chuck for this part. To do so:

- Check the **Use Chuck** box
- Click the **Chuck Data** button to enter the **Chuck Data** dialog
- Enter the parameters indicated below:

**Chuck OD:** 7.0 (175)      **Chuck ID: 3.1** 3.1 (78.0)  
**Chuck Length (Lc):** 4.0 (100)      **Jaw Length (Lj):** 1.5 (37.5)  
**Jaw Height (Hj):** 1.75 (43.75)      **Jaw Width (Wj):** 1.0 (25.0)  
**Number of Jaws:** 3      **First Jaw Angle:** 0

Once you are satisfied the **Chuck Data** dialog appears as shown below, click the **<OK>** button to return to the **4-Axis Rotary Settings dialog**



- Click the **<OK>** button to close 4-Axis Rotary Table Settings dialog and return to the setup dialog.
- Click the **<Close>** button to close the **Setup** dialog.

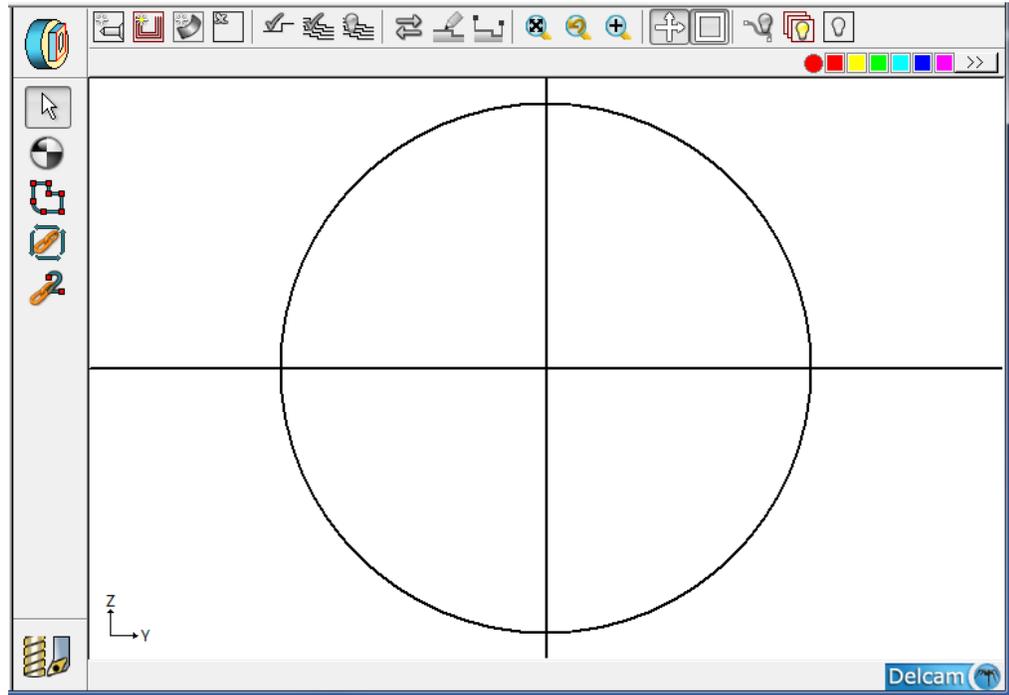


**Note:** If at any time the menu commands become "grayed out" and unusable, this indicates that the window you are working in has become inactive. This will be evident by the top bar of your window (where the name of the window is displayed) becoming gray. To reactivate the window, click anywhere within your face window.

## Creating Geometry in CAD Mode

In this section, you will create the geometry for the hexagonal flats.

- 1 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.
- 2 Choose **Show Boundaries** from the **View** menu. A circle should now appear on the screen that indicates the boundary of the one-inch diameter of the stock



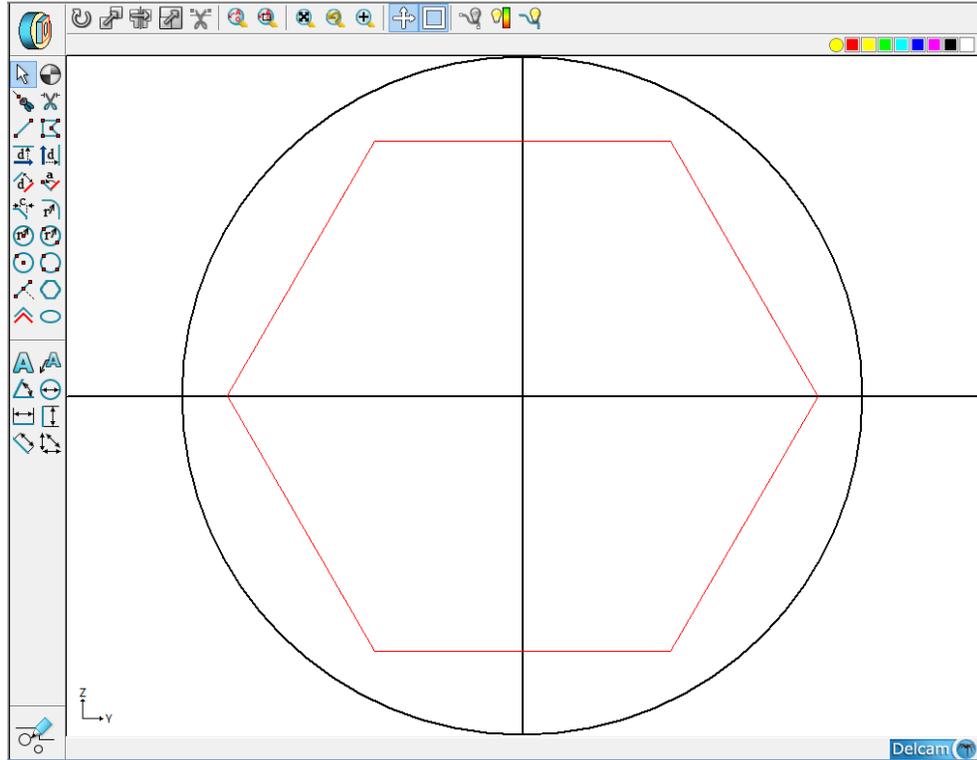
- 3 To create geometry, click on the **CAD/CAM switch** in the lower left-hand corner of the window. The icon will change its appearance from a drill bit to a pencil.  

- 4 Choose the **Hexagon** icon from the icon bar on the left side of the screen.  

- 5 After selecting this icon a field will open at the bottom of the screen prompting you to enter a flat to flat distance.

6 Type .750 (19.00) and press <Enter>

This will draw a Hexagon with a .750 (19) flat to flat distance as shown below:



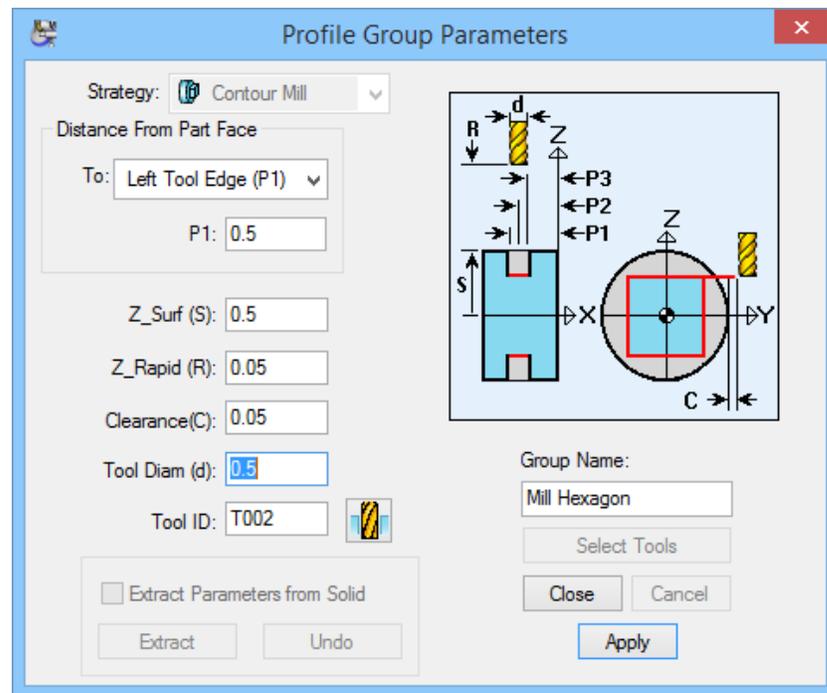
Next you will proceed to machine this hexagon in the **CAM** mode.

## Creating the Hexagonal Profile

In this section, you will define the profile to machine the hexagon on the end of the cylinder.



- 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 in Mill Hexagon.
- 4 Choose **Left Tool Edge** and enter 0.500 (12) for **P1**.
- 5 Enter 0.5 (12) for **Tool Diamter** Click on the **<Select Tools>** button. The endmill icon will appear to the right of the **Tool ID** box.



Now that you have completed setting up the profile group, take a moment to study this dialog, as it is unique to others you have encountered in PartMaker for 2 ½ axis milling.

**Z\_Surf:** The distance from the center of the cylinder to the edge of the diameter. PartMaker calculates this parameter automatically based on the value you enter for the OD of the stock in the **Setup** dialog. For the tutorial workpiece this is **1.0/2** or **.500 (25/2 or 12.5)**.

**P1 P2 or P3** Is the distance in from the face of the work piece to the location on the tool.

- 6 When you are satisfied that your **Profile Group Parameters** dialog matches the one above, click the **<Close>** button.

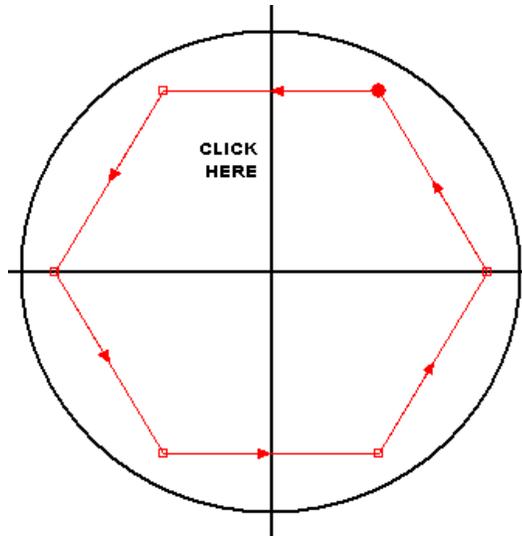
After closing you should see the group symbol for Mill Hex appear in the upper right of the screen. Notice the group symbol is an empty box because you have not yet assigned the part feature to the geometry.



The next step is to develop the tool path.



- 1 Select the **Chain Geometry** icon.
- 2 Move your cursor above the top line of the hex and slightly to the left of the vertical axis line as indicated by the words **Click Here**.
- 3 Click your mouse button and the screen should appear as shown below:



The arrows display the direction of the tool path. The flats will be machined in this order. If you wish to machine the flats in a different order, using the **Selection** icon, click anywhere on the tool path to select it. Choose **Reverse** from the **Edit** menu and the arrows displayed on the tool path will change direction.

### Saving Your Work



You have finished the first part face for the 4th axis tutorial. At this point save your work by choosing **Save Job File As** from the **File** menu. In the file name box type in a name and select **<Save>**.

The next step will be to program the .925" (23.00) octagonal profile located 2.25" (56.00) from the end of the workpiece. You will therefore have to create a second face for the octagon.

## Creating the Octagonal Profile

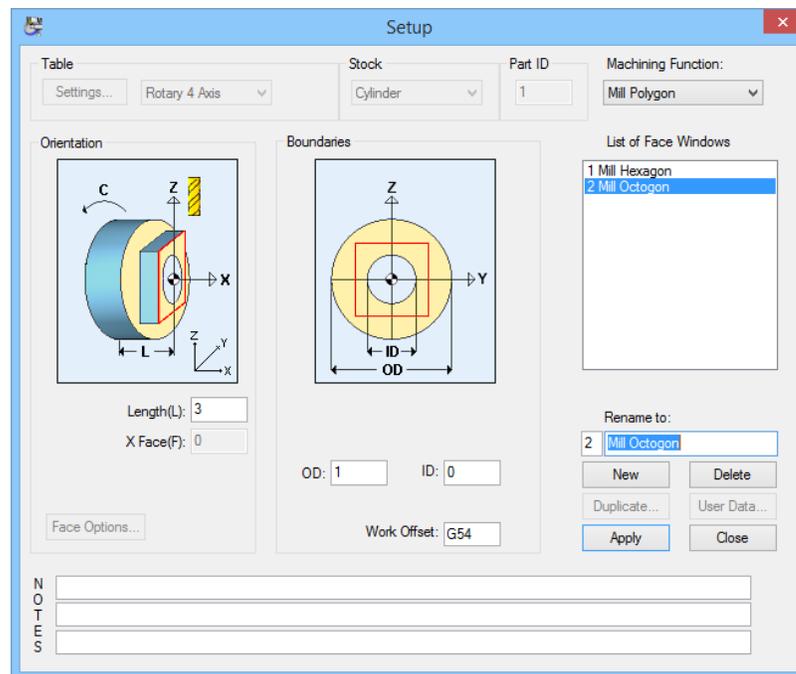
The next profile to create will be the .925" (23.00) octagonal profile located 2.25" (56.00) from the end of the workpiece.

### Setting Up the New Face Window

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

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box.
- 2 Click the **<New>** button to create a new face window.
- 3 Type Mill Octagon in the Rename To: field and press **<Apply>**.
- 4 From the **Machining Function** drop down menu select **Mill Polygon**.

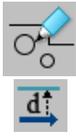
When completed, your completed **Setup** dialog should appear as shown below:



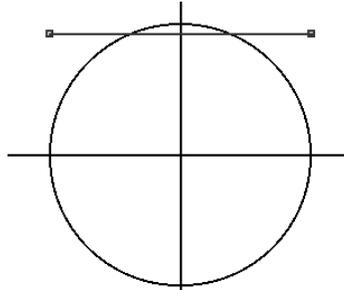
- 5 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current face window. Click the **<Close>** button to close the **Setup** dialog.
- 6 Switch to the new window you created by choosing **Mill Octagon** from the **Window** menu.
- 7 Select **Show Axes** from the view menu.
- 8 Select **Show Boundaries** from the view menu.

## Creating Geometry in CAD Mode

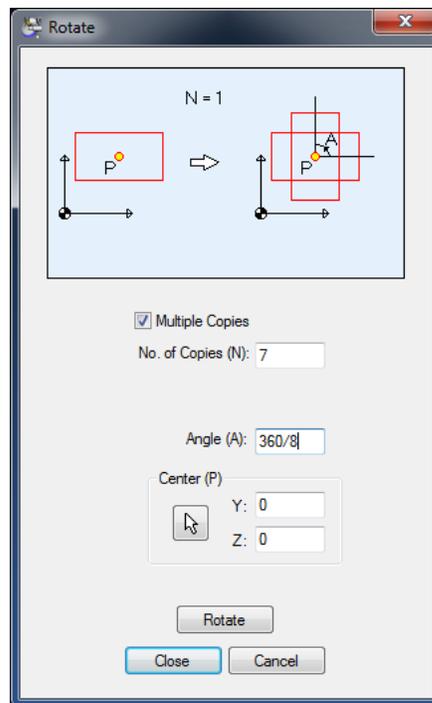
In this section, you will create the geometry for the eight holes.



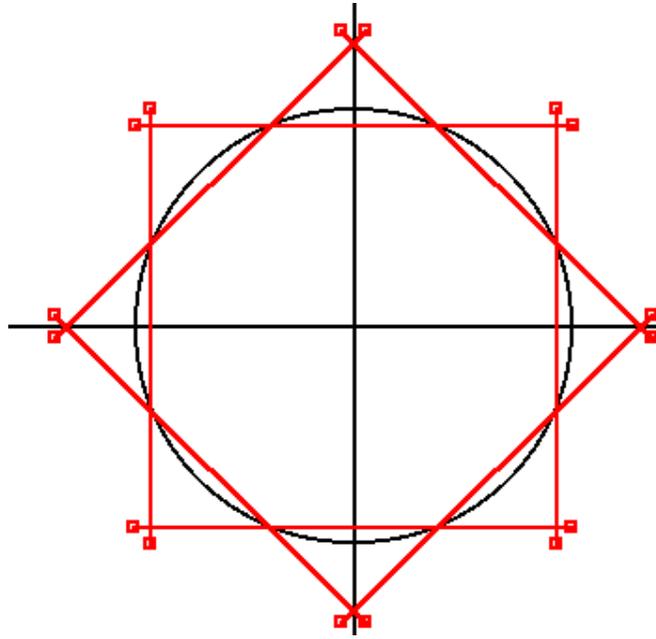
- 1 Go to **CAD** mode by clicking on the CAD/CAM switch.
- 2 Select the Line Parallel to the Horizontal Axis icon.
- 3 Type in **.925/2 (23/2)** in the field which appears in the lower left corner of the window and press **<Enter>**. You should see the following:



- 4 Choose the **Selection** icon and click the line you just created.
- 5 Select **Rotate** from the **Edit > Transform** menu to create seven more copies to complete the octagon.
- 6 Complete the **Rotate** dialog box as shown below and click **<Rotate>** and **<Close>**.



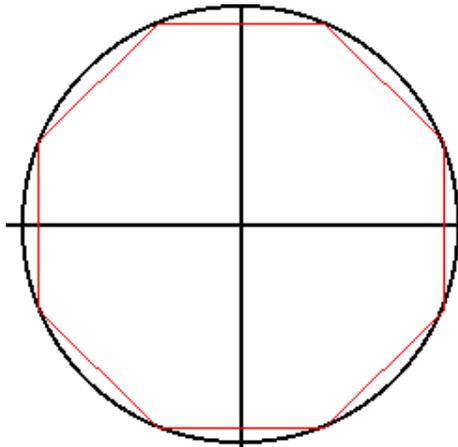
Your window should now appear as the one below:



Next, trim the ends of each line.



- 1 Select the **Remove** icon.
- 2 Click your cursor near the end of each line until only the octagon is visible as shown below:



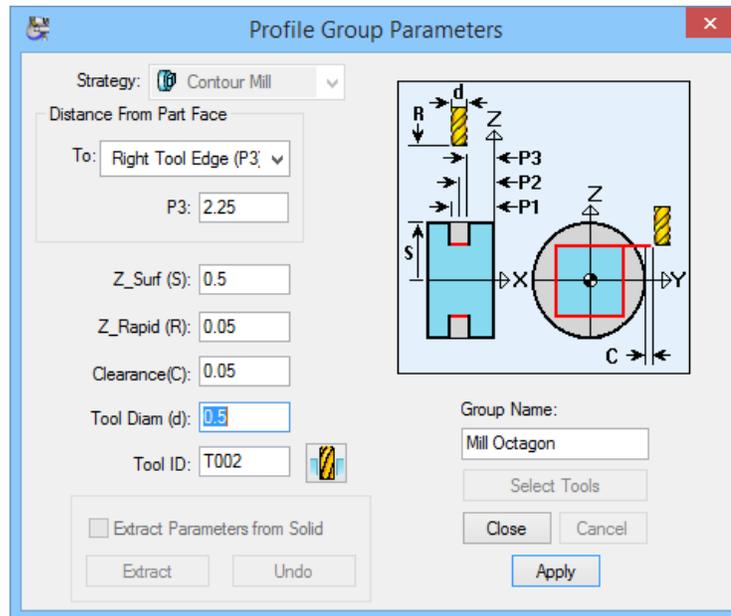
Save your work by pressing **<CTRL+S>**

## Creating the Octagonal Profile

In this section, you will define the profile to machine the octagonal profile.



- 1 Click the CAD/CAM switch such that it appears as drill bit. This will put you back into the CAM mode.
- 2 Choose New Profile Group from the Part Features menu.
- 3 In the Group Name: box type in Mill Octagon.
- 4 Choose **Right Tool Edge** and enter 2.25 (56.00) for **P3**.
- 5 Enter 0.5 (12.00) for **Tool Diameter**.
- 6 Click on the **<Select Tools>** button. The endmill icon will appear to the right of the **Tool ID** box.

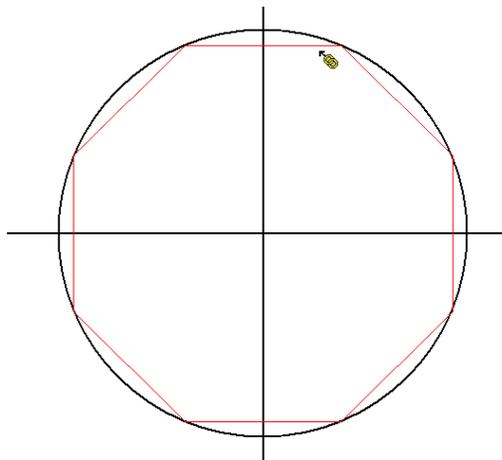


- 7 When you are satisfied that your **Profile Group Parameters** dialog matches the one above, click the **<Close>** button.

The next step is to develop the tool path.



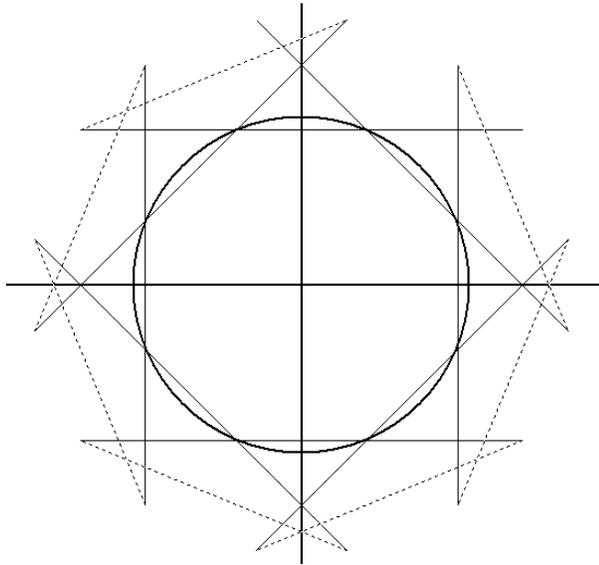
- 1 Select the **Chain Geometry** icon.
- 2 Click on the geometry along the top of the octagon to the right of the vertical axis line as shown below.



## Verifying the Tool Path

You can now verify the tool path of the operation you have just programmed.

- 1 Choose **Verify Work Group Tool Path** from the **Part Features** menu. You should see the following:



- 2 Choose **Hide Work Group Tool Path** from the Part Features menu.
- 3 Save your work with using **<CTRL+S>**.

## Creating Indexed Holes Using Mill Diameter Index

This section of the tutorial incorporates the concept of 'unwrapping' a cylinder and programming it as a flat surface.

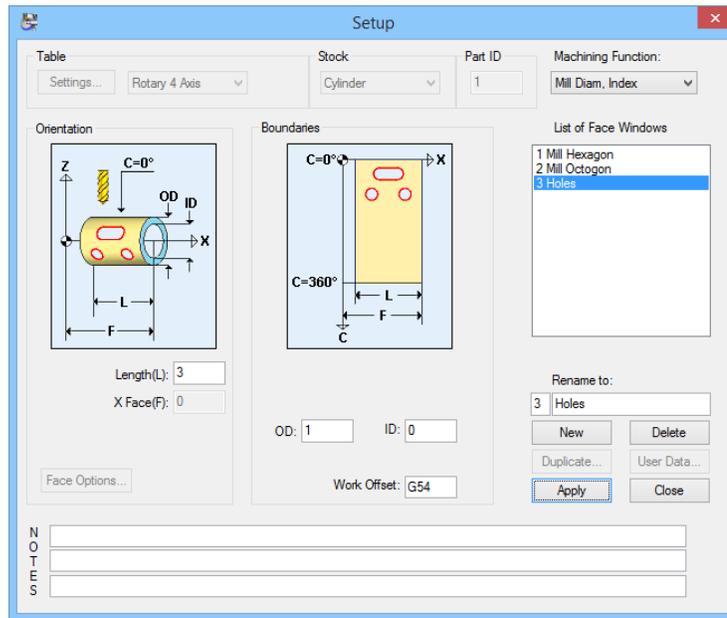
To understand this concept, imagine the outside diameter (OD) of the cylinder as a paper wrapper on a tin can. Slit that wrapper and peel it off the can and lay it front side up on the table. This is how PartMaker handles cylindrical (OD) machining. Instead of using X and Y coordinates, programming is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

### Setting Up the Face Window

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

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box.
- 2 Click the **<New>** button to create a new face window.
- 3 From the Machining Function drop down menu select Mill Diam, Index.
- 4 Type **Holes** in the **Rename To:** field and press **<Enter>**.

When completed, your **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 Switch to the new window you created by choosing **Holes** from the **Window** menu.
- 7 Select **Show Axes** from the view menu.
- 8 Select **Show Boundaries** from the view menu.

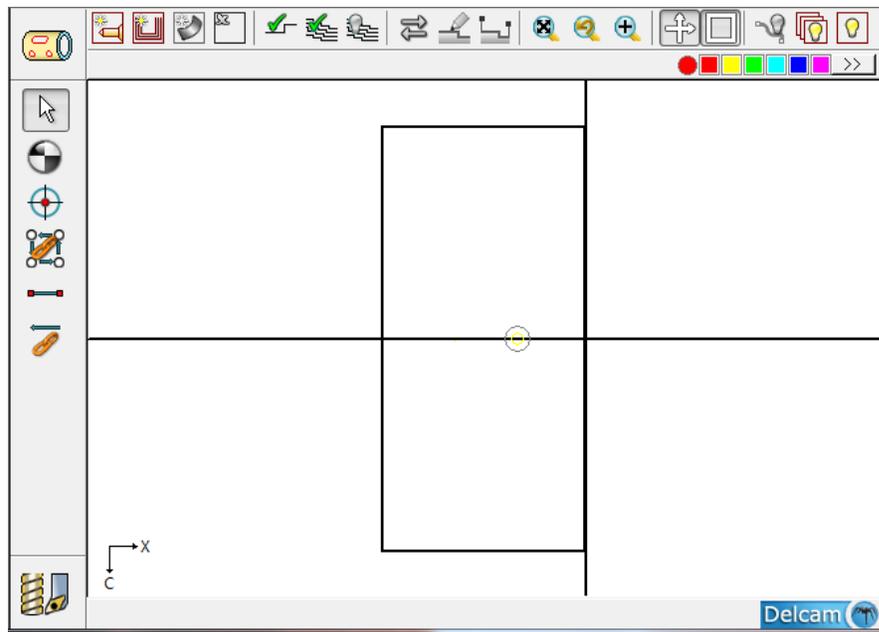
## Creating Geometry in CAD Mode

In this section, you will create the geometry for the eight holes.



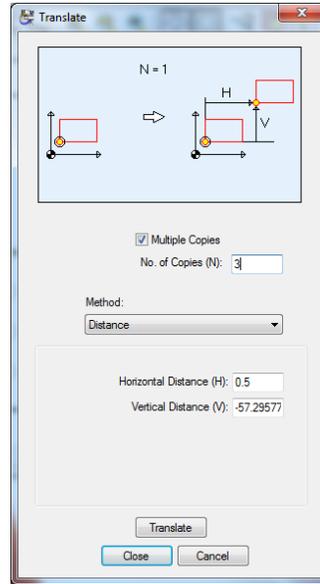
- 1 Go to **CAD** mode by clicking on the CAD/CAM switch.
- 2 Select the **Circle with a Known Radius and Center** icon. This will allow you to draw the first of the eight 0.1875 (5.00) diameter holes shown on the part drawing.
- 3 Enter a value for the radius of **.1875/2 (5.00/2)** in the field at the bottom of the face Window press **<Enter>**.
- 4 Select the **XC coordinates** icon from the Snap Mode icon bar.
- 5 In the lower left of the window you are prompted for the X axis location and the angle where the first hole begins on the circumference of the work. Enter **-1.0 (-25.00)** for X and **zero (0)** for C and press **<Enter>**. A hole should now appear on the negative side (left) side of the vertical axis and on the C-axis line.

Your window should look like the one below:



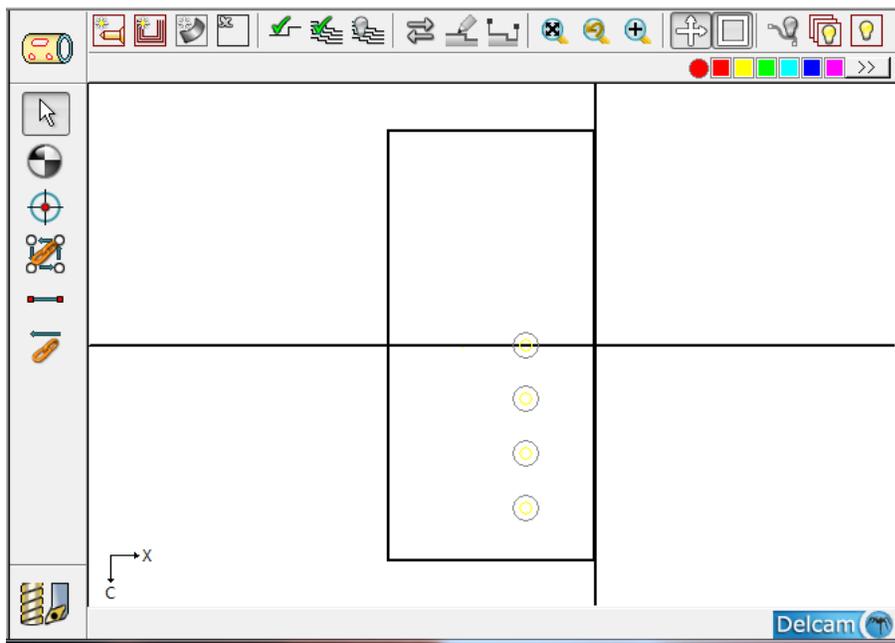
- 6 To draw the remainder of the holes choose the **Selection** icon and select the circle that you drew on your screen by clicking on it.

- 7 After the circle is highlighted, select **Translate** from the **Transform** submenu found under the **Edit** menu. Enter parameters as shown here:



The horizontal distance is set to zero (0) to show the holes will only be translated along the C axis in increments of 90 degrees.

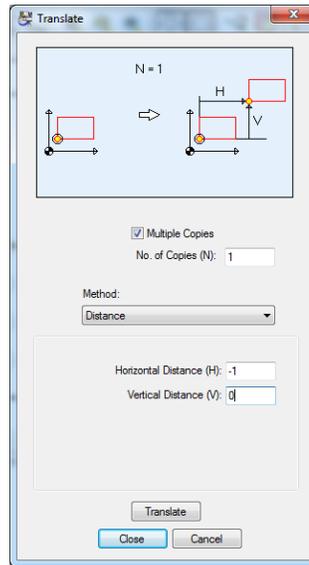
- 8 Once you have entered these parameters, click on **<OK>**. Your screen should now appear as shown below:



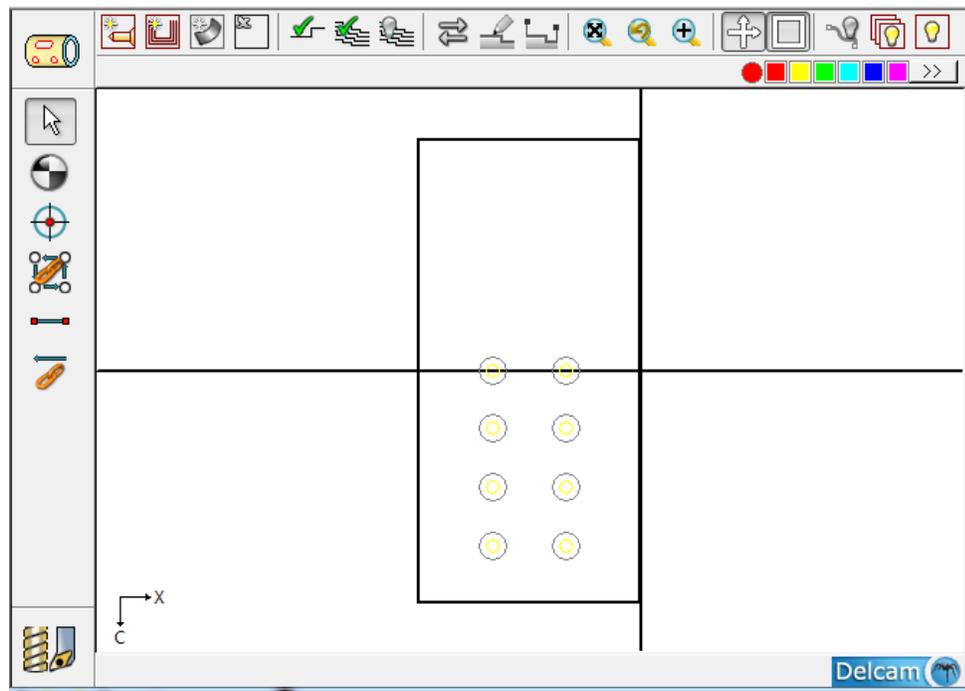
Next you will translate all four holes in the X-axis by 1.0" (25.00mm) in the negative direction.

- 1 Click anywhere within the window to deselect the holes you just translated.
- 2 Choose **Select All** from the **Edit** menu. All four holes should be highlighted.

- 3 Select **Translate** from the **Transform** submenu found under the **Edit** menu and enter the parameters as shown here. (metric users, enter -25 for X Distance)



- 4 Once you have entered these parameters, click on **<OK>**. Your screen should now appear as shown below:



- 5 Click anywhere in the **Face Window** to deselect the holes.
- 6 Now that the geometry for the holes has been created you will next define the tool path.

First, you must save your work. You can do so in one of two ways:

Choose the **Save** command from the **File** menu.



or

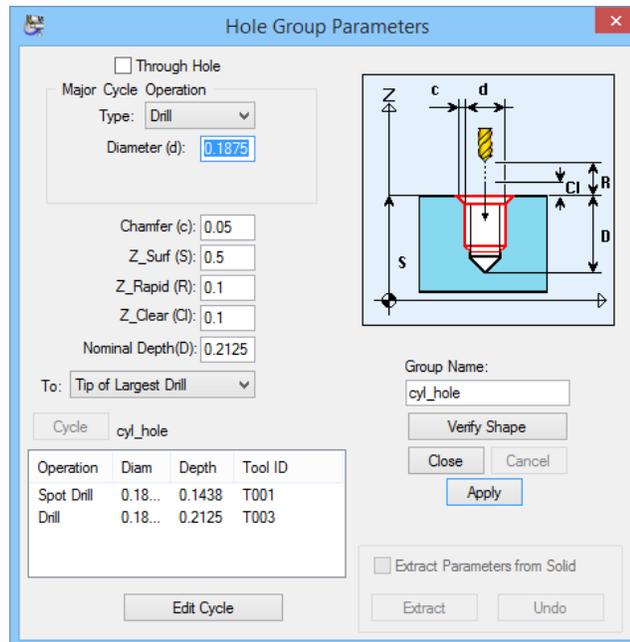
Hold down the **<Ctrl>** key and the **<S>** key simultaneously.

## Creating the Indexed Holes

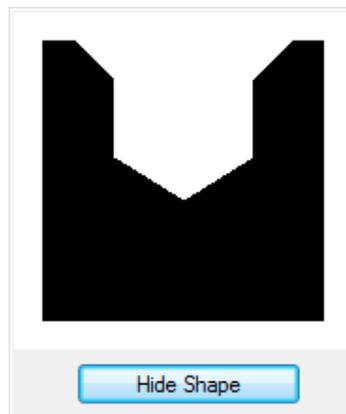
In this section, you will define the holes to be machined along the outside of the cylinder.



- 1 Click the CAD/CAM switch such that it appears as cutting tools.
- 2 Select New Hole Group from the Part Features menu.
- 3 Uncheck the **Through Hole** box.
- 4 From the **Type:** drop down menu, select Drill.
- 5 Enter a **Diameter** of 0.1875 (5.00)
- 6 Enter a **Chamfer** of 0.05 (1.0)
- 7 Enter a **Nominal Depth** of 0.2125 (5.00)
- 8 In the **To:** drop down menu, select **Tip of Largest Drill**
- 9 Click the **<Cycle>** button. In doing so, PartMaker found an existing cycle from the **Cycles** database where the major operation was a **.1875 (5.00)** diameter drill. The completed dialog should appear as shown below:



- 10 Click the **<Verify Shape>** button to see the shape of your hole

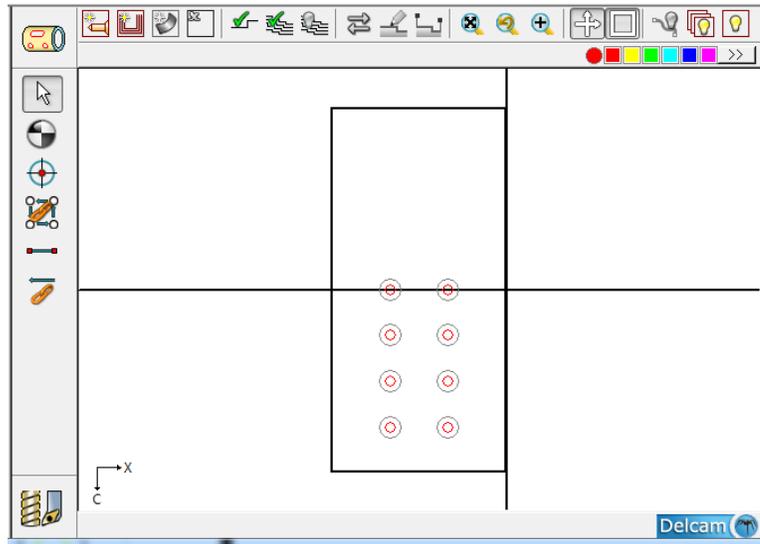


- 11 Once you are satisfied that your dialog and hole shape appear as the ones above, click **<Hide Shape>** and **<Close>** to exit the dialog.

You will specify the holes you want to machine.



- 1 Select the **Chain Circle** icon.
- 2 Click your cursor somewhere near the first hole you want to machine. In this case you will select the lower right hole in the window as shown below:



The holes you have chained should now be highlighted in the **Sample Color**.



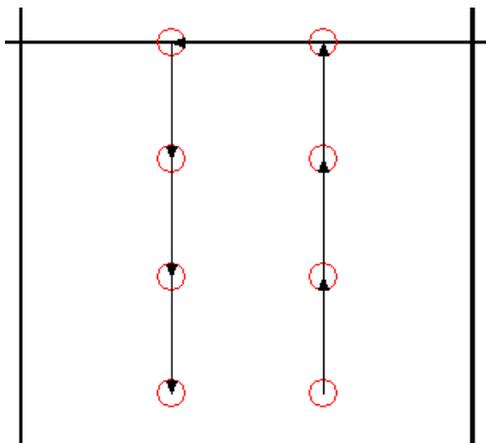
**Note:** If all the holes are not selected when performing the chaining operation described here, you may not have unchecked the **In Part Boundaries** box in the **Preferences** dialog. To do so select **Preferences** from the **View** menu.

### Verifying the Tool Path

You can now verify the tool path of the operation you have just programmed.



- 1 Click anywhere within the **Face Window** to un-highlight the holes.
- 2 Select **Verify Work Group Tool Path** from the **Part Features** to see the tool path displayed below:



- 3 When you are satisfied that your toolpath appears as the one shown, choose **Hide Work Group Tool Path** from the **Part Features** to hide the tool path displayed.



Now save your work by pressing **<CTRL+S>**. You have finished the third part face for the 4<sup>th</sup> axis tutorial. The next step will be to program the four .125" (3.5) slots.

## Creating Helical Channels Using Mill Cylinder

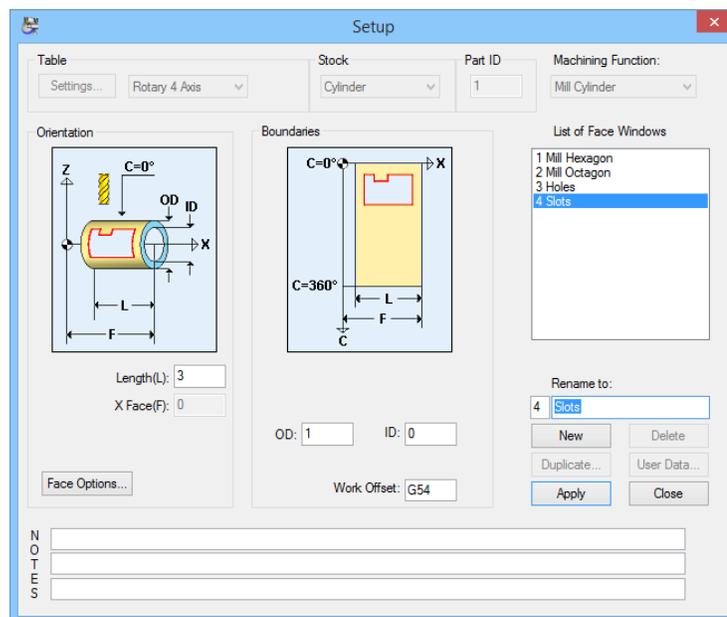
PartMaker's Mill Cylinder machining function supports cylindrical milling, the ability to simultaneously feed in the X and C axes. This machining function also incorporates the concept of 'unwrapping' a cylinder and programming it as a flat surface as discussed previously.

### Setting Up the Face Window

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

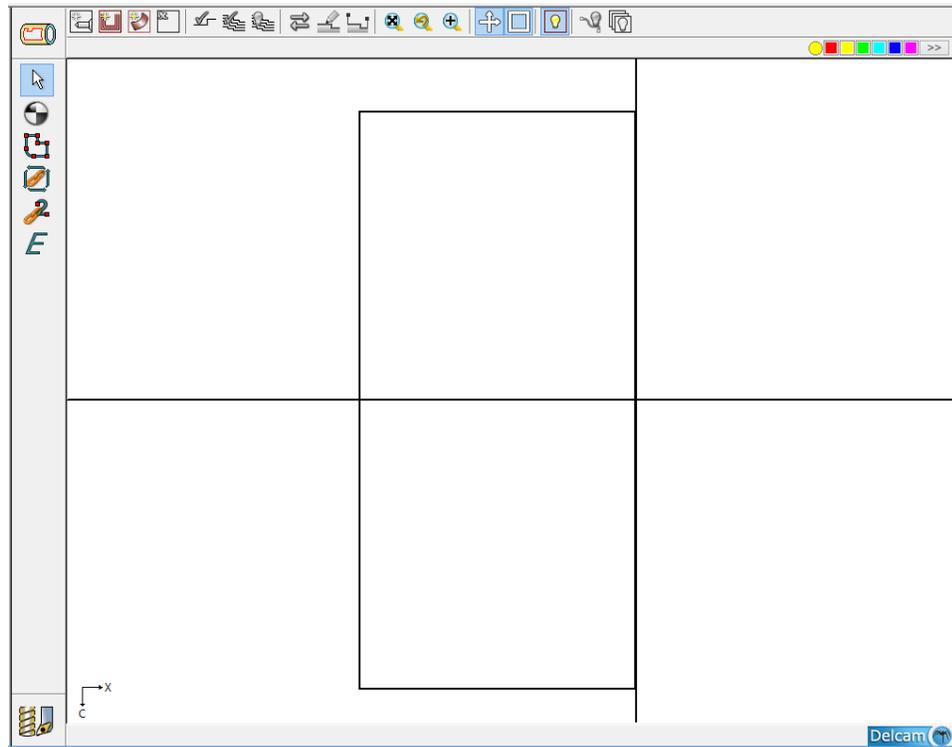
- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box.
- 2 Click the **<New>** button to create a new face window.
- 3 From the Machining Function drop down menu select Mill Cylinder
- 4 Type **Slots** in the **Rename To:** field and press **<Enter>**.

When completed, your **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 Switch to the new window you created by choosing **Slots** from the **Window** menu.
- 7 Select **Show Axes** from the **View** menu.

- 8 Select **Show Boundaries** from the **View** menu. Your screen should now appear as shown below:



### Creating Geometry in the CAD Mode

To quickly define geometry for the slots you can use the holes created in the previous section. Because the slots run between the centers of the holes, instead of drawing this geometry again, you will copy the hole geometry created in the previous face window to the one you are working with now.

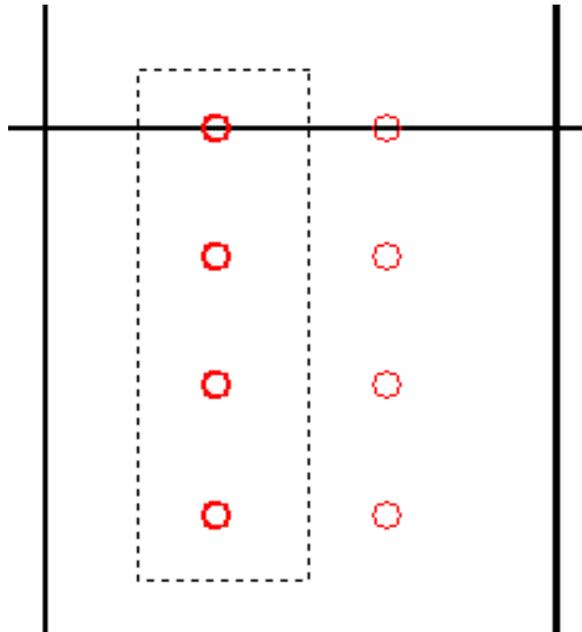


- 1 Select **Holes** from the **Window** menu.
- 2 Click the CAD/CAM switch to enter CAD mode
- 3 Choose **Select All** from the **Edit** menu. All the holes should now be highlighted.
- 4 Choose **Copy** from the **Edit** menu.
- 5 Return to the **Slots** face window by choosing **Slots** from the **Window** menu.
- 6 In CAD mode in the **Slots** face window, select **Paste** from the **Edit** menu.

Now you will shift one set of holes by 90 degrees.

- 1 Move your cursor slightly above the top set of holes and between them. Holding your mouse button down drag your cursor down and to the left. As you do this you create a dotted box around the left set of holes.

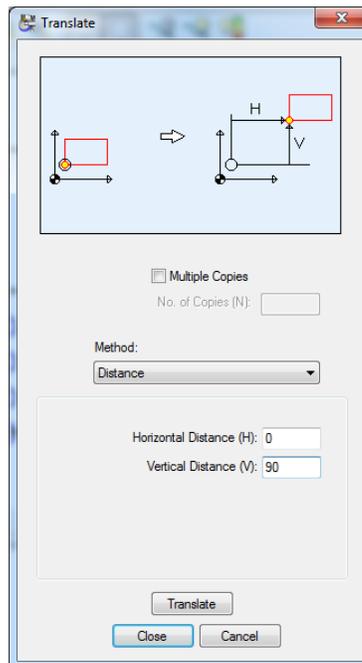
- 2 Once the box is around all four holes on the left release the mouse button. Your screen should look like the one below. After releasing the mouse button the holes on the left will be highlighted.



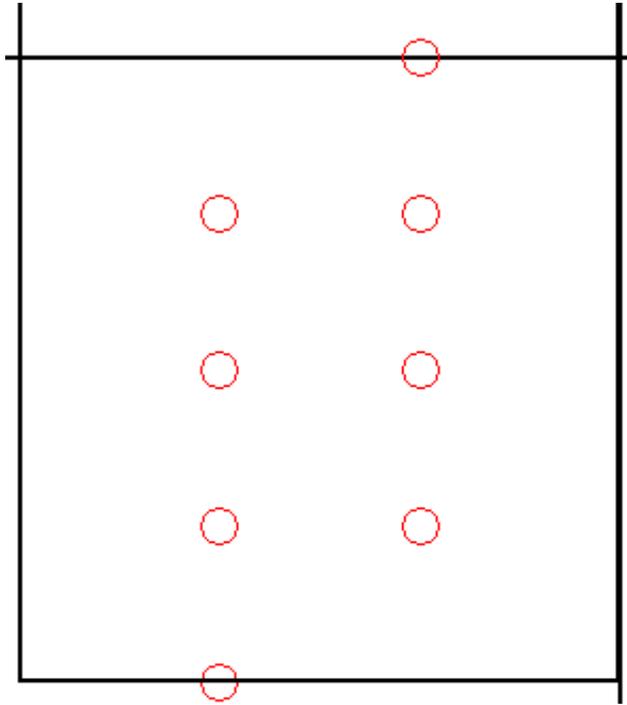
**Note:** If the bottom row of holes appears to be off the screen and is unselectable, choose **Zoom Out** from the **View** menu.

You will now shift the selected circles down so that you can create diagonal lines between the centers of the holes.

- 1 Choose **Translate** from the **Transform** submen found under **Edit** menu. Set the parameters as shown below:



- 2 Once you have entered these parameters, click on **<Translate>**. Your screen should now appear as shown below:



- 3 Click anywhere within the **Face Window** to deselect the circles that were just translated.
- 4 Change back to the CAM mode by clicking CAD/CAM switch.

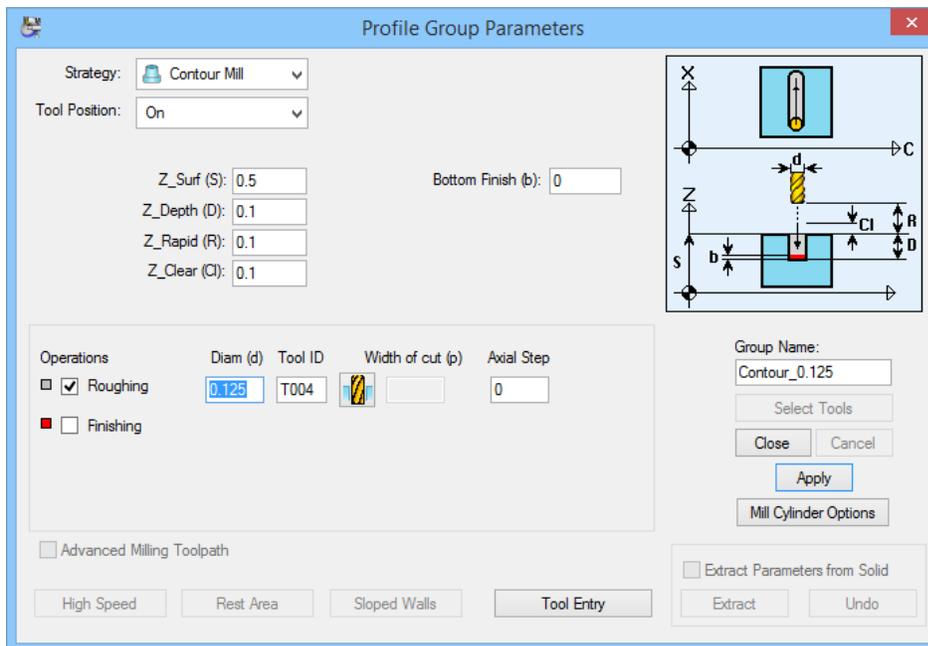


## Creating the Helical Channels

You can now use the translated circles to more easily define the tool path for the helical slots.

- 1 Select **New Profile Group** from the **Part Features** menu.
- 2 From the **Strategy:** drop down menu, select **Contour Mill**.
- 3 From the **Tool Position:** drop down menu, select **On**.
- 4 Enter a **Z\_Depth (D):** of 0.098 (2.5)
- 5 Enter a **Diam (d)** of 0.125 (3.5)
- 6 In the **Group Name** box type in **Contour\_0.125 ( 3.5)**.
- 7 After having entered these parameters, click the **<Select Tools>** button to call up the .125 (3.5) end mill.

Your completed dialog should appear as shown below:



- 8 Once your screen looks like the one above click the **<Close>** button to exit **the Profile Group Parameters** dialog

Next, you will define the tool motion.

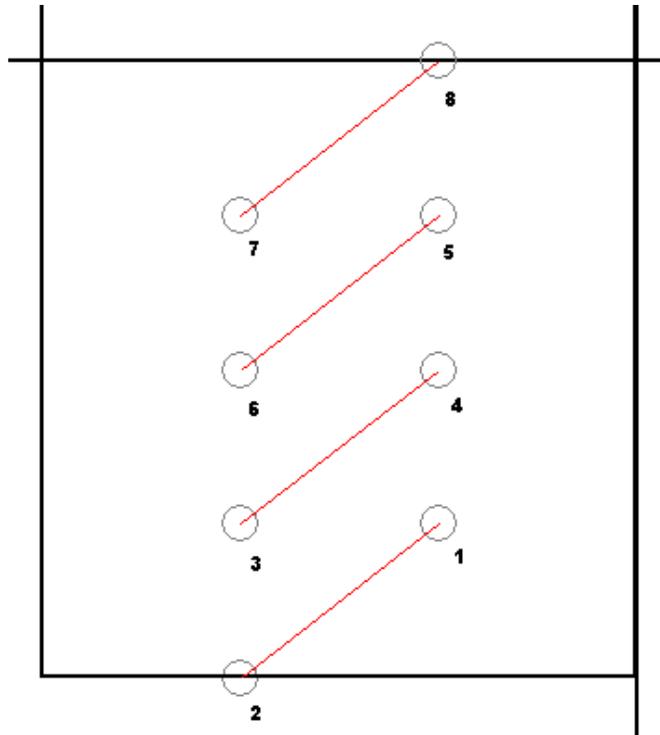


- 1 Select the **Define Profile** icon. This icon will allow you to define the tool motion for the slots from point to point.
- 2 Select the **Center of Circle** snap mode icon from the snap mode icon bar at the top of the face window. This will allow you to use the centers of the circles you defined as guides for the creation of the slots. Your cursor should now appear as a circle with an arrow attached to it.
- 3 Click on the lowest right hand hole, the one labeled #1 below. Move your cursor down and click on the bottom left hole lying directly on the X axis, #2 in the window.



- 4 Choose the **Selection** icon to deselect the profile you just created.
- 5 Once you have done so, repeat steps 1 to 4 above for holes 3 and 4, 5 and 6, and 7 and 8 respectively as shown in the picture below.

Your screen should appear as shown below:

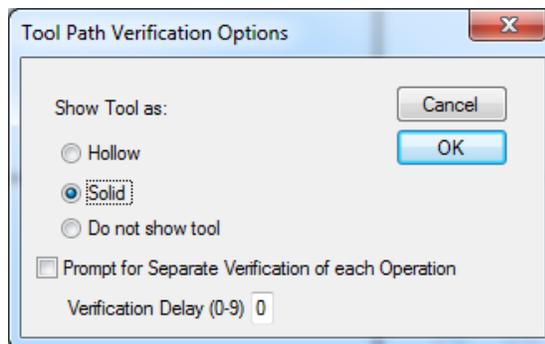


### Verifying the Tool Path

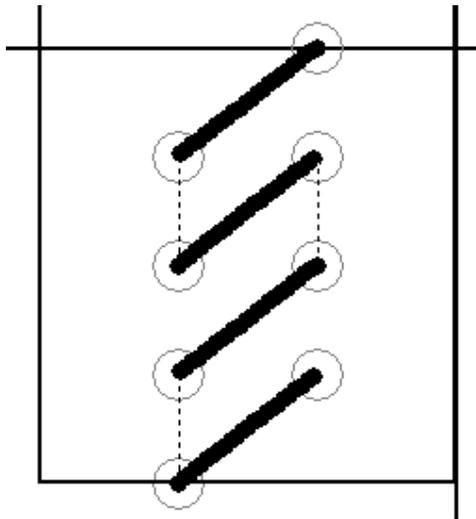
You can now verify the tool path of the operation you have just programmed.

This time try a different type of verification process.

- 1 Choose Verification Options from the Part Features menu.
- 2 In the **Tool Path Verification Options** dialog select the **Solid** radio button and click **<OK>**.



- 3 Choose **Verify Work Group Tool Path** from the **Part Features** menu. PartMaker should show the tool path verified as shown below:



- 4 When you are satisfied that your screen looks like the one above, choose **Hide Work Group Tool path** from the part features menu and press **<Ctrl-S>** to save your work.

## Creating Offset Holes Using Mill 4 Axis Plane

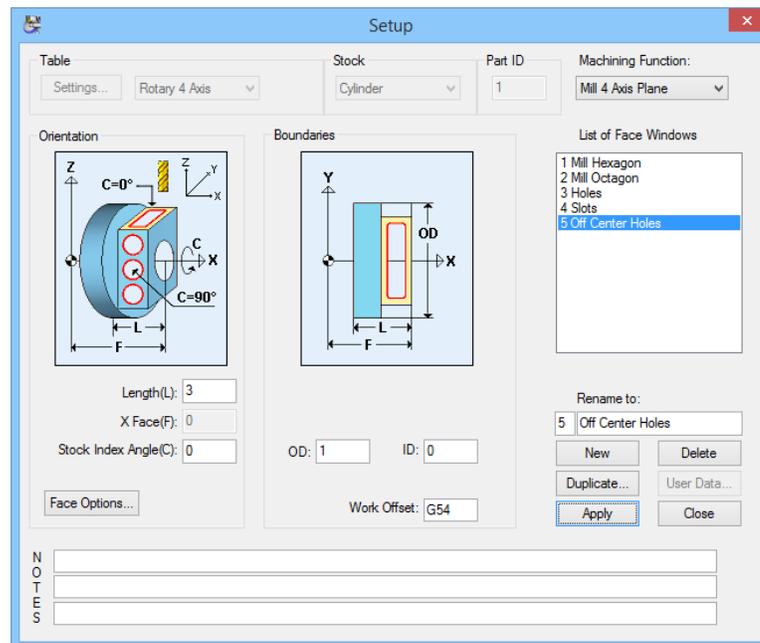
This section of the tutorial you will use a spot drill and a drill to create the three .125' (3.00) diameter holes .200' (5.00) from the face of the part and arranged .300" (7.5) apart.

### Setting Up the Face Window

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

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box.
- 2 Click the **<New>** button to create a new face window.
- 3 From the Machining Function drop down menu select Mill 4 Axis Plane
- 4 Type Off Center Holes in the Rename To: field and press <Enter>.

When completed, your **Setup** dialog should appear as shown below:

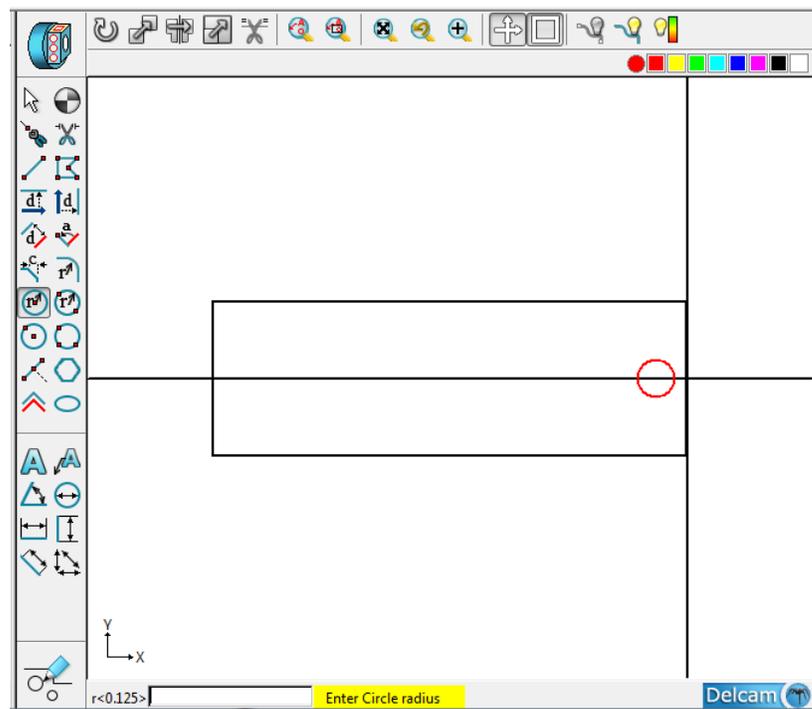


## Creating Geometry in CAD Mode

In this section, you will create the geometry for the three holes.

-  1 From the View menu, choose **Show Axes**.
-  2 From the View menu, choose **Show Boundaries**.
-  3 Go to **CAD** mode by clicking on the CAD/CAM switch.
-  4 Select the **Circle with a Known Radius and Center** icon. This will allow you to draw the first of the three 0.125 (3/2) diameter holes.
-  5 Enter a value for the radius of **.125/2 (3/2)** in the field at the bottom of the face Window press **<Enter>**.
-  6 Select the **XY coordinates** icon from the Snap Mode icon bar.
- 7 In the lower left hand corner of the window you are prompted for the X and Y location for the first hole. Enter **-.200 (-5.00)** for X and **zero (0)** for Y and press **<Enter>**. A circle should now appear on the negative side (left) side of the vertical axis and on the Y-axis line.

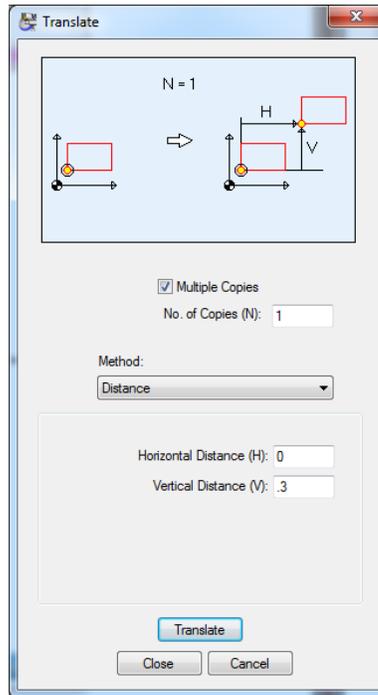
Your window should look like the one below:



-  8 To draw the remainder of the holes choose the **Selection** icon and select the circle that you drew on your screen by clicking on it.

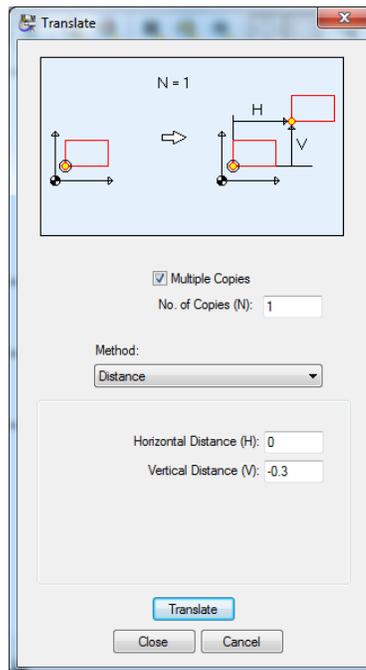


- 9 After the circle is highlighted, select **Translate** from **Transform** submen found under the **Edit** menu. Enter parameters as shown here and click **<Translate>**: (metric users enter a Y Distance of 7.5)

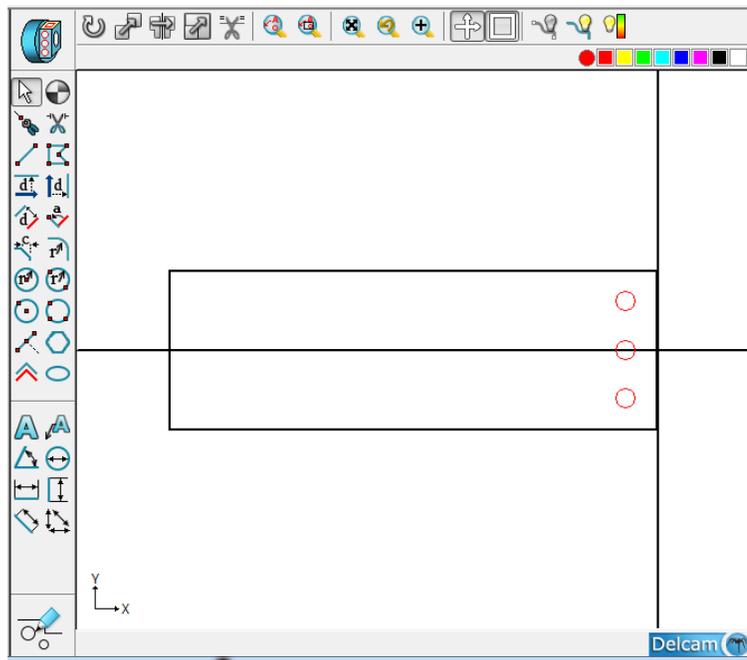


Now you will translate the first circle you created  $-0.300$  ( $-7.5$ ) in Y. To do so:

- 10 Select the first circle you drew using the **Selection** icon.
- 11 Select **Translate** from the **Transform** submen found under the **Edit** menu again. Enter parameters as shown here and click **<Translate>**: (metric users enter a Y Distance of 7.5)



Your window should look like the one below:

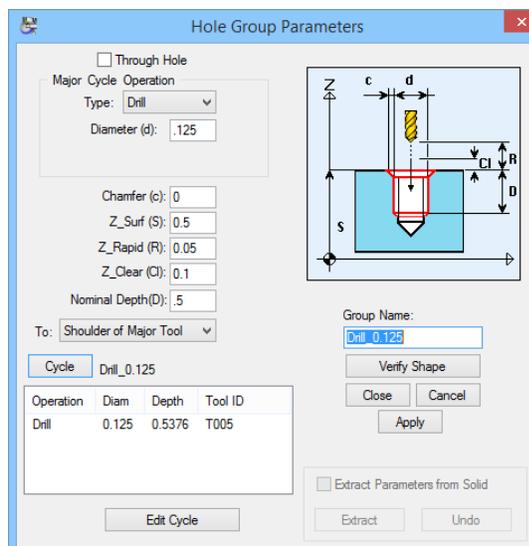


## Creating the Off Center Holes

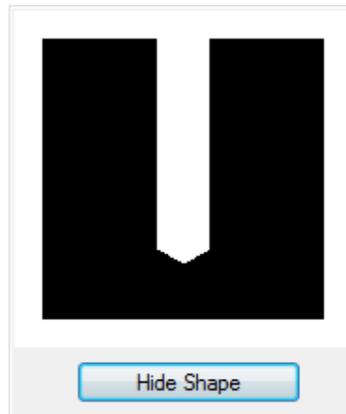
In this section, you will define the holes to be machined on the top of the part.



- 1 Click the CAD/CAM switch such that it appears as a drill bit.
- 2 Select New Hole Group from the Part Features menu.
- 3 Uncheck the **Through Hole** box.
- 4 From the **Type:** drop down menu, select Drill.
- 5 Enter a **Diameter** of 0.125 (3.00)
- 6 Enter a **Nominal Depth** of 0.500 (12.00)
- 7 In the **To:** drop down menu, select **Shoulder of Major Tool**
- 8 Click the **<Cycle>** button. In doing so, PartMaker found an existing cycle from the **Cycles** database where the major operation was a **.125 (3.00)** diameter drill. The completed dialog should appear as shown below:



- 9 Click the **<Verify Shape>** button to see the shape of your hole.

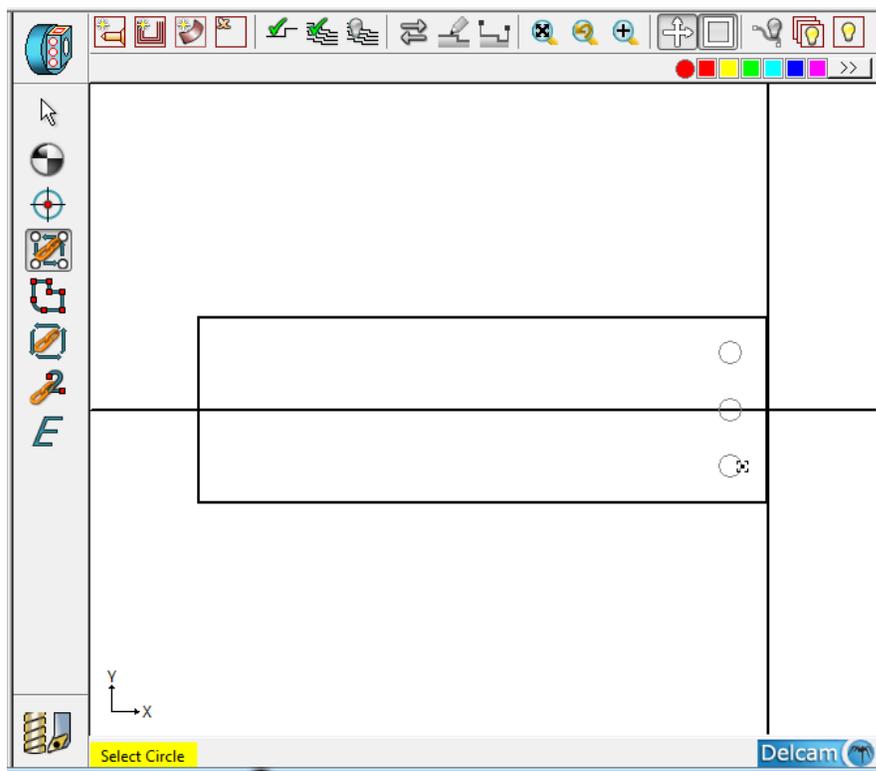


- 10 Once you are satisfied that your dialog and hole shape appear as the ones above, click **<Hide Shape>** and **<Close>** to exit the dialog.

You will now specify the holes you want to machine.



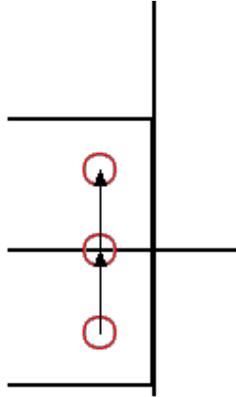
- 1 Select the **Chain Circle** icon.
- 2 Click your cursor somewhere near the first hole you want to machine. In this case you will select the lower hole in the window as shown below:



## Verifying the Tool Path

You can now verify the tool path of the operation you have just programmed.

- 1 Click anywhere within the **Face Window** to un-highlight the holes.
- 2 Select **Verify Work GroupTool Path** from the **Part Features** to see the tool path displayed below:



- 3 When you are satisfied that your toolpath appears as the one shown, choose **Hide Work GroupTool Path** from the **Part Features**,



Now save your work by pressing **<Ctrl+S>**. You have finished the 4<sup>th</sup> axis tutorial part.

## Creating an NC Program

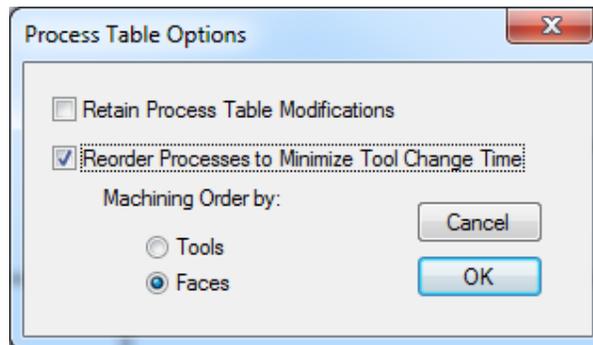
At this point, you have completed creating the five faces for the tutorial part. You are now prepared to take the final steps in developing your machine specific Part Program.

### 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 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.



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

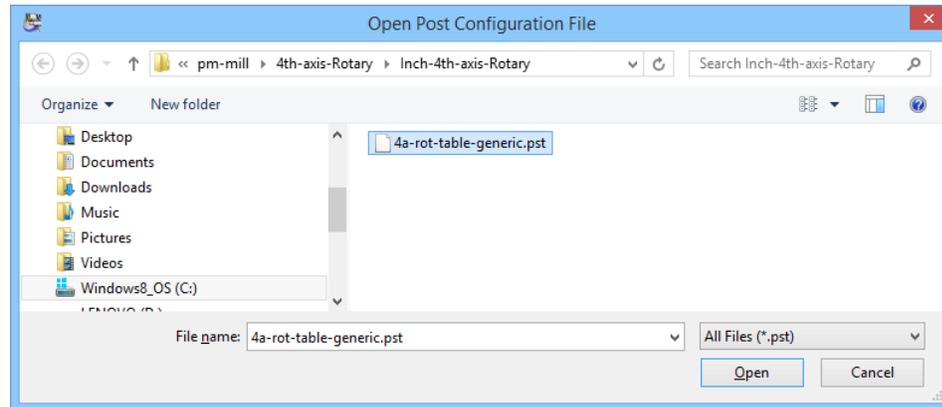
| Proc ID | Tool ID | Tool No. | Tool Name      | Proc Task    | Group   | Face             | Feed    | Speed   | Time(min) | Status |
|---------|---------|----------|----------------|--------------|---------|------------------|---------|---------|-----------|--------|
| P01     | T002    | 1        | End Mill_1/2   | Contour Roug | Mill F  | Mill Hexagon     | 35.7upm | 4577rpm | 0.28      | ✓      |
| P02     | T002    | 1        | End Mill_1/2   | Contour Roug | Mill C  | Mill Octagon     | 35.7upm | 4577rpm | 0.34      | ✓      |
| P03     | T001    | 2        | Spot-Drill     | DRILL        | cyl_hc  | Holes            | 20.7upm | 3054rpm | 0.12      | ✓      |
| P04     | T003    | 3        | Drill_0.1875   | DRILL        | cyl_hc  | Holes            | 24.1upm | 5000rpm | 0.13      | ✓      |
| P05     | T004    | 4        | End Mill_0.125 | Contour Roug | Contc   | Slots            | 9.9upm  | 5000rpm | 1.50      | ✓      |
| P06     | T005    | 5        | Drill .125     | DRILL        | Drill_C | Off Center holes | 14.1upm | 5000rpm | 0.15      | ✓      |

Material File: Alu\_allc.mdb Total Time: 3.87 min.

## Selecting a Post Processor

Having generated a Process Table you must now select a Post Configuration File before you can output an NC program.

- 1 Choose Post Config File = ? from the **Job Optimizer** menu.
- 2 Select the **4a-rot-table-generic.pst** postprocessor from the **4th-axis-Rotary** directory as shown below.



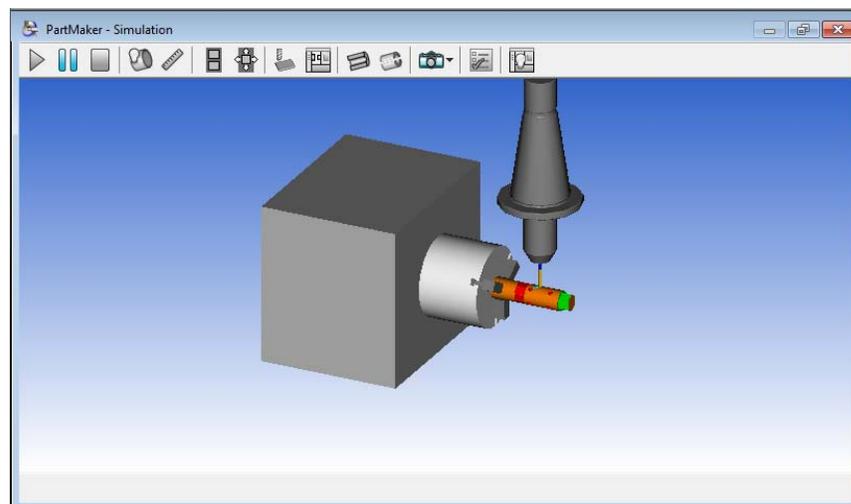
- 3 Click on the post processor and click the **<Open>** button to load the post processor.

## Simulating the Process

Now that the Process Table has been generated you can simulate the entire machining operation for the part

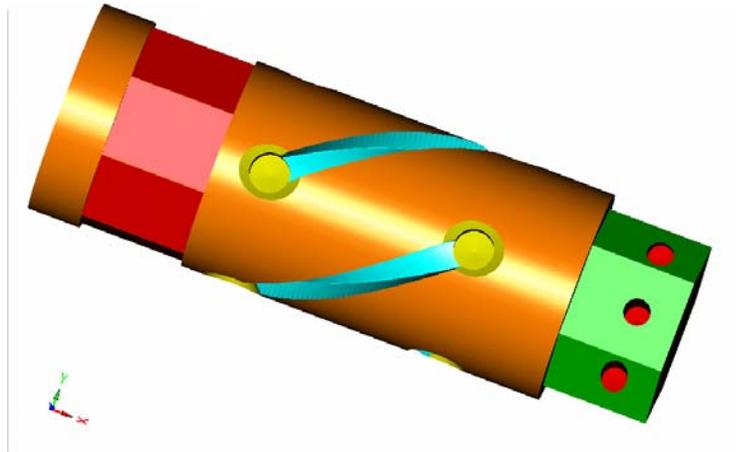


- 4 Click the **Simulation** button at the bottom left hand portion of the Process Table.
- 5 Press the **<Space Bar>** or choose **Simulation** from the **Simulation** menu. Below a picture of the simulation in process is shown:





Once simulation has completed, you can choose **Show Finished Part** from the **Simulation** menu to view a completed solid model of the finished part.



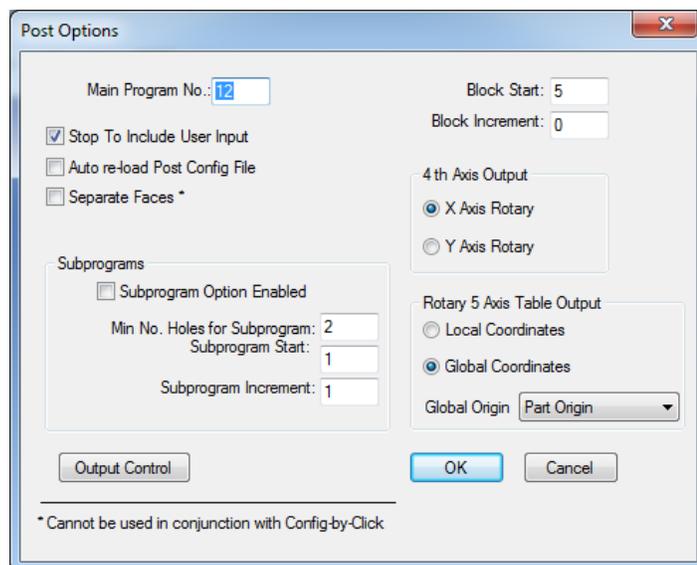
Shown above is the finished part in the Simulation window.

## Generating an NC Program

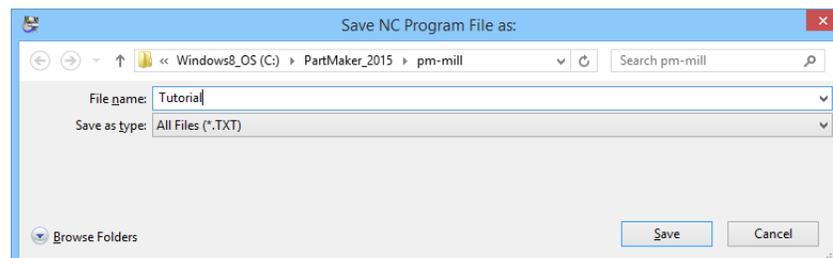
Having selected a Post Configuration file, you are now ready to generate an **NC Program**.



- 1 Choose Generate NC Program from the Job Optimizer menu.

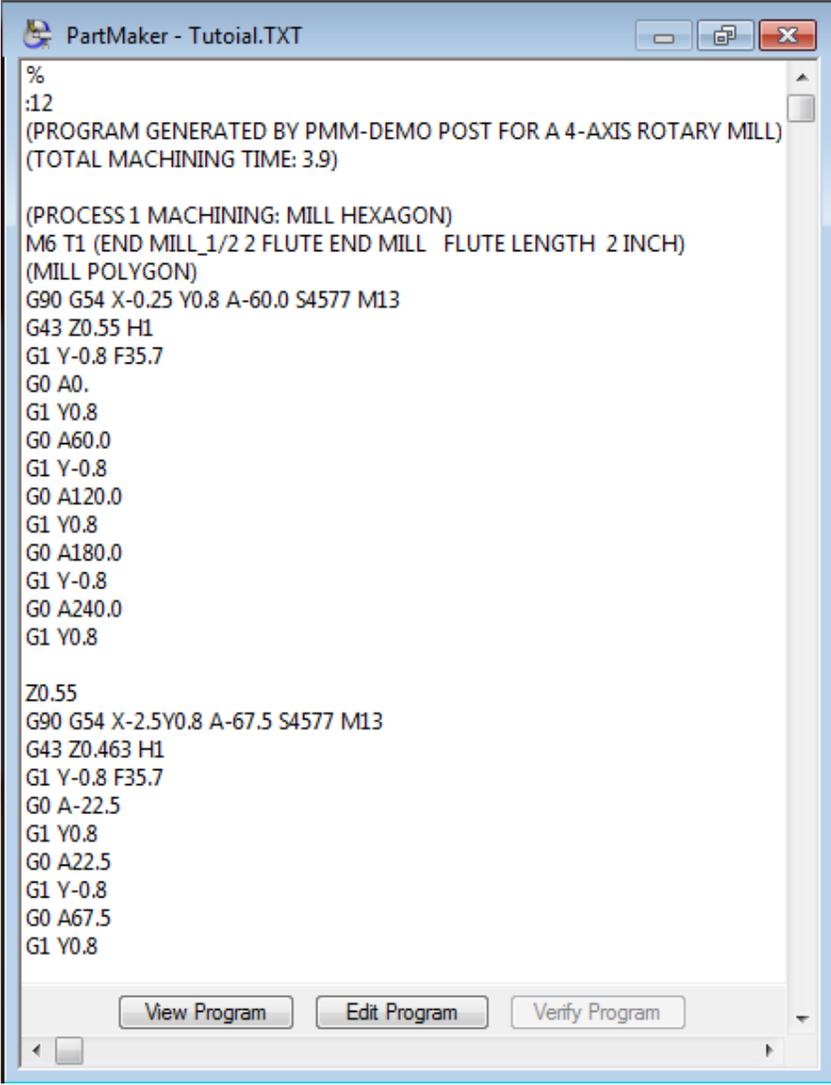


- 2 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog.
- 3 In this dialog, enter a name for your program in the **File Name:** field. Here we have entered the Tutorial.



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

The completed **NC Program** is shown below:



```
%
:12
(PROGRAM GENERATED BY PMM-DEMO POST FOR A 4-AXIS ROTARY MILL)
(TOTAL MACHINING TIME: 3.9)

(PROCESS 1 MACHINING: MILL HEXAGON)
M6 T1 (END MILL_1/2 2 FLUTE END MILL FLUTE LENGTH 2 INCH)
(MILL POLYGON)
G90 G54 X-0.25 Y0.8 A-60.0 S4577 M13
G43 Z0.55 H1
G1 Y-0.8 F35.7
G0 A0.
G1 Y0.8
G0 A60.0
G1 Y-0.8
G0 A120.0
G1 Y0.8
G0 A180.0
G1 Y-0.8
G0 A240.0
G1 Y0.8

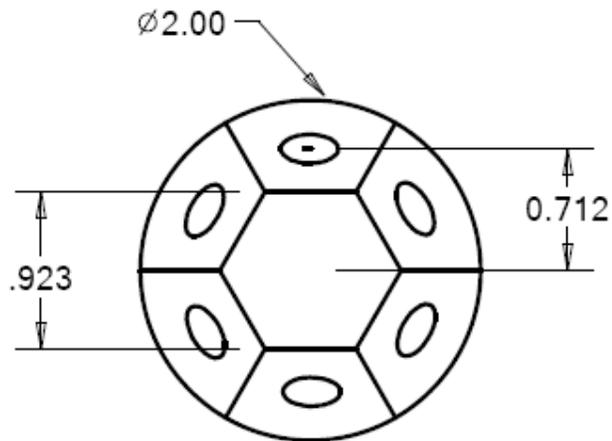
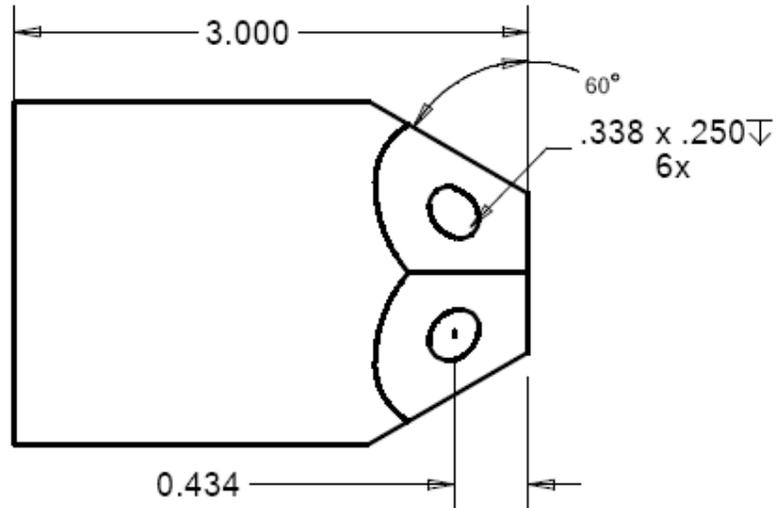
Z0.55
G90 G54 X-2.5Y0.8 A-67.5 S4577 M13
G43 Z0.463 H1
G1 Y-0.8 F35.7
G0 A-22.5
G1 Y0.8
G0 A22.5
G1 Y-0.8
G0 A67.5
G1 Y0.8
```

View Program Edit Program Verify Program

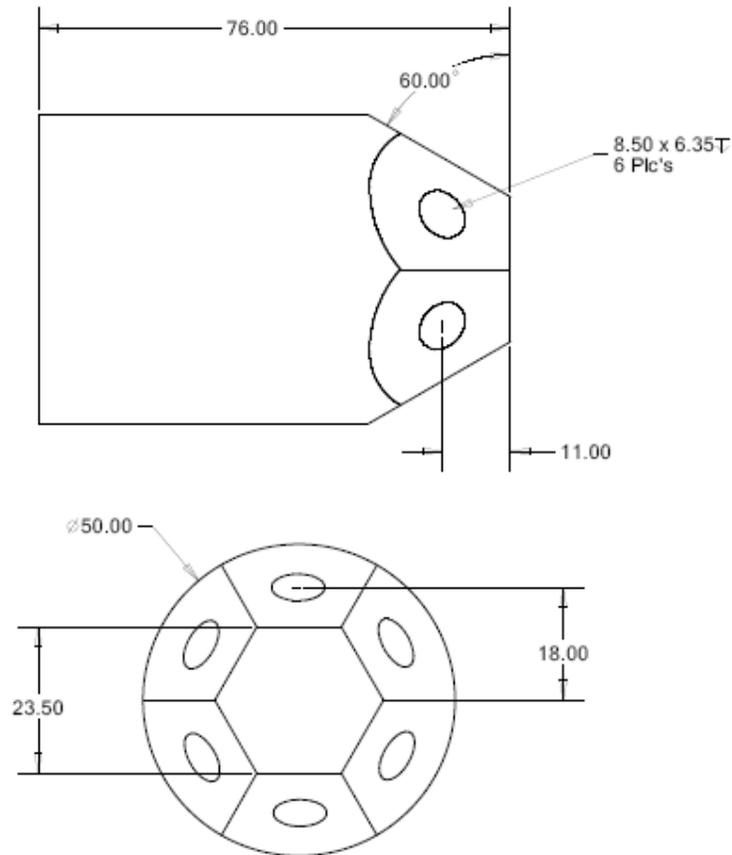
# Appendix F: The 5-Axis Milling Tutorial Part

The drawing below shows a part print of the part you will be programming in this tutorial:

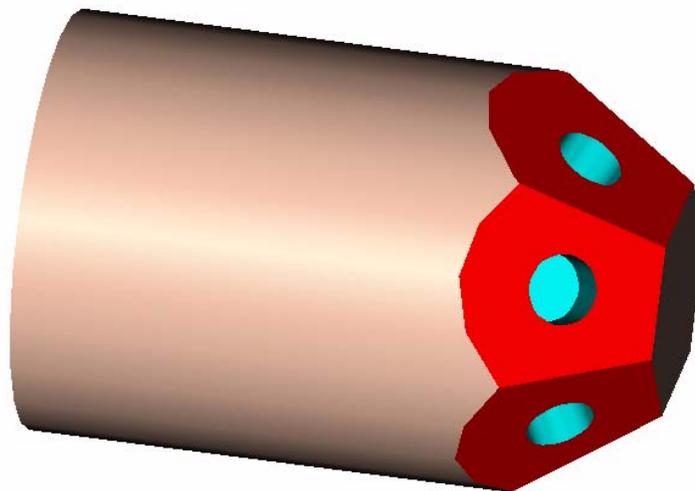
**Inch Units**



**Metric Units**



For this tutorial, you will work with an imported Solid Model. A graphic of the solid model is shown below:



**Important!** To complete this tutorial, you must have licensed the PartMaker Production Milling Module (PMM), the PartMaker Parasolid Import Module and the PartMaker 5<sup>th</sup> Axis Milling Module.

## How you'll create the 5 Axis Mill tutorial part

You will complete the following steps in working through this tutorial:

- 1 Importing the solid file
- 2 Transfer geometry for a 5 axis plane
- 3 Perform milling operations in the chosen plane
- 4 Duplicate step 3 for the other faces
- 5 Generate the Process Table
- 6 Simulate the machining of the part
- 7 Generate an NC program

### Introduction

The first part of any programming exercise with PartMaker's 5 Axis milling module is to decide which machining functions will be used.

The machining functions used in this tutorial are as follows:

#### Mill 5 Axis Plane

Use an end mill to create hexagonal flats on the end of the part and mill .3386" (8.50) diameter hole .250 (6.35) inches deep.

This tutorial will help you learn the steps to go through in using PartMaker to program a PartMaker Mill part that requires 5-axis milling. This tutorial uses a solid model.

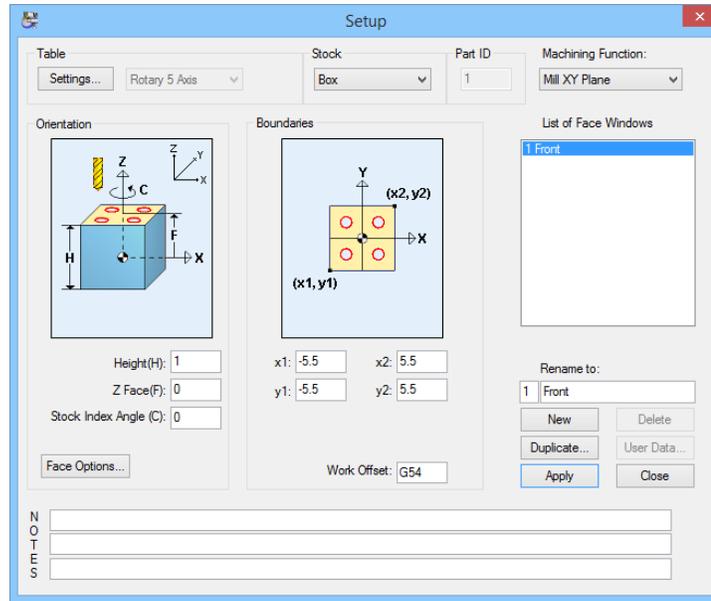
### Getting Started

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 tutorial for the 5 Axis Milling part can be found in the following directories:

- |                               |   |
|-------------------------------|---|
| <b>Completed Job File:</b>    | C:\PartMaker\PM-Mill\5-axis-demo\5-axis-job.JOB |
| <b>Completed Tools File:</b>  | C:\PartMaker\PM-Mill\5-axis-demo\5-axis-job.TDB |
| <b>Completed Cycles File:</b> | C:\PartMaker\PM-Mill\5-axis-demo\5-axis-job.CDB |

## Starting PartMaker

Start the PartMaker Milling application. 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. This dialog will be explained in greater depth starting on page F-10.

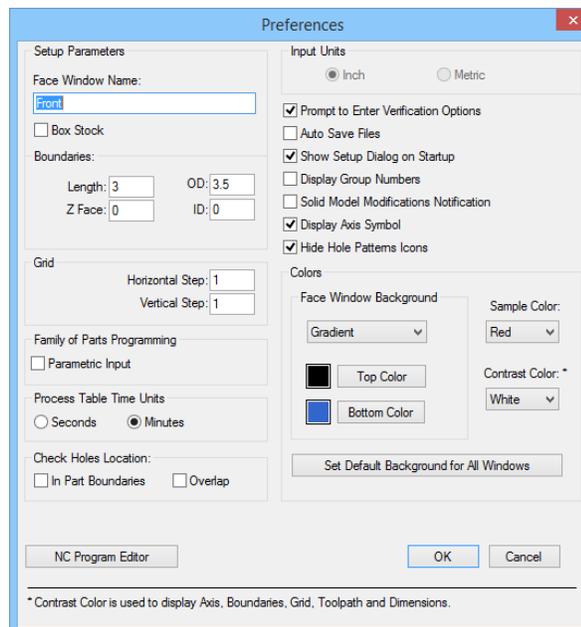


## Setting Preferences for the 5-Axis Exercise

In this section, you will setup the necessary preferences for this exercise.

- 1 Choose **Preferences** from the **View** menu.
- 2 Check the **Display Axis Symbol** item. It may already be checked. Checking this box will display the axes you are working with in the lower left hand corner of each Face Window.

Your completed **Preferences** dialog should appear as the one shown below:

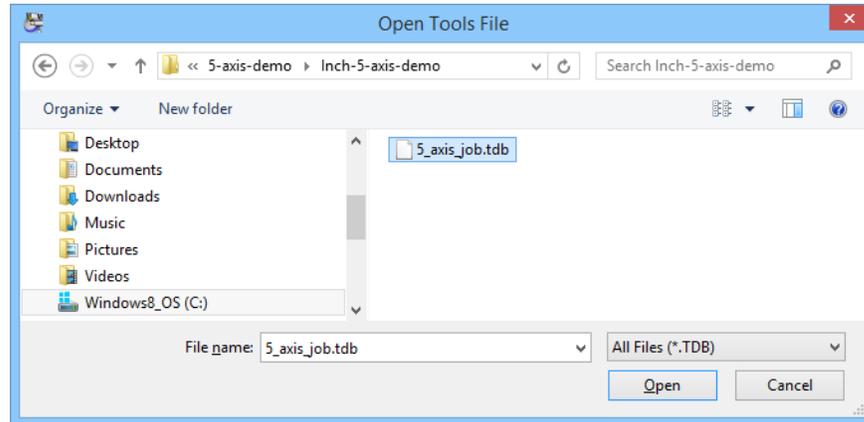


- 3 Click **<OK>** to close the **Preferences** dialog.

## Opening the Tools file

The tools library for the 5-Axis Programming Exercise has already been created. To open the tools library:

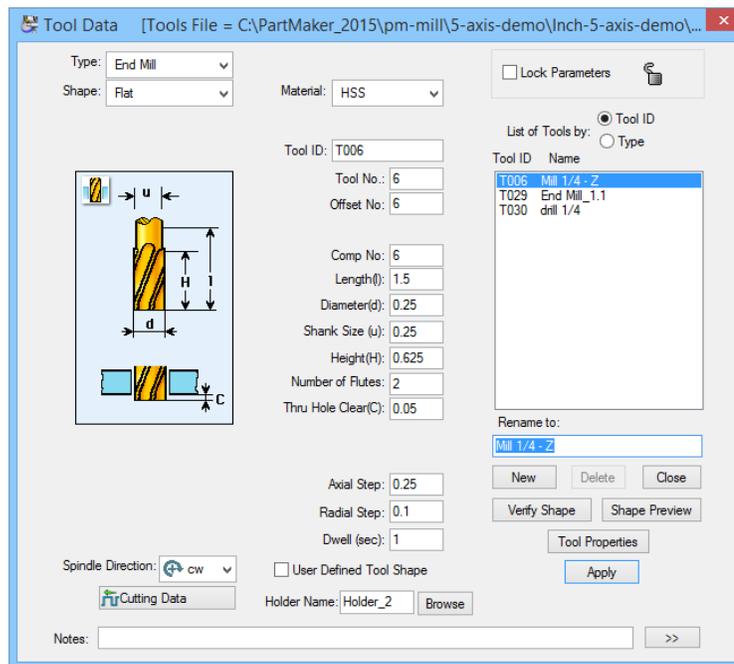
- 1 Choose **Open Tools File** from the **File** menu
- 2 Open the **5\_axis\_job.tdb** from the C:\PartMaker\PM-Mill\5-axis-demo\5-axis-job.TDB directory as shown below:



Once you have loaded the tools database, you can view the tools that you will be using for this programming exercise. To do so:



- 3 Choose **Tools** from the **ToolMinder** menu. This tools file contains five tools. The first of these appears below:

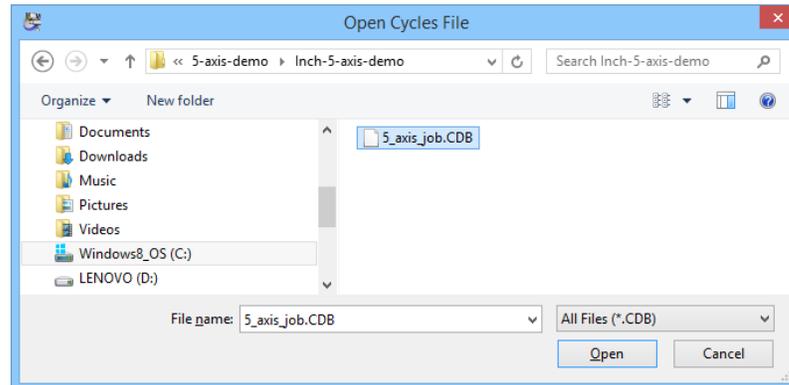


- 4 Click the **<Close>** button to close the **Tool Data** dialog.

## Opening the Cycles file

At this point, you should be familiar with the concept of Cycles in PartMaker. Cycles are sequences of repetitive operations that go into machining a hole. The cycles library for the 5-Axis Programming Exercise has already been created. To open the cycles file:

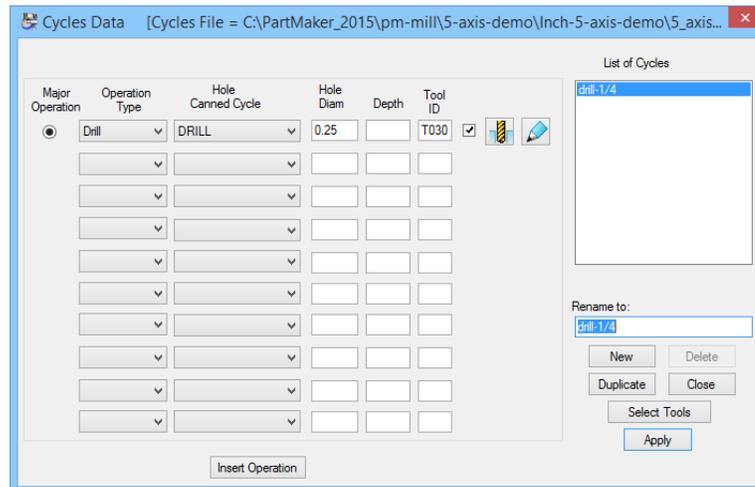
- 1 Choose **Open Cycles File** from the **File** menu.
- 2 Open the **5\_axis\_job.CDB** from the C:\PartMaker\PM-Mill\5-axis-demo directory as shown below



You can view the cycle that you will be using for this programming exercise. To do so:



- 1 Choose **Cycles** from the **ToolMinder** menu.



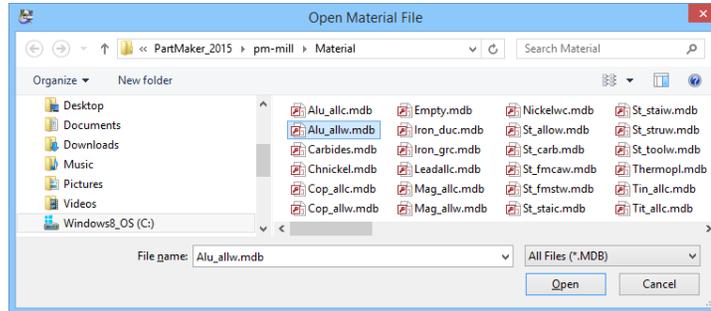
We will use the drill-1/4 cycle to create the six drilled holes on the work piece.

- 2 Click the **<Close>** button to close the **Cycles Data** dialog.

## Select a Material for the Part

The work piece in this exercise will be machined from **aluminum alloy wrought**. To load this material:

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



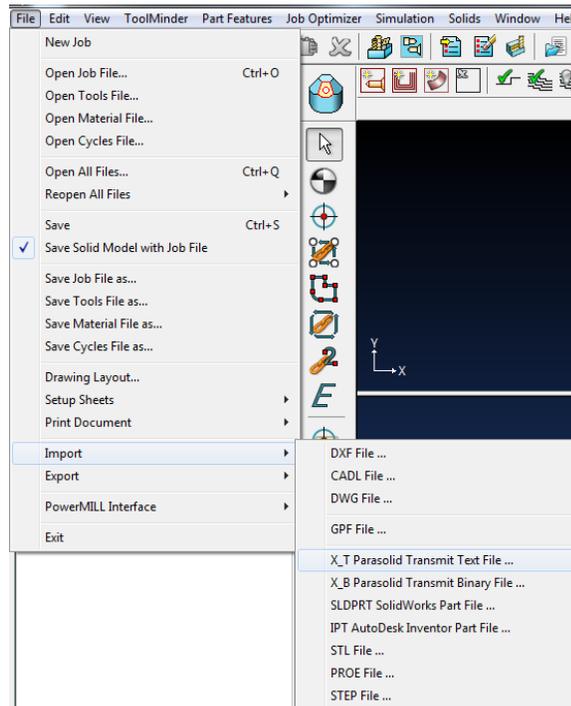
You have now loaded the tools, cycles, and material files that were previously developed for the 5 Axis tutorial. PartMaker has also provided a completed job file. At this point you will not open the job file. You will be guided through a step-by-step process to develop a job file for the tutorial.

## Importing the Solid Model

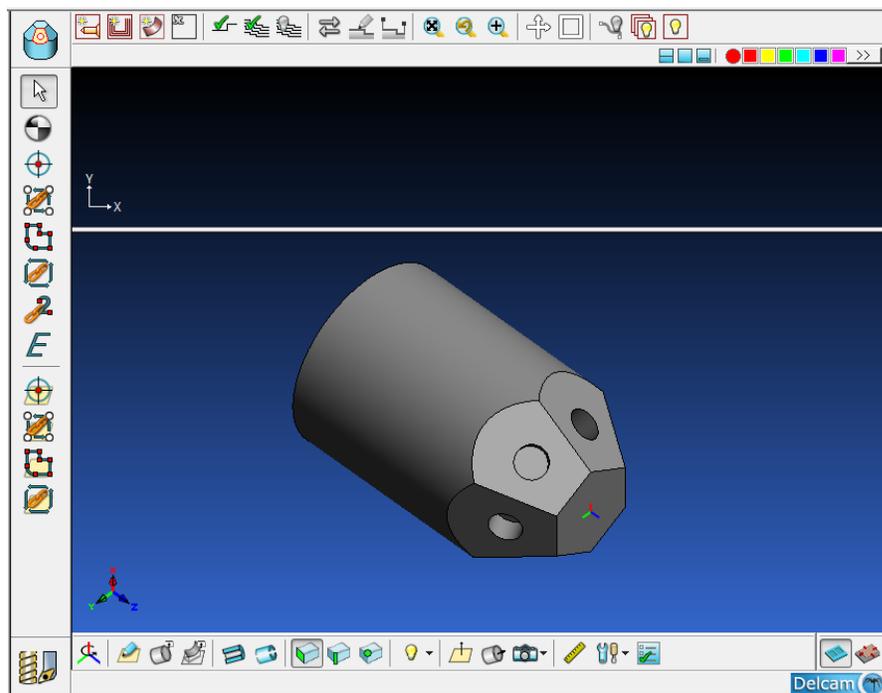
**Note:** We will use a Parasolid .x\_t formatted file for this exercise. The .x\_t file is called 5\_axis\_job.x\_t and is located in the C:\PartMaker\Pm-Mill\5-axis-demo directory.



- 1 Choose **Import X\_T Parasolid Transmit Text file** from the **File** menu or select the Import button on the main menu tool bar.
- 2 Select 5\_axis\_demo.x\_t from the C:\PartMaker\Pm-Mill\ 5-axis-demo directory.



Your screen should appear as below after you import the 5-axis solid model:

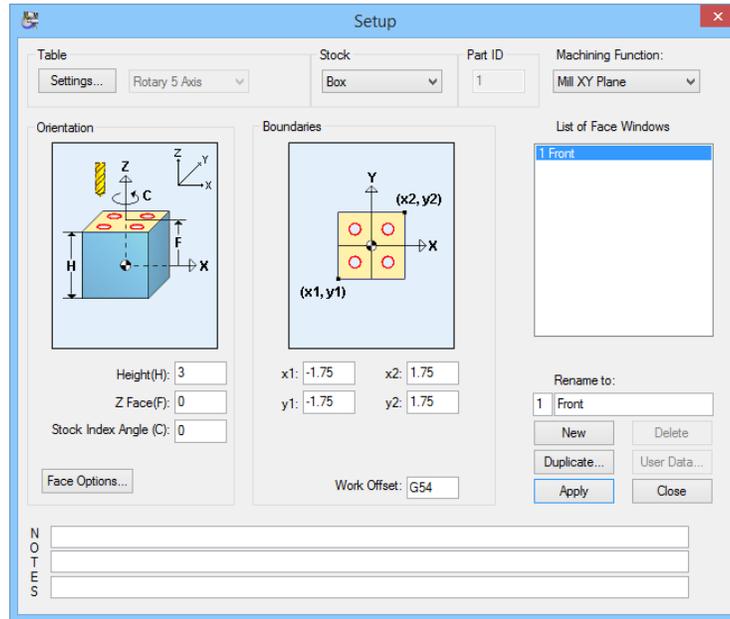


## Setting Up the Face Window

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

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

The default setup screen will appear as shown below.



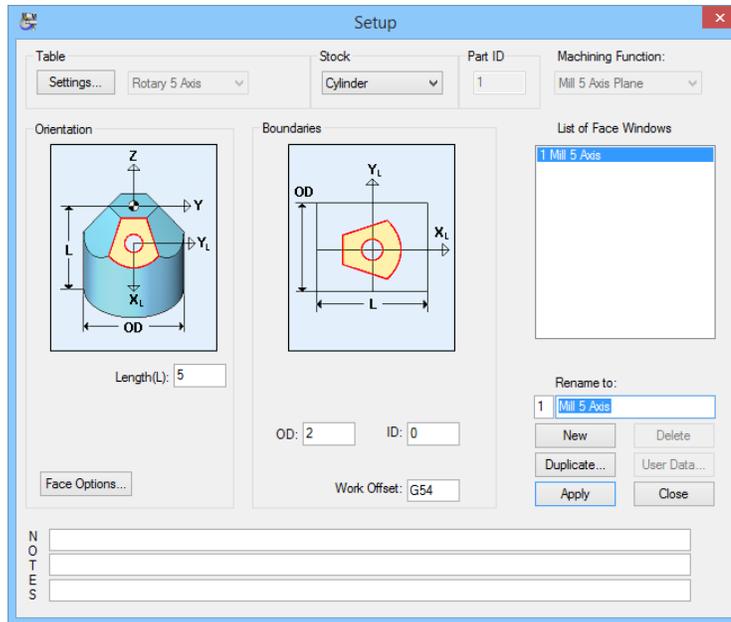
- 2 From the **Table** drop down menu select **Rotary 5 Axis**.
- 3 From the **Stock** drop down menu select **Cylinder**.
- 4 From the Machining Function drop down menu select Mill 5 Axis Plane.
- 5 Rename the current face window by clicking on **Front** in the **List of Face Windows**: field and type **Mill 5 AXIS** in the **Rename To:** field and click the **<Apply>** button.
- 6 Now enter the parameters listed below in the appropriate fields:

**Length (L):** 3.0 (76.0)

**OD** 2.0 (50.0)

**ID** 0 (0)

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



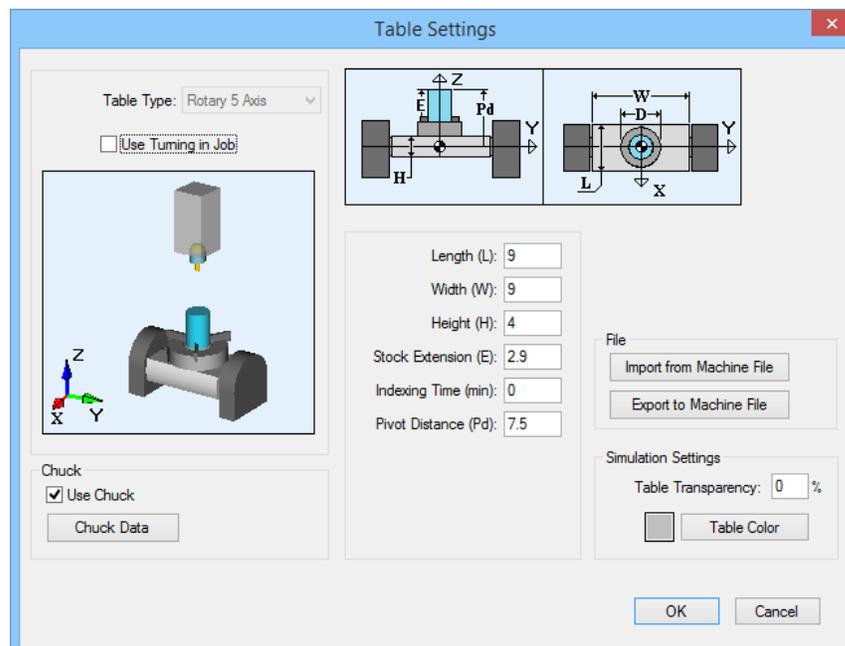
- 7 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current face window.

## Setting Up the 5-axis Table

At this point you will setup the 5-Axis Rotary Table being used in this exercise.

The **Settings** dialog is accessed for the **Table Type: 5 Axis Rotary** by clicking the **Settings** button. The **Settings** dialog is used to define the dimensions and appearance of the machining table in PartMaker's 3D machining simulation for both block and cylinder stock.

A description of all of the fields for the **5 Axis Rotary Table Settings** dialog is located in Chapter 2 of this manual.



To setup the rotary 5-axis table:

- 1 From the **Table** section of the **Setup** dialog, click the **Settings** button to enter the **5-Axis Rotary Table Settings** dialog.
- 2 Enter the values shown above on the **5 Axis Rotary Table Settings** dialog.
- 3 When you are satisfied that your **5 Axis Rotary Table Settings** dialog appears as the one above, click the **<OK>** button.
- 4 Click the **<Close>** button to close the **Setup** dialog.

## Saving Your Work

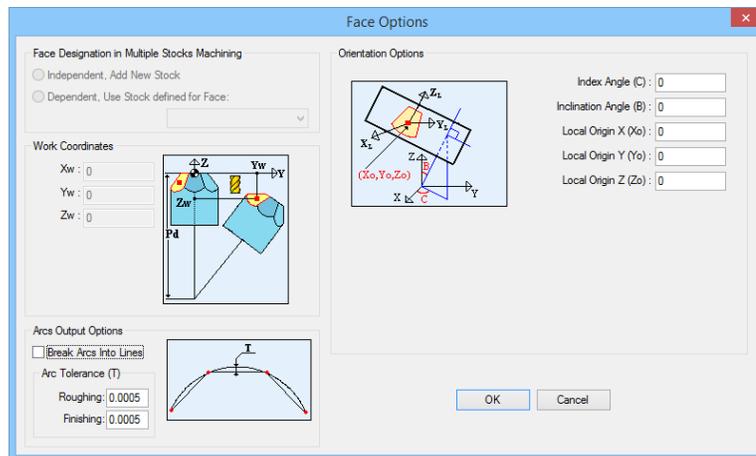


At this point save your work by choosing **Save Job File As** from the **File** menu. In the file name box type in a name and select **<Save>**. Be sure to give your file a name you will remember.

## Transfer geometry for a 5 axis plane

In this section you will transfer the geometry of the part from the solid model and establish the angular orientation of the 5-axis rotary table during machining.

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box
- 2 Click the **Face Options** button in the lower left area of the **Setup** dialog box
- 3 The default **Face Options** dialog will appear as shown below.



A description of all of the fields for the **Face Options** dialog is located in Chapter 2 of this user manual.

- 4 Check the **Break Arcs Into Lines** box in the **Arc Output Options** section of this dialog.



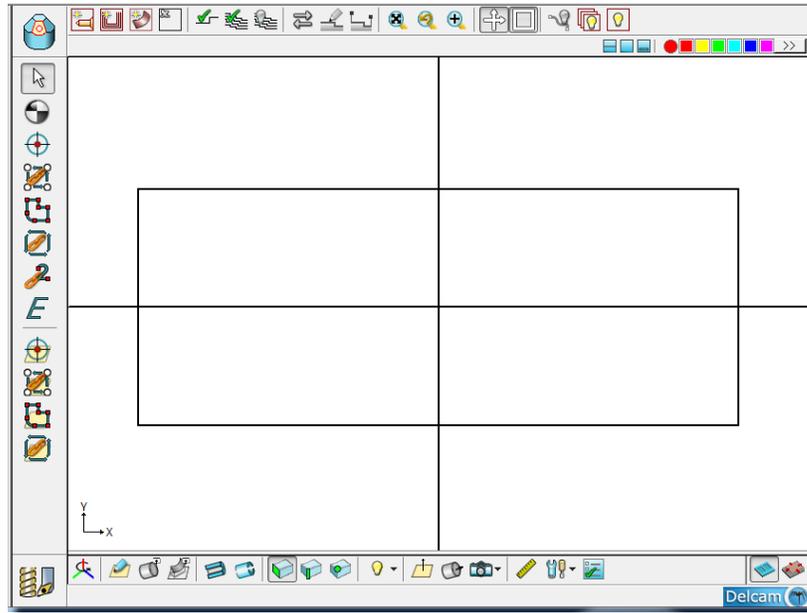
**Important:** Checking this box breaks arcs into small lines. Doing so may be necessary for some 5-axis milling machines with rotary fixtures because they do not support interpolation (i.e. G2's and G3's) in arbitrary 5-axis planes.

- 5 Click **<OK>** to close the **Face Options** dialog.
- 6 Click **<Close>** to close the **Setup** dialog.
- 7 Click the CAD/CAM switch to enter the CAD mode.
- 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.

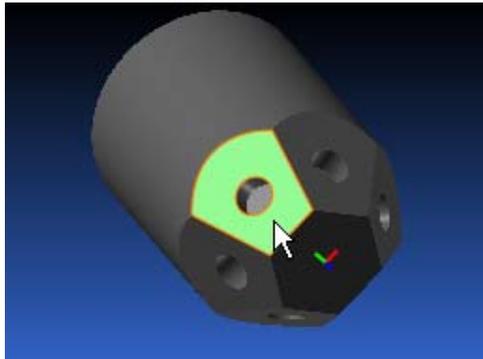




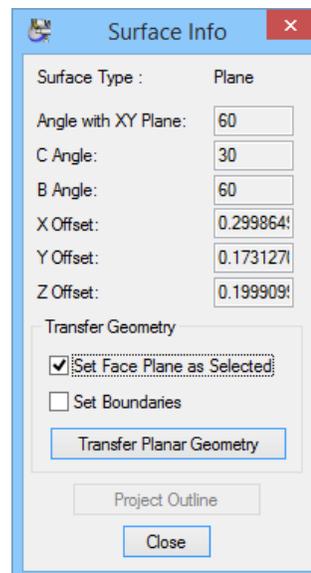
- 9 Choose **Show Boundaries** from the **View** menu. A rectangle should now appear on the screen that indicates the boundaries of the stock.



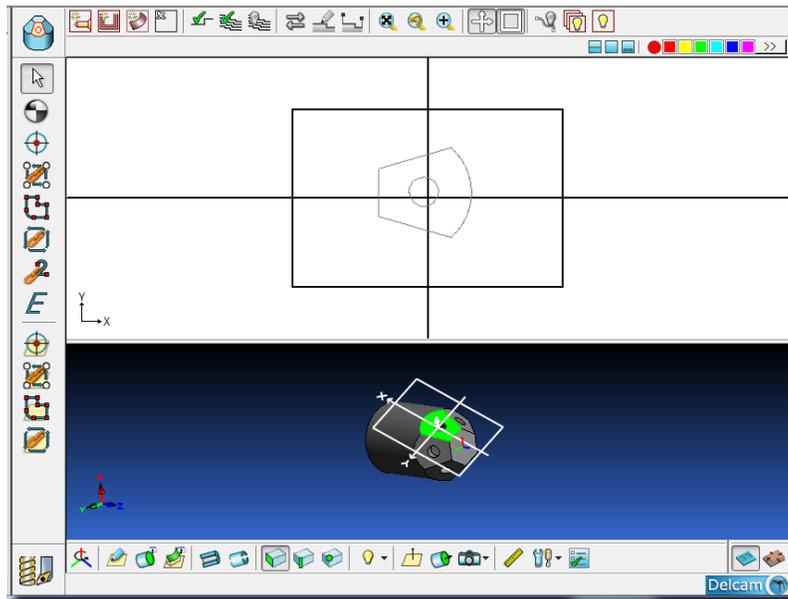
- 10 From the Window Menu, choose **Arrange Windows** to display the Solid Model.
- 11 Double click on the surface as shown below:



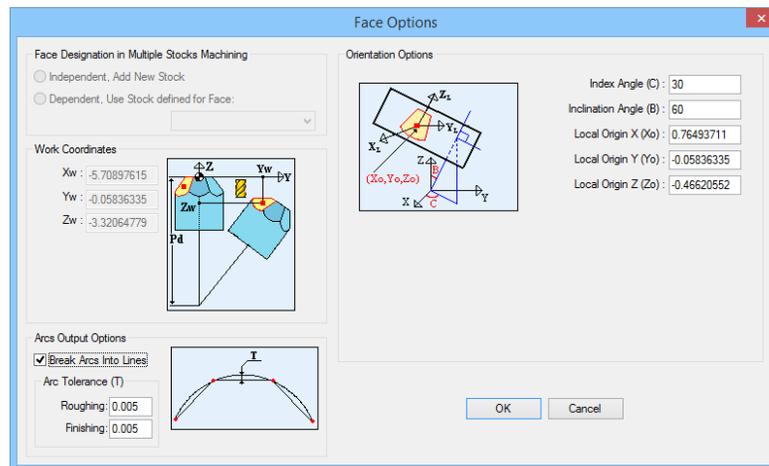
Doing so will open the **Surface Info** dialog as shown below:



- 12 Click the **Transfer Planer Geometry** button to transfer the required geometry to the Mill 5 Axis Face Window. Click anywhere to deselect. Your face window will appear as shown below:



**Note:** Transferring the geometry from the solid model will also set the 5 axis milling parameters for this face window. You may return to the **Face Options** dialog accessed from **Setup** dialog to view this. The transferred parameters can be viewed in the **Orientation Options** section of this dialog.



The 5 axis milling parameters found in the **Orientation Options** section of this dialog are defined as follows:

**Index Angle (C):** This field represents the rotary angle to which the stock is indexed on the 5-axis table.

**Inclination Angle (B):** This field represents the angle about the Z axis at which the machining plane is oriented. The tool will be perpendicular to this plane.

**Local Origin (Xo)/Local Origin (Yo)/Local Origin (Zo):** These fields represent the co-ordinates for the local origin created for the planar face to be machined. These represent coordinates with respect to the work piece's global coordinate system origin after the work piece is indexed by the Index Angle (C).

## Perform Milling Operations for the Chosen Plane

In this section you will create the part features and tool paths necessary to complete machining this plane.



**Note:** Milling operations are performed the same as any other PartMaker Milling face window and will use Z-oriented tools.

### Creating the Contour Mill Profile

In this section, you will define the profile to machine the flat surface on the edge of the cylinder.



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 in **Contour\_1**

4 Choose Contour Mill from the **Strategy:** drop down menu.

5 Choose **On** from the **Tool Position:** drop down menu.

6 Enter a Value for **Z\_Surf** that would be the maximum thickness of stock you will be cutting. In this case enter 1.



**Note:** The face windows Z datum surface has been set to the surface that was used to bring the geometry into the face window. (Z zero surface for this window)

7 Enter the same value for the **Z\_Depth** as you did for Z\_Surf, i.e. 1. Doing so will result in the final cut will being taken at the Z surface of the geometry that was used to bring the geometry into the face window from the solid model.

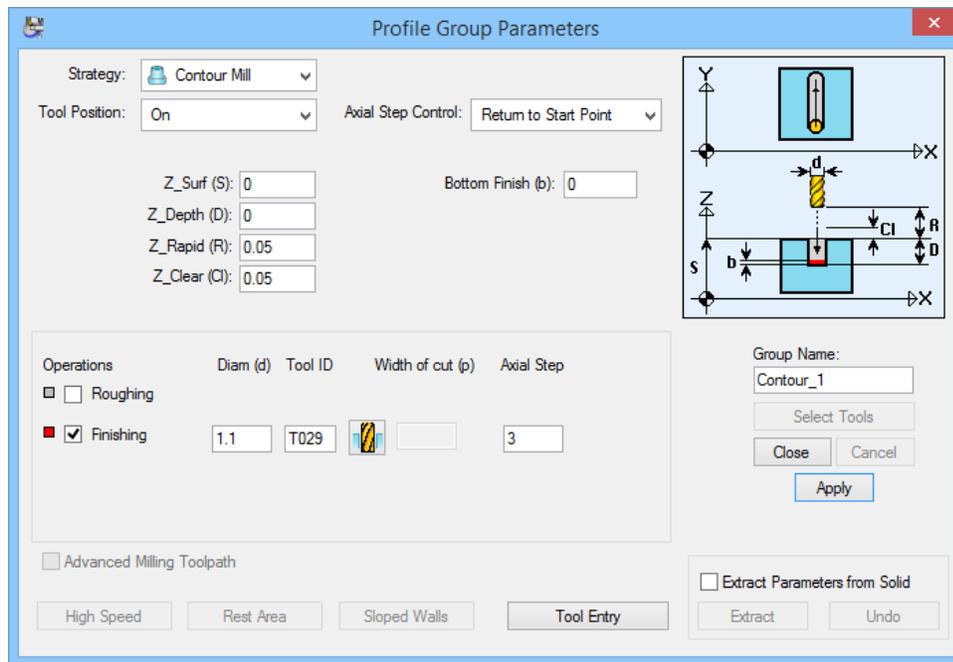
8 Enter 1.1 for **Tool Diameter** number. Click on the **<Select Tools>** button. The endmill icon will appear to the right of the **Tool ID** box.

**Note:** In this exercise we will be only taking one pass.

9 Uncheck **Roughing** and check **Finishing**.

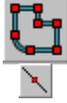
10 Enter 0 in the **Axial Step** field.

The completed dialog should appear as shown below:

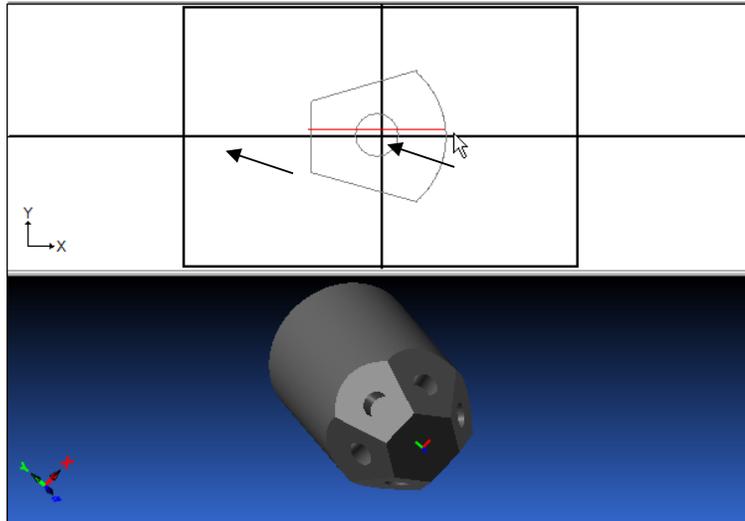


- 11 When you are satisfied that your **Profile Group Parameters** dialog matches the one above, click the **<Close>** button.

## Develop the Tool Path

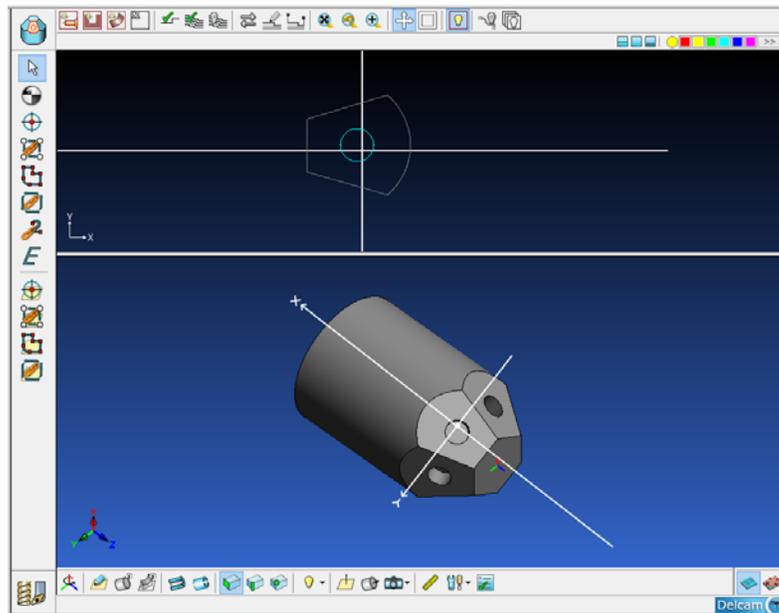


- 1 Select the **Define Profile** icon.
- 2 Click on the **Middle of an Element Snap Mode** icon located across the top of the Face Window.



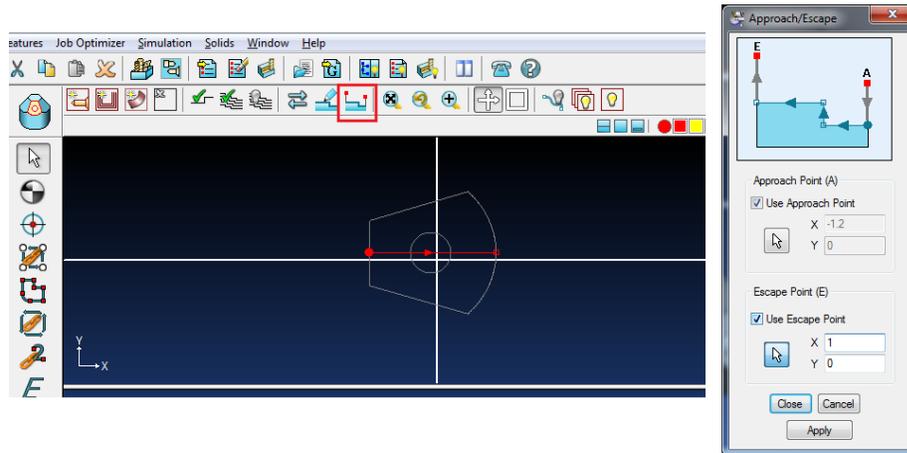
- 3 Select the left side of the shape by clicking the left mouse button as indicated above.
- 4 Now select the right side of the shape by clicking the left mouse button as indicated above.
- 5 Your screen should now have a tool profile on it. This will mill the flat surface on one of the flats.

Your window should look like the one below:



- 6 Choose the **Selection** icon and select the tool path that you created on your screen by clicking on it.

- 7 Select **Approach, Escape** from the **Part Features** drop down menu after the tool path is highlighted. Enter the parameters as shown here: This will allow the end mill to start and exit its path off of the work piece. Approach – 1.2 (-30) , 1 (25).

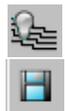
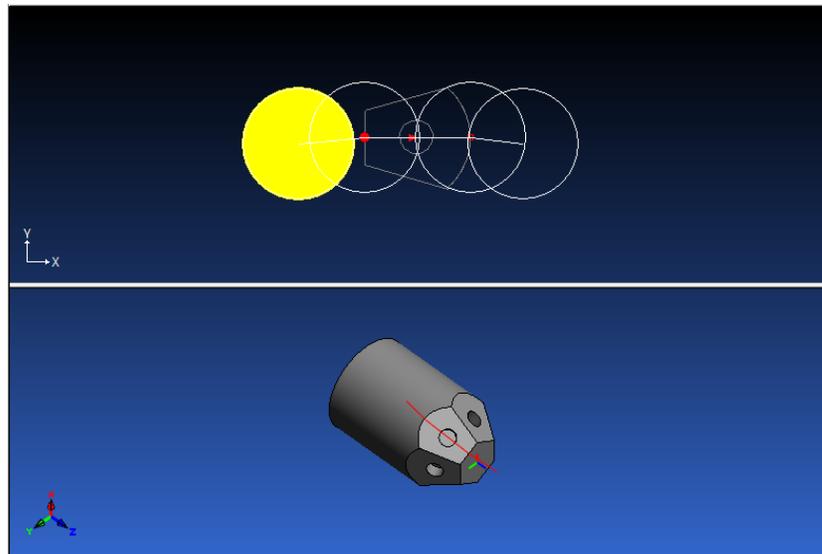


**Note:** Approach, Escape Command; this command allows to enter coordinates for either approach or escape point for a selected profile.

### Verifying the Tool Path

You can now verify the tool path of the operation you have just programmed.

- 1 Click anywhere within the **Face Window** to un-highlight the tool path.
- 2 Select **Verify Work Group Tool Path** from the **Part Features** to see the tool path displayed below:



- 3 When you are satisfied that your toolpath appears as the one shown, choose **Hide Work Group Tool Path** from the **Part Features** to hide the tool path.
- 4 Now save your work by pressing **<CTRL+S>**. You have finished the contour mill of the flat face.

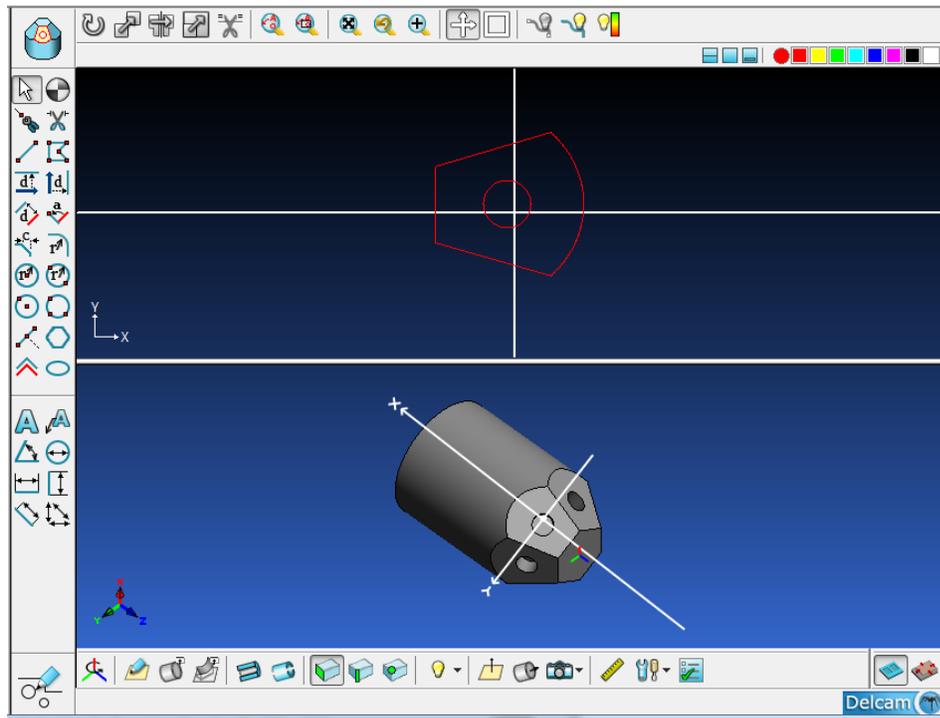
The next step will be to program the interpolated hole.

## Modifying Geometry in CAD Mode

In this section, you will edit the geometry for the interpolated hole.



- 1 Go to **CAD** mode by clicking on the CAD/CAM switch.
- 2 Choose the selection icon and select the circle in the screen. It should now be highlighted.
- 3 Choose **Divide Equal** from the **Edit** Menu as shown below to break the circle into 2 pieces of geometry.



- 4 Click anywhere in the Face Window to deselect.

## Creating the Milled Pocket

In this section, you will create the milled hole.



- 1 Go to **CAM** mode by clicking on the CAD/CAM switch.
- 2 Select a new color from the color palet located at the top right of your screen.

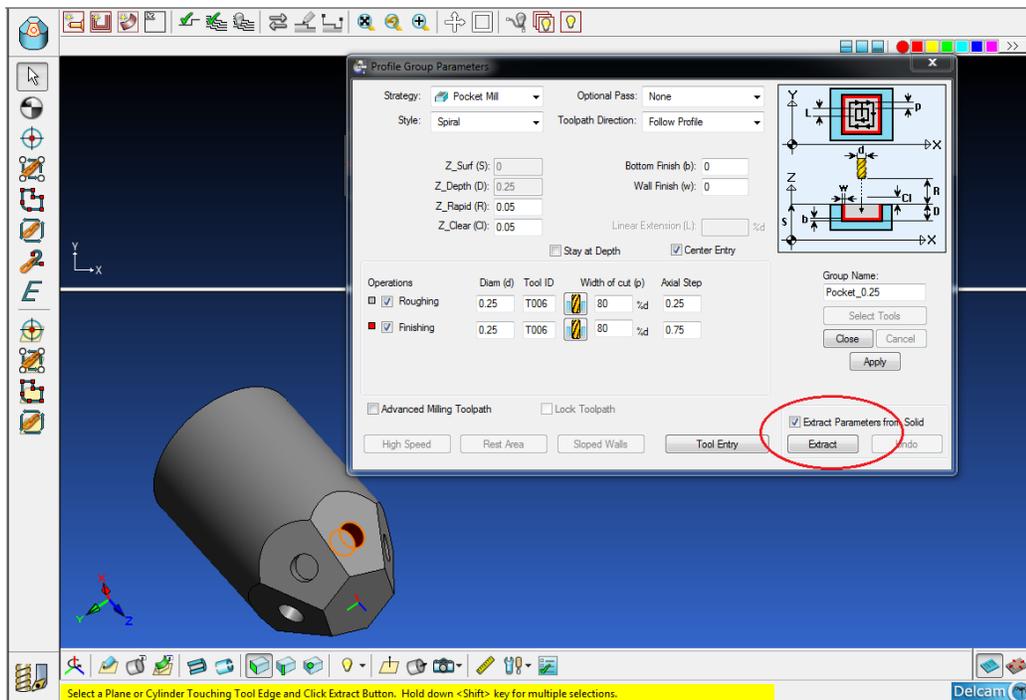


- 3 Select New Profile Group from the Part Features menu.
- 4 Choose **Pocket Mill** from the **Strategy:** drop down menu.
- 5 Click on the **Spiral** radio button.
- 6 Enter .010 (.250) for **Wall Finish (w):** and 0 for **Bottom Finish (b):**
- 7 Enter .250 (6.00) for **Tool Diamter** number. Click on the **<Select Tools>** button. The endmill icon will appear to the right of the **Tool ID** box.



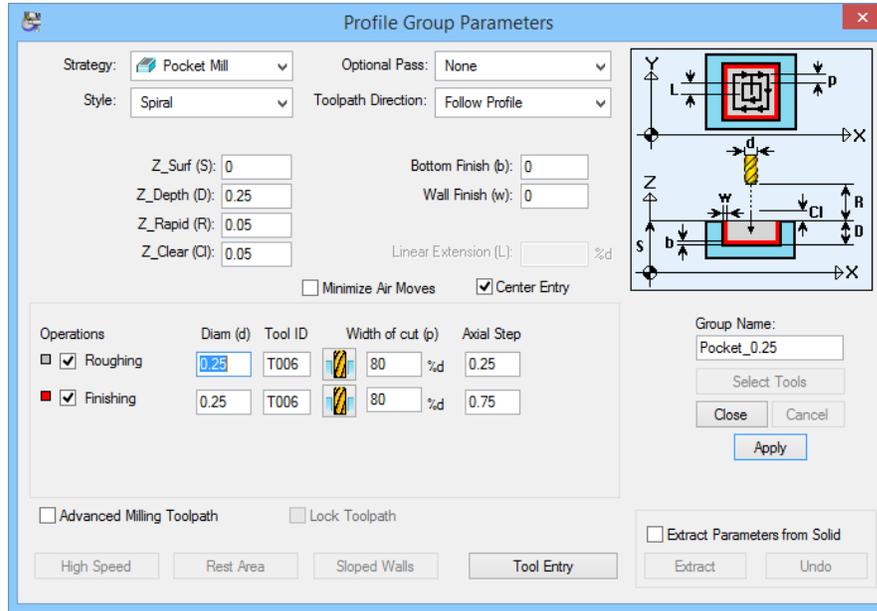
**Note:** In this exercise we will be taking two passes.

- 8 Check Roughing and Finishing.
- 9 In the Group Name: box type Pocket\_0.25 (6.00)
- 10 Select the check box for Extract Parameters from Solid.
- 11 Select the side wall of the hole as shown below:



- 12 Press the Extract button and the depth will be entered.

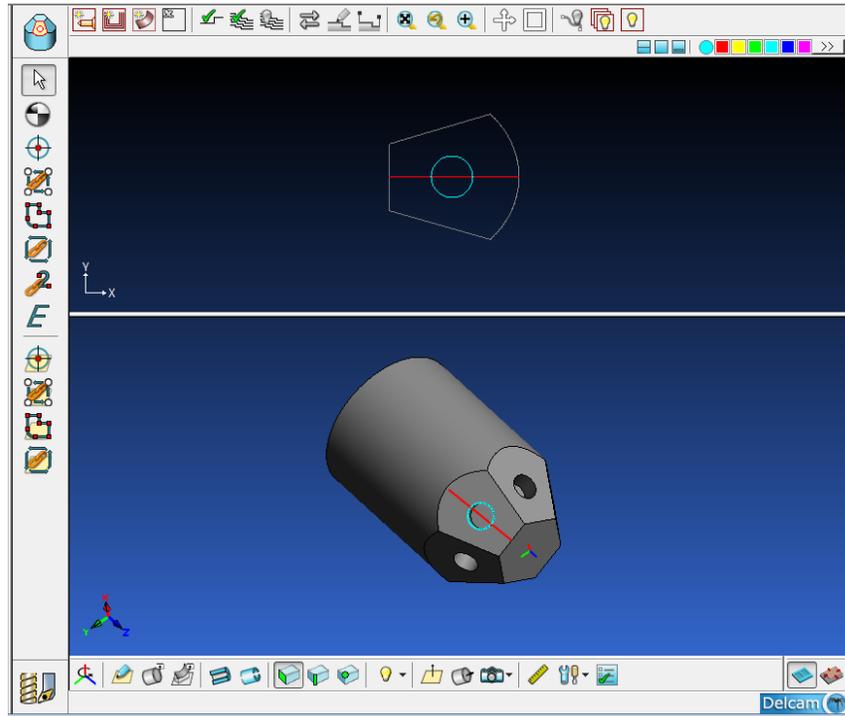
- 13 Once you are satisfied that your dialog appears as the ones below, click **<Apply>** and **<Close>** to exit the dialog.



You will specify the hole you want to machine. To do so:



- 1 Select the **Chain Geometry** icon to select the hole on the PartMaker mill window  
or  
Use the Select Geometry on Solid Model icon to select the hole on the solid.
- 2 Click your cursor somewhere near the hole you want to machine. In this case you will select the upper right side of the hole this will cause the profile to be in a counter clockwise rotation.



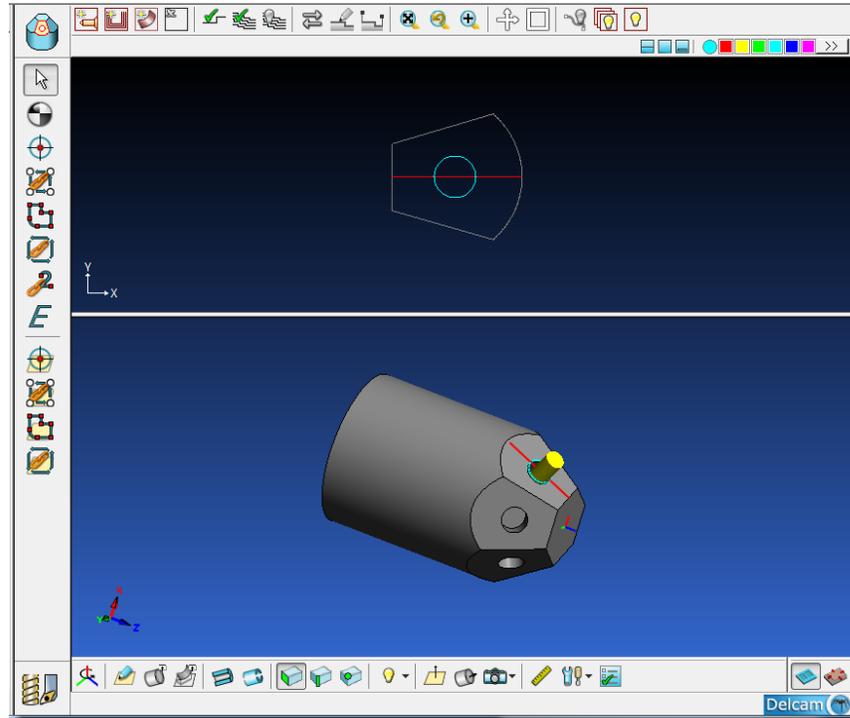
The hole you have chained should now be highlighted in the **Sample Color**.

## Verifying the Tool Path

You can now verify the tool path of the operation you have just programmed.



- 1 Click anywhere within the **Face Window** to un-highlight the holes.
- 2 Select **Verify Work GroupTool Path** from the **Part Features** to see the tool path displayed below:



- 3 When you are satisfied that your toolpath appears as the one shown, choose **Hide Work GroupTool Path** from the **Part Features** to hide the tool path displayed.

Now save your work by pressing **<CTRL+S>**. You have finished the surface and pocket milling operations for the 5 axis tutorial.

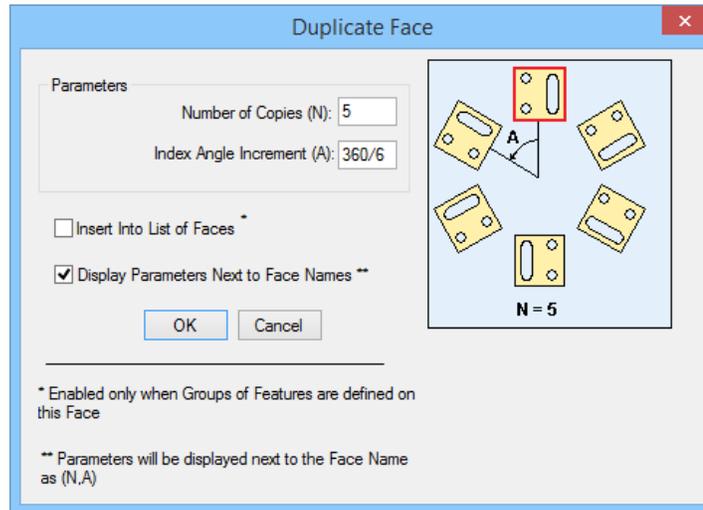
The next step will be to duplicate this window.

## Duplicating the Face Window

In this section, you will duplicate the Mill 5-Axis window you have created by using **Duplicate** function in the **Setup** dialog. You will make 5 copies of your original window at 60 degree increments. To do so:

- 1 Choose **Setup** from the **View** menu to view the **Setup** dialog box.
- 2 Click the **<Duplicate>** button to open the Duplicate Face dialog.
- 3 Enter 5 in the Number of Copies (N): field
- 4 Enter 360/6 in the Index Angle Increment (A): field.
- 5 Uncheck Insert Into List of Faces.
- 6 Check on Display Parameters Next to Face Names.

The **Duplicate** Face dialog should appear as shown below:



- 7 Click **<OK>** to duplicate the faces.

You will now see displayed beside the Mill XY name the number of copies (5) and the increment angle (60) as shown below. No new windows have been listed.



- 8 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.

## Creating the NC Program

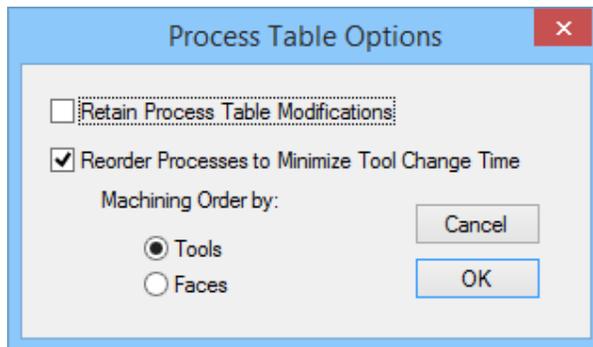
At this point, you have completed creating the faces for the tutorial part. You are now prepared to take the final steps in developing your machine specific Part Program.

### 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 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.



- 2 Click on <OK> to display the process table shown below:

| Proc ID | Tool ID | Tool No. | Tool Name    | Proc Task      | Group       | Face        | Feed    | Speed   | Time(min) |
|---------|---------|----------|--------------|----------------|-------------|-------------|---------|---------|-----------|
| P01     | T029    | 29       | End Mill_1.1 | Contour Finish | Contour_1   | Mill 5 Axis | 33.3upm | 2080rpm | 0.24      |
| P02     | T006    | 6        | Mill 1/4 - Z | Pocket Rough   | Pocket_0.25 | Mill 5 Axis | 19.8upm | 5000rpm | 0.31      |
| P03     | T006    | 6        | Mill 1/4 - Z | Pocket Finish  | Pocket_0.25 | Mill 5 Axis | 19.8upm | 5000rpm | 0.31      |

Material File: Alu\_allc.mdb Total Time: 1.45 min.

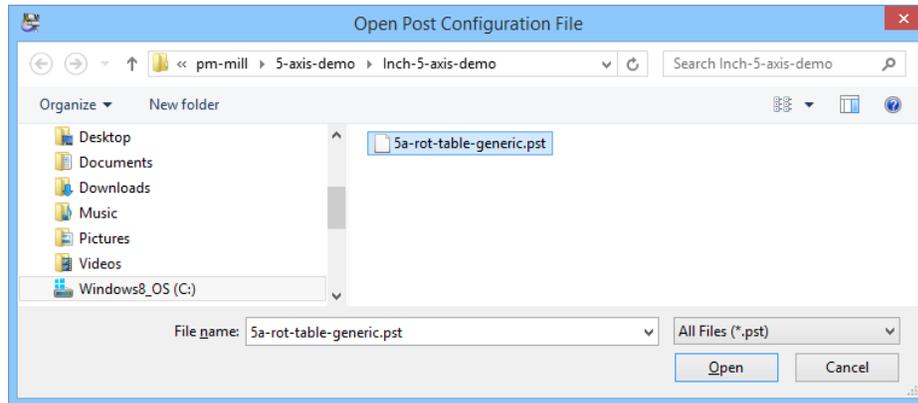


**Note:** Only three processes are listed in the process table but each of the six faces will be machined. The overlapping rectangles  to the left of the Mill 5-Axis in the Face column indicate the existence of duplicated window.

## Selecting a Post Processor

Having generated a Process Table you must now select a Post Configuration File before you can output an NC program.

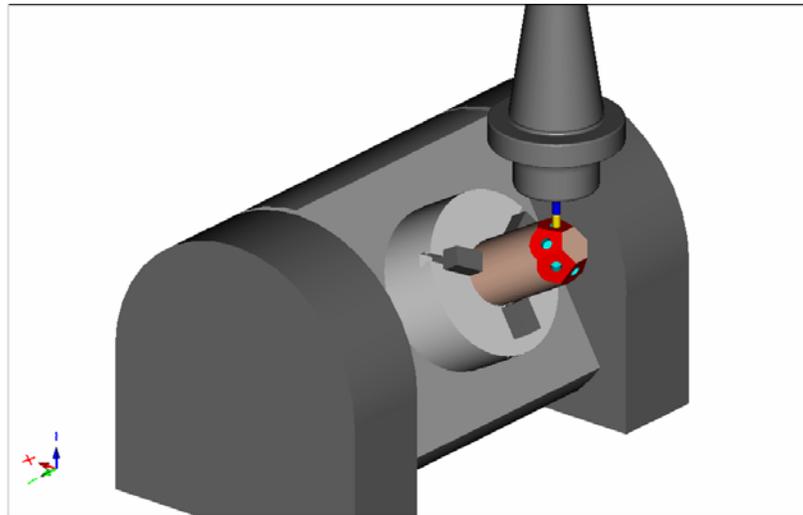
- 1 Choose Post Config File = ? from the Job Optimizer menu.
- 2 Select the **5a-rot-table-generic.pst** postprocessor from the **5-axis- demo** directory as shown below.



## Simulate the Machining of the Part

Now that the Process Table has been generated you can simulate the entire machining operation for the part

- 1 Click the **Simulation** button at the bottom left hand portion of the Process Table.
- 2 Press the Space Bar or choose **Simulation** from the **Simulation** menu.



Shown above is the part in the Simulation window.

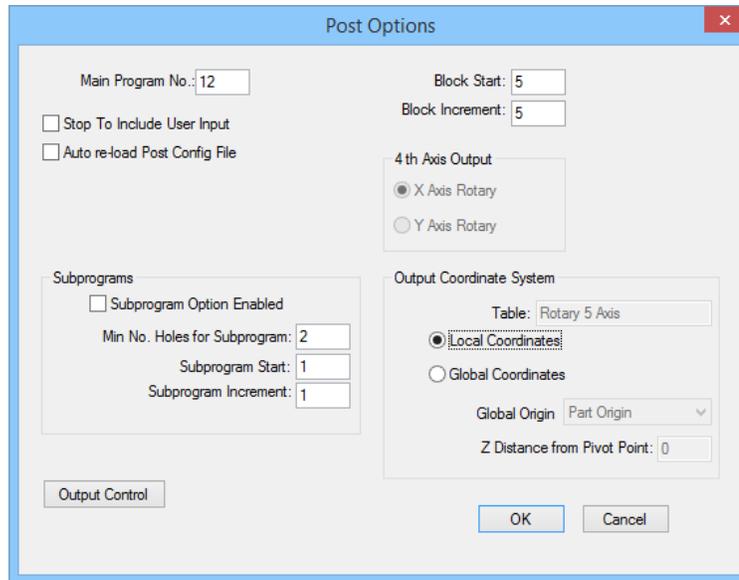
- 3 From the **Simulation** menu choose show finished part to show the completed part.

## Generate the NC program

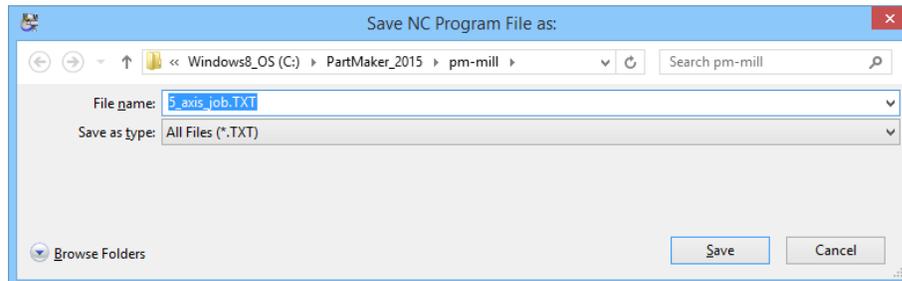
Having selected a Post Configuration file and you are now ready to generate an **NC Program**.



- 1 Choose Generate NC Program from the Job Optimizer menu.

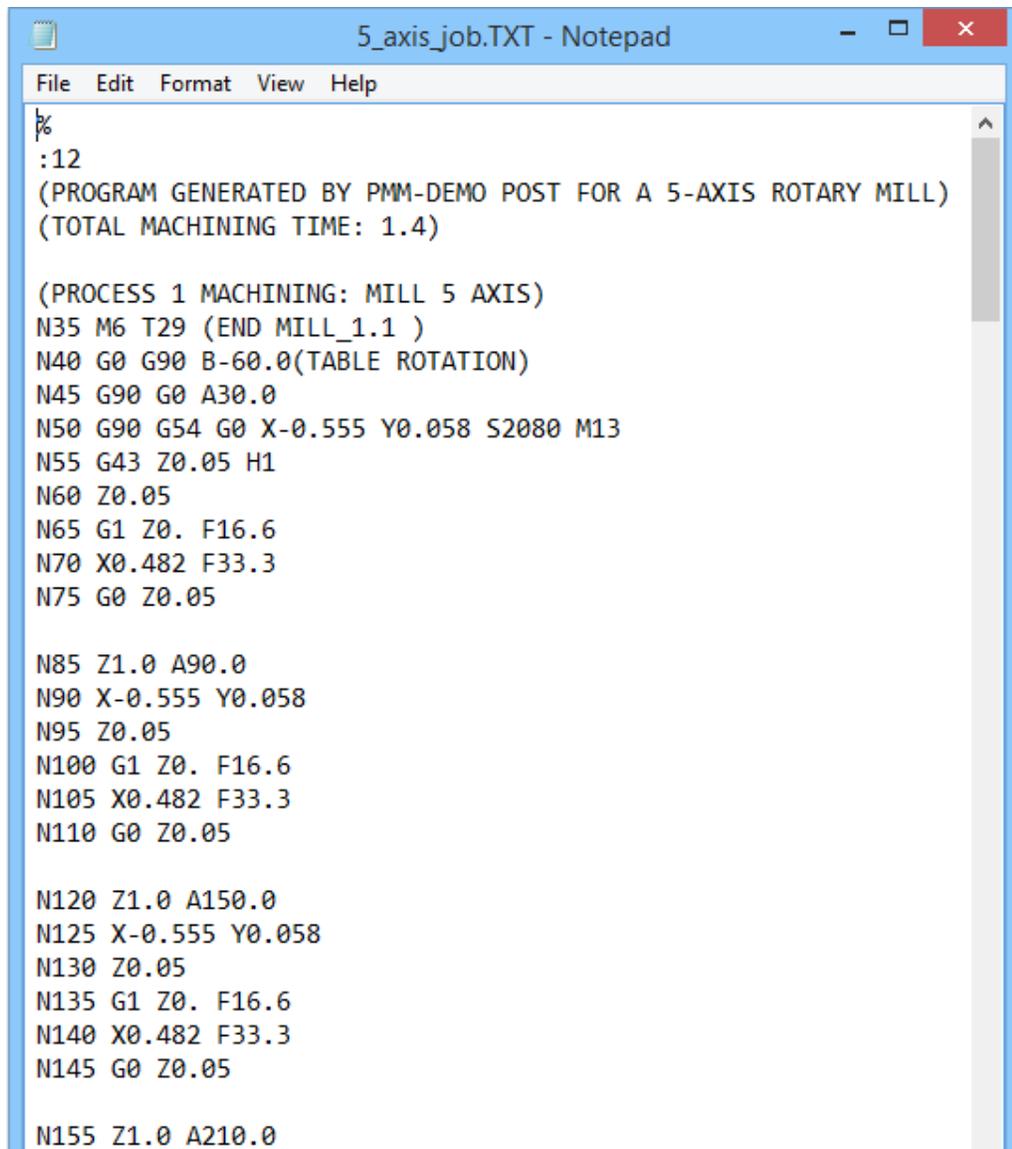


- 2 If your machine requires NC-code coordinates in the local inclined plane, select **Local Coordinates** from the 5 Axis Output section of the Post Options dialog. If your machine requires global coordinates, select **Global Coordinates**.
- 3 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog.
- 4 In this dialog, enter a name for your program in the **File Name:** field.



- 5 Click **<Save>** to generate the NC program.

The completed **NC Program** is shown below:



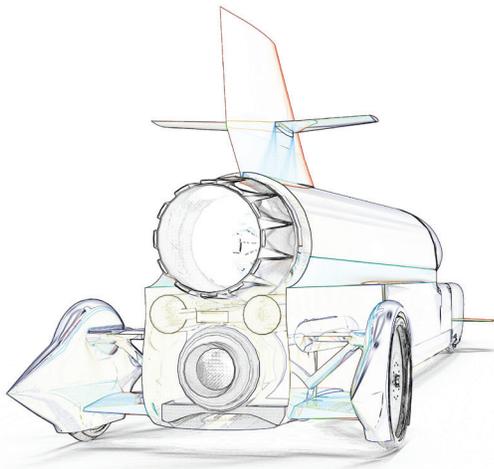
```
File Edit Format View Help
%
:12
(PROGRAM GENERATED BY PMM-DEMO POST FOR A 5-AXIS ROTARY MILL)
(TOTAL MACHINING TIME: 1.4)

(PROCESS 1 MACHINING: MILL 5 AXIS)
N35 M6 T29 (END MILL_1.1 )
N40 G0 G90 B-60.0(TABLE ROTATION)
N45 G90 G0 A30.0
N50 G90 G54 G0 X-0.555 Y0.058 S2080 M13
N55 G43 Z0.05 H1
N60 Z0.05
N65 G1 Z0. F16.6
N70 X0.482 F33.3
N75 G0 Z0.05

N85 Z1.0 A90.0
N90 X-0.555 Y0.058
N95 Z0.05
N100 G1 Z0. F16.6
N105 X0.482 F33.3
N110 G0 Z0.05

N120 Z1.0 A150.0
N125 X-0.555 Y0.058
N130 Z0.05
N135 G1 Z0. F16.6
N140 X0.482 F33.3
N145 G0 Z0.05

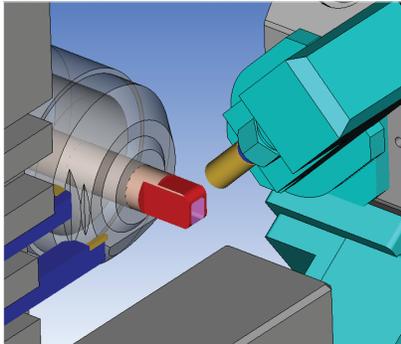
N155 Z1.0 A210.0
```



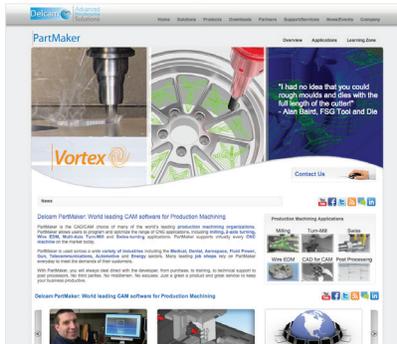
# PartMaker 2015

World-leading CAM software for production machining

[www.partmaker.com](http://www.partmaker.com)



PartMaker 2015



[www.partmaker.com](http://www.partmaker.com)



[www.delcam.tv](http://www.delcam.tv)

Unlock the power of Delcam software



## LEARNING ZONE

Visit [www.delcam.tv/lz](http://www.delcam.tv/lz) and see how Delcam software could benefit your business!



Powering your productivity

### Delcam

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

### PartMaker Inc.

550 Pinetown Road, Suite 470, Ft. Washington PA 19034 USA  
T: +215 643 5077 | E: [info@partmaker.com](mailto:info@partmaker.com)  
W: [www.partmaker.com](http://www.partmaker.com)

To contact your local reseller, visit [www.delcam.com/resellers](http://www.delcam.com/resellers)

