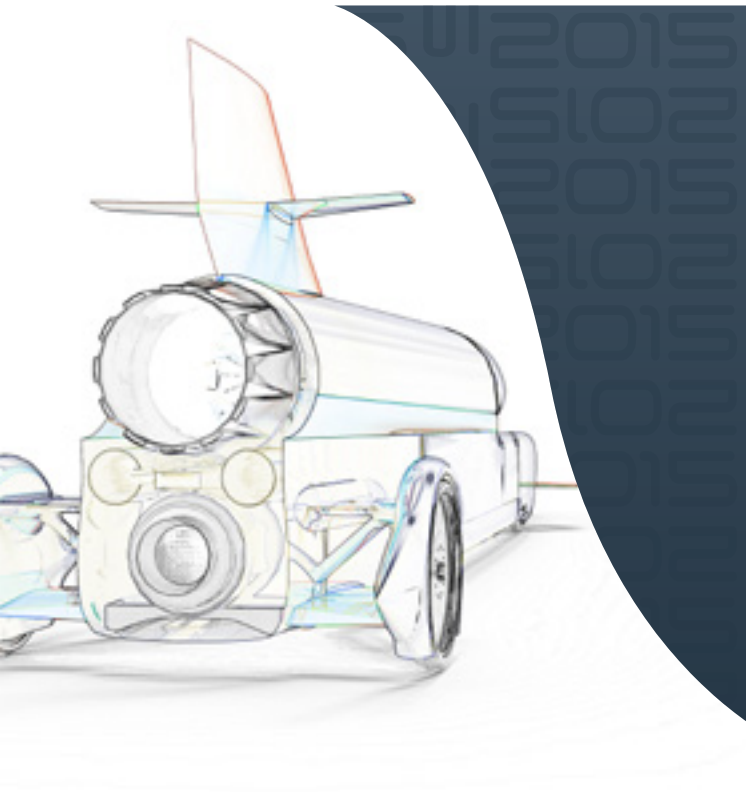


Delcam Advancec Manufacturing Solutions

PartMaker 2015

PartMaker SwissCAM Supplement

Version 2015 or Higher

User Guide

PartMaker Supplement – Chapter 1
Part 2 - Plug

This educational supplement to the PartMaker SwissCAM manual is intended to provide additional training to the end user.

Creating the Plug in this tutorial introduces PartMaker procedures step by step for ID turning, milling, threading, grooving and cross drilling. This tutorial will give many examples of the type of programming features PartMaker has, but is not limited to.

PartMaker Supplement - Chapter 2
Part 3 - Reamer

This educational supplement to the PartMaker SwissCAM manual is intended to provide additional training to the end user.

Creating the Reamer in this tutorial introduces PartMaker procedures step by step for sequencing turning and milling operations in the process table. This tutorial will give many examples of the type of programming features PartMaker has, but is not limited to.

PartMaker 2015

User Manual

User Guide/PartMaker SwissCAM Supplemental

Important User Notices

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

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

Acknowledgements

Patents

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

PartMaker 2015. Published on 08 December 2014

Contents

Chapter 1: PartMaker® SwissCAM™ Programming Tutorial Part 2 – Plug

Introduction

This tutorial is designed to help you learn the steps to go through in using **PartMaker SwissCAM** to program Swiss-type lathe parts.

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 gives 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.

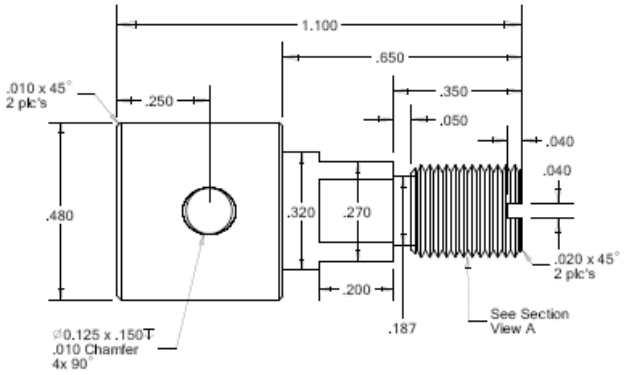
How you will create Part 2 – Plug using SwissCAM

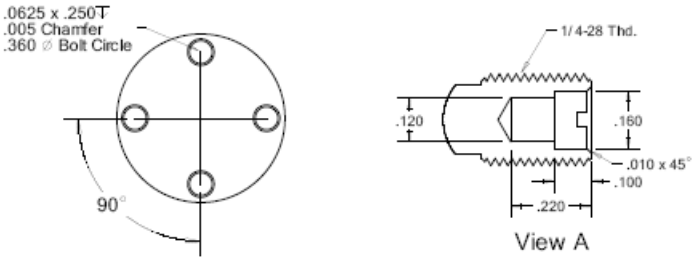
Here are the major steps you will follow to create the Swiss tutorial Part 2:

- Start PartMaker SwissCAM
- View tool information for the part
- View cycle information for the part
- Select a material for the part
- Setup defaults for Swiss style manufacturing
- Define the stock boundaries for the Face Window
- Open new **Face Windows** for each operation type (Turning and Milling)
- Create geometry
- Perform turning operations on the main spindle
- Perform a cut-off operation
- Perform Milling on the main spindle
- Perform cross drilling on the main spindle
- Perform face drilling & tapping using driven tools on the sub-spindle
- Create the Process Table
- Synchronize machining operations on the main and sub-spindles
- Simulate the Machining Process
- Generate an NC Program

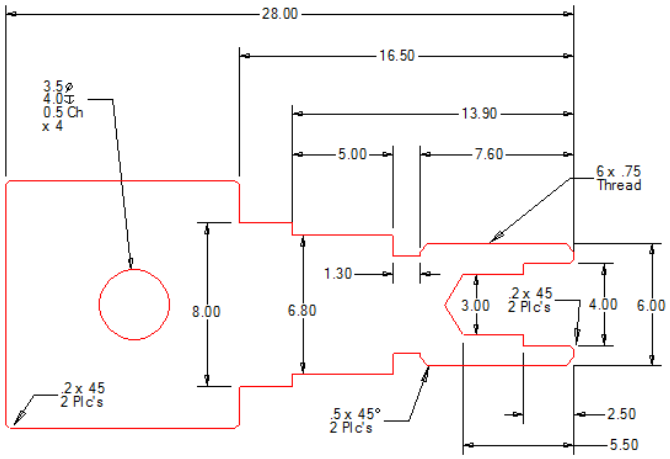
The drawings below show the part to be programmed:

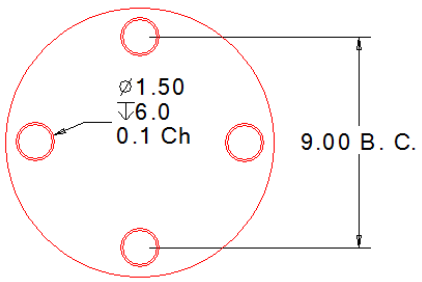
Inch Dimensions



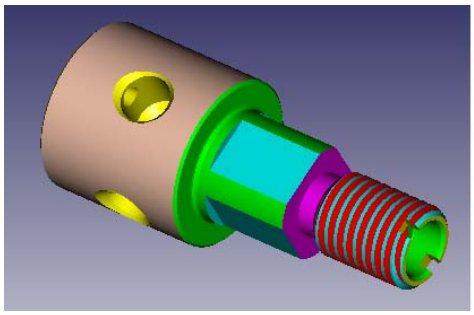


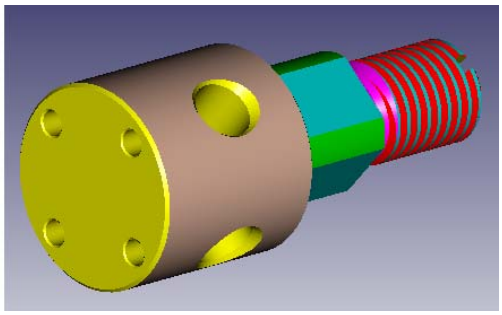
Metric Dimensions

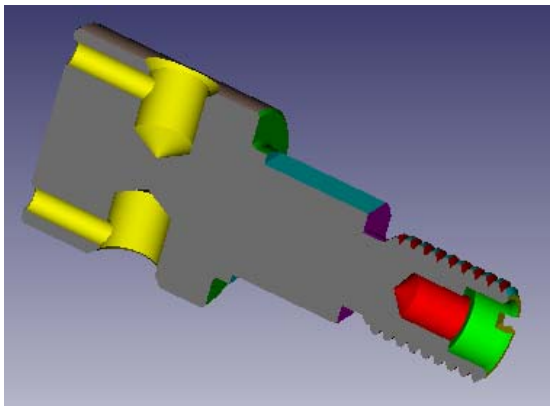




When you complete this tutorial, you will have a part that looks like the pictures below.







Getting Started

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

The machining functions used in this tutorial are as follows:

Turn – Main Spindle:

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

Mill ZY – Main Spindle:

Use a slotting mill to create a slot in the face of the part.

Mill Polygon – Main Spindle:

Use an end mill to create four flats .270 (6.8) wide towards the center of the part.

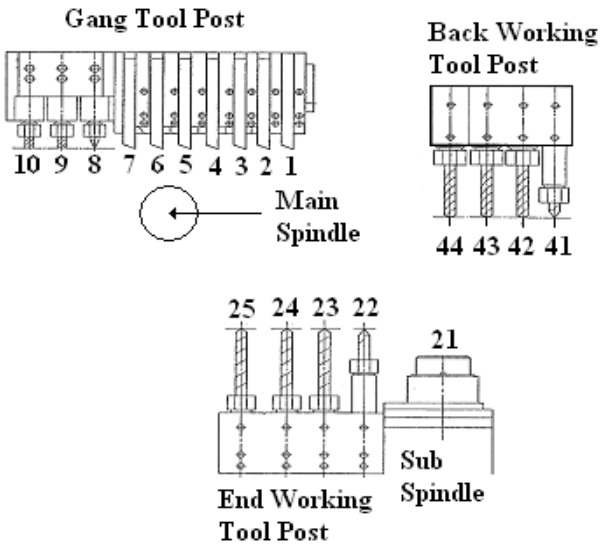
Mill Diameter Index – Main Spindle:

Use cross drilling tools to create holes in the side of the part.

Mill End Index – Sub Spindle:

Use drilling tools to create holes in the back face of the part.

Programming Swiss-type lathe operations with **PartMaker** is performed in a unique manner. No matter what the orientation of the part in the guide bushing or sub spindle, **PartMaker** programs all Swiss operations as if being machined in a conventional lathe, i.e. all programming is carried out from right to left. For this tutorial you are going to use a machine layout like the one shown below.



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

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

Completed Job File:

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\swiss2.job

Completed Tools File:

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\swiss2.tdb

Completed Cycles File:

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\swiss2.cdb

Machine Data File:

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\Swiss-demo.mch

Post Configuration Files:

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\Swiss-demo.pst

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\Swiss-demo.sub

DXF files

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\

Main_Spindle_Turn.DXF

Mill_Diam_Index.DXF

Mill_End_Index.DXF

Mill_Polygon.DXF

Mill_ZY.DXF

Solid Model Files:

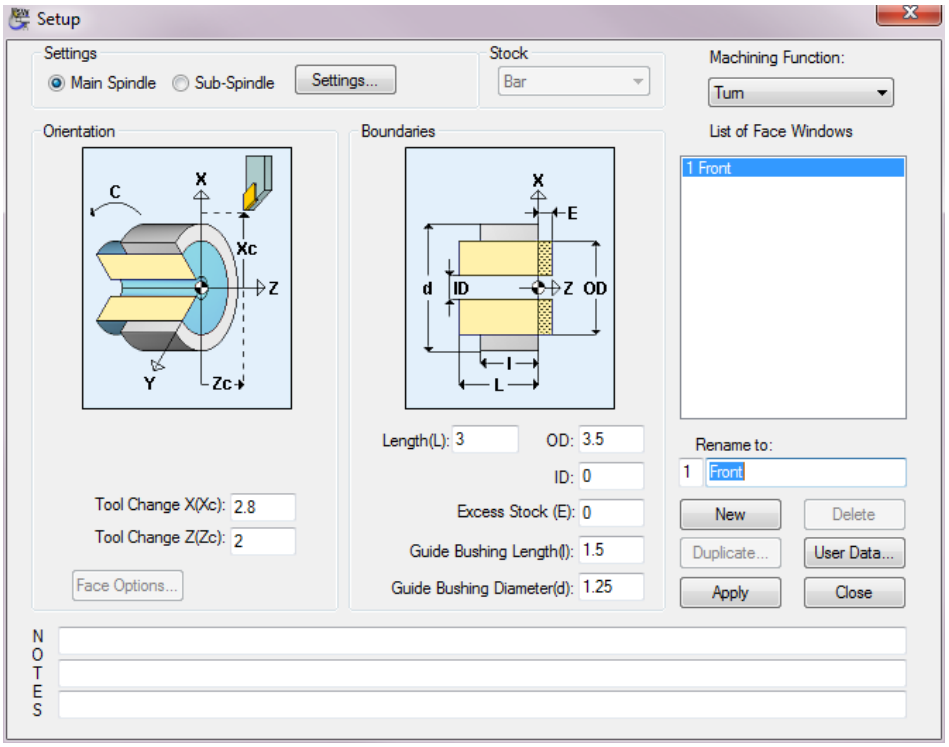
C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\swiss2.x_t

C:\PartMaker\PM-Swiss\Swiss Tutorial 2\Inch-Swiss Tutorial 2\swiss2.SLDPRT

Starting PartMaker SwissCAM

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

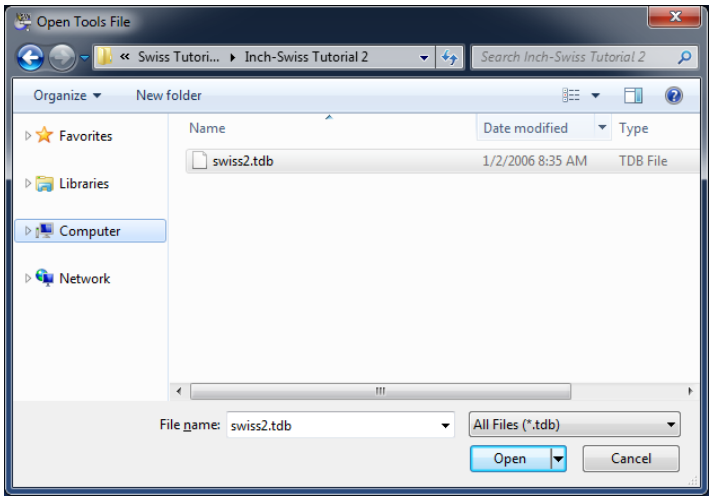
1. Double Click the **SwissCAM** icon on your desktop or choose **SwissCAM** from the **PartMaker** Group accessed through Programs under the Windows Start button.
2. An icon containing the **PartMaker** logo will appear for a few moments. You will then see the **Setup** dialog shown below. You will use this dialog to describe your part later in this tutorial. Now click the **<Close>** button.



View the Tool Information

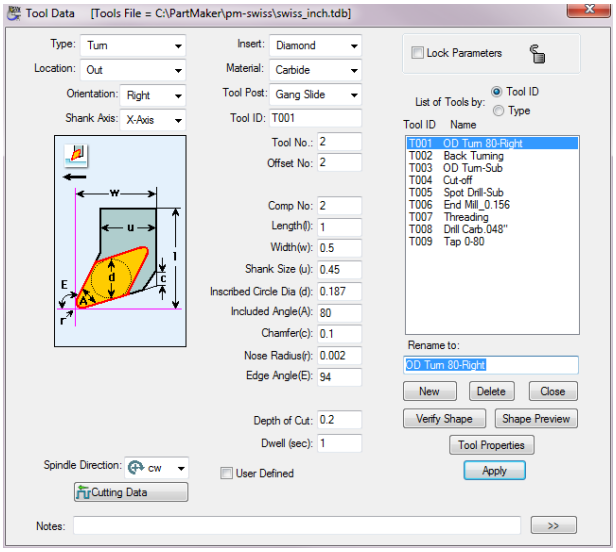
The tool library for the Swiss tutorial Plug has already been created. To view the tools:

1. Open the Tools file by choosing **Open Tools** File from the **File** menu.
2. Select the **Swiss Tutorial 2 >> Inch-Swiss Tutorial 2** (for metric use Metric-Swiss Tutorial 2) directory and click **<Open>**.
3. Select the **Swiss2.tdb** (Swiss2_metric) file and click **<Open>**.



4. Choose **Tools** from the **ToolMinder** menu, or Toolbox Icon:

The tools to be used in the Swiss Part 2 tutorial are shown below:



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****. See Chapter 2 of the PartMaker SwissCAM manual for more information on creating your own tools.*

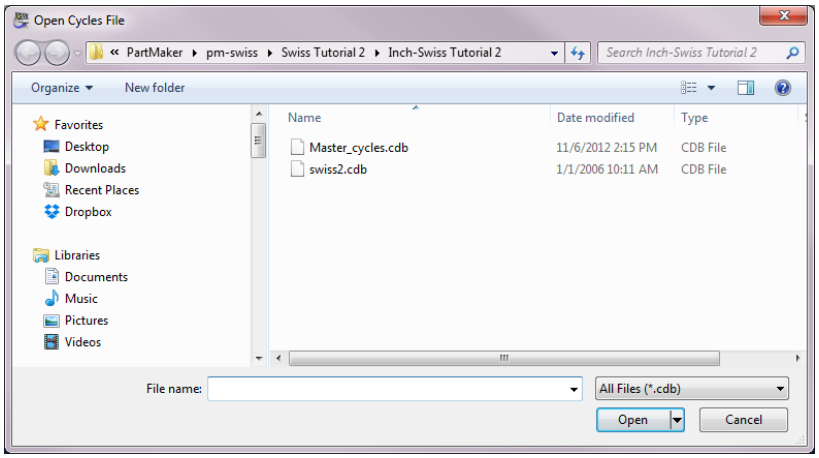
5. Click **<Close>** button to close the Tools dialog.

View the Cycles Information

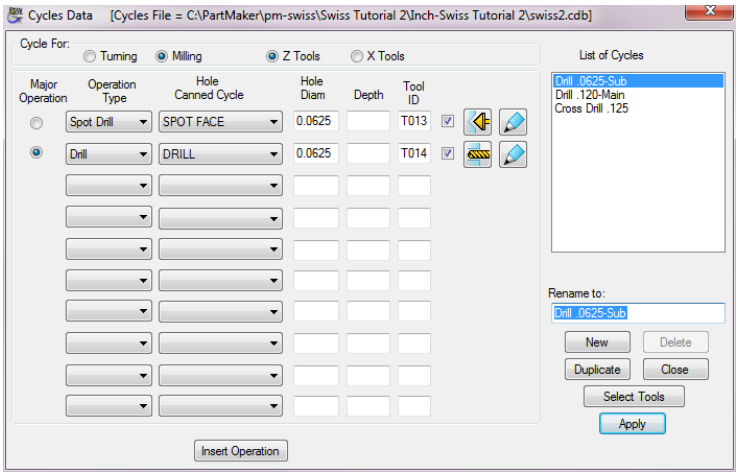
At this point, you should be familiar with the concept of **Cycles** in **PartMaker**. **Cycles** are sequences of repetitive operations used to machine a hole. The **Cycles library** for the Swiss tutorial has already been created.

You can view the cycle that you will be using for this programming exercise. To do so:

1. Open the Cycles file by choosing **Open Cycles File** from the **File** menu.
2. Select the Swiss Tutorial 2 >> Inch-Swiss Tutorial 2 directory and click <Open>.
3. Select the **Swiss2.cdb** file and click **<Open>**.



4. Choose **Cycles** from the **ToolMinder** menu or icon:



You will use several cycles to create holes on Part 2 in this Tutorial.

If you are not familiar with the concept of creating and working with cycles, please refer to Chapter 2 of the PartMaker SwissCAM User Guide in the section titled **Cycles Database** now.

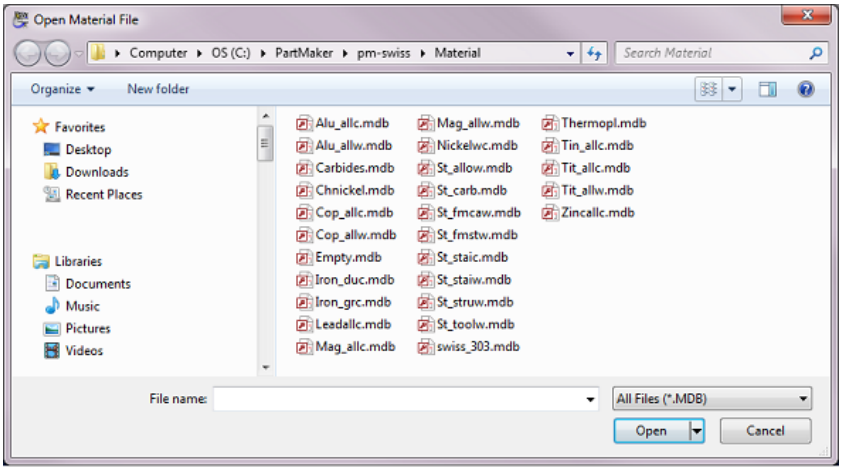
5. Click **<Close>** to close the **Cycles** dialog.

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 Select **st_fmstw.mdb** as shown below.



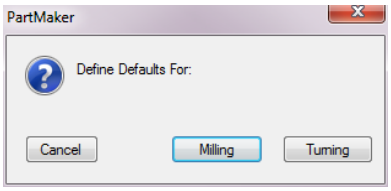
3 Click **<Open>** to open the **Material** file.

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

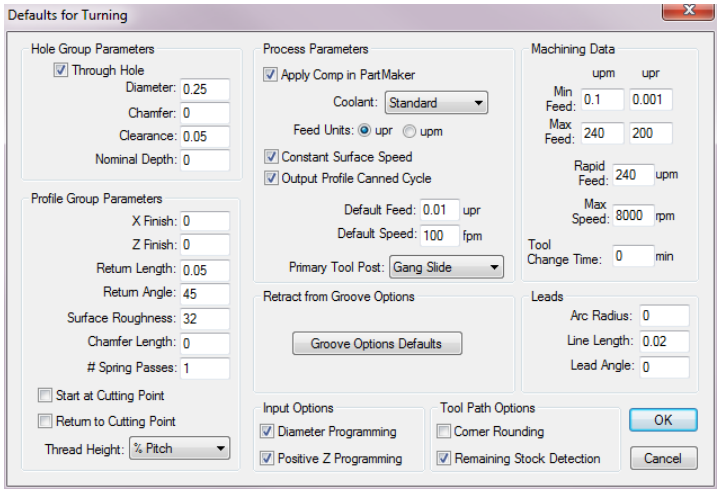
Setting Defaults For Turning

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

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



2. Click **<Turning>** button to access the **Defaults for Turning** dialog.



Note: *In Swiss machining, in order to avoid pulling stock back into the guide bushing it may be desirable not to start and return to a* ***Cutting Point*** *when programming in* ***PartMaker****. Therefore these normally are* ***not*** *checked by default. If required by a particular profile you can check these* ***ON*** *in the* ***Profile Group Parameters*** *for any process.*

3. Check the **Diameter Programming** checkbox which will allow you to define all geometry and feature programming in terms of diameter in a **Turn Face Window**, thus alleviating the need to divide measurements in half when programming or drawing on the X-axis.

   Note that **Positive Z Programming** box is checked. This tutorial is done with drawing and programming using positive Z style. This allows the Z-axis to be treated as it would be on the Swiss machine. This tutorial is created using the **Positive Z Programming** method.

4. Set a default **Line Length** for your programmed Leads. For this tutorial the value should be .020 (1.0). In the **Line Length:** field, enter **.020 (1.0)**

5. **Thread Height.** The following options are available:

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

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

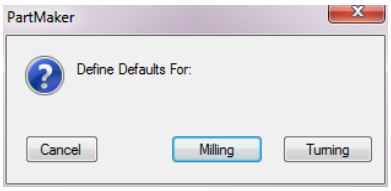
6. Click <**OK**>.

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

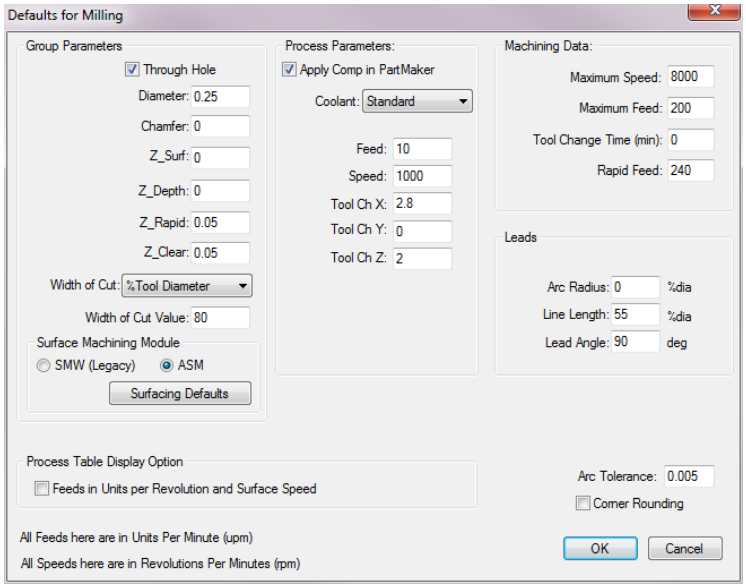
Setting Defaults For Milling

Before beginning the Swiss-tutorial, you will have to set certain milling defaults to facilitate the programming process:

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



2. Click **<Milling>** to access the **Defaults for Milling** dialog.



3. Set the default values for your programmed Leads. In Milling the defaults for **Arc Radius and Line Length** become a percentage of the diameter of the milling tool. For the Swiss Tutorial Part 2 you will set the value for **Arc Radius** to 0%, the **Line Length** to 55% and the **Lead Angle** to 90 deg.

4. **Width of Cut**. This parameter specifies whether the "**Width of Cut**" field in the Mill Profile Group Parameters Dialog will be defined in terms of percentage of tool diameter or an absolute value. Uncheck **Corner Rounding** option at the bottom right.

   Note: *Setting a generally applicable leads as a default will save you a great deal of time when programming Swiss parts because you will not have to re-enter the same data from feature to feature.*

Once you have looked at the tools and cycles databases, selected a material for the job, and setup the defaults for your job as shown above, you can move on to begin programming the Swiss Tutorial 2 Part.

Creating a Turning Operation - Main Spindle using the Turn Machining Function

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

Setting Up the Face Window

In this section, you will set up the Face Window for the turning operations on the main spindle.

To do so:

1. Choose **Setup** from the **View** menu to view the Setup dialog (or click the Setup Icon in the upper left hand corner of the screen).

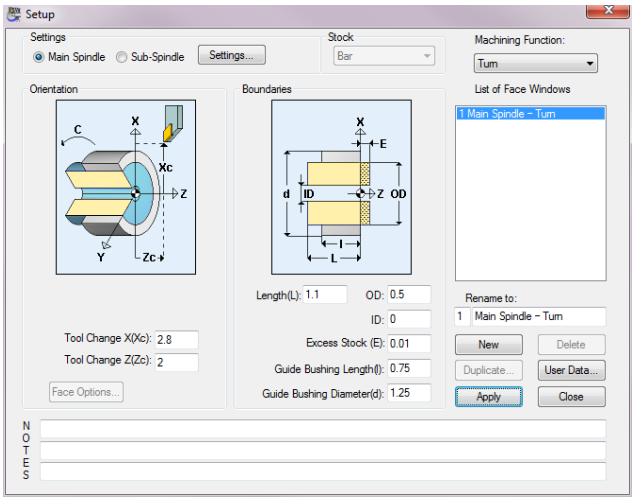
   In this dialog, you will define the machining methods, part boundaries, and location of the stock with respect to the programming origin.

   Note: *The Main Spindle radio button is selected indicating this window will be used for Main Spindle processes.*

2. Rename the current **Face Window** by clicking the **Rename to:** field and enter **Main Spindle – Turn**.
3. Click the **<Apply>** button.
4. Now enter the parameters listed below in the appropriate fields on the Setup window:

Length (L):	1.1	(28.0)
OD:	0.5	(12.7)
ID:	0	(0)
Excess Stock (E):	0.01	(0.1)
Guide Bushing Length (l):	0.75	(20.0)
Guide Bushing Diameter (d):	1.25	(25.0)
Tool Change X(Xc):	2.8	(15.0)
Tool Change Z (Zc):	2.0	(0)

5. When you are satisfied that your **Setup** dialog appears as the one below, click **<Apply>**. This will apply the parameters to the current **Face Window**.

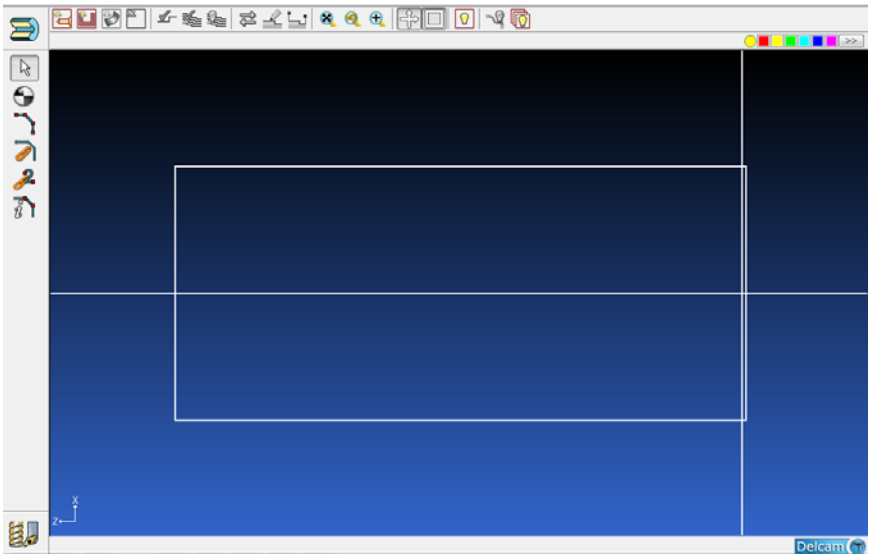


6 Click **<Close>** to close the **Setup** dialog.

7 Click anywhere in the **Face Window** to activate it.

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

9 Choose **Show Boundaries** from the **View** menu, or select the icon. This will display the boundaries of the part as you described in the **Setup** dialog.



Tip: *You can get a better view of your part by zooming out (Shown Above). The best way to do so is to perform a FULL VIEW, then roll your mouse wheel just a little bit to move the stock away from the edges of the window.*

Tip: *If at any time the menu commands become grayed out and unusable, this indicates that the* ***Face Window*** *you are working in has become inactive. This will be evident if the top bar of your window (where the name of the window is displayed) has become gray. To reactivate the window, click anywhere within your* ***Face Window****.*

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

Give your job file the name Plug.

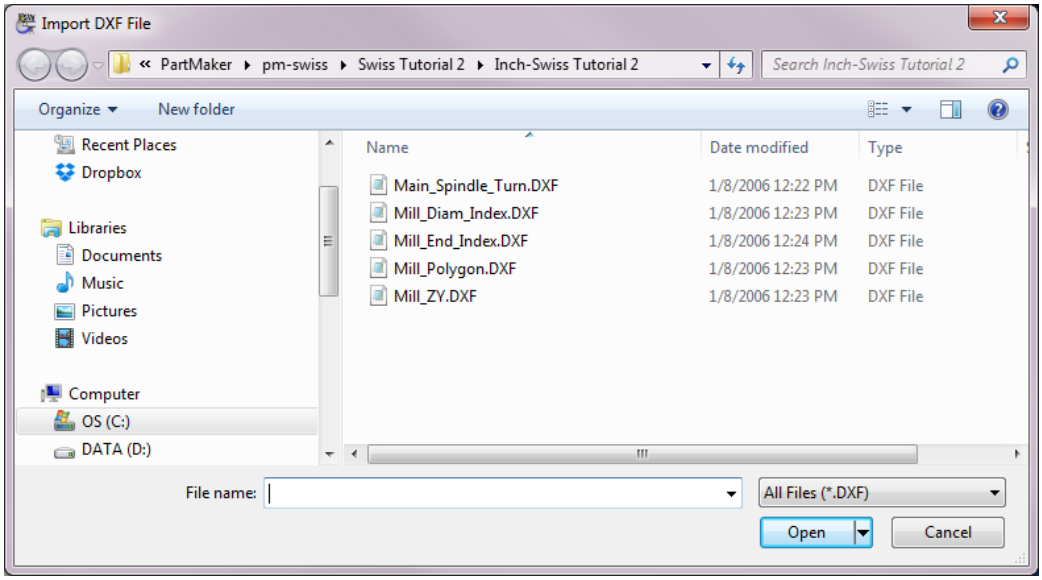
Creating Geometry in CAD Mode

In this section, you will create the geometry for the turning operations to be performed on the main spindle. If you wish to draw the geometry, skip to Method 2 on the next page.

Method 1: Import a .DXF file

1 Choose **Import DXF File** from the **File** menu or click on the Import icon and select DXF.

2 Select the **Swiss Tutorial 2 >> Inch-Swiss Tutorial 2** (Metric-Swiss Tutorial 2, for metric) directory.



3 Select Main_Spindle_Turn.DXF.

4 Click <**Open**>.

Note: *If you do not want to create the geometry for the Swiss tutorial in* ***PartMaker****, a .DXF file for each window is supplied in the* ***Swiss_Tutorial2*** *directory.*

Method 2: Create Geometry in PartMaker

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

1 Click <**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.

CAD **CAM**

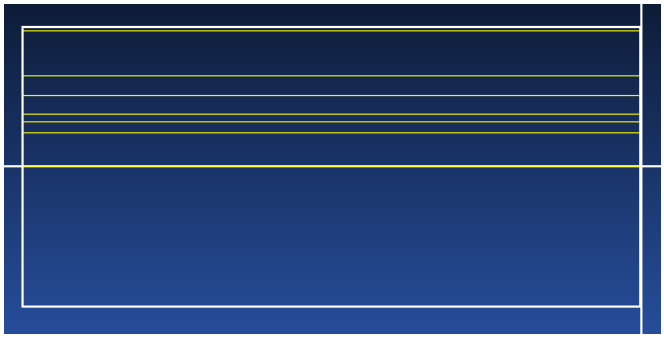

2 Select the **Line Parallel Horizontal Axis** Icon from the drawing icons on the left side of the window. This will be used to create Horizontal Parellel lines at specific coordinates.

At the Bottom of the window, a input field will appear.

3 Input into this field the value which you need to place horizontal lines in our drawing for the turning operations. Follow the print for reference. After each entry, press the **<Enter>** key to place a horizontal line on the screen at the coordinate input.

Input 0 (0)	Press <Enter>
Input 0.12 (3.0)	Press <Enter>
Input 0.16 (4.0)	Press <Enter>
Input 0.187 (4.75)	Press <Enter>
Input 0.25 (6.0)	Press <Enter>
Input 0.32 (8.1)	Press <Enter>
Input 0.48 (12.2)	Press <Enter>

You should have a drawing which appears below.



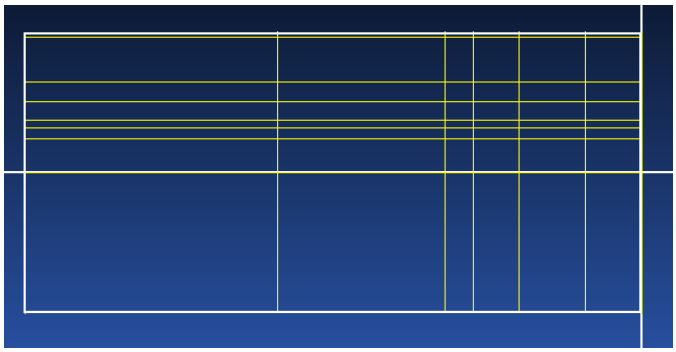
4 Select the **Line Parallel Vertical Axis** Icon from the drawing icons on the left side of the window. This will be used to create Vertical Parallel lines at specific coordinates.

5 Enter into this field the value which you need to place vertical lines in our drawing for the turning operations. Follow the print for reference. After each entry, press the **<Enter>** key to place a vertical line on the screen at the coordinate input.

Input 0 (0)	Press <Enter>
Input 0.10 (2.5)	Press <Enter>
Input 0.22 (5.5)	Press <Enter>
Input 0.30 (7.62)	Press <Enter>
Input 0.35 (9.0)	Press <Enter>
Input 0.65 (16.5)	Press <Enter>
Input 1.1 (28.0)	Press <Enter>

6 After entering these coordinates, Use the **View Menu** and choose **Hide Boundaries** , then **Hide Axis** to get a better view of the CAD drawing.

You should have a drawing which appears below.

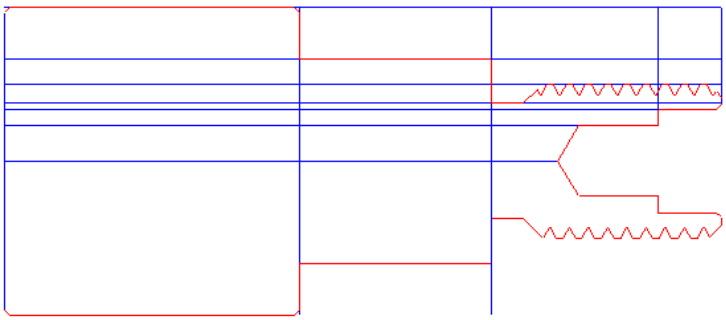


7 Select the **Remove** Icon to delete the lines not needed.

Note: *The* ***Remove*** *Icon (scissors) will cut a line to it's closest intersection with another geometry element. Please see the user manual for more information on the* ***Remove*** *Icon.*

Working from Right to Left, you want to trim away lines that have nothing to do with the actual contour of the part. For purpose of clarity, you can see below the contour outline in red with the drawing lines in blue.

Tip: *If you can not tell which element is the correct element, you can choose the* ***Selection*** *Icon (the arrow in the upper left) and double click on any element to see the geometric location of that element.*



8 Continue trimming away lines until you final contour appears as seen below.

Note that the line at X = .25 (6), which represents the top of the threads should remain.

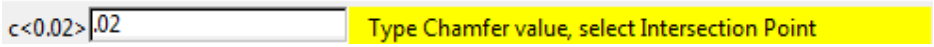
As best applies, use the REMOVE ICON or highlight a line, then press **DELETE** or **BACKSPACE** to remove any totally unwanted lines.

With very little effort, your drawing should appear as seen here below:

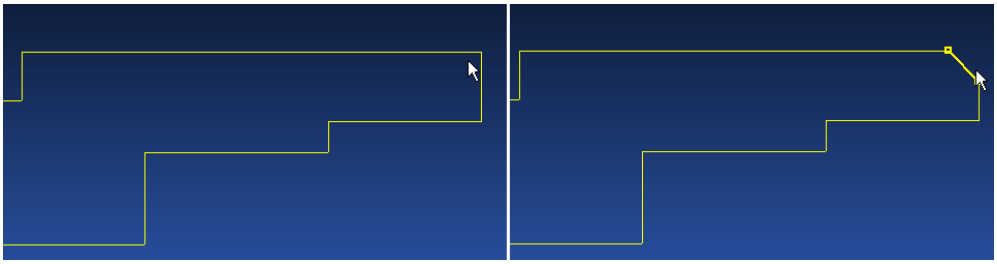
You will now place a 0.02 x 45 (.5 x 45) degree chamfer on the front of the part.

1 To do so choose the **Chamfer** Icon.

2 A field will then open up at the bottom of the window prompting you to enter the size of the chamfer. Enter a value of 0.02 (.5) and press **<Enter>**. The mouse pointer will also change to a new arrow with a **C** in it.



3 Place the point of the new mouse icon on the inside of the corner you want the chamfer to be placed and click the mouse to insert a chamfer.



4 Next, enter 0.01 into the field at the bottom of the screen for the 3 other chamfers. You will click the inside of each corner that requires the 0.01 chamfer.

Note: *You do not have to enter the 0.01 into the field for each chamfer.* ***PartMaker*** *will remember the last setting and use it each time you make a chamfer.*

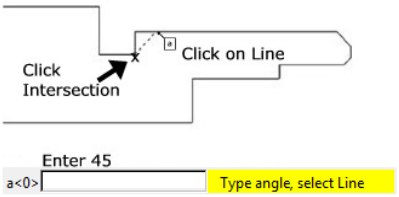
1 Create a 45 degree angle from the groove up to the threaded area. Choose the **Line on an Angle** Icon.

2 First you will decide how to choose a start point. For this in the upper part of the screen click the **Closest Intersection Snap Mode**.

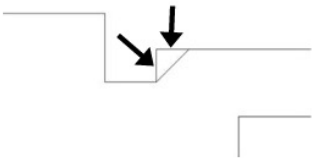
3 Click at the intersection at the bottom right hand corner of groove as shown below.

4 Enter 45 into the field at the bottom of the screen for the degrees of the angle.

5 Click the line you are drawing this angled line to.



6 Now that the line is created, use the **Remove** Icon to cut away the geometry not required, as indicated by the arrows.



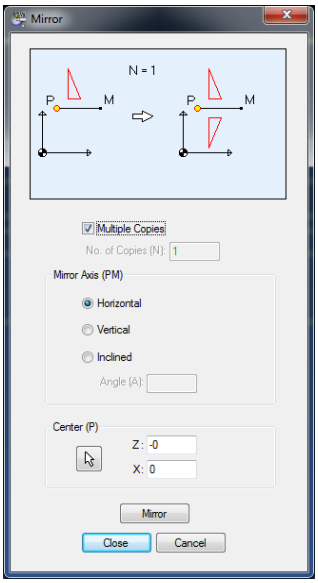
The drawing should then look like this.

You will now mirror the selected geometry to have a complete part on the screen. To do so:

1 Select the geometry you have created by clicking on the **Selection** Icon and choosing **Select All** from the **Edit** menu. All the geometry should be highlighted on the screen.

Note: *This mirroring is not required but will help you better visualize the part in the CAM mode and will help with clearances on internal processes.*

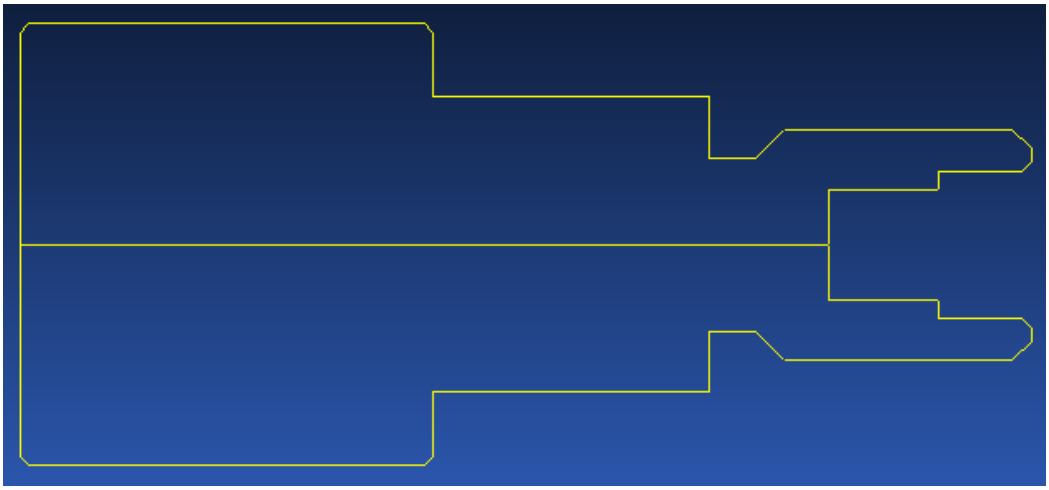
2 To mirror, select the whole geometry and choose **Mirror** from the **Edit** menu or select the icon. Complete the **Mirror** dialog as shown below:



3 When you have completed the dialog, click **<Mirror>** and <**Close**>.

4 Click anywhere in the window with your mouse pointer to deselect the geometry.

The completed drawing should appear as below:



Note: *Once you finish drawing your geometry for the Main Spindle Turn Face Window, you can either continue drawing the geometry for other Face Windows or switch to creating cutting profiles for each Face Window. For this tutorial you will create geometry and cutting profiles before moving to the next Face Window.*

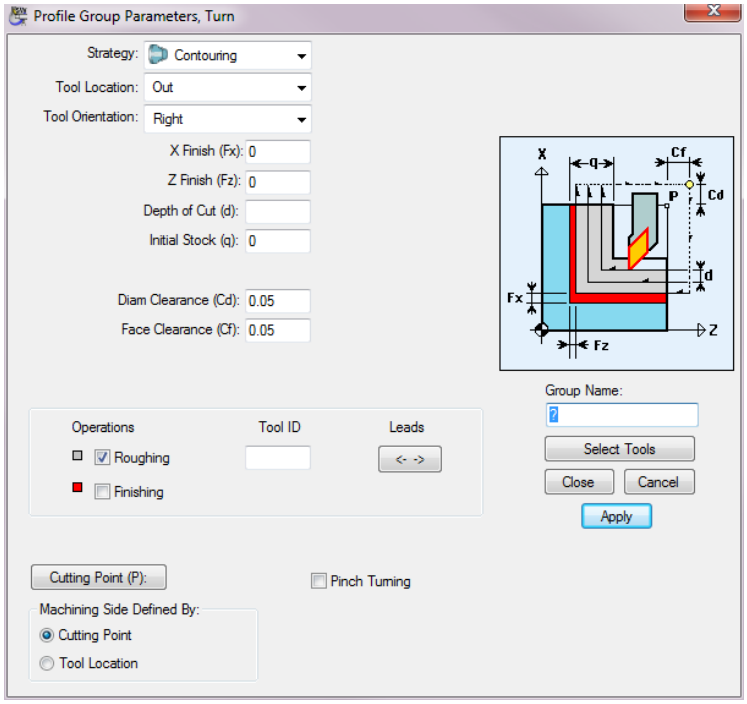
Defining the Profile for Face Turning

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

1 Click **<CAD/CAM>** switch to go to CAM mode.

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

3 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Group Parameters** dialog. When selected, you should see a dialog which appears as the one below:

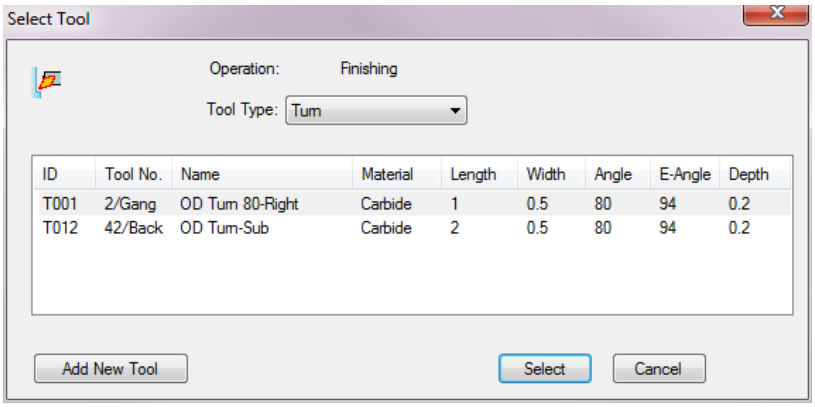


4 From the **Strategy** menu, select **Contouring** (it should be chosen by default).

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

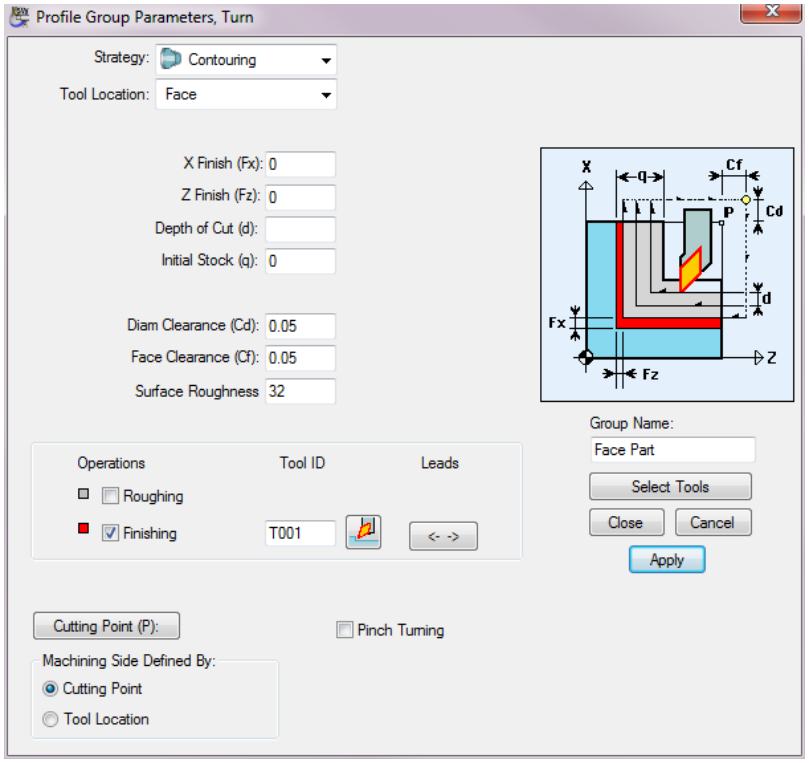
6 Uncheck the **Roughing Tool ID**: box this will automatically cause the **Finishing Tool ID**: to be checked.

7 Click <**Select Tools**> and select the appropriate tool. PartMaker will present you a choice of two tools, one is mounted on the main spindle and the other is on the sub spindle as shown below:



8 In this case you are doing main spindle work, so choose **T001** and click **<Select>**. You are choosing T001 because this tool has been setup up for work on the main spindle and the **Tools Database** has been designated as **Gang Tool**.

9 In the field called **Group Name**, enter **Face Part**. Your dialog should now appear as shown below:

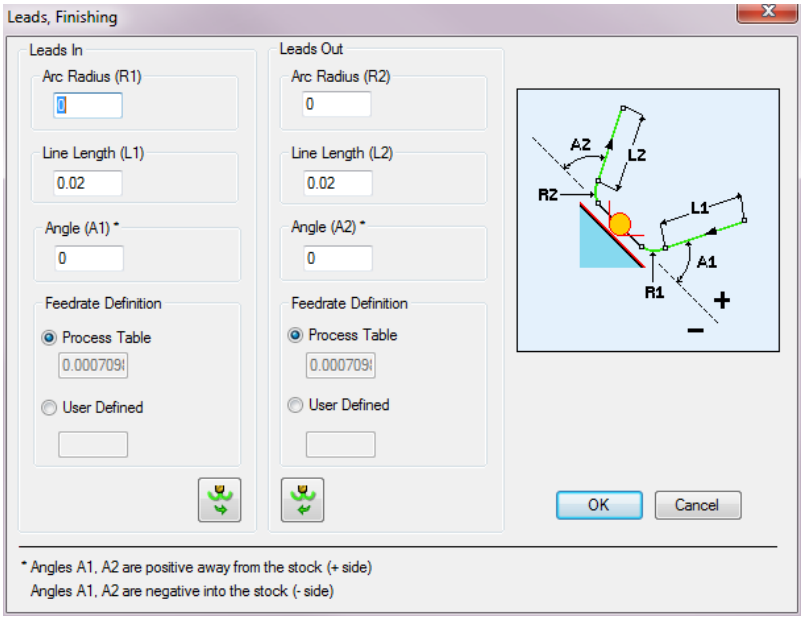


Leads In/Leads Out

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

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

1. The **Leads In** and **Out** dialog box combine the settings for controlling the movement of the tool as it approaches (Leads In) and leaves (Leads Out) the stock.
2. Set the **Leads In** values for the operation.
3. You can also set the values from one **Leads** to the other by selecting the Copy Leads button.



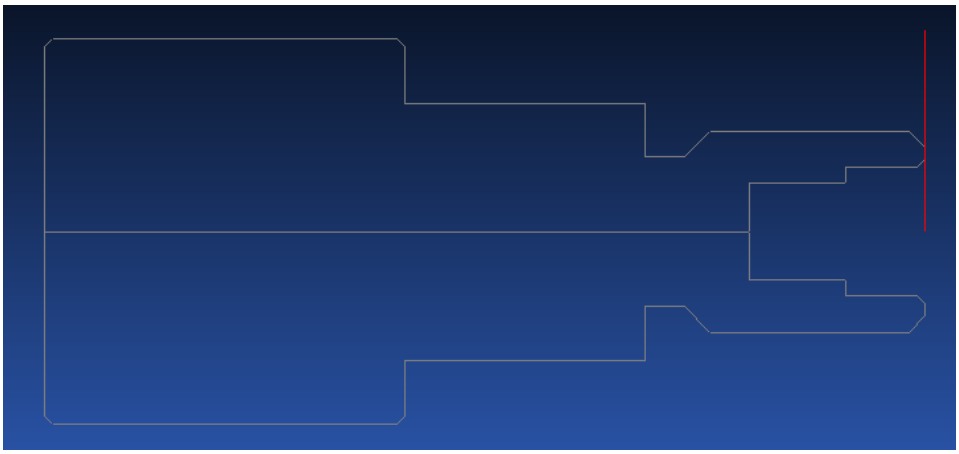
Press **<OK>** to return to the **Profile Group Parameters** dialog.

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

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

Notice, that by choosing a **Tool Location** of **Face**, **PartMaker** automatically applied your facing tool path to the part geometry.

Your completed profile should appear as below:



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

Tool Path Verification

Once a tool path is defined, you can verify its shape to gauge the accuracy of the cut you have defined. In **PartMaker SwissCAM**, for programming being done on the **Main Spindle** in a **Face Window** of **Machining Function Turn**, you have the option of performing two types of tool path verification:

- **Fixed headstock** simulation

  Or

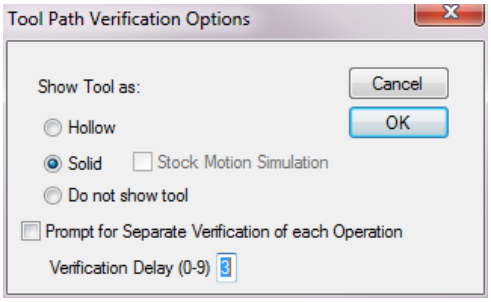
- **Sliding headstock** simulation (Stock Motion Simulation)

Each type of verification has its relative merits. Fixed headstock simulation is excellent when focusing on the cut being made by a particular tool path. The sliding headstock simulation, referred to as **Stock Motion Simulation**, is excellent for seeing how your part will move through the guide bushing during machining and to assure that the Tool Shifts have been set properly.

Note: *When performing, the* ***Stock Motion Simulation****, a guide bushing of the OD and Length described in the* ***Setup*** *dialog will be shown.*

Fixed Headstock Simulation

1. Select Verify Work Group Tool Path from the Part Features menu. You will be presented with the Tool Verification Options dialog.
2. Check the **Solid** radio button
3. Enter a delay between 0 and 9 (3 is a good starting number). The higher the number, the slower the verification will run.



4. Click **<OK>** and you will see your tool path verified.

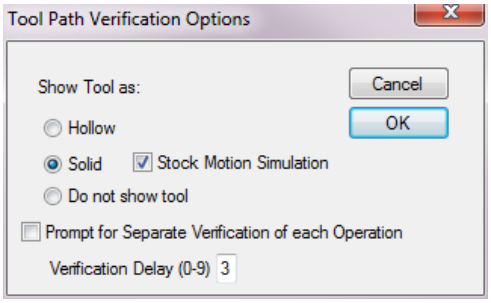
5. Once you are satisfied with your tool path, choose **Hide Work Group Tool Path** from the **Part Features** menu.

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

Sliding Headstock Simulation

Note: *Sliding Headstock Simulation is only possible in a Face Window of Machining Function type Turn used for Main Spindle operations.*

1. Select Verify Every Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options Menu.
2. Check the **Solid** radio button and the Stock Motion Simulation.
3. Enter a delay between 0 and 9 (3 is a good starting number). The higher the number, the slower the verification will run.



4. Click **<OK>** and you will see your tool path verified with the stock moving and a Guide bushing drawn on the screen.

Note: *you may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*

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

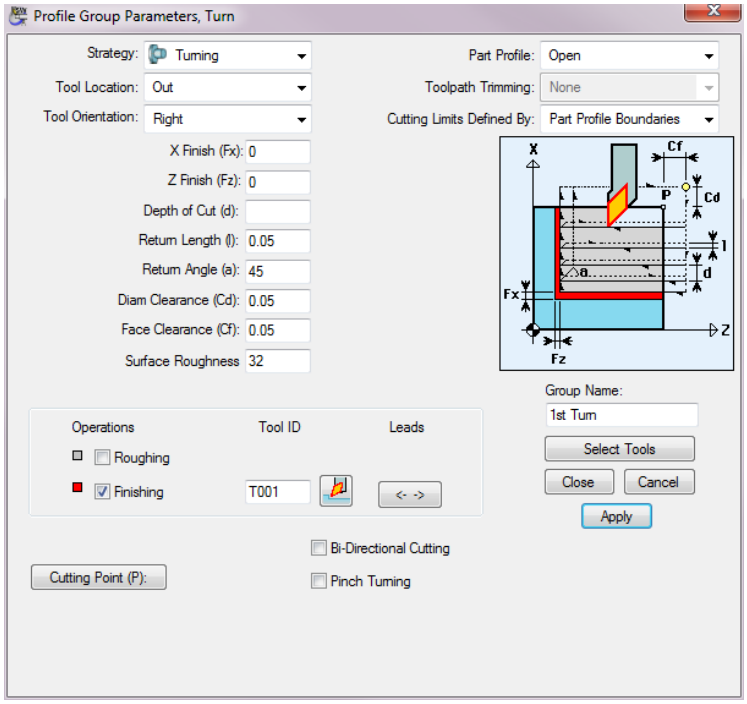
Defining the 1st OD turning profile

1. Click anywhere in the **Face Window** to make it active.
2. Choose red from the **Color Bar** in the upper right corner. 
3. Choose **New Profile Group** from the **Part Features** menu, or icon to display the **Profile Group Parameters** dialog.

4. From the **Strategy** menu, select **Turning** (The turning setting allows PartMaker to stop from Overcutting on certain profiles).
5. From the **Tool Location** menu, select **Out**.
6. From the **Tool Orientation** menu, select **Right**.
7. Remove the check from the **Roughing Tool ID**: which will automatically cause the **Finishing Tool ID**: to be checked.
8. Click <**Select Tools>** and select the appropriate tool.
9. In this case you are doing main spindle work, so choose **T001** and click **<Select>**. You are choosing T001 because this tool has been setup up for work on the main spindle and in the Tool Database has been designated as a **Gang Tool**.
10. In the upper right corner, select **Part Profile Boundaries** from the **Cutting Limits defined By:** drop-down list.

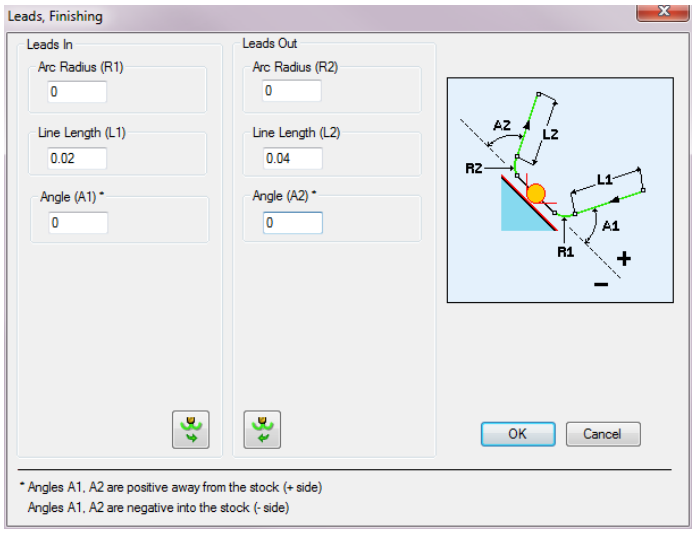
    Note: *Selecting Part Profile Boundaries moves the* ***Cutting Limits*** *to the edges of the part profile. For more information see* ***Cutting Limits*** *in the user manual or Help files.*

11. In the field called **Group Name**, enter **1st Turn**. Your dialog should now appear as shown below:



Leads

1 Now you will set the lead in and lead out for your profile. Click the **Leads** button and **Leads** dialog will appear. Set the line lengths as they appear below:



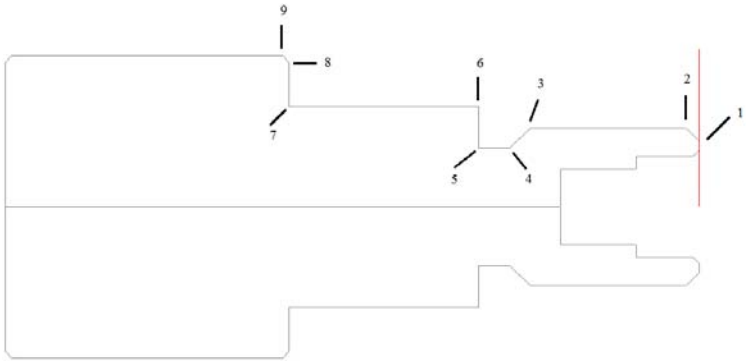
2 Click **<OK>** when done.

3 You have now finished entering the parameters for the **Face Part Profile Group**. Click **<Apply>** and **<Close>** to accept all changes.

Assign The Profile

1 Select the **Define Profile Icon** and then the **Closest Intersection Snap Mode** icon

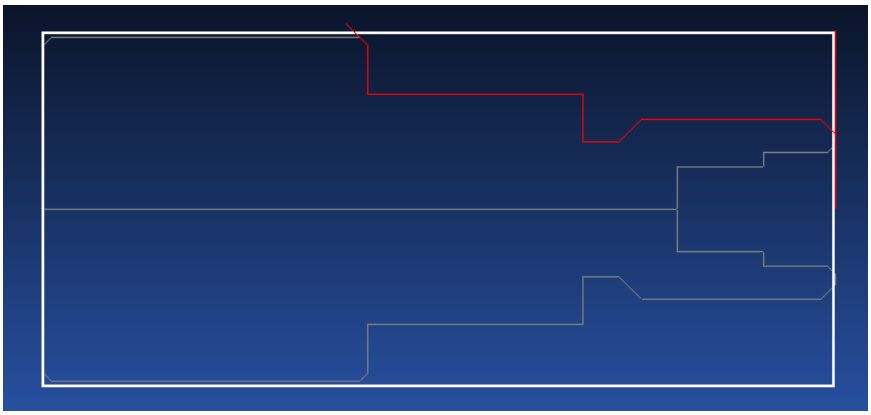
2 Click near the intersections on the screen in the order indicated below to create the profile. You will notice as you click from point to point that the profile is being created.



3 After clicking the last intersection, select the **Screen Snap Mode** icon.

4 Now click on the screen out side of the part boundaries.

5 Click the **Selection** Icon on the left side of the screen. This will end the profile. Your completed profile should appear as shown here:



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

Tool Path Verification

Fixed Headstock Simulation

1 Select Verify Work Group Path from the Part Features menu.

2 Check the **Hollow** radio button

3 Click **<OK>** and you will see your tool path verified.

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

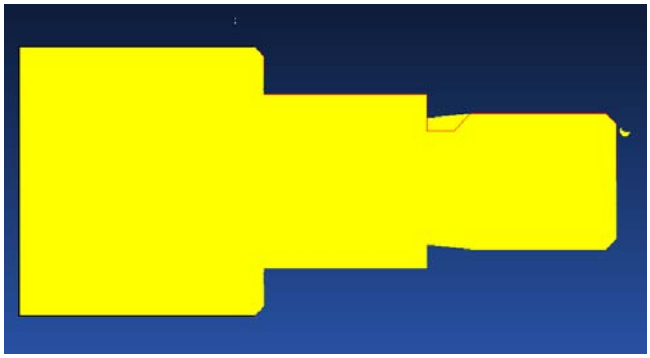
Sliding Headstock Simulation

1 Select **Verify Every Tool Path** from the **Part Features** menu. You will be prompted with the **Tool Verification Options** menu.

2 Check the **Solid** radio button.

3 Click **<OK>** and you will see your tool path verified with the stock moving and a Guide bushing drawn on the screen.

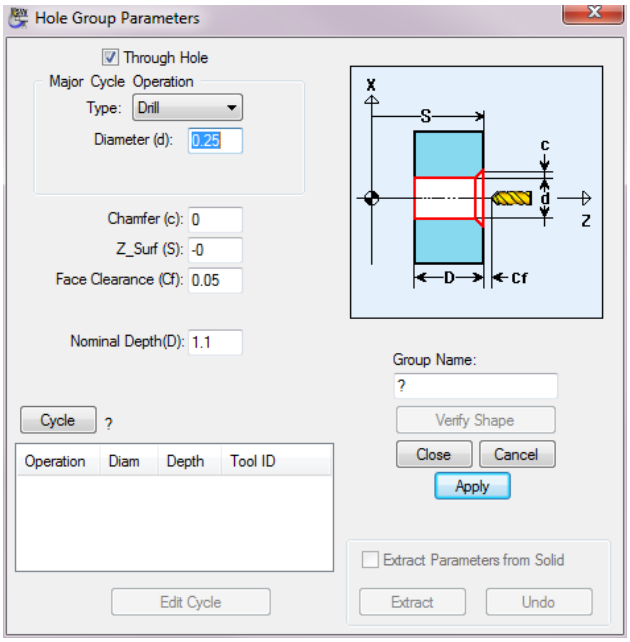
Note: *You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*



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

Defining a Drilling Operation

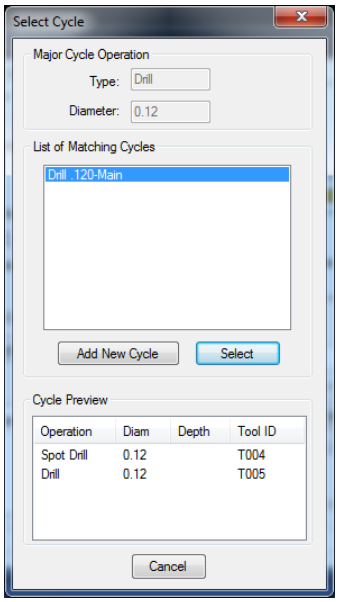
1. Click anywhere in the **Face Window** to make it active.
2. Choose Cyan from the **Color Bar** in the upper right side. 
3. Choose **New Hole Group** from the **Part Features** menu to display the **Hole Group Parameters** dialog. When selected, you should see a dialog which appears as the one below:



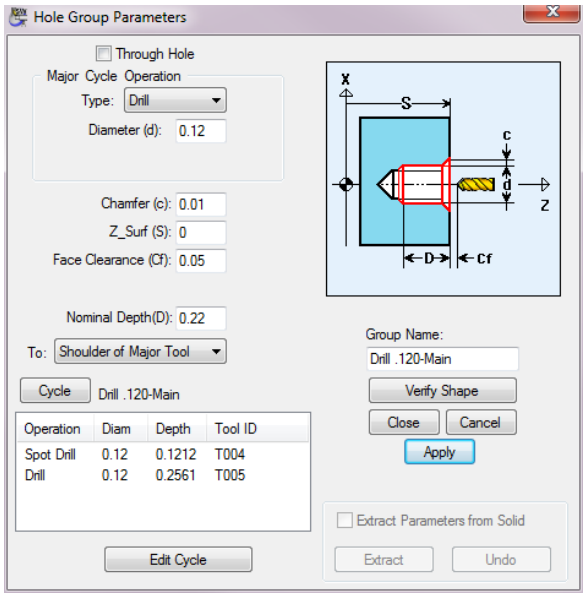
4. Uncheck the **Through Hole** box.
5. Under **Major Cycle Operation** set the **Type** of Operation to **Drill**.
6. Under **Major Cycle Operation** enter .120 (3.0) in the **Diameter** field.

   Note: *The **Major Cycle Operation** parameters are used by **PartMaker** to search the **Cycles Database** for an appropriate cycle.*
7. Enter 0.010 (1.0) into the **Chamfer (C)** field. **PartMaker** will calculate the depth of a Spot Drill according to the size of the Chamfer entered and the angle of the spot drill.
8. Enter 0 into the **Z_surf (S)** field.
9. Enter .050 (1.0) into the **Face Clearance (Cf)** field.
10. Enter .220 (5.6) into the **Nominal Depth (D)** field.
11. In the **To:** drop down menu under **Nominal Depth (D)**, select **Shoulder of Major Tool** to indicate where **PartMaker** will calculate the depth of the tools.

12 Click <**Cycle**> and **PartMaker** will search the **Cycles Database** for the appropriate cycle to create the desired hole by clicking **<Select>**.



13 You will notice a cycle is chosen and the tools and calculated depths are listed in the field in the lower left of the **Hole Group Parameters**. The dialog should now appear as below.



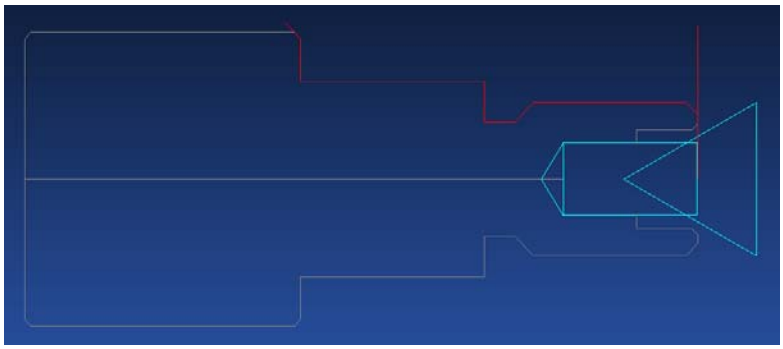
14 Click **<Verify Shape>**. This will display a graphical representation of the cycle.

15 Click **<Hide Shape>** when done, returning to the **Hole Group Parameters**.

16 Click **<Apply>** to accept all the information.

17 Click **<Close>** to close the Hole Group Parameters.

On the Screen you will now see a Profile of a Spot Drill and a Drill. Creating a Hole Group in a Turning window allows PartMaker to create the tool paths automatically from the information provided.



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

Tool Path Verification

Fixed Headstock Simulation

1 Select **Verify Work Group Tool Path** from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.

2 Check the **Hollow** radio button

3 Click **<OK>** and you will see your tool path verified.

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

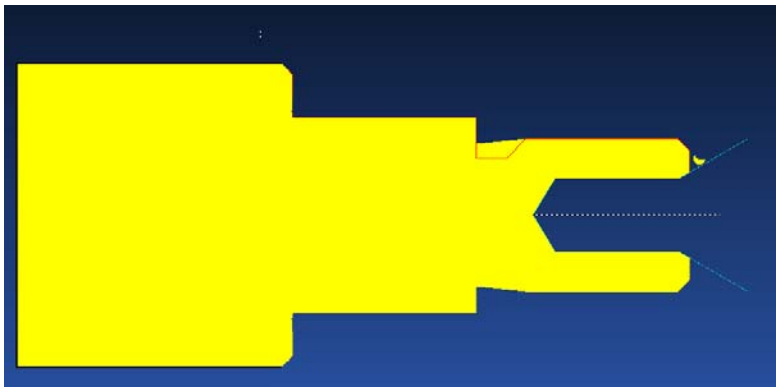
Sliding Headstock Simulation

1 Select Verify Every Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.

2 Check the **Solid** radio button.

3 Click **<OK>** and you will see your tool path verified with the stock moving and a Guide bushing drawn on the screen.

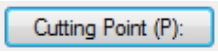
***Note**: You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*



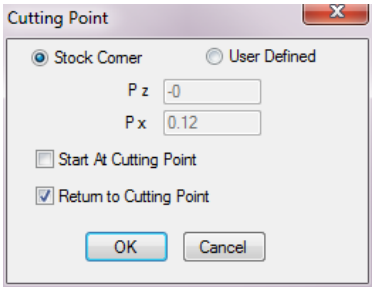
4 Once you are satisfied with your tool path, choose **Hide Every Tool Path** from the **Part Features** menu.

Defining the ID Boring Operation

1. Click anywhere in the **Face Window** to make it active.
2. Choose yellow from the **Color Bar** in the upper right corner. 
3. Choose New Profile Group from the Part Features menu.
4. From the **Strategy** menu, select **Contouring**.
5. From the **Tool Location** menu, select **In**.
6. Enter 0.0 into the **Diameter Clearance (CD)** field.
7. Remove the check from the **Roughing Tool ID**: which will automatically cause the **Finishing Tool ID**: to be checked.
8. Click **<Select Tools>** and select the appropriate tool.
9. Click **<Cutting Point (P):>** button to display the Cutting Point dialog.



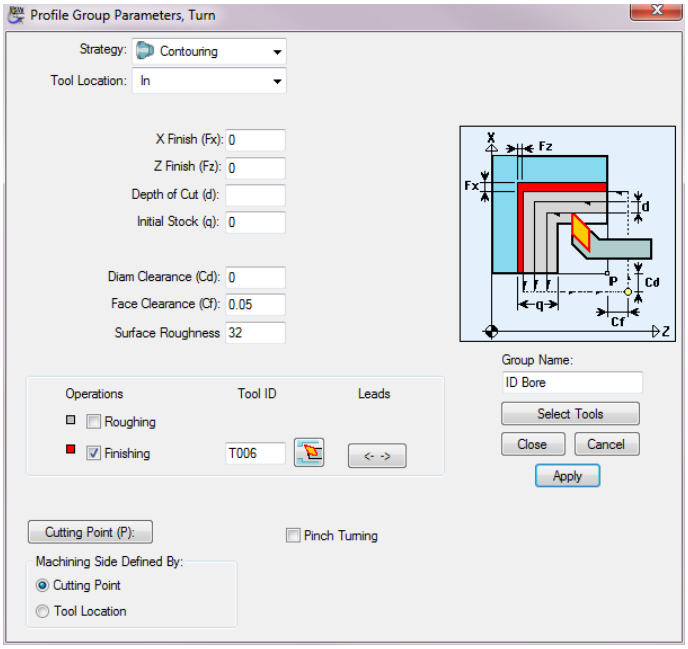
10. Check Return to Cutting Point.



***Note**: Placing a check by **Return to Cutting Point** will cause the tool to rapid to the defined cutting point after the **lead out** has been finished.*

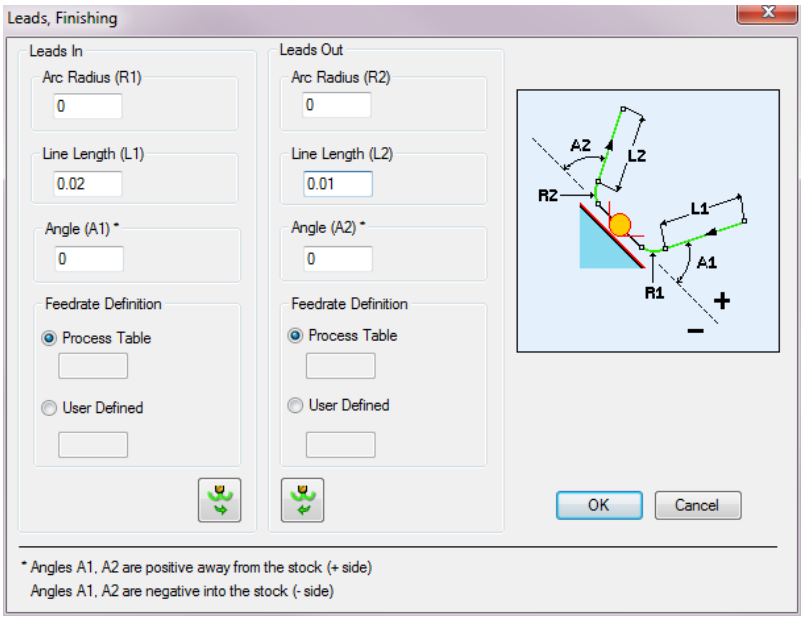
11. In the field called **Group Name:**, enter **ID Bore**.
12. Click **<Apply>** when done.

The dialog should now appear as below.



Leads

1 Next you would set the **Lead in** and **Lead out** for your profile. Click the **Leads** button and **Leads** dialog will appear. Set the line Lengths as they appear below:

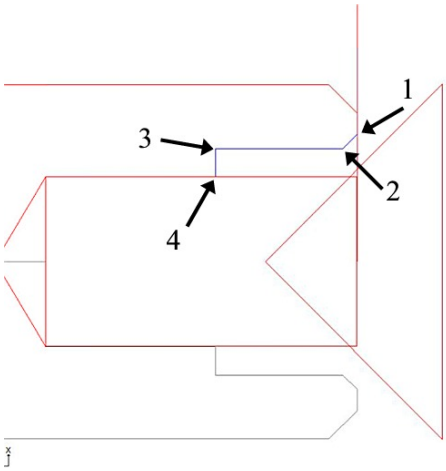


2 You have now finished entering the parameters for the **Face Part Profile Group**. Click **<Apply>** and **<Close>** to accept all changes.

Assigning the Profile to the Geometry

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

2 From the top of the screen, select the **End of an Element Snap Mode** icon. To define the tool path, click your left mouse on positions 1 through 4 as shown in the diagram below:

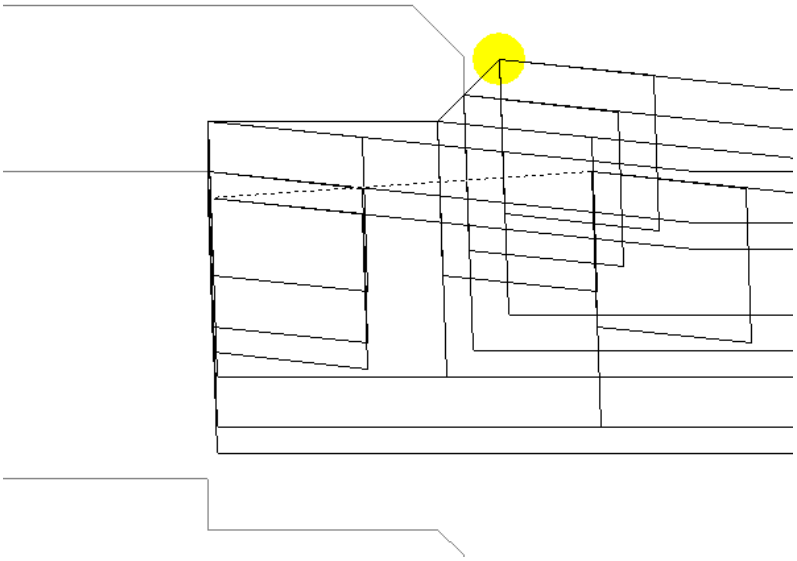


After clicking the last intersection, select the **Selection** Icon on the left side of the screen. This will end the profile.

Save your work

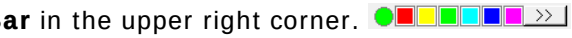
Tool Path Verification

Use either method described above to make sure the tool paths you have defined are correct.

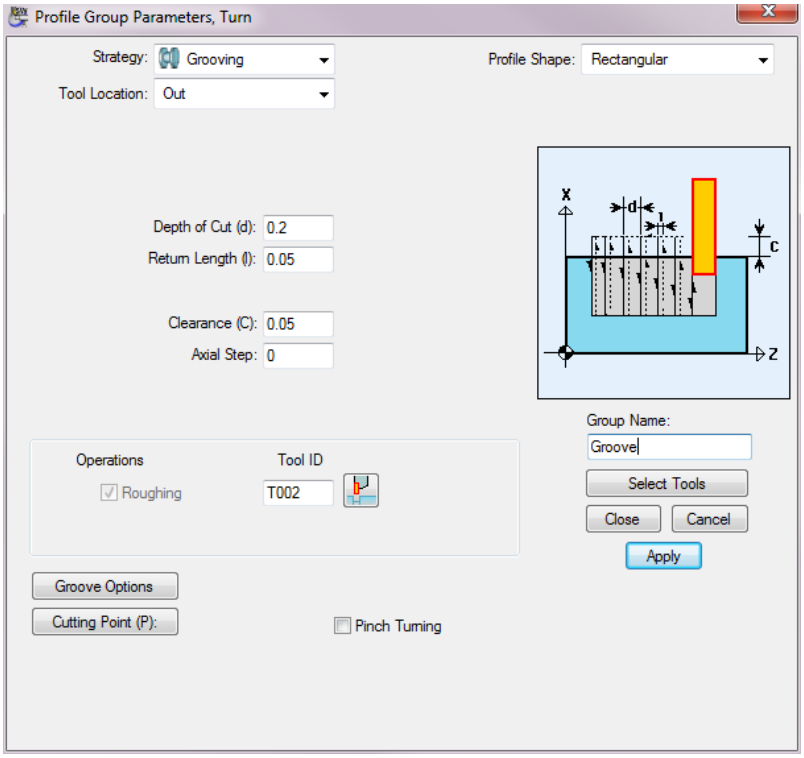


Once you are satisfied with your tool path, choose **Hide Every Tool Path** from the **Part Features** menu.

Defining a Grooving Operation

1. Click anywhere in the **Face Window** to make it active.
2. Choose green from the **Color Bar** in the upper right corner. 
3. Choose New Profile Group from the Part Features menu to display the Profile Group Parameters dialog.

4. From the **Strategy** menu, select **Grooving**.
5. From the **Tool Location** menu, select **Out**.
6. Set Profile Shape to **Rectangular** in the upper right hand corner of the dialog.
7. Click <**Select Tools**> and **PartMaker** will prompt you to choose between the grooving style tools available in the **Tools Database**.
8. In this case you are doing main spindle work, so choose **T002** and click **<Select>**. This tool was previously set up in the **Tools Database** for you to use.
9. Set the **Depth of Cut (d)** to .2 (5), **Return Length** to 0.05 (0.1), and **Clearance** to 0.05 (1.2)
10. In the field called **Group Name**, enter **Groove**. Your dialog should now appear as shown below:

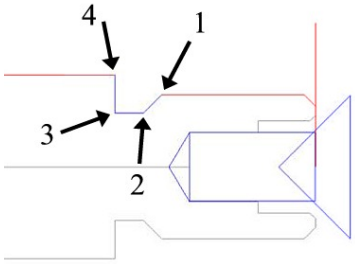


You have now finished entering the parameters for the Groove Profile Group.

11. Click <**Apply**> and <**Close**> to accept all changes.

Assigning the Profile to the Geometry

1. From the left-hand side of the screen select the **Define Profile** icon.
2. From the top of the screen, select the **Closest Intersection Snap Mode**. To define the tool path, click your left mouse on positions 1 through 4 as shown in the diagram below:



3. After clicking the last intersection, click the **Selection** Icon on the left side of the screen. This will end the profile.

4. Save your work.

Tool Path Verification

Fixed Headstock Simulation

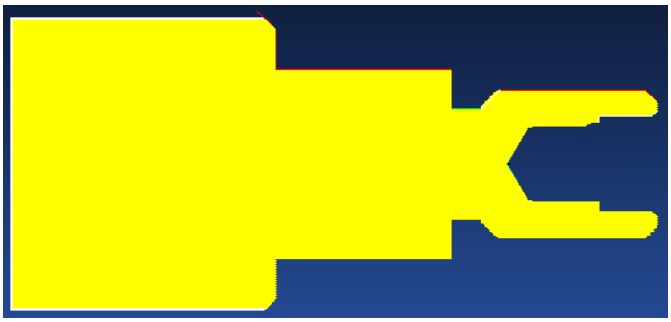
1. Select Verify Work Group Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Hollow** radio button.
3. Click <**OK**> and you will see your tool path verified.

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

Sliding Headstock Simulation

1. Select Verify Every Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Solid** radio button, and enter a delay between 0 and 9 (3 is a good starting number). The higher the number, the slower the verification will run.
3. Click <**OK**> and you will see your tool path verified.

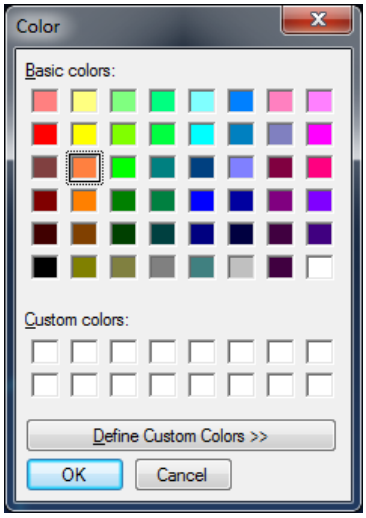
   ***Note**: You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*



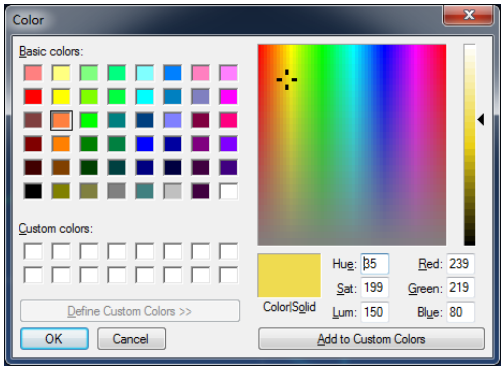
4. Once you are satisfied with your tool path, choose **Hide Every Tool Path** from the **Part Features** menu.

Defining a Thread Profile

1. Click anywhere in the **Face Window** to make it active.
2. Choose a new color from the **Color Bar** which is in the upper right corner. If you are running out of colors, click on the icon as shown below.

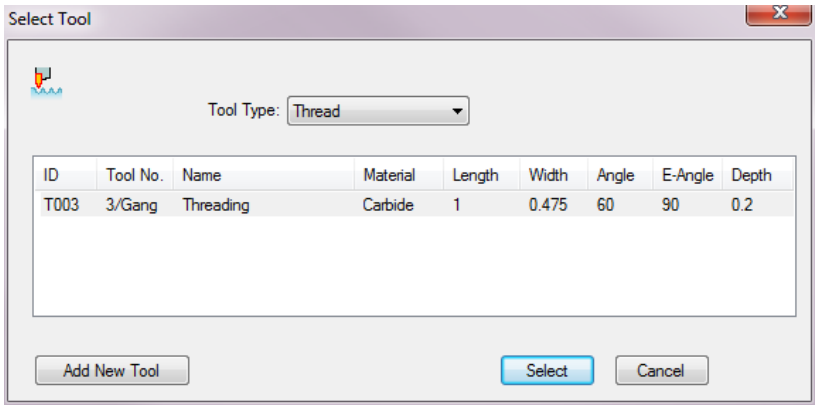


*Note: For additional colors you can click on <**Define Custom Colors**> and pick a different color.*

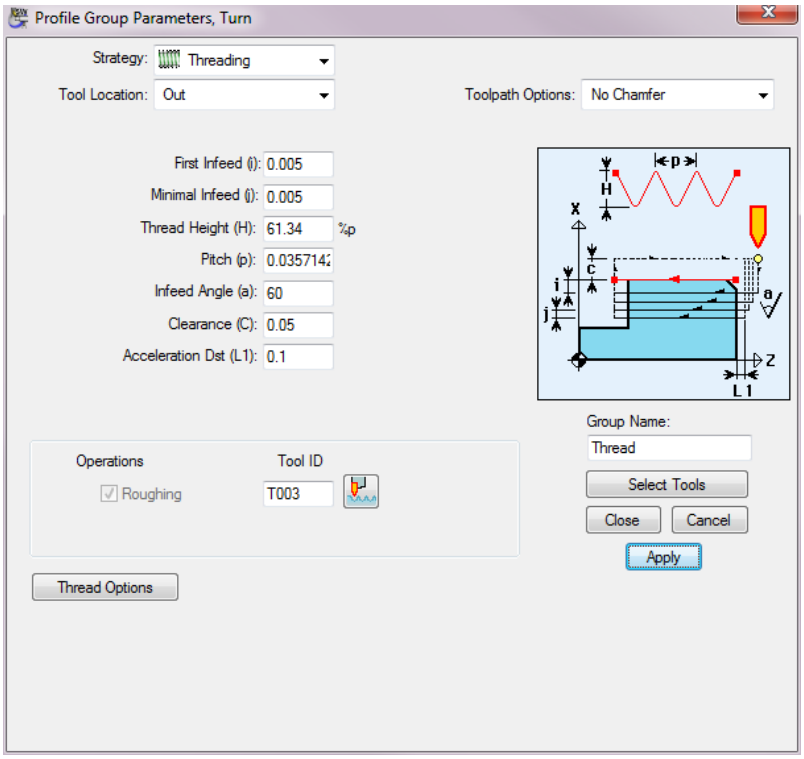


3. Choose New Profile Group from the Part Features menu to display the Profile Group Parameters dialog.
4. From the **Strategy** menu, select **Threading**.
5. From the **Tool Location** menu, select **Out**.
6. Select **No Chamfer** from the **Toolpath Options:** drop-down in the upper right hand corner of the dialog.
7. Enter 0.005 (.1) in the **First Infeed (i)** field.
8. Enter 1/28 (.75) in the **Pitch (p)** field.
9. Enter 0.05 (1.0) in the **Clearance (C)** field.
10. Enter 0.1 (1.0) in the **Acceleration Dst (L1)** field.

11 Click <**Select Tools**> and select the appropriate tool.



12 In the field called **Group Name**, enter **Thread**. Your dialog should now appear as shown below:



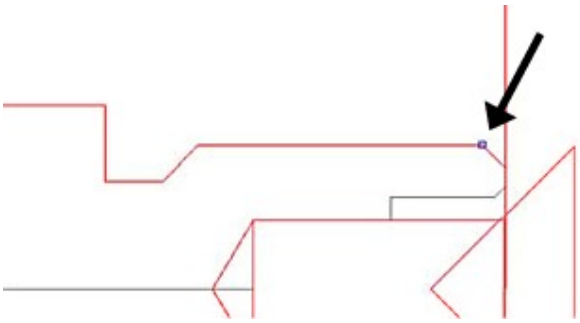
Note: *The Thread Height = 61.34% of the pitch is standard for ANSI inch and ISO / DIN 60º thread form.*

13 You have now finished entering the parameters for the **Thread** Profile Group. Click <**Apply**> and <**Close**> to accept all changes.

Assigning the Profile to the Geometry

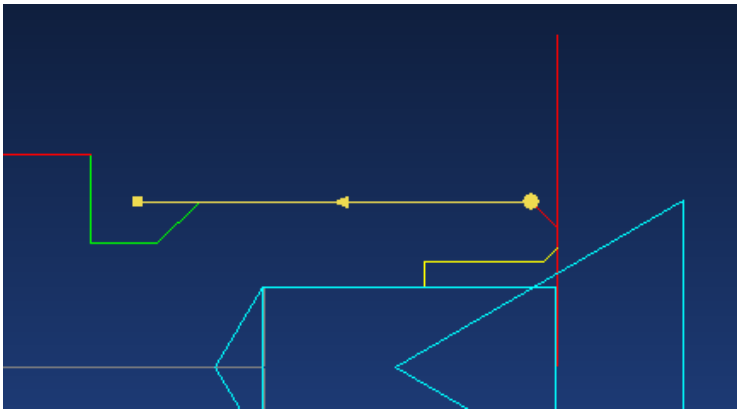
1. You will now create the profile for this process. Select the **Define Profile Icon** from the icons on the left side of the screen.

2. Select the **Closest Intersection Snap Mode** icon from the upper left.
3. Click near the intersection where the major diameter and threads begin for the start of the thread profile.



4. Now select **ZX Coordinates Snap Mode** icon from the upper left.
5. Enter .315 (8.0) in the lower left **Z** field. Do not change the X field.

6. Press <**Enter**> to accept.
7. Select the **Selection Icon** in the upper left. The Thread Profile is now complete.



Save your work.

Tool Path Verification

Fixed Headstock Simulation

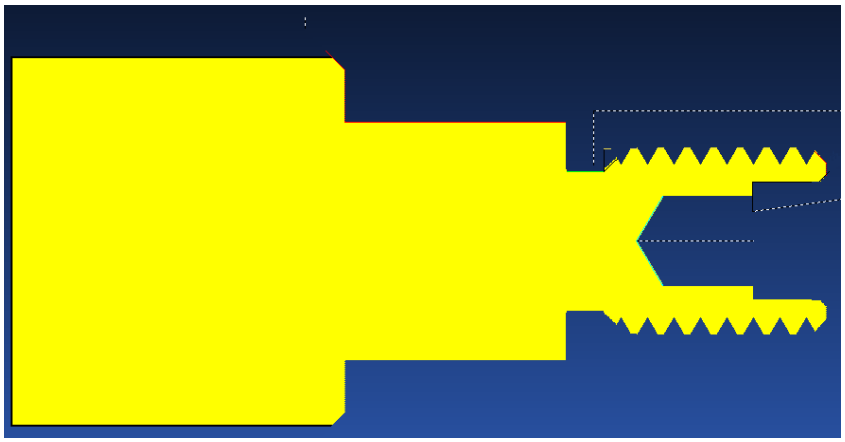
1. Select **Verify Work Group Tool Path from the Part Features** menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Hollow** radio button.
3. Click <**OK**> and you will see your tool path verified.

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

Sliding Headstock Simulation

1. Select **Verify Every Tool Path from the Part Features** menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Solid** radio button.
3. Click <**OK**> and you will see your tool path verified.

   ***Note**: You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*

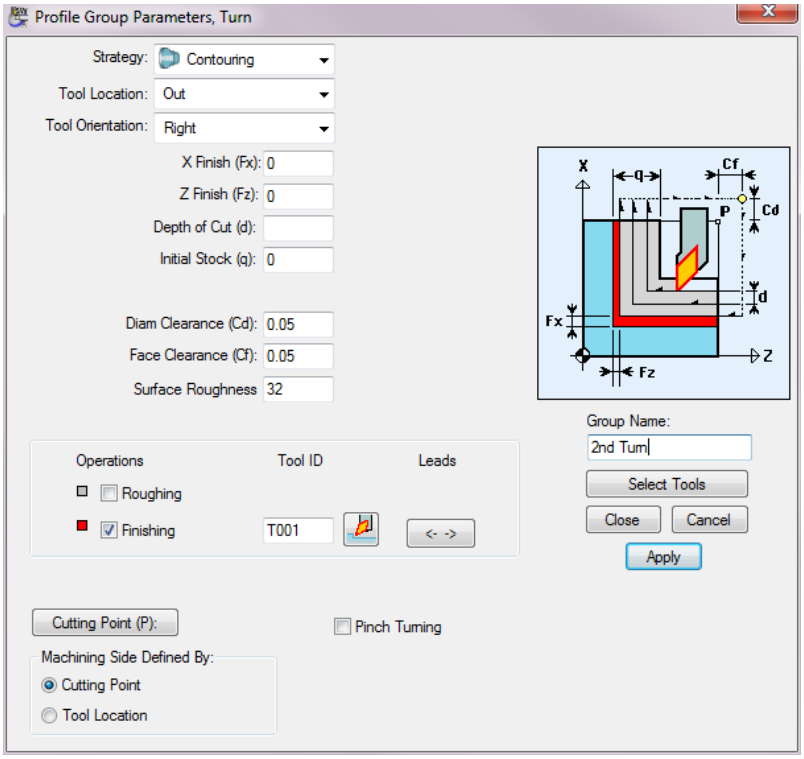


4. Once you are satisfied with your tool path, choose **Hide Every Tool Path** from the **Part Features** menu.

Defining the 2nd Turning Operation

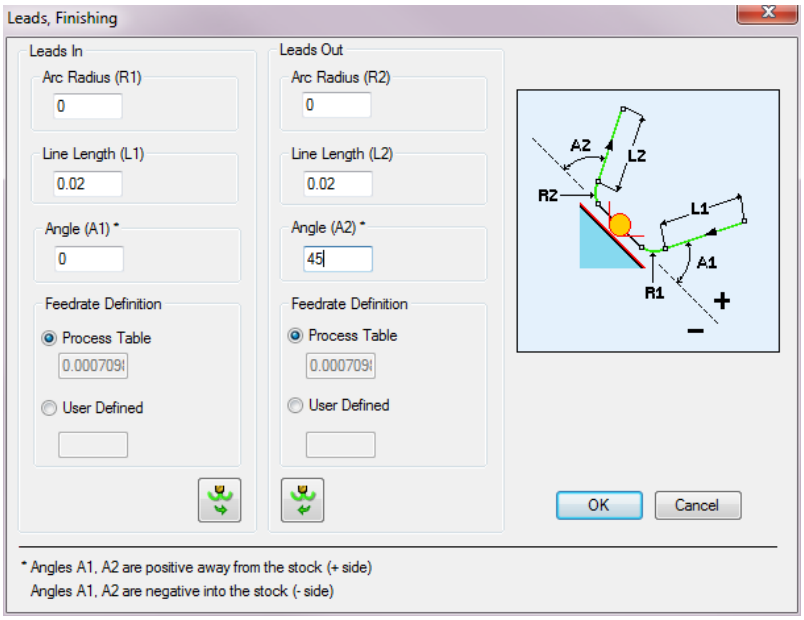
1. Click anywhere in the **Face Window** to make it active.
2. Choose green from the **Color Bar.**

3. Choose New Profile Group from the Part Features menu to display the Profile Group Parameters dialog.
4. From the **Strategy** menu, select **Contouring**.
5. From the **Tool Location** menu, select **Out**.
6. From the **Tool Orientation** menu, select **Right**.
7. Uncheck the **Roughing Tool ID**: box which will automatically cause the **Finishing Tool ID**: to be checked.
8. Click **<Select Tools>** and select the appropriate tool.
9. In this case you are using the same tool as in the 1st Turn process, so choose **T001** and click **<Select>**.
10. In the field called **Group Name**, enter **2nd Turn**. Your dialog should now appear as shown below:



Leads

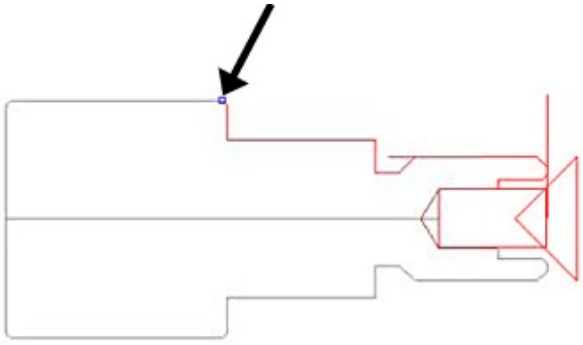
1. Now you will set the lead in and lead out for your profile. Click the **Leads** button and **Leads** dialog will appear. Set the line Lengths as they appear below:



2. Click <**OK**> when done.
3. You have now finished entering the parameters for the **2nd Turn** Profile Group. Click <**Apply**> and <**Close**> to accept all changes.

Assigning the Profile to the Geometry

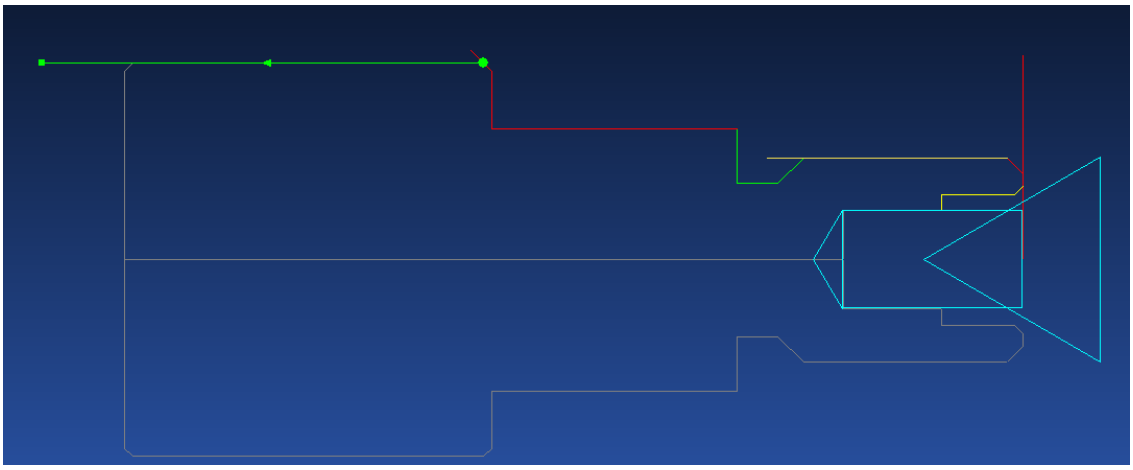
1. You will now create the profile for this process. Select the **Define Profile Icon** from the icons on the left side of the screen.
2. Select the **Closest Intersection Snap Mode** icon from the upper left.
3. Click near the intersection where the 2nd Turn profile will start.



4. Now select **ZX Coordinates Snap Mode** icon from the upper left.
5. Enter 1.2 (30.50) in the lower left **Z** field. Do not change the X coordinate value.

6. Press <**Enter**> to accept.
7. Select the **Selection** Icon in the upper left.

The 2nd Turn Profile is now complete.



Save your work.

Tool Path Verification

Use either method described before to make sure the tool paths you have defined are correct.

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

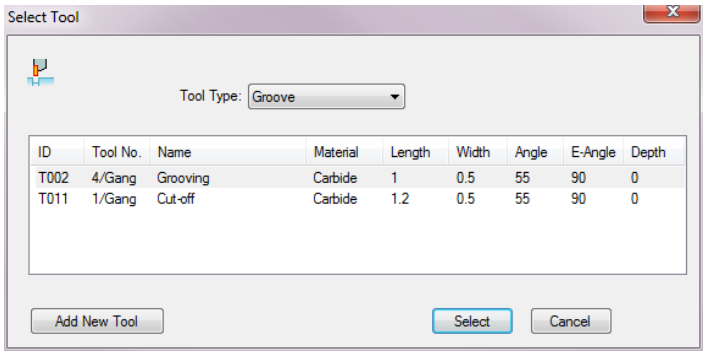
Defining the Cut Off Operation

1. Click anywhere in the **Face Window** to make it active.
2. Choose Magenta from the **Color Bar.**

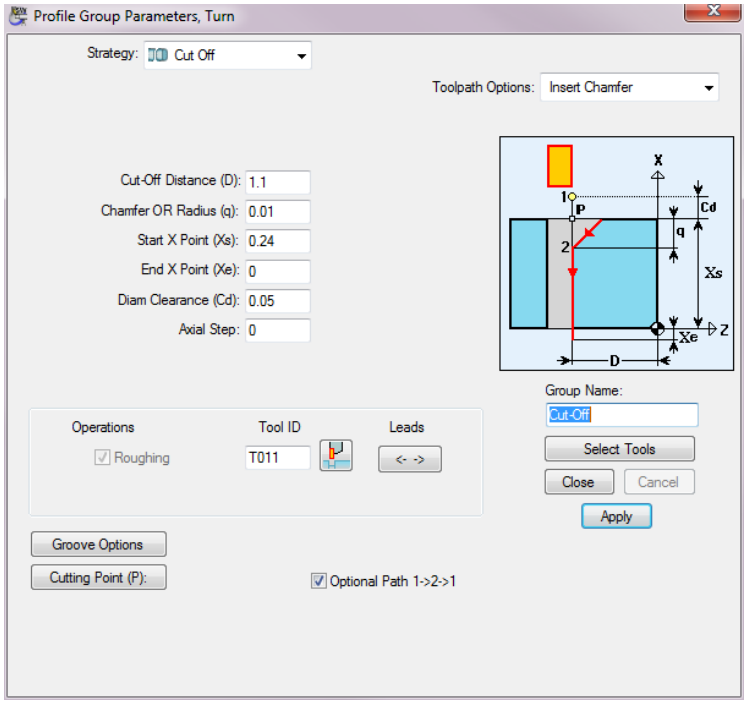
3. Choose New Profile Group from the Part Features menu to display the Profile Group Parameters dialog.

4. From the **Strategy** menu, select **Cut-Off**.
5. Select **Insert Chamfer** from **Toolpath Options** in the upper right hand corner of the dialog.
6. Enter .01 (.2) into the **Chamfer or Radius (q)** field.
7. Enter .240 (6.0) into the **Start X Point (Xs)** field. This is a RADIAL value
8. Click <**Select Tools>** and select the appropriate tool.

   In this case you are doing main spindle work, so choose **T011** and click **<Select>**. You are choosing T011 because it is set up as the Cut Off tool.

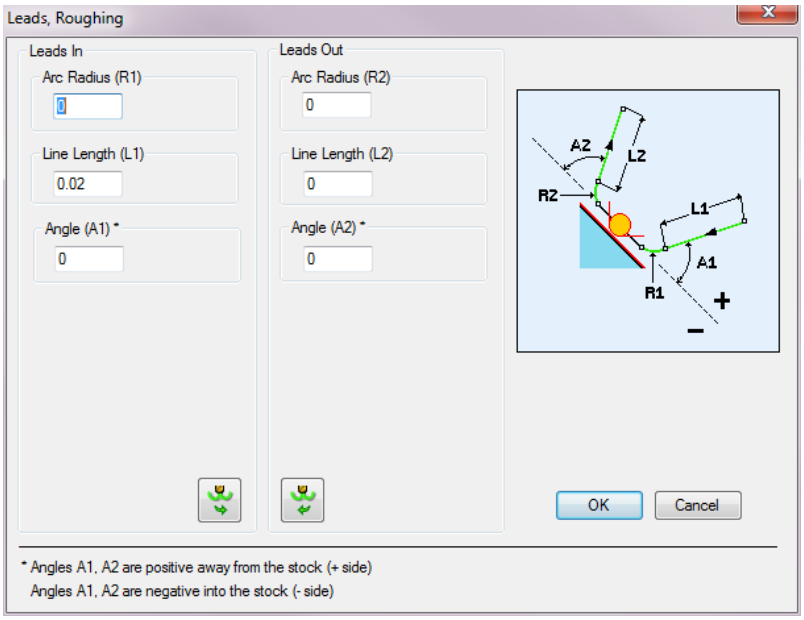


9. Check **Optional Path 1->2->1**. This will cause the Cut Off tool to rough out material before creating a chamfer.
10. In the field called **Group Name**, enter **Cut Off**. Your dialog should now appear as shown below:



Leads In/ Leads Out

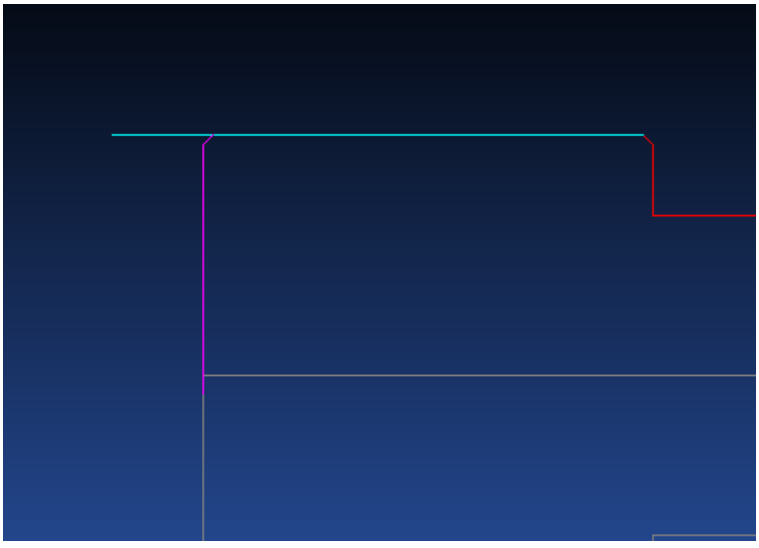
1. Now you will set the lead in and lead out for your profile. Click the **Leads** button and the **Leads** dialog will appear. Set the line Lengths as they appear below:



2. Click **<OK>** when done.

3. You have now finished entering the parameters for the **Cutoff Profile Group**. Click **<Apply>** and **<Close>** to accept all changes.

   ***Note**: After closing the **Profile Group Parameters**, **PartMaker** automatically created the profile on the screen based on the information given.*



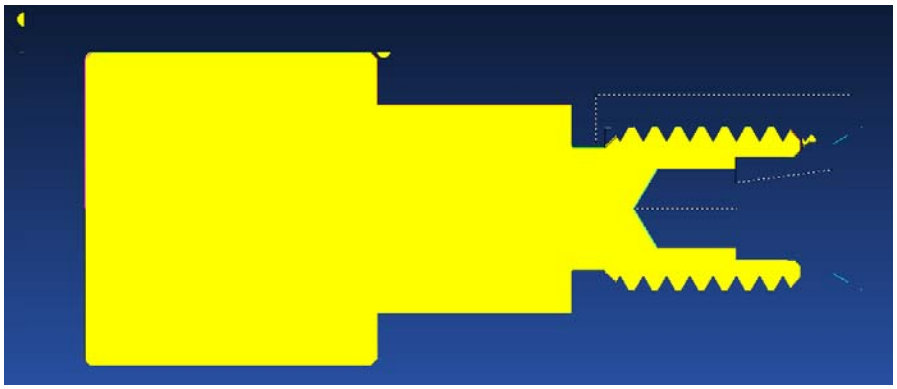
Save your work.

Tool Path Verification

Sliding Headstock Simulation

1. Select **Verify Every Tool Path from the Part Features** menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Solid** radio button.
3. Enter a delay of 2.
4. Click <**OK**> and you will see your tool path verified.

   ***Note**: You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*



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

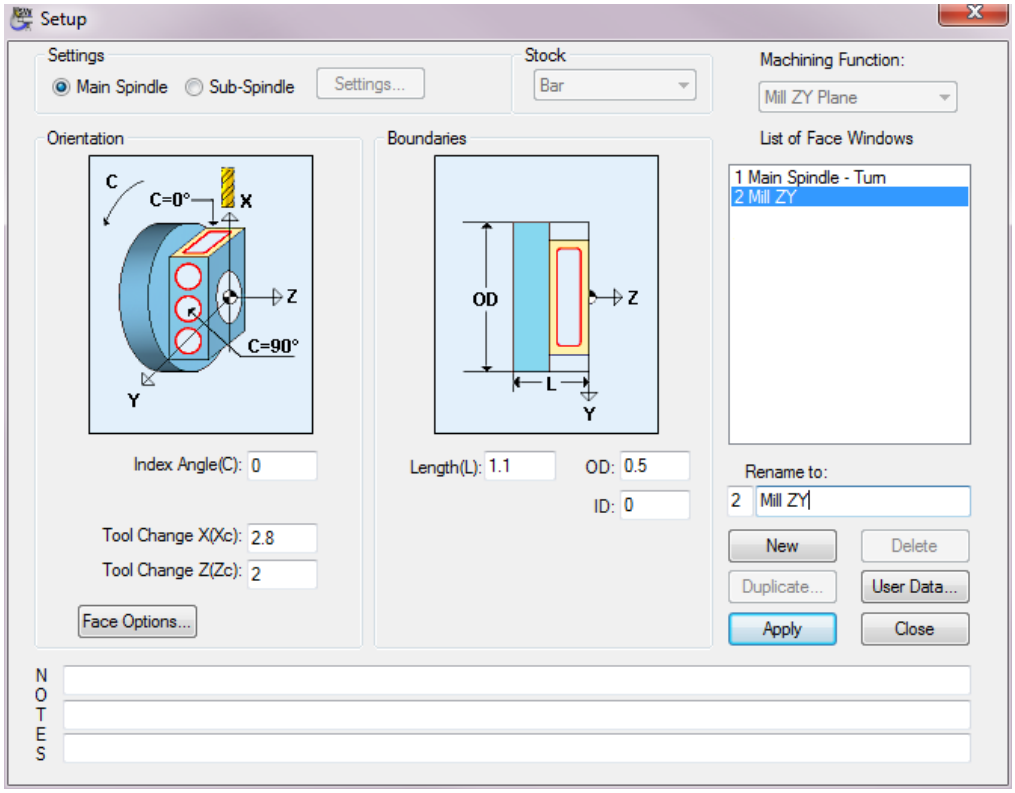
Milling a Slot on the Main Sub-spindle Using the Mill ZY Face Window

This section of the tutorial will instruct you in the steps necessary to create the slot on the front of the part by performing milling in a **Mill ZY Face Window**.

Setting Up the Face Window

1. Choose **Setup** from the **View** Menu.
2. Click <**New**> to create a new **Face Window**.
3. Under Machining Function choose Mill ZY Plane.
4. Enter **Mill ZY** into the **Rename to**: field.

   Your Setup dialog should appear as shown below.



5. Click <**Apply**> and <**Close**> when finished.

Creating the Geometry for the Slot Milling Operation

1. Click <**CAD/CAM>** switch in the lower left to switch to CAD mode.

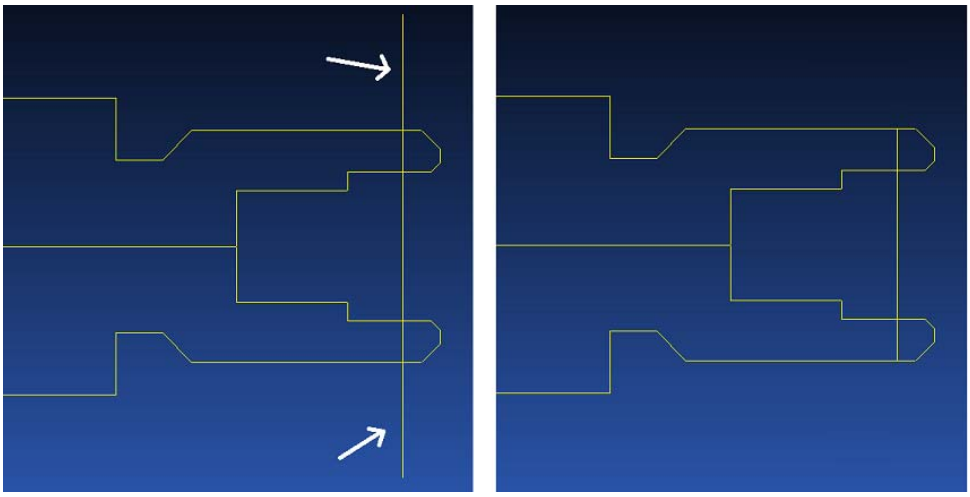
   Copy the geometry already drawn in the **Main Spindle – Turn Face Window**. To do so:

2. Select Main Spindle – Turn from the Window menu.
3. Click <**CAD/CAM>** switch in the lower left corner of **Main Spindle – Turn Window** to switch to CAD mode.
4. Choose **Select All** from the **Edit** menu or use the shortcut <**Ctrl-A>** to select all the geometry in the window.
5. Select **Copy** from the **Edit** menu or use the shortcut <**Ctrl-C>** to copy the geometry.
6. Select **Mill ZY** from the **Window** menu to switch to the **Mill ZY Face Window**.
7. Select **Paste** from the **Edit** Menu or use <**Ctrl-V>** to paste the geometry into the **Mill ZY Face Window**.
8. Click anywhere in the **Face Window** to deselect the geometry and make the window active.

9. You now need to create a line for the Tool Path to follow. Select the **Line Parallel Vertical Axis Icon** on the left side of the screen.
10. Enter .04 (.1) in the lower left field.

11. Press <**Enter>** to create the vertical line.

12. Next select the **Remove Icon** on the left side of the screen.
13. Click Near the upper and lower portions of the vertical line you created to trim the line.

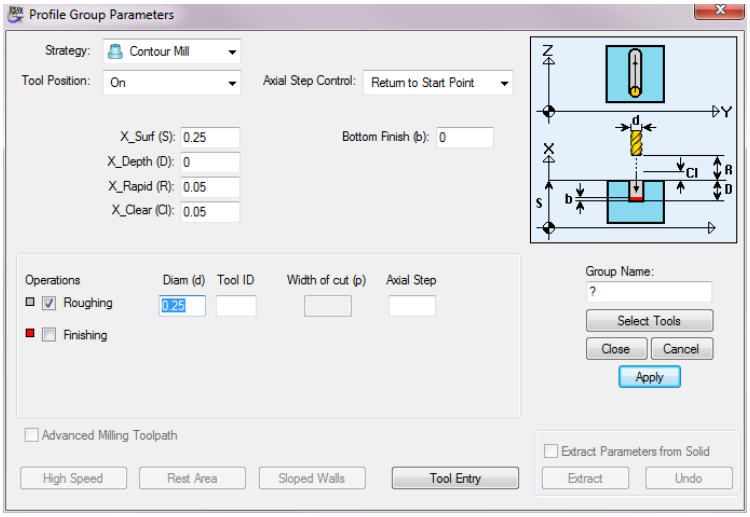


14. Click the **Selection** Icon in the upper left.

Defining the Slot Milling Operation

1 Click the **CAD/CAM** switch in the lower left to switch to CAM mode.

2 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Group Parameters** dialog. When selected, you should see a dialog which appears as the one below:



3 From the **Strategy** menu, select **Contour Mill**.

4 From the **Tool Position** menu, select **Left**.

5 Enter .125 (3.1) into the **X_Surf (S)** field.

6 Enter 0.145(3.7) into the **X_Depth (D)** field.

7 Uncheck from the **Roughing Tool ID**:, this will automatically cause the **Finishing Tool ID**: to be checked.

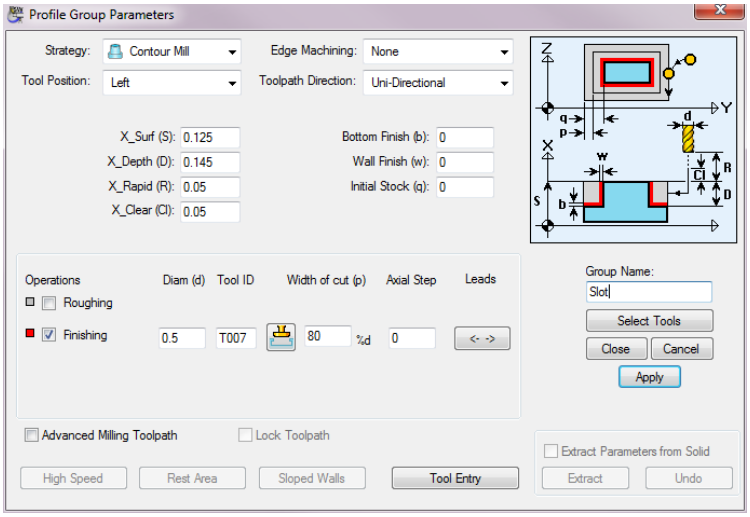
8 Enter .5 (20.0) into the field under **Diam (d)**.

9 Enter **T007** into the field under **Tool ID**. This is the Slotting Tool entered into the **Tools Database** for this operation.

10 Click the <**Apply**> to selelct the slitting saw

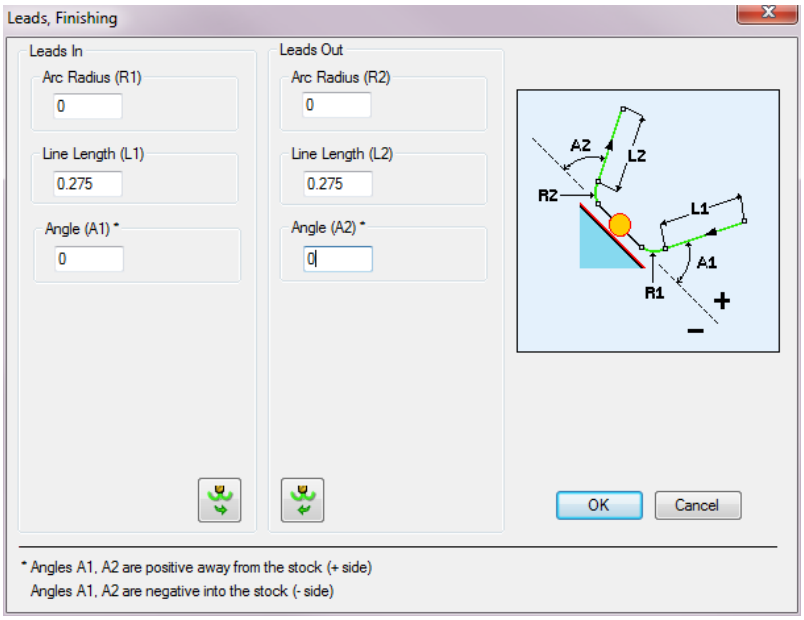
11 In the field called **Group Name**, enter **Slot**.

Your dialog should now appear as shown below:



Leads In/ Leads Out

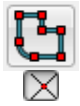
1 Now you will set the lead in and lead out for your profile. Click the **Leads** button and the **Leads** dialog will appear. Set the line Lengths as they appear below:



2 Click **<OK>** when done.

3 You have now finished entering the parameters for the **Slot Profile Group**. Click **<Apply>** and **<Close>** to accept all changes.

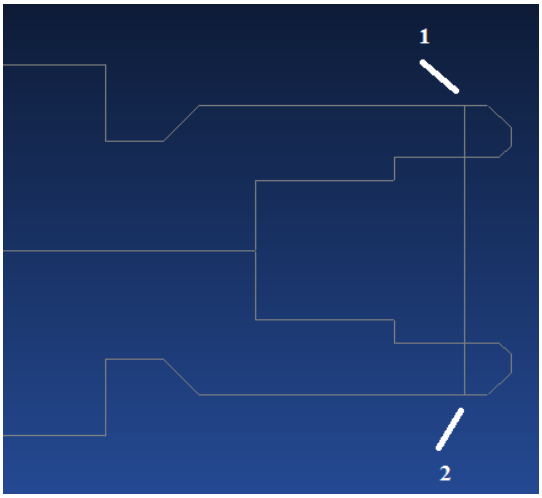
Assigning the Profile to the Geometry



1 Select the **Define Profile Icon** from the icons on the left side of the screen.

2 Select the **Closest Intersection Snap Mode** icon from the upper left.

3 Click near the intersections on the screen in the order indicated below to create the profile. You will notice as you click from point to point that the profile is being created.

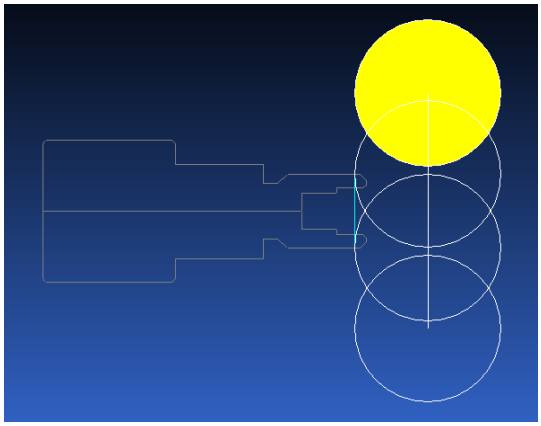


4 After clicking the last intersection, select the **Selection Icon** on the left side of the screen. This will end the profile.

5 Save your work

Tool Path Verification

1. Select Verify Work Group Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Hollow** radio button and enter a delay between 0 and 9 (3 is a good starting number). The higher the number, the slower the verification will run.
3. Click <**OK**> and you will see your tool path verified.



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

Milling the Square on the Main Spindle

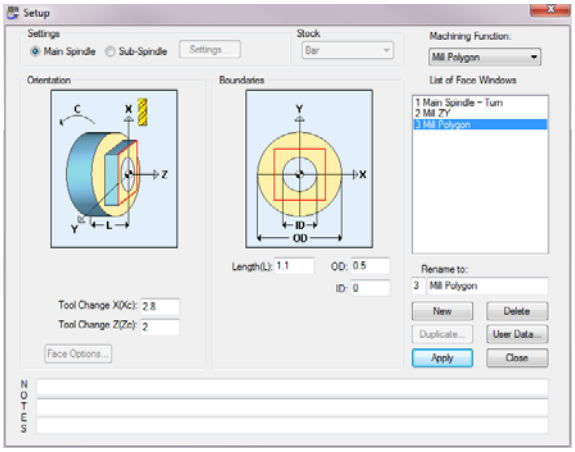
This section of the tutorial will instruct you in the steps necessary to perform milling in a **Mill Polygon Face Window**.

***Note**: All Milling Style Face Windows use **Radius** not Diameter style programming.*

Setting Up the Face Window

1. Choose **Setup** from the **View** Menu.
2. Click <**New**> to create a new **Face Window**.
3. Under Machining Function choose Mill Polygon.
4. Enter **Mill Polygon** into the **Rename to**: field.

   Your Setup dialog should appear as shown below.



5. Click <**Apply**> and <**Close**> when finished.

Creating Geometry in CAD Mode

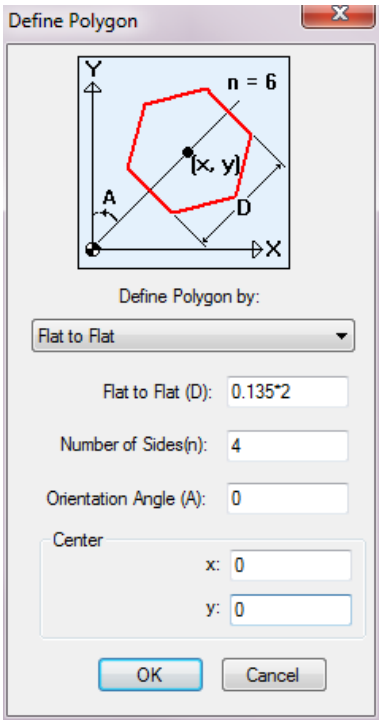
In this section, you will create the geometry for the square profile to be milled.

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

2. Select the **Polygon Icon**.

   After doing so, a dialog will open prompting you to enter a distance and a number of sides.

3. Enter a distance of **.135*2 (3.375*2)** and **4** for the number of sides. Click the **<OK>** button.



Define Polygon By:

Flat to Flat (D). Allows you to specify the distance between parallel sides.

Center to Flat (d). Allows you to specify the distance between center and a side.

Outer Circle Radius(R). Allows you to specify the radius of outer circle.

Side Length (L). Allows you to specify the side length.

Number of Sides (n). Specifies the number of sides in polygon.

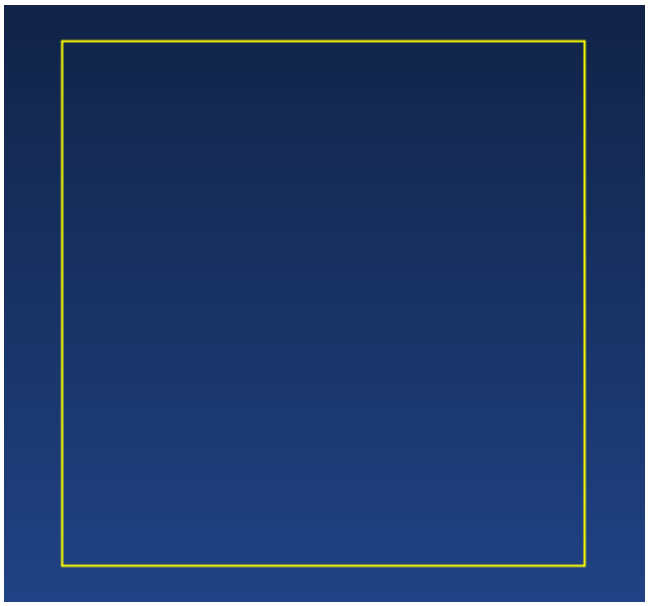
Orientation Angle (A). Specifies the polygon angle with horizontal axis.

Center:

z: Specifies the horizontal coordinate of the polygon center.

x: Specifies the vertical coordinate of the polygon center.

Your screen should appear as shown below:



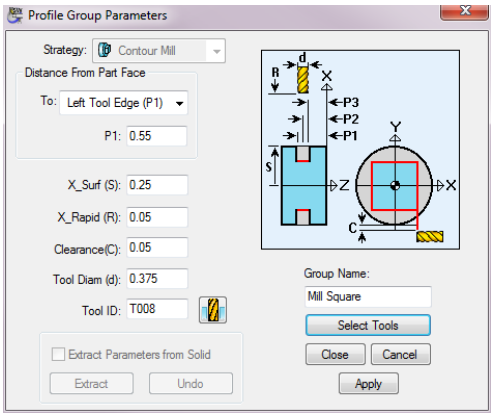
Next, you will proceed to machine this square in **CAM** mode.

Creating the Square Profile

In this section, you will define the profile to machine the hexagonal flats on the work piece.

1. Click the CAD/CAM switch which will put you back into the CAM mode.
2. In CAM mode, select red from the **color bar**.

3. Select New Profile Group from the Part Features menu.
4. In the Group Name box, type **Mill Square**.
5. Choose **Left Tool Edge (P1)** in the **To:** drop down menu to define the **Distance From Part Face** from which the hex will be milled.
6. Enter .550 (14.0) for **P1**, representing the distance from the part face machining will occur.
7. Enter 0.375 (10.0) for **Tool Diameter**.
8. Click on the **<Select Tools>** button. An end mill icon will appear to the right of the **Tool ID** box. The completed **Profile Group Parameters** dialog should appear as shown below:



Now that you have completed setting up the profile group, take a moment to study this dialog, as it is unique to other **PartMaker** Milling Profile Group Parameters dialogs you have encountered. Understand the following unique parameters:

Distance From Part Face

To: Left Tool Edge (P1): Specifies the distance from the face of the part to the left edge of the tool.

To: Tool Center (P2): Specifies the distance from the face of the part to tool center.

To: Right Tool Edge (P3): Specifies the distance from the face of the part to the right edge of the tool.

X_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 Swiss tutorial 2 workpiece this is **0.5/2** or **.25 (6.35).**

X_Rapid (R): Specifies the distance between the bottom tip of the tool and the part surface when tool performs Rapid Moves.

Clearance (C): Specifies the distance between the bottom tip of the tool and the part surface when a tool starts feeding into the part.

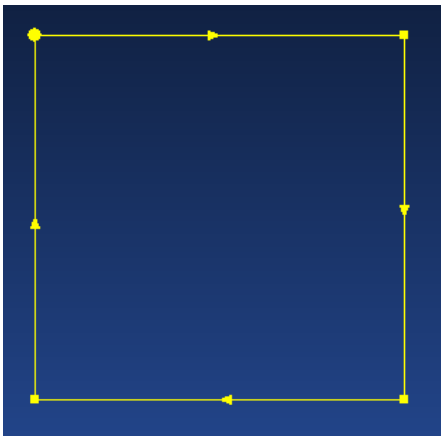
Tool Diam (d): Specifies diameters of tool.

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

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

1. Select the **Chain Geometry Icon**.
2. Move your cursor above the top line of the square and slightly to the left of the horizontal axis line as indicated below.

   Click your mouse button and the screen should appear as shown below:

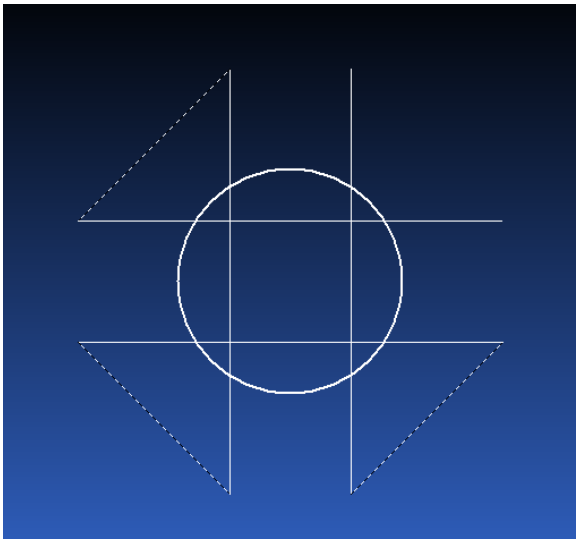


The arrows display the direction of the tool path. The flats will be machined in this direction. If you wish to machine the flats in the opposite direction, using the **Selection** icon, click anywhere on the tool path to select it. Choose **Reverse** from the **Edit** menu and the arrows displayed on the tool path will change direction.

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

Tool Path Verification

1. Select Verify Work Group Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Hollow** radio button and enter a delay between 0 and 9 (3 is a good starting number). The higher the number, the slower the verification will run.
3. Click <**OK>** and you will see your tool path verified.



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

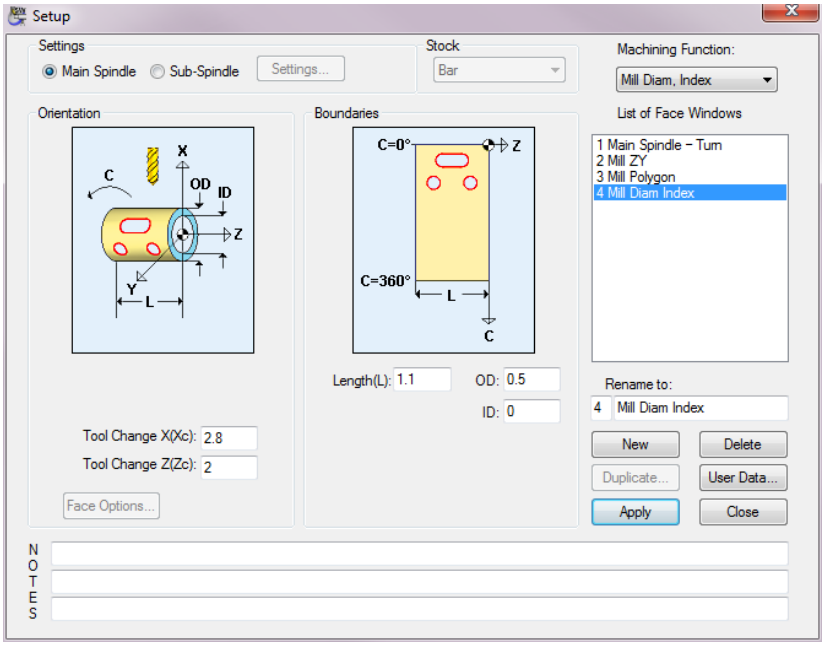
Drilling Cross (X Oriented) Holes - Main Spindle Using the Mill Diameter Index Machining Function

This section of the tutorial will instruct you in the steps necessary to perform milling in a **Mill Diameter Index Face Window**.

Setting Up the Face Window

1. Choose **Setup** from the **View** Menu.
2. Click <**New**> to create a new **Face Window**.
3. Under Machining Function choose Mill Diam. Index.
4. Enter **Mill Diam Index** into the **Rename to**: field.

Your Setup dialog should appear as seen here.



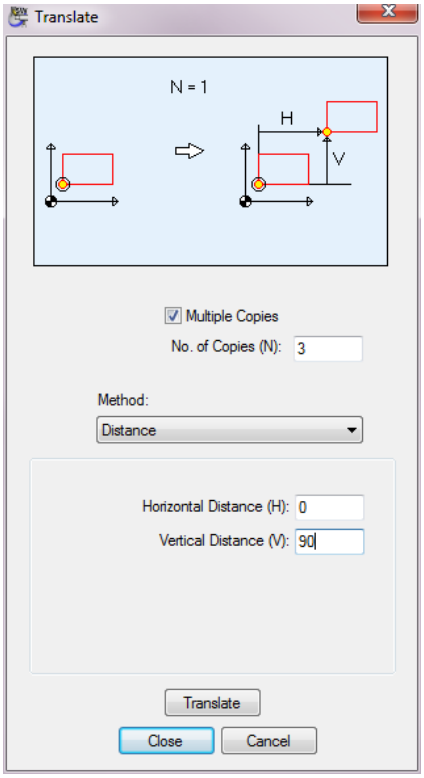
5. Click <**Apply**> and <**Close**> when finished.

Creating the Geometry for the Cross Drill Operation

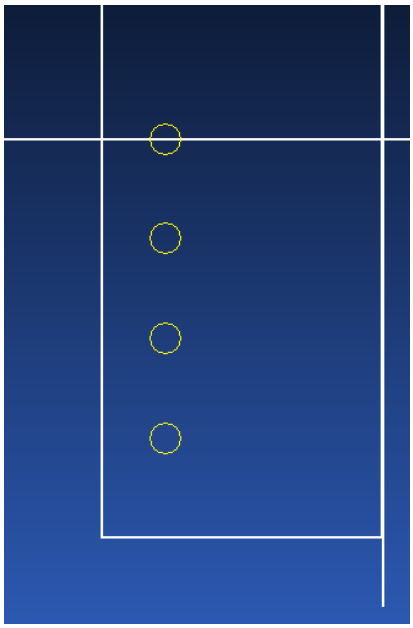
1. Click <**CAD/CAM>** switch in the lower left to switch to CAD mode.
2. Choose **Show Boundaries** from the **View** Menu.
3. Choose **Show Axis** from the **View** Menu.
4. Select the **Circle with a Known Radius and Center** Icon from the left side of the screen.
5. Enter .0625 (1.6) into the field at the bottom of the screen.

6. Press <**Enter>** to accept the input.
7. Enter .850 (21.60) in the **Z** field at the bottom of the screen.
8. Enter 0 in the **C** field at the bottom of the screen.

9. Press <**Enter>** to accept the input and place a circle on the screen.
10. Choose the **Selection** Icon on the left side of the screen.
11. Click the circle to select it.
12. Choose Translate from the Edit > Transform menu.
13. Check **Multiple Copies** option and enter 3 into the **No. of Copies**: field.
14. Enter 0 into the **Horizontal Distance (H)**: field.
15. Enter 90 into the **Vertical Distance (V)**: field.



16 Click <**Translate**> and **<Close>** to accept the inputs values and make 3 more circles in the window.

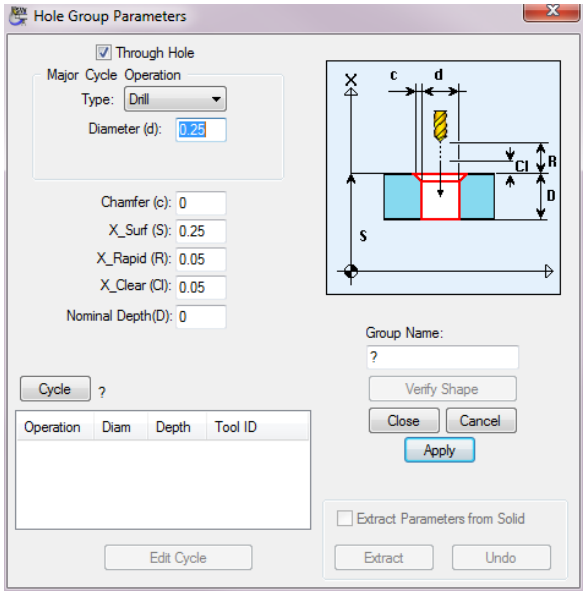


17 Click anywhere in the Face Window to deselect the circles

Defining the Cross Drilling Operation

1 Click the green square in the **Color Bar**

2 Choose **New Hole Group** from the **Part Features** menu to display the Hole **Group Parameters** dialog. When selected, you should see a dialog which appears as the one below:



3 Uncheck the **Through Hole** box.

4 Under **Major Cycle Operation** set the Type of Operation to **Drill**.

5 Under **Major Cycle Operation** enter .125 (3.175) into the **Diameter** field.

***Note**: The **Major Cycle Operation** parameters are used by **PartMaker** to search the **Cycles Database** for an appropriate cycle.*

6 Enter .01 (.5) into the **Chamfer (C)** field. **PartMaker** will calculate the depth of a Spot Drill according to the size of the Chamfer indicated and based on the angle of the chamfering tool.

7 Enter .240 (6.0) into the **X_Surf (S)** field.

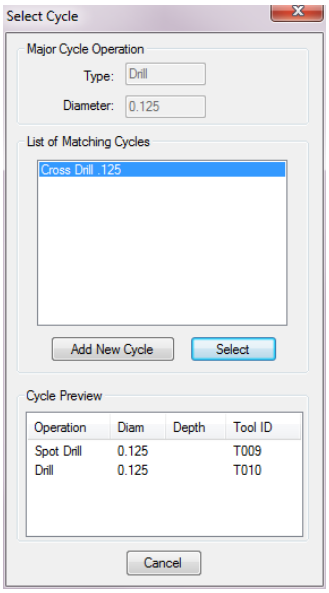
8 Enter .1 (2.5) into the **X_Rapid (R)** field>

9 Enter .05 (1.0) into the **X_Clear (Cl)** field.

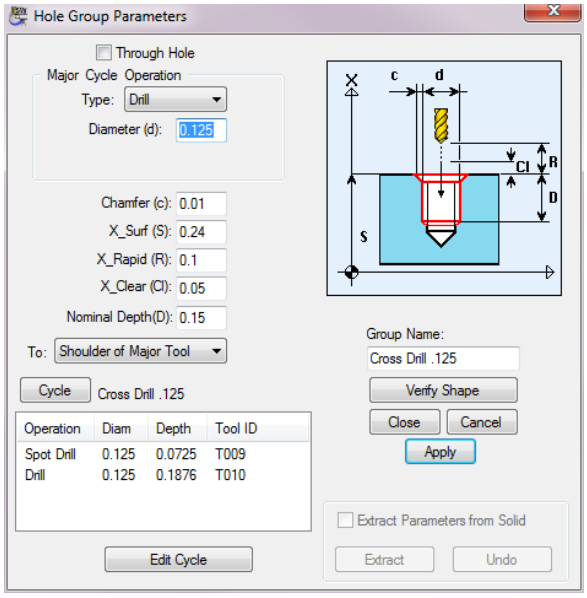
10 Enter .150 (3.8)into the **Nominal Depth (D)** field.

11 In the **To:** field under **Nominal Depth (D)**, select **Shoulder of Major Tool** to indicate where **PartMaker** will calculate the depth of the tools.

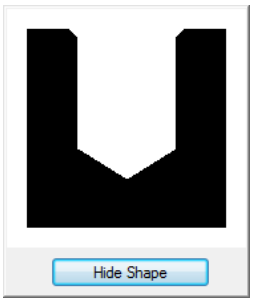
12 Click the **Cycle** button and **PartMaker** will select the appropriate cycle to create the desired hole by clicking **<Select>**. The selected cycle should be **Cross Drill .125 (Drill 3.5 X)**.



13 The dialog should now appear as below.



14 Review the dialog, then click <**Verify Shape**>. This will display a graphical representation of the cycle.

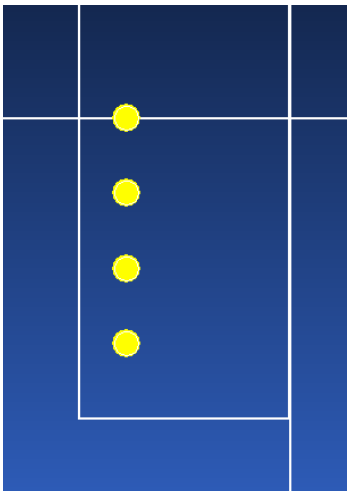


15 Click <**Hide Shape**> when done, returning to the **Hole Group Parameters**.

16 Click <**Apply**> and <**Close**> to close the Hole Group Parameters dialog.

Select the Holes

1. From the left hand tool bar, select the **Chain Circle** icon.
2. Click on any one of the four circles and PartMaker will select the other three. After doing so, your screen should appear as shown below:

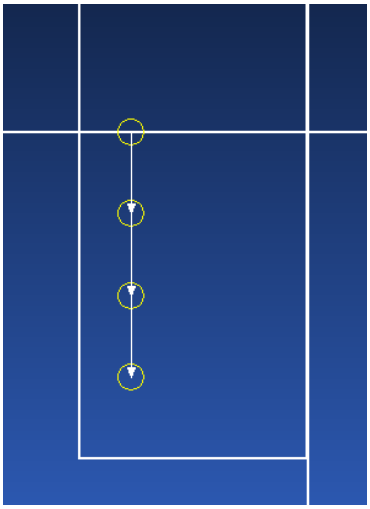


3. Choose the **Selection** Icon to finish selecting circles.
4. Save your work.

Tool Path Verification

1. Select Verify Work Group Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.
2. Check the **Hollow** radio button.
3. Click <**OK**> and you will see your tool path verified.

   ***Note**: You may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.*



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

Drilling the Face Holes - Sub Spindle Using Mill End Index Machining Function

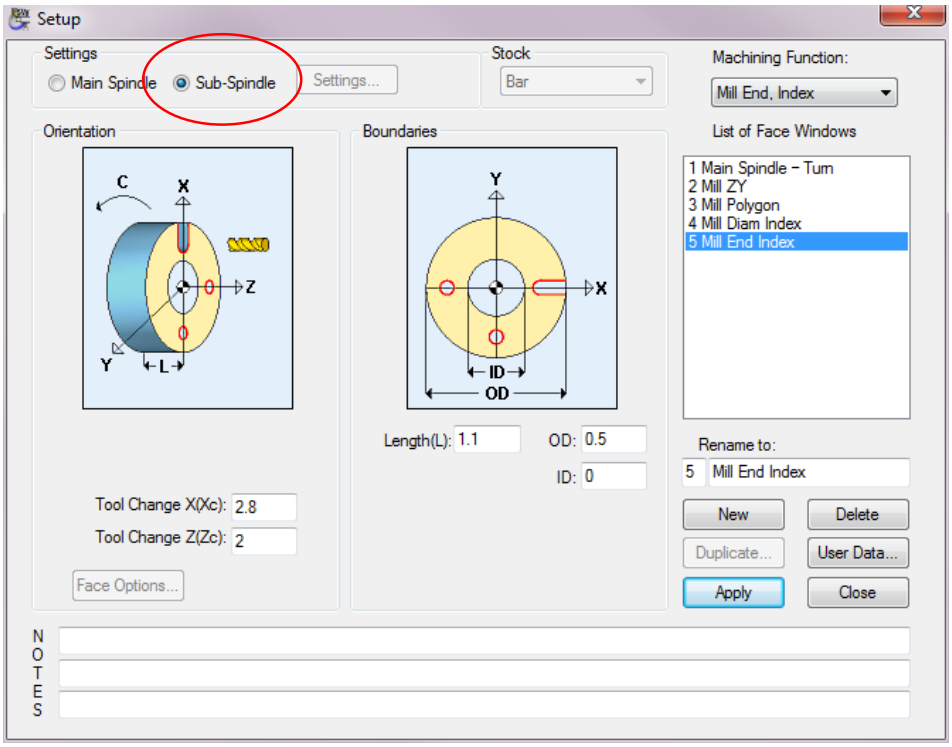
This section of the tutorial will instruct you in the steps necessary to perform drilling in a **Mill End Index Face Window** on the Sub Spindle.

Note: *In* ***PartMaker*** *all programming is done using the same convention whether it is done on the main spindle or sub spindle. In the following window, you will be looking at the end of the part as it would look in the Sub Spindle.*

Setting Up the Face Window

1. Choose **Setup** from the **View** menu.
2. Click <**New**> to create a new **Face Window**.
3. Select **Sub Spindle** (red circle) as this will be an operation carried out on the Sub Spindle.
4. Under Machining Function choose Mill End Index.
5. Enter Mill End Index into the **Rename to**: field. Click **<Apply>.**

Your Setup dialog should now look like this.



6. Click <**Close>** when finished.

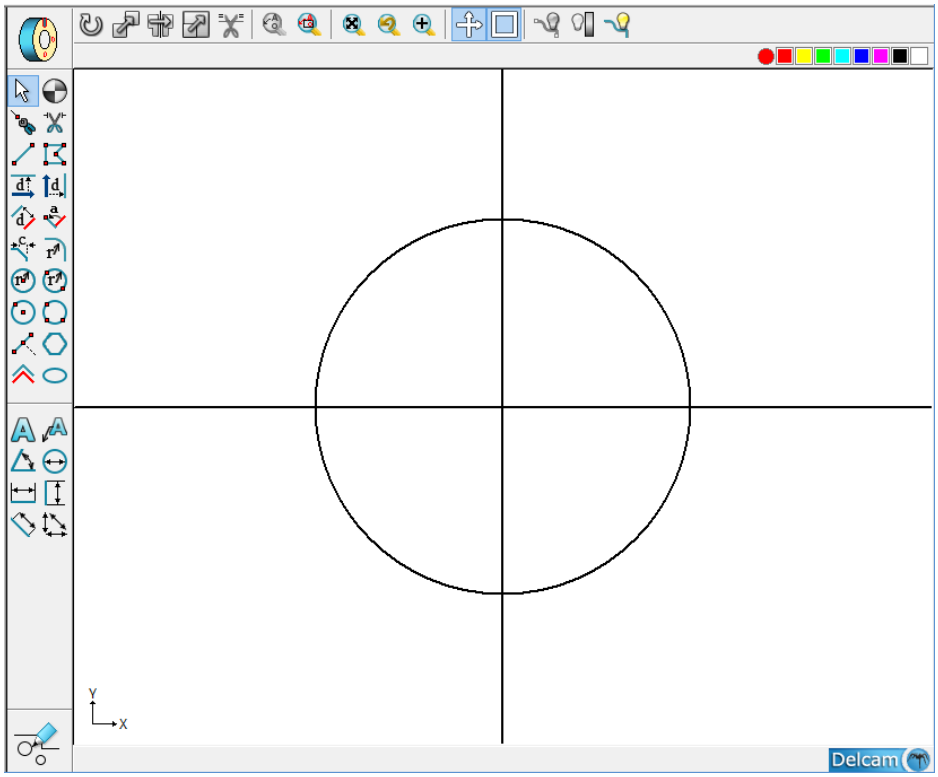
Creating the Geometry for the Face Drill Operation

1 Click <**CAD/CAM>** switch in the lower left to switch to CAD mode.

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

3 Choose **Show Axis** from the **View** menu.

***Note**: The boundaries are round because you are looking at the end of the part.*



4 Select **Circle with a Known Radius and Center Icon** from the left side of the screen.

5 Enter 0.0625/2 = .03125 (.75) into the field at the bottom of the screen.

6 Press <**Enter>** to accept the input.

7 Select the **XY Coordinates Snap Mode Icon** in the upper left of the screen.

8 Enter .180 (4.5) in the **X** field at the bottom of the screen.

9 Enter 0 in the **Y** field at the bottom of the screen.

10 Press <**Enter**> to accept the input and place a circle on the screen .180 from the center of the part.

11 Choose the **Selection** Icon on the left side of the screen.

12 Click on the circle you just drew to select it.

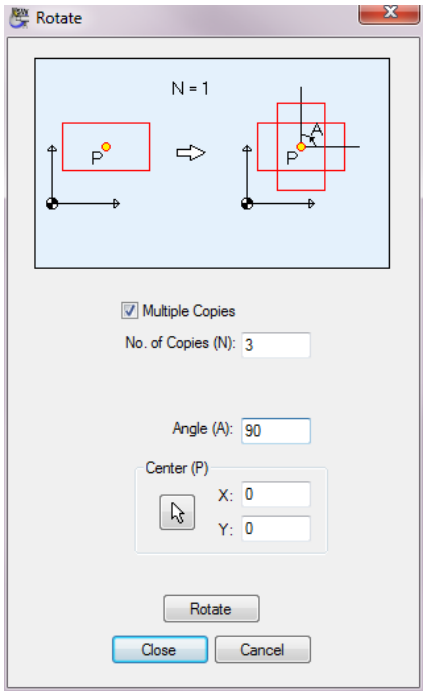
13 Choose **Rotate** from the **Edit > Transform** menu.

14 Check **Multiple Copies** option and enter 3 into the **No. of Copies**: field.

15 Enter 90 into the **Angle**: field.

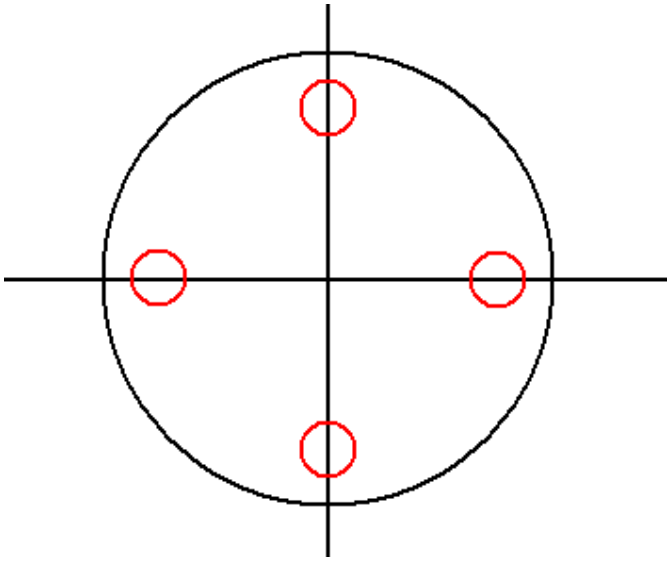
16 Enter 0 into the **X**: field indicating the rotation point.

17 Enter 0 into the **Y**: field indicating the rotation point.



18 Click <**Rotate**> and **<Close>** to accept the input values and place 3 more circles in the window.

Your screen will appear as below:

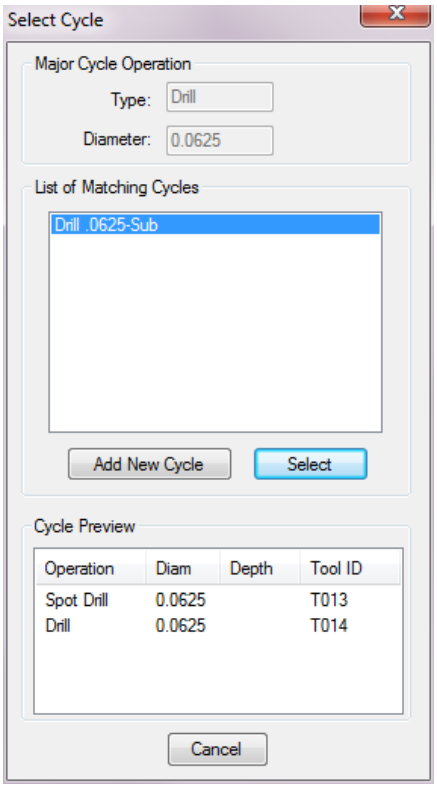


Defining the Face Drilling Operation for the Sub Spindle

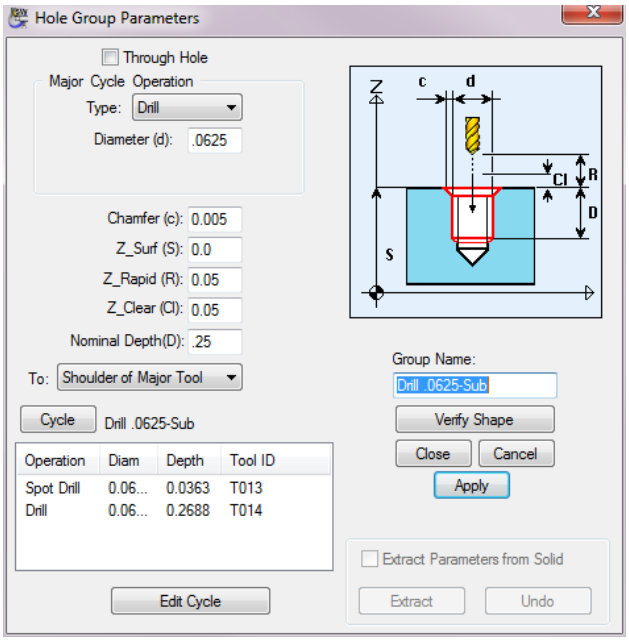
1. Click <**CAD/CAM>** switch in the lower left to switch to CAM mode.

2. Choose **New Hole Group** from the **Part Features** menu to display the Hole **Group Parameters** dialog.
3. Uncheck the **Through Hole** box.
4. Under **Major Cycle Operation** set the Type of Operation to **Drill**.
5. Under **Major Cycle Operation** enter .0625 (1.5) into the **Diameter** field.

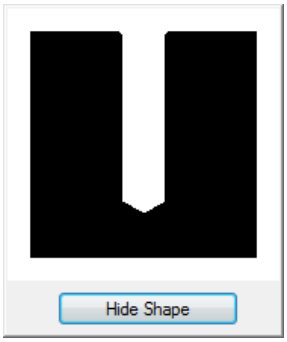
   Note: *The* ***Major Cycle Operation*** *parameters are used by* ***PartMaker*** *to search the* ***Cycles Database*** *for an appropriate cycle.*
6. Enter 0.005 (.1) into the **Chamfer (C)** field. **PartMaker** will calculate the depth of a Spot Drill according to the size of the Chamfer indicated.
7. Enter 0.0 into the **Z_Surf (S)** field.
8. Enter .250 (6.35) into the **Nominal Depth (D)** field.
9. In the **To** field under **Nominal Depth (D)**, select **Shoulder of Major Tool** to indicate where **PartMaker** will calculate the depth of the tools.
10. Click the **Cycle** button and **PartMaker** will select the appropriate cycle to create the desired hole by clicking **<Select>. T**he selected cycle should be **Drill .0625-Sub (Drill_1.5 z sub)**.



11 You will notice a cycle is chosen and the tools and calculated depths are listed in the field in the lower left of the **Hole Group Parameters**. The dialog should now appear as below.



12 Review the dialog, then Click <**Verify Shape**>. This will display a graphical representation of the cycle.



13 Click <**Hide Shape**> when done, returning to the **Hole Group Parameters**.

14 Click <**Apply**> to accept all the information.

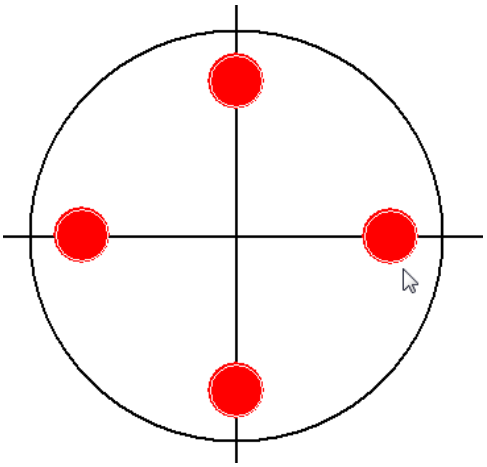
15 Click **<Close>** to close the Hole Group Parameters.

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

1 From the left hand tool bar, select the **Chain Circles Icon**.

2 Click on any one of the four circles and PartMaker will select the other three.

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



3 Select the **Selection** Icon on the left side of the screen.

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

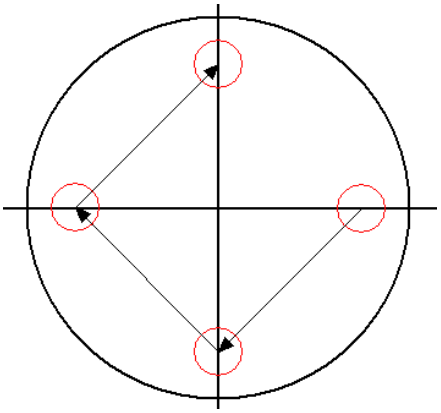
Tool Path Verification

1 Select Verify Work Group Tool Path from the Part Features menu. You will be prompted with the Tool Path Verification Options menu.

2 Check the **Hollow** radio button.

3 Click <**OK>** and you will see your tool path verified. Note: you may have to move and/or zoom out the part on the screen to view the simulation satisfactorily.

Note: The arrows indicate order of holes.



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

Creating an NC Program

At this point, you have completed creating the three faces for the Swiss tutorial part. You are now prepared to take the final steps in developing your part program.

The steps required for process plan development and program creation are described using the “generic” Fanuc-style Swiss-lathe post processor that comes standard with your **PartMaker** SwissCAM software. Synchronization and other processing related issues may vary slightly for different machine makers, but the steps below are instructive to help you build an understanding of process development in **PartMaker** SwissCAM.

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

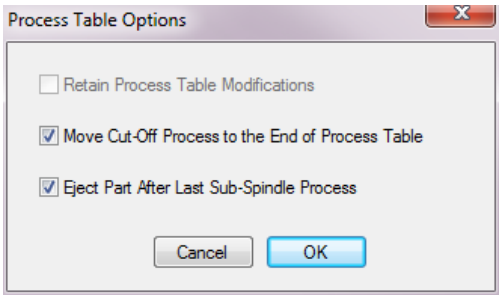
Generating a Process Table

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

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

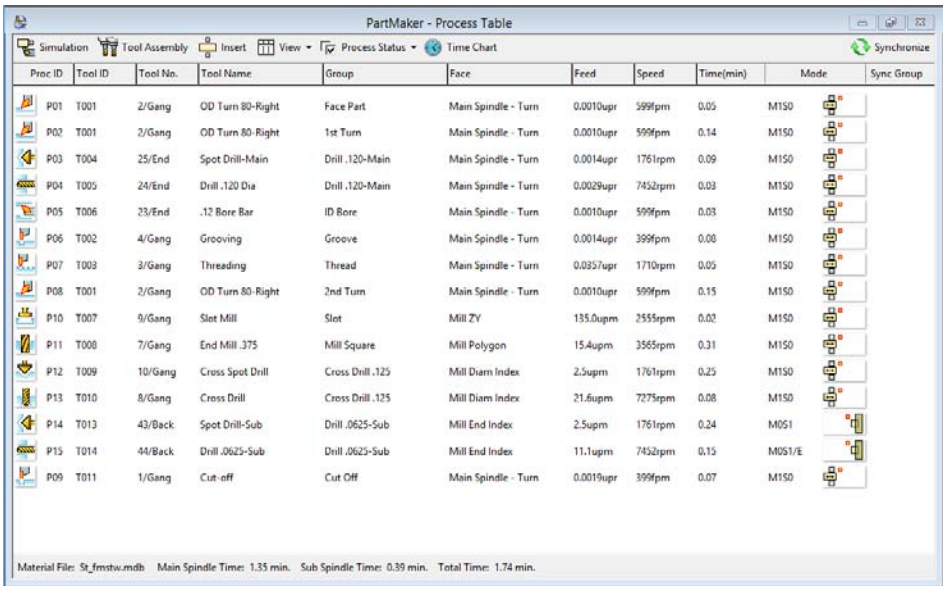
To create the Process Table:

1. Choose **Generate Process Table** from the **Job Optimizer** menu. Upon doing so, you will be prompted with the dialog below:

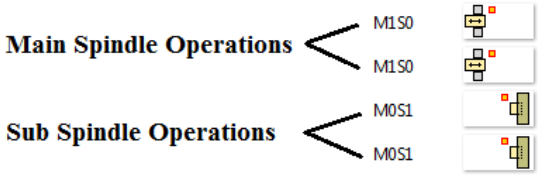


2. Leave the boxes in the **Process Table Options** dialog checked and click on **<OK>**.

The **Process Table** will be display as below.



*Notice the buttons on the right side of the screen. These open the **Set Modes** dialog for each process. These buttons also visually indicate the primary spindle for each process. You will use these buttons later to set **Synchronization**.*



Visual Process Synchronization and PartMaker Specific Modes

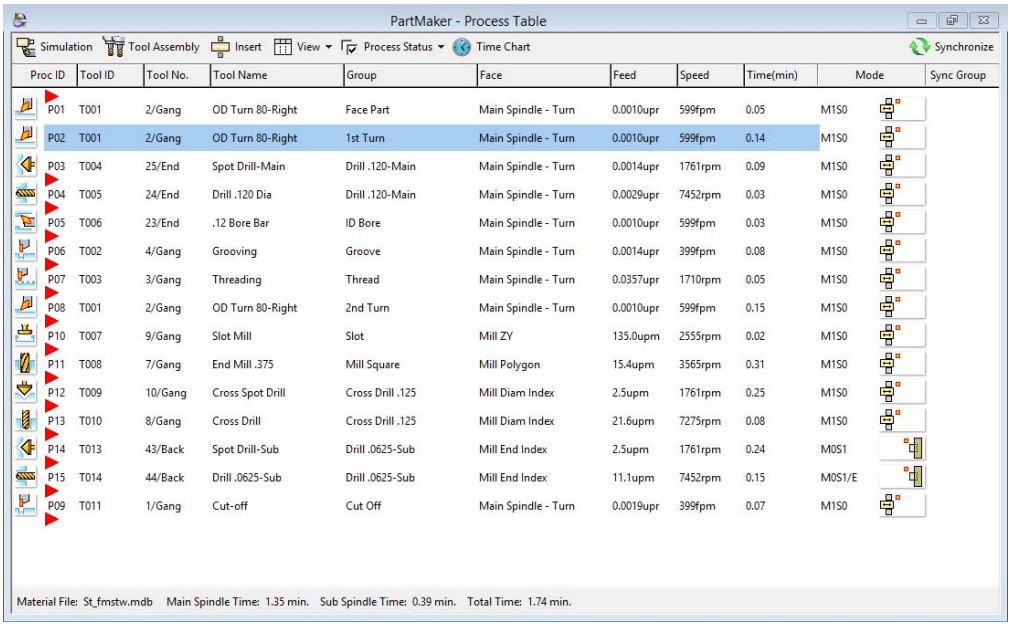
When generating a Process Table, **PartMaker** starts by choosing the processes in the first **Face Window** you have created in the **Setup** dialog and executing all the part features in that **Face Window** in the order they have been created. The software then moves onto the next **Face Window**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

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

More information on the **Set Modes** dialog can be found in Chapter 2, **PartMaker Fundamentals** of the SwissCAM User Guide. In addition, more information on synchronization supported by your machine can be found in the post processor reference guide that can be generated on demand. After select your post processor under the Job Optimzer menu, .choose **Job Optimizer > Create Post Processor Reference Guide...** and follow the prompts.

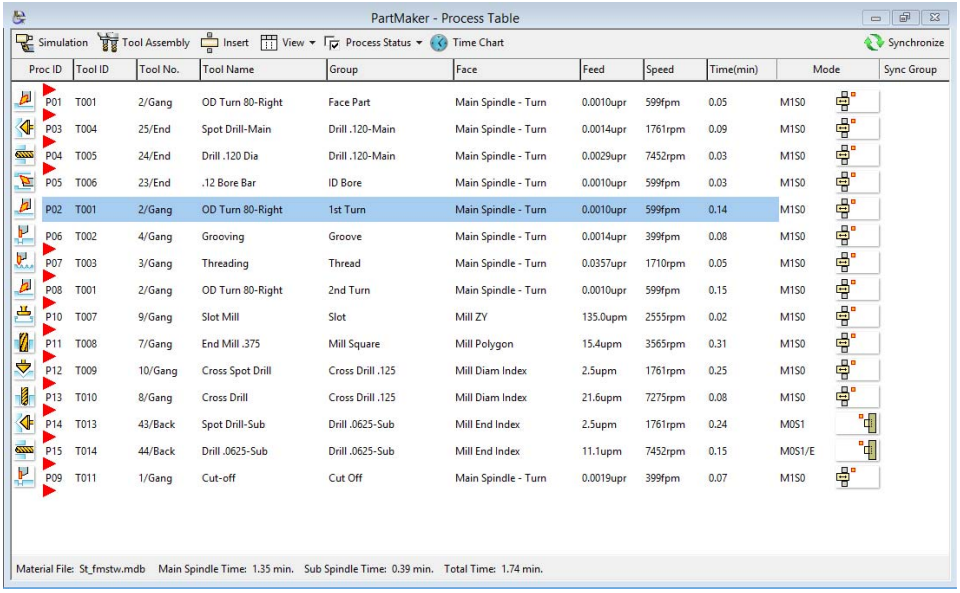
Arranging the Processes in the Process Table

1 Click in the middle of the screen on **Process P02** to select the **1st Turn** Process. This is indicated by highlighting it as the selected process.



2 You need to move this process to happen after **Process P05 ID Bore**. While **Process P02 1st Turn** is highlighted, click on the **Arrow** between **Process P05 ID Bore** and **Process P06 Groove** as shown.

The Process Table should now appear as below:

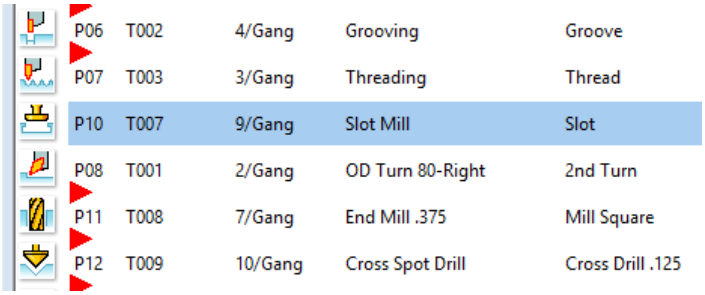


3 Next you need to move **Process P10 Slot** up above **Process P08 2nd Turn**. Click in the middle of the screen on **Process P10 Slot** to select it.

P06	T002	4/Gang	Grooving	Groove	Main Spindle - Turn	0.0014upr	399fpm	0.08	M1S0	
P07	T003	3/Gang	Threading	Thread	Main Spindle - Turn	0.0357upr	1710rpm	0.05	M1S0	
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn	Main Spindle - Turn	0.0010upr	599fpm	0.15	M1S0	
P10	T007	9/Gang	Slot Mill	Slot	Mill ZY	135.0upm	2555rpm	0.02	M1S0	
P11	T008	7/Gang	End Mill .375	Mill Square	Mill Polygon	15.4upm	3565rpm	0.31	M1S0	
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125	Mill Diam Index	2.5upm	1761rpm	0.25	M1S0	
P13	T010	8/Gang	Cross Drill	Cross Drill .125	Mill Diam Index	21.6upm	7275rpm	0.08	M1S0	

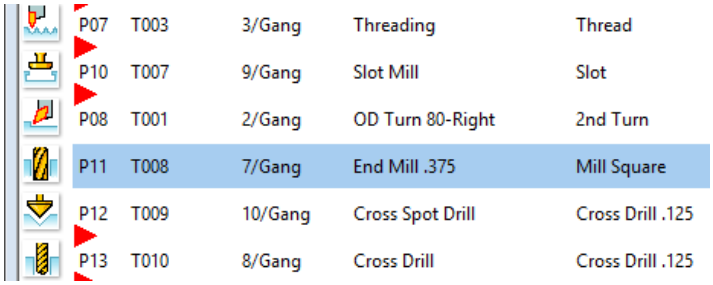
4 Click on the Arrow between Process P07 Thread and Process P08 2nd Turn to move Process P10 Slot above Process P08 2nd Turn.

The Process Table should appear as below:



P06	T002	4/Gang	Grooving	Groove
P07	T003	3/Gang	Threading	Thread
P10	T007	9/Gang	Slot Mill	Slot
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn
P11	T008	7/Gang	End Mill .375	Mill Square
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125

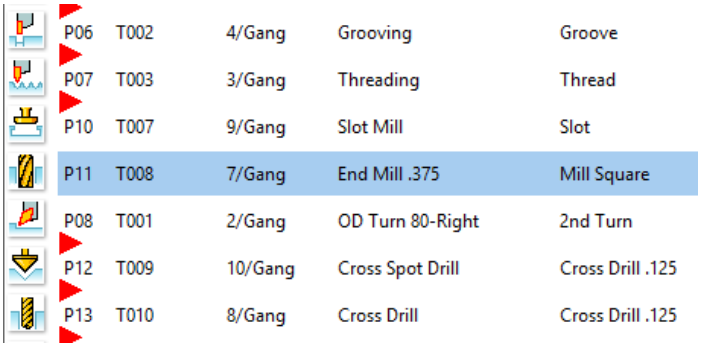
5 Next you need to move **Process P11 Mill Square** up above **Process P08 2nd Turn**. Click in the middle of the screen on **Process P11 Mill Square** to select it.



P07	T003	3/Gang	Threading	Thread
P10	T007	9/Gang	Slot Mill	Slot
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn
P11	T008	7/Gang	End Mill .375	Mill Square
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125
P13	T010	8/Gang	Cross Drill	Cross Drill .125

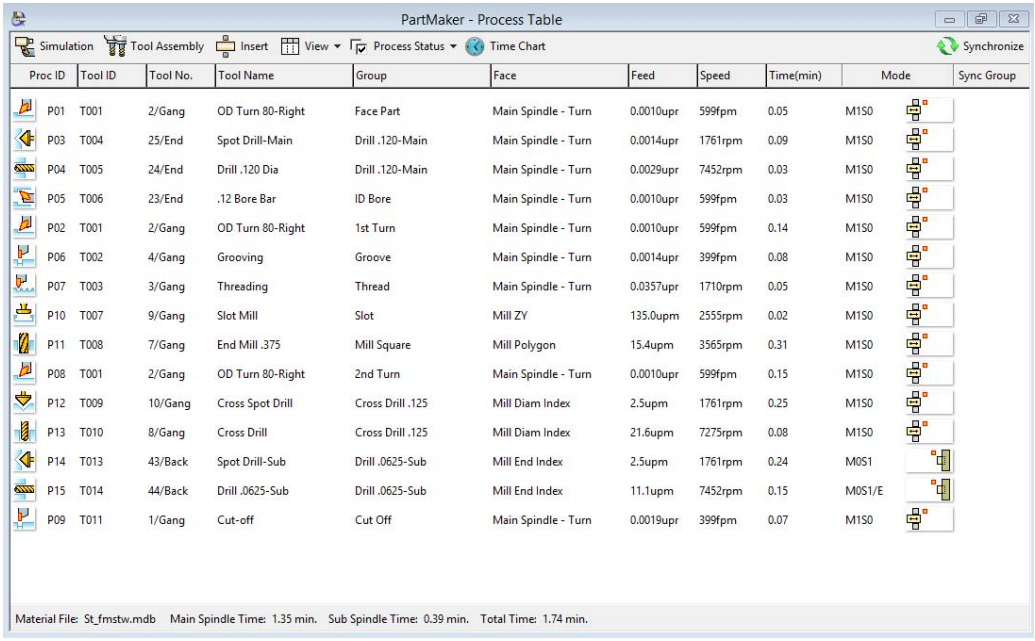
6 Click on the Arrow between Process P10 Slot and Process P08 2nd Turn to move Process P11 Mill Square above Process P08 2nd Turn.

The Process Table should now appear as below:



P06	T002	4/Gang	Grooving	Groove
P07	T003	3/Gang	Threading	Thread
P10	T007	9/Gang	Slot Mill	Slot
P11	T008	7/Gang	End Mill .375	Mill Square
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125
P13	T010	8/Gang	Cross Drill	Cross Drill .125

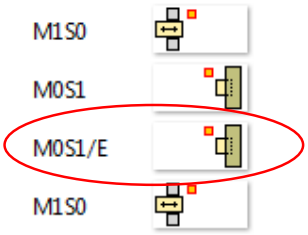
The full Process Table should appear as below:



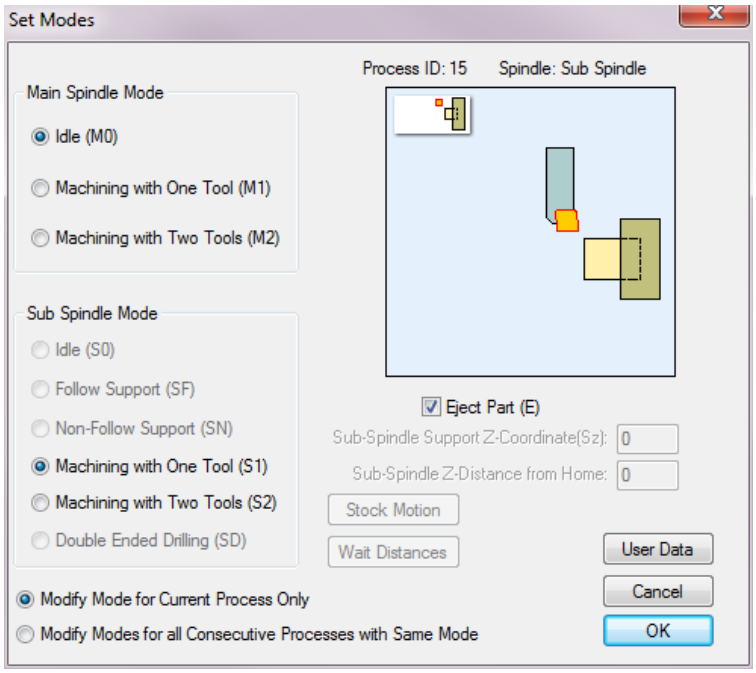
Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T001	2/Gang	OD Turn 80-Right	Face Part	Main Spindle - Turn	0.0010upr	599fpm	0.05	M1S0	
P03	T004	25/End	Spot Drill-Main	Drill .120-Main	Main Spindle - Turn	0.0014upr	1761rpm	0.09	M1S0	
P04	T005	24/End	Drill .120 Dia	Drill .120-Main	Main Spindle - Turn	0.0029upr	7452rpm	0.03	M1S0	
P05	T006	23/End	.12 Bore Bar	ID Bore	Main Spindle - Turn	0.0010upr	599fpm	0.03	M1S0	
P02	T001	2/Gang	OD Turn 80-Right	1st Turn	Main Spindle - Turn	0.0010upr	599fpm	0.14	M1S0	
P06	T002	4/Gang	Grooving	Groove	Main Spindle - Turn	0.0014upr	399fpm	0.08	M1S0	
P07	T003	3/Gang	Threading	Thread	Main Spindle - Turn	0.0357upr	1710rpm	0.05	M1S0	
P10	T007	9/Gang	Slot Mill	Slot	Mill ZY	135.0upm	2555rpm	0.02	M1S0	
P11	T008	7/Gang	End Mill .375	Mill Square	Mill Polygon	15.4upm	3565rpm	0.31	M1S0	
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn	Main Spindle - Turn	0.0010upr	599fpm	0.15	M1S0	
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125	Mill Diam Index	2.5upm	1761rpm	0.25	M1S0	
P13	T010	8/Gang	Cross Drill	Cross Drill .125	Mill Diam Index	21.6upm	7275rpm	0.08	M1S0	
P14	T013	43/Back	Spot Drill-Sub	Drill .0625-Sub	Mill End Index	2.5upm	1761rpm	0.24	M0S1	
P15	T014	44/Back	Drill .0625-Sub	Drill .0625-Sub	Mill End Index	11.1upm	7452rpm	0.15	M0S1/E	
P09	T011	1/Gang	Cut-off	Cut Off	Main Spindle - Turn	0.0019upr	399fpm	0.07	M1S0	

Setting the Eject Process

1. On **Process P15** Drill .0625-Sub go to the right side of the screen and find the last sub spindle process.
2. Click <**Set Modes**> (buttons on right of each process) for **Process P15** Drill .0625-Sub.



3. Find the **Eject Part** (E) check box inside the **Set Modes** dialog.



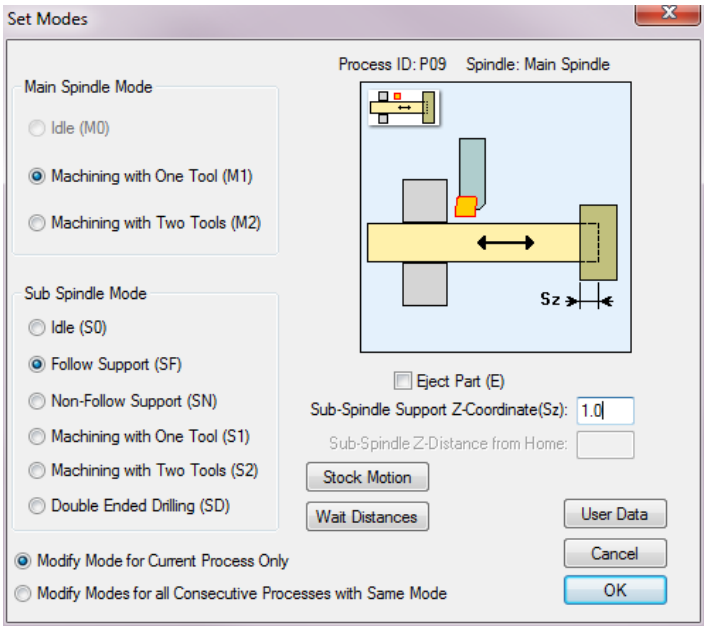
4. Click <**OK**> when done.

Setting the Cut-off Process for Sub Spindle Support

1. Select Process **P09 Cut Off.**

P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125	Mill Diam Index	4.6upm	3318rpm	0.14
P13	T010	8/Gang	Cross Drill	Cross Drill .125	Mill Diam Index	21.5upm	7255rpm	0.08
P14	T013	43/Back	Spot Drill-Sub	Drill .0625-Sub	Mill End Index	4.6upm	3318rpm	0.17
P15	T014	44/Back	Drill .0625-Sub	Drill .0625-Sub	Mill End Index	11.9upm	8000rpm	0.14
P09	T011	1/Gang	Cut-off	Cut-Off	Main Spindle – Turn	0.0019upr	399fpm	0.07

2. Click on the Mode Switch for the cutoff process.
3. Under **Sub Spindle Mode**, select **Follow Support (SF)**. Notice the graphics change according to the mode.
4. Enter 1.0 into the **Sub-Spindle Support Z-Coordinate (Sz)** field as shown below.



5. Click <**OK**> when done.

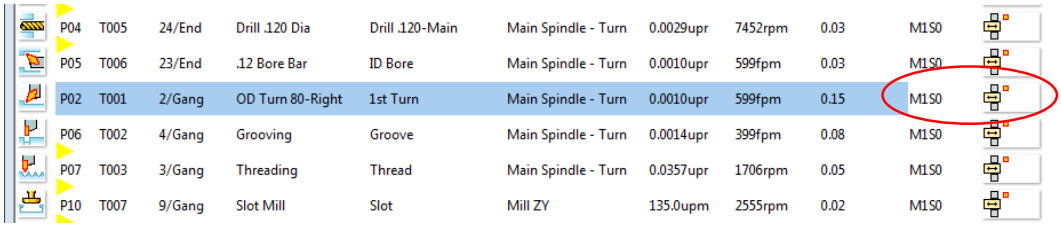
Synchronizing Main and Sub Spindle Processes in the Process Table

In this section you are going to synchronize **Main Spindle** processes and **Sub Spindle** processes to run at the same time to optimize the machine usage. The following operations will be run "simultaneously".

Main Spindle Processes Gang Tools	**Sub Spindle** Processes Back Tools
P02 1st Turn	P14 Drill .0625-Sub (Drill 1.5-Sub)
P06 Groove	P15 Drill .0625-Sub (Drill 1.5-Sub)
P07 Thread	
P10 Slot	
P11 Mill Square	
P08 2nd Turn	
P12 Cross Drill .125 (Cross Drill 3.5)	
P13 Cross Drill .125 (Cross Drill 3.5)	

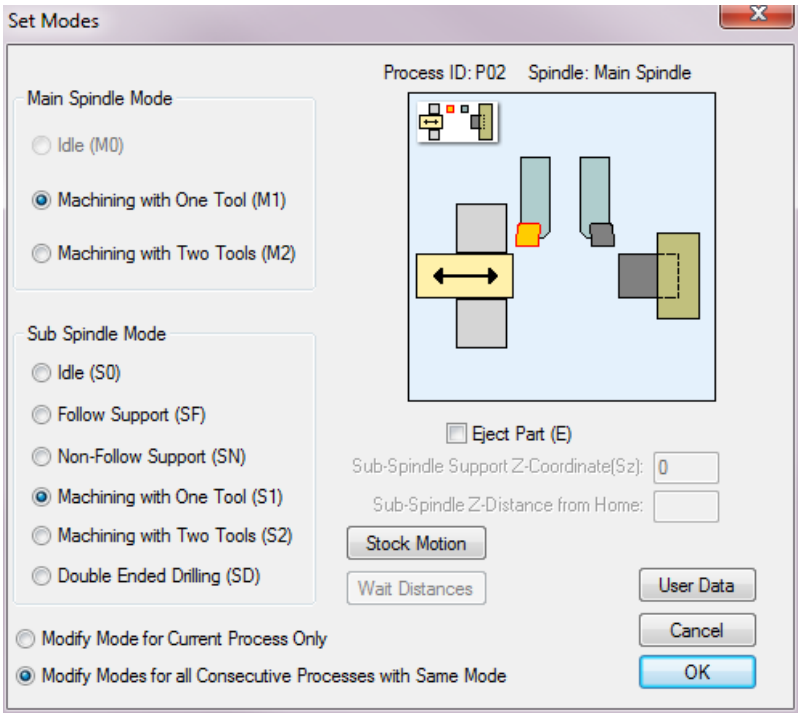
Note: *Notice processes P01, P03, P04 and P05 are NOT being synchronized with the operations on the sub spindle. The reason for this is that processes P03, P04 and P05 are being carried out by the End Working Tool Post and CANNOT work at the same time as the tools in processes P14 and P15, which are mounted on the Back Working Tool Post. The machine layout assumed for this part on page 3 of this tutorial precluded End Working and Back Working Tool Posts from being used simultaneously.*

1 On **Process P02 1st Turn** go to the right side of the screen and click <**Set Modes>**.



2 Under **Sub Spindle Mode**, select **Machining with One Tool (S1)**. This will allow **PartMaker** to **Synchronize** Sub Spindle processes with this process. Notice the graphics change according to the mode.

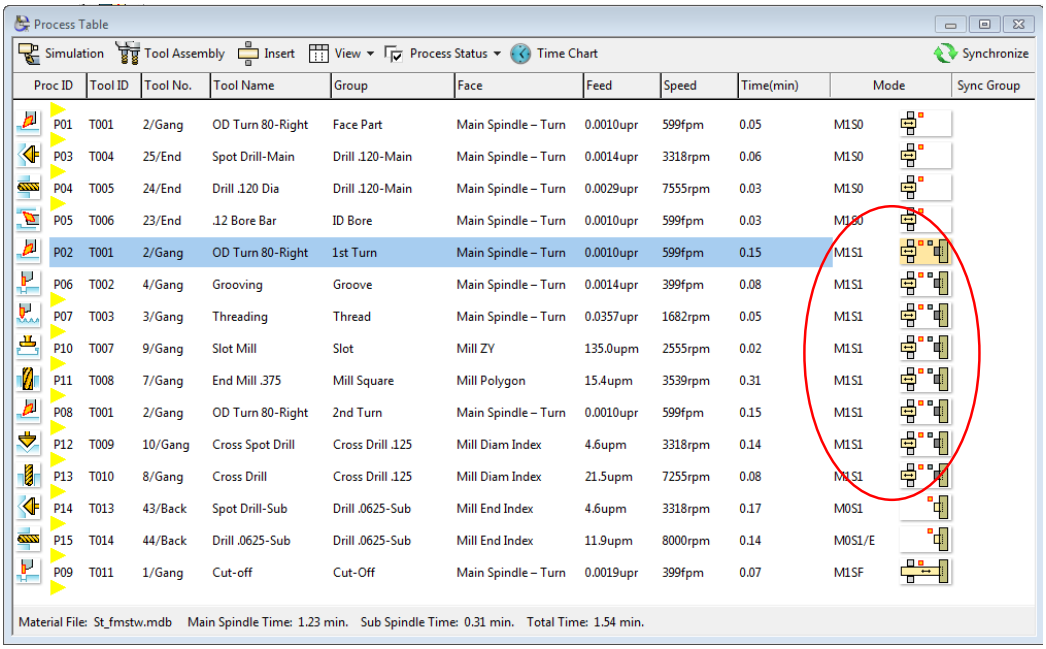
3 At the bottom of the dialog, select **Modify Modes for all Consecutive Processes with Same Mode**. This will set all the modes following the Process you are setting that are **M1S0** (Main Spindle One Tool, Sub Spindle Idle) to the new mode **M1S1** (Main Spindle One Tool, Sub Spindle One Tool).



4 Click <**OK**> when done.

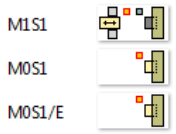
The Process Table should appear as below:

Notice the **Set Modes Buttons** to the right have changed for all the processes following **Process P02** 1st Turn that were **M1S0** to **M1S1**.

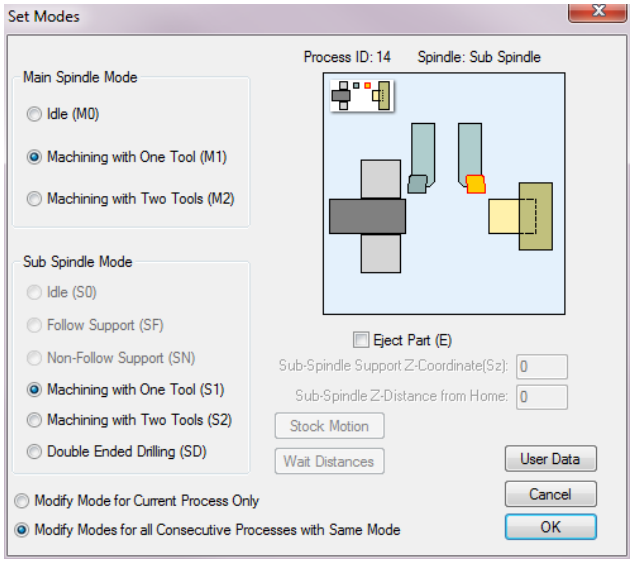


Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T001	2/Gang	OD Turn 80-Right	Face Part	Main Spindle – Turn	0.0010upr	599fpm	0.05	M1S0	
P03	T004	25/End	Spot Drill-Main	Drill .120-Main	Main Spindle – Turn	0.0014upr	3318rpm	0.06	M1S0	
P04	T005	24/End	Drill .120 Dia	Drill .120-Main	Main Spindle – Turn	0.0029upr	7555rpm	0.03	M1S0	
P05	T006	23/End	.12 Bore Bar	ID Bore	Main Spindle – Turn	0.0010upr	599fpm	0.03	M1S0	
P02	T001	2/Gang	OD Turn 80-Right	1st Turn	Main Spindle – Turn	0.0010upr	599fpm	0.15	M1S1	
P06	T002	4/Gang	Grooving	Groove	Main Spindle – Turn	0.0014upr	399fpm	0.08	M1S1	
P07	T003	3/Gang	Threading	Thread	Main Spindle – Turn	0.0357upr	1682rpm	0.05	M1S1	
P10	T007	9/Gang	Slot Mill	Slot	Mill ZY	135.0upm	2555rpm	0.02	M1S1	
P11	T008	7/Gang	End Mill .375	Mill Square	Mill Polygon	15.4upm	3539rpm	0.31	M1S1	
P08	T001	2/Gang	OD Turn 80-Right	2nd Turn	Main Spindle – Turn	0.0010upr	599fpm	0.15	M1S1	
P12	T009	10/Gang	Cross Spot Drill	Cross Drill .125	Mill Diam Index	4.6upm	3318rpm	0.14	M1S1	
P13	T010	8/Gang	Cross Drill	Cross Drill .125	Mill Diam Index	21.5upm	7255rpm	0.08	M1S1	
P14	T013	43/Back	Spot Drill-Sub	Drill .0625-Sub	Mill End Index	4.6upm	3318rpm	0.17	M0S1	
P15	T014	44/Back	Drill .0625-Sub	Drill .0625-Sub	Mill End Index	11.9upm	8000rpm	0.14	M0S1/E	
P09	T011	1/Gang	Cut-off	Cut-Off	Main Spindle – Turn	0.0019upr	399fpm	0.07	M1SF	

5 On **Process P14** Drill .0625-Sub go to the right side of the screen and click <**Set Modes**>.



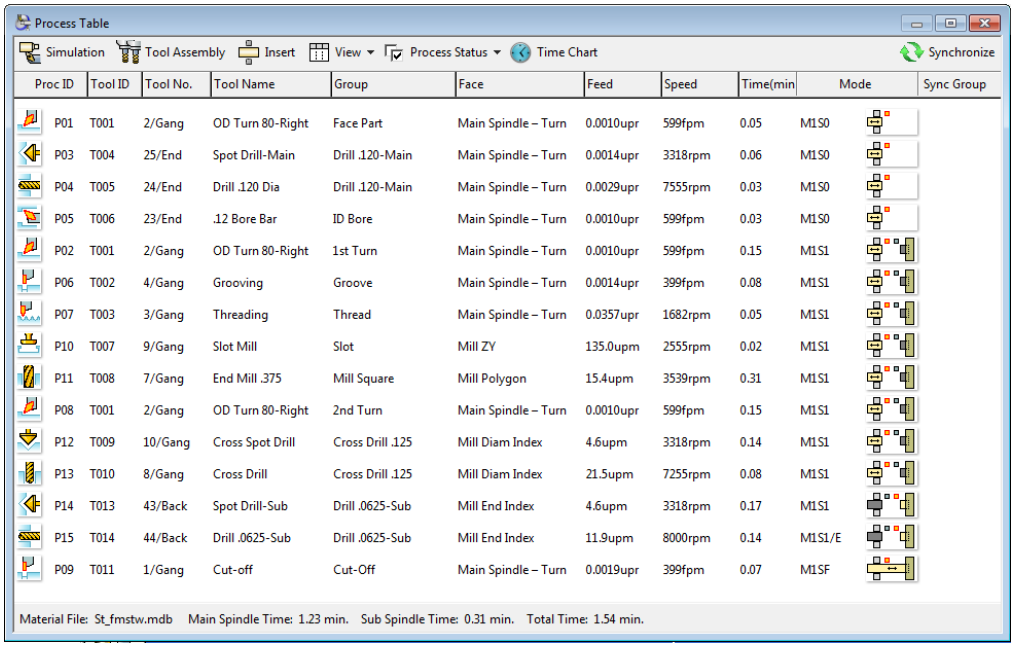
6 Under **Main Spindle Mode**, select **Machining with One Tool (S1)**. This will allow **PartMaker** to **Synchronize** Main Spindle processes with this process. Notice the graphics change according to the mode.



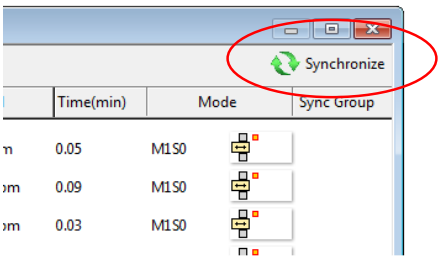
7 At the bottom of the dialog, select **Modify Modes for all Consecutive Processes with the Same Mode**. This will set all the modes following the Process you are setting that are **M0S1** (Main Spindle Idle, Sub Spindle One Tool) to the new mode **M1S1** (Main Spindle One Tool, Sub Spindle One Tool).

The Process Table should appear as below:

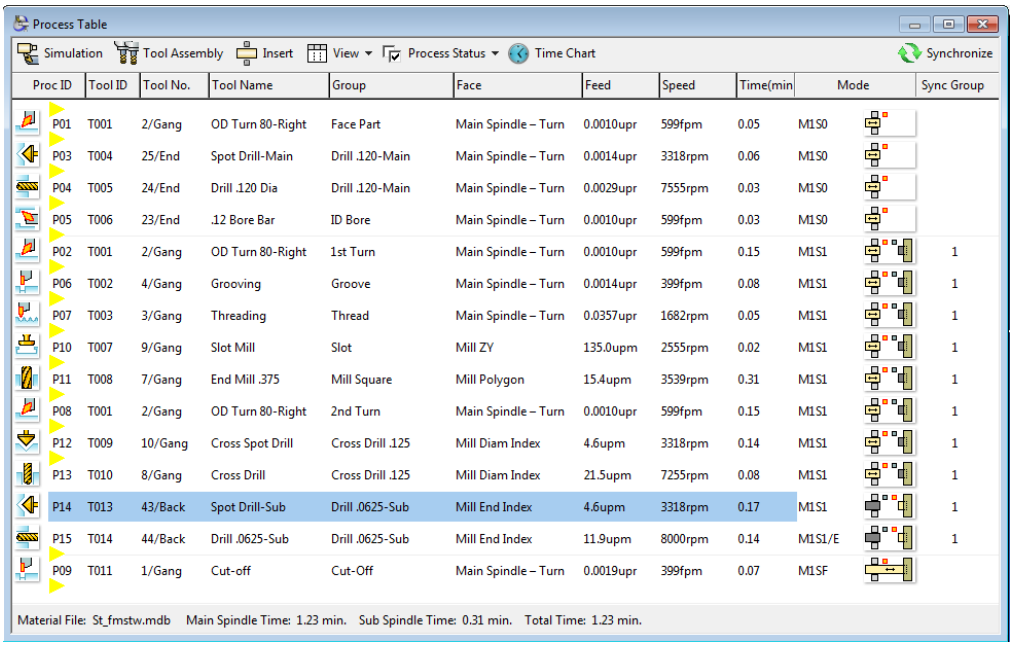
Notice the **Set Modes Buttons** to the right have changed for all the processes following **Process P14** Drill 1.5-Sub that were **M0S1** to **M1S1**.

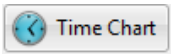


8 Click <**Synchronize**> in the upper right to create the synchronized groups.

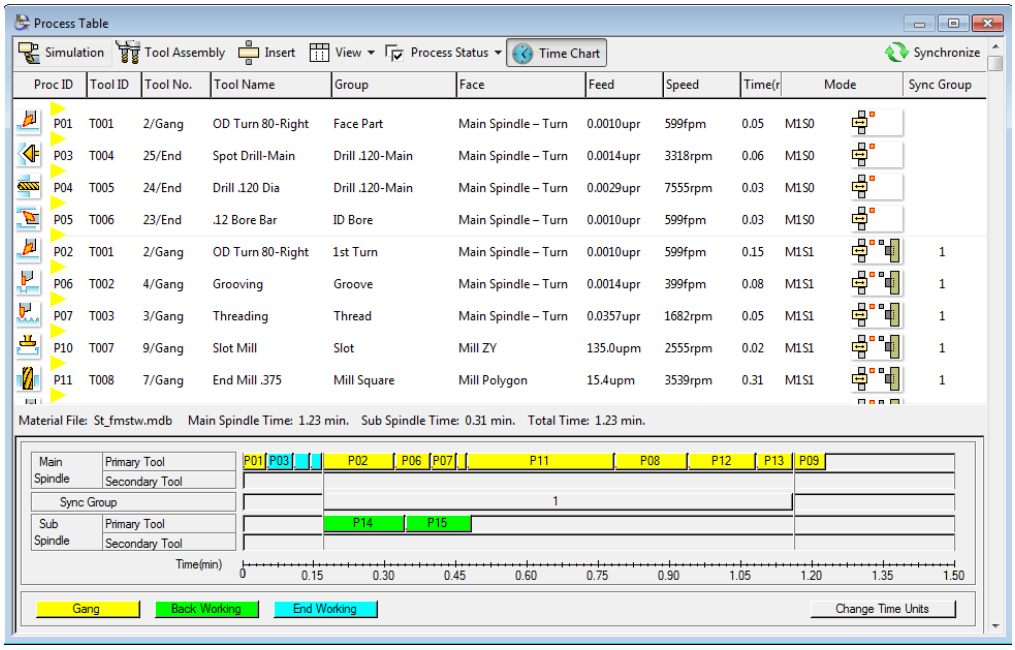


Notice the **Sync** Group number on the right hand side of the Process Table.



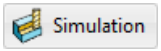


9 Click < **Time Chart**> on the top of the Process Table to display the Time Chart. The Time Chart displays the “overlapping” operations, the result of your process synchronization; graphically showing just how much sub-spindle machining time you are “getting for free.”



Simulating the Machining Process

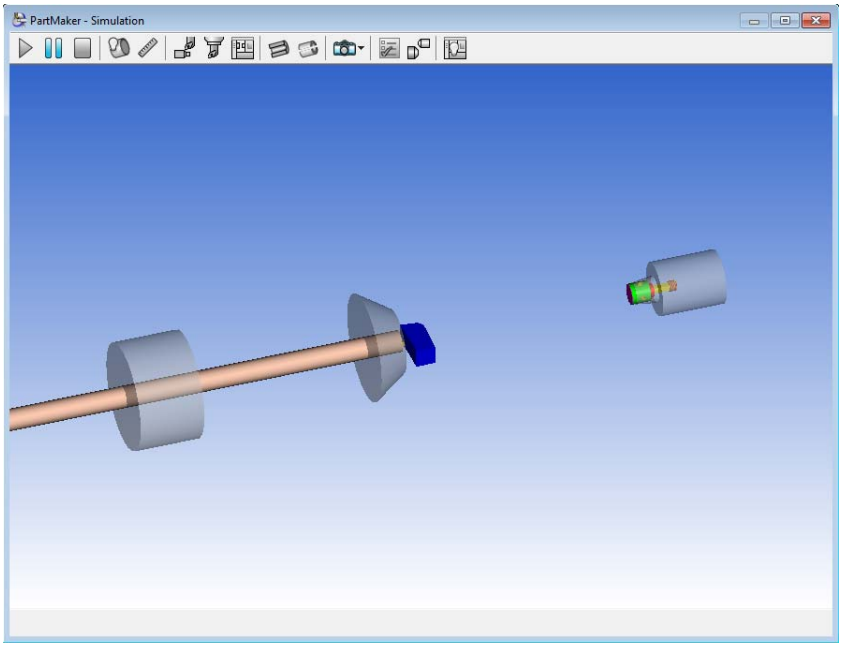
Having completed generating the Process Table, synchronizing operations and loading a post processor, you can now simulate the processes. The simulation will show your part being cut just how it would on your machine tool. Simulation will show synchronization between your Main and Sub-Spindles. It will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



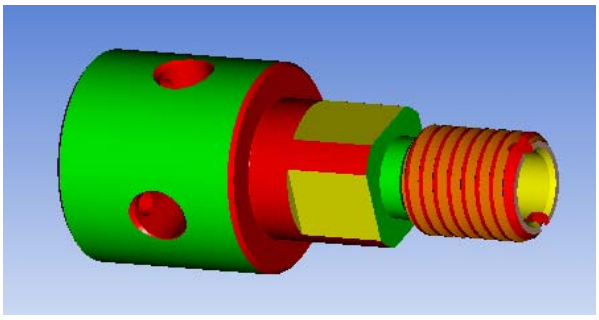
1 Click the <**Simulation**> button at the bottom left hand portion of the Process Table.

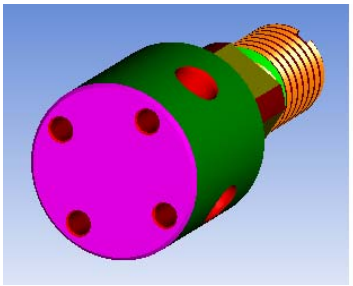
2 Press the Space Bar or choose **Simulation** from the **Simulation** menu to begin the simulation.

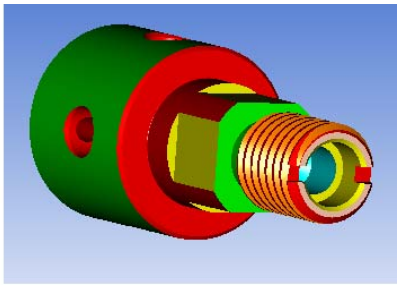
Viewing Basic Simulation



When you are done viewing the simulation, select **Show Finished Part** from the **Simulation** menu.

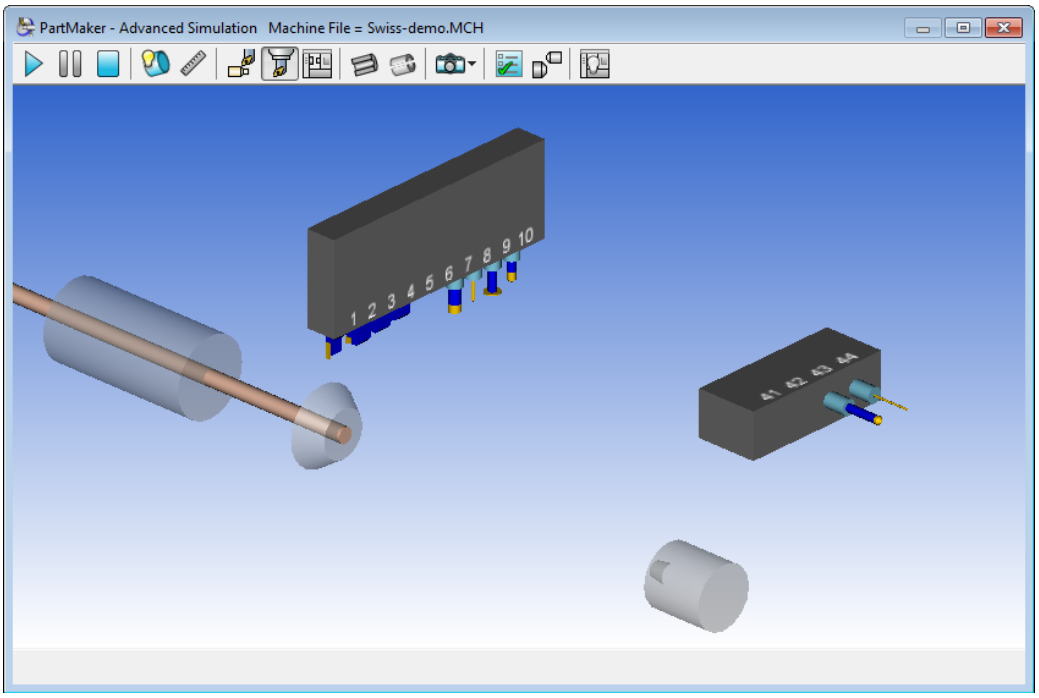






Viewing Advanced Simulation

Press the <**Spacebar**> to start simulation.



Note: In ADVANCED simulation, tools are shown held in PARAMETRIC tool holder blocks that represent the standard locations on Swiss machines: Gang, Turret, Front, Back

Rotating, Zooming and Sectioning the part works exactly the same as it does in Basic Simulation.

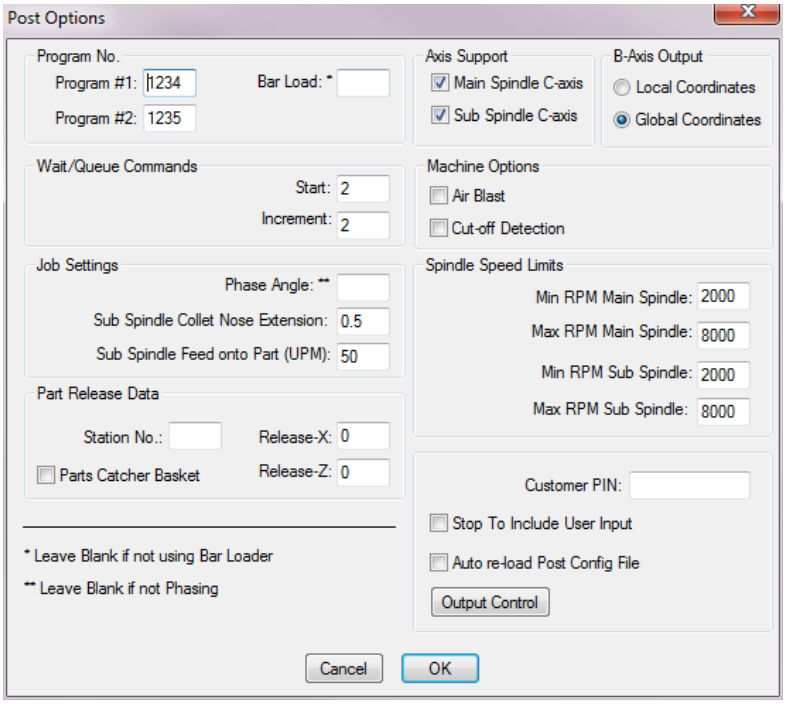
Generating an NC Program

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

1 Choose Generate NC Program from the Job Optimizer menu.

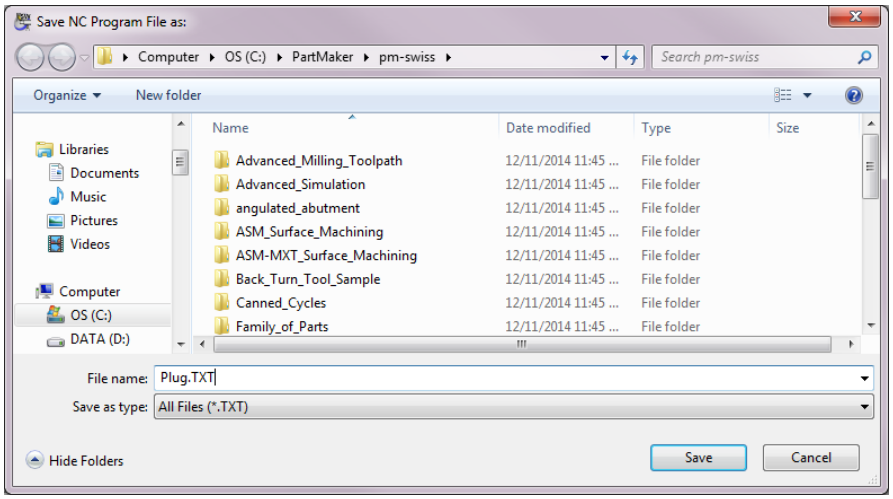
Note: *Choosing the* ***Generate NC Program*** *command from the* ***Job Optimizer*** *menu has the same effect as clicking the* ***Sync Group*** *button on the Process Table if* ***Sync Groups*** *have not yet been created.*

2 You will next see the **Post Options** dialog. Set the **Post Options** dialog as shown below:

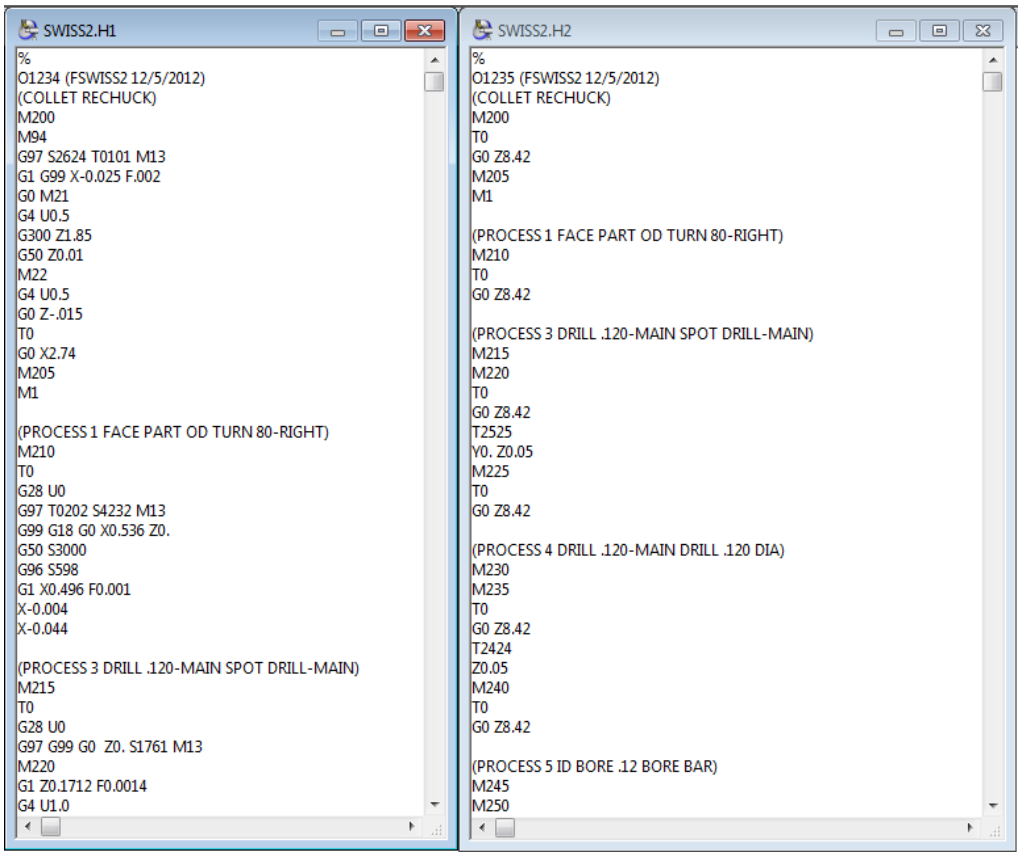


In the **Post Options** dialog you set certain parameters for assignment during post processing. Please refer to your machine specific addendum for more information on how each of the fields in this dialog pertain to your Swiss turning equipment.

3 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog. In this dialog, enter a name for your program. Here you should enter **"Plug.TXT"**.



4 Click **<Save>** to generate the NC program. You will see your programs as shown below:



SWISS2.H1

```
%
O1234 (FSWISS2 12/5/2012)
(COLLET RECHUCK)
M200
M94
G97 S2624 T0101 M13
G1 G99 X-0.025 F.002
G0 M21
G4 U0.5
G300 Z1.85
G50 Z0.01
M22
G4 U0.5
G0 Z-.015
T0
G0 X2.74
M205
M1

(PROCESS 1 FACE PART OD TURN 80-RIGHT)
M210
T0
G28 U0
G97 T0202 S4232 M13
G99 G18 G0 X0.536 Z0.
G50 S3000
G96 S598
G1 X0.496 F0.001
X-0.004
X-0.044

(PROCESS 3 DRILL .120-MAIN SPOT DRILL-MAIN)
M215
T0
G28 U0
G97 G99 G0  Z0. S1761 M13
M220
G1 Z0.1712 F0.0014
G4 U1.0
```

SWISS2.H2

```
%
O1235 (FSWISS2 12/5/2012)
(COLLET RECHUCK)
M200
T0
G0 Z8.42
M205
M1

(PROCESS 1 FACE PART OD TURN 80-RIGHT)
M210
T0
G0 Z8.42

(PROCESS 3 DRILL .120-MAIN SPOT DRILL-MAIN)
M215
M220
T0
G0 Z8.42
T2525
Y0. Z0.05
M225
T0
G0 Z8.42

(PROCESS 4 DRILL .120-MAIN DRILL .120 DIA)
M230
M235
T0
G0 Z8.42
T2424
Z0.05
M240
T0
G0 Z8.42

(PROCESS 5 ID BORE .12 BORE BAR)
M245
M250
```

You have just completed programming Swiss_Tutorial2 – Plug.

Chapter 2: PartMaker® SwissCAM™ Programming Tutorial Part 3 – Reamer

Introduction

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

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 gives 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.

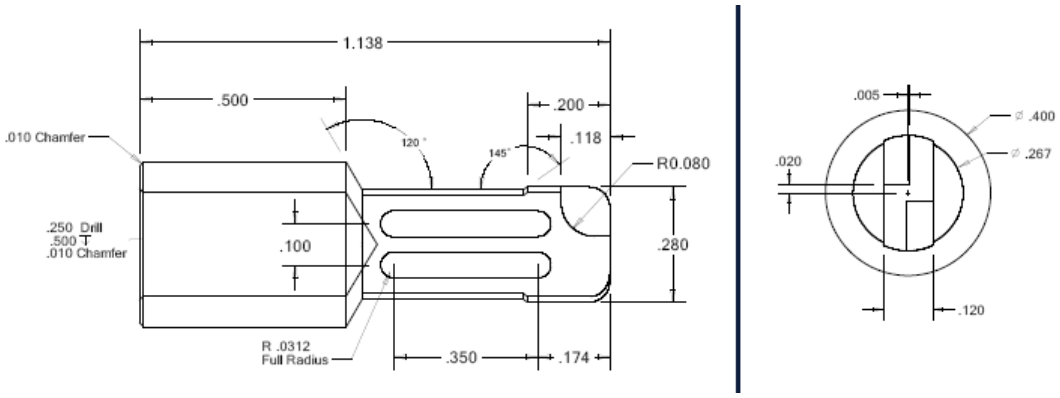
How you will create Part 3 – Reamer using SwissCAM

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

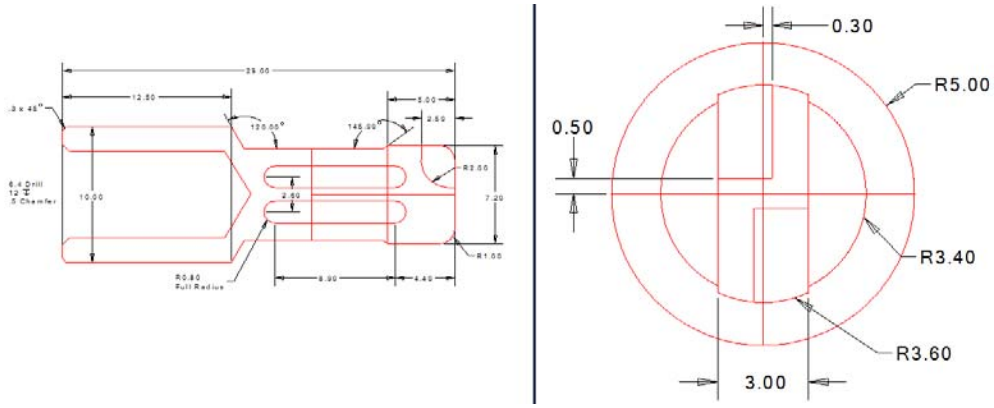
- Start PartMaker
- View tool information for the part
- View cycle information for the part
- Select a material for the part
- Setup defaults for Swiss turning
- Define the stock boundaries for the Face Window
- Open a new Face Window
- Create geometry
- Perform turning operations on the main spindle
- Perform milling operations in a ZY plane
- Duplicating the Mill ZY window
- Perform drilling operations on the sub or pick-off spindle
- Create a Process Table
- Synchronize machining operations on the main and sub-spindles
- Simulate the Machining Process
- Generate an NC Program

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

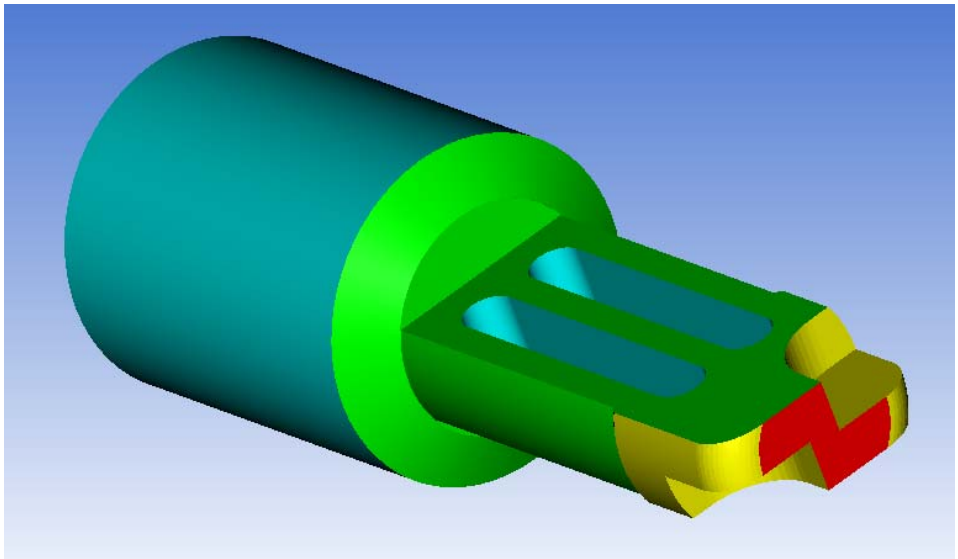
Inch Dimensions



Metric Dimensions



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



Getting Started

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

The machining functions used in this tutorial are as follows:

Turn – Main Spindle:

Use turning tools to perform a variety of Swiss-turning operations on the main spindle including facing, OD turning, and pick-off.

Mill ZY Plane – Main Spindle:

Use a 3/8 End mill to mill the part flat.

Use a .156 End mill to mill the .08 Radius and the two Slots.

Use a .0625 End mill to mill the two Slots.

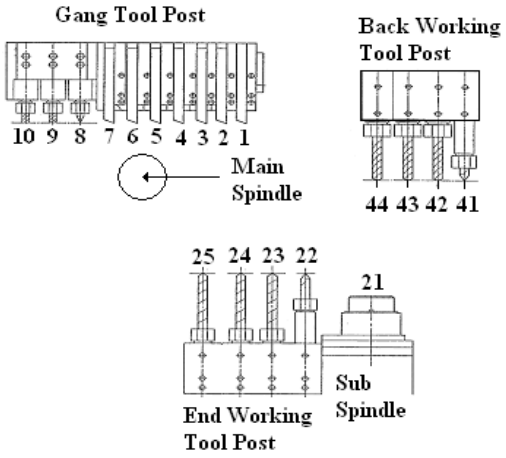
Duplicating a window:

Using the **Duplicate** command to copy the features into another window.

Turn – Sub Spindle:

Use on centerline drilling tools to create a hole on the sub spindle.

Programming Swiss-type lathe operations with **PartMaker** is performed in a unique manner. No matter what the orientation of the part in the guide bushing or sub spindle, **PartMaker** programs all Swiss operations as if being machined in a conventional lathe, i.e. all programming is carried out from right to left. For this tutorial you are going to use a machine layout like the one shown below.



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

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

Completed Job File:

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss3.job

Completed Tools File:

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss3.tdb

Completed Cycles File:

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss3.cdb

Machine Data File:

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss-demo.mch

Post Configuration File:

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss-demo.pst

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\Swiss-demo.sub

DXF files

C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\mill_zy.dxf

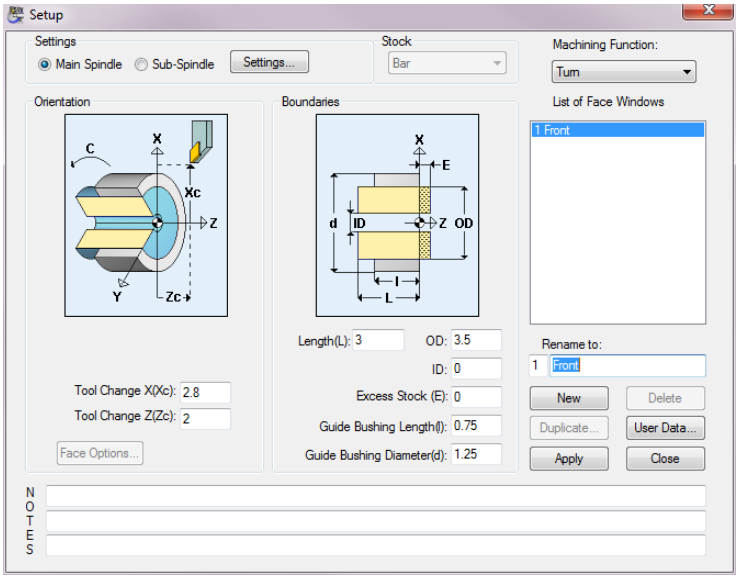
C:\PartMaker\pm-swiss\Swiss Tutorial 3\Inch-Swiss Tutorial 3\turn.dxf

Starting PartMaker SwissCAM

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

1. Double click on the SwissCAM icon on your desktop or choose SwissCAM from the PartMaker Group accessed through Programs under the Windows Start button.
2. 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.

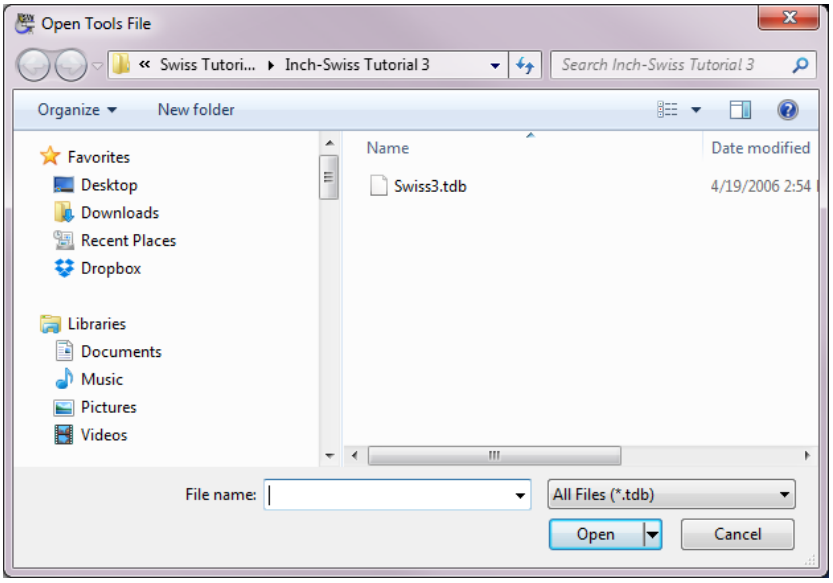
   This dialog will be explained in greater depth starting on page 9.



View the Tool Information

The tool library for the Swiss tutorial has already been created. To view the tools:

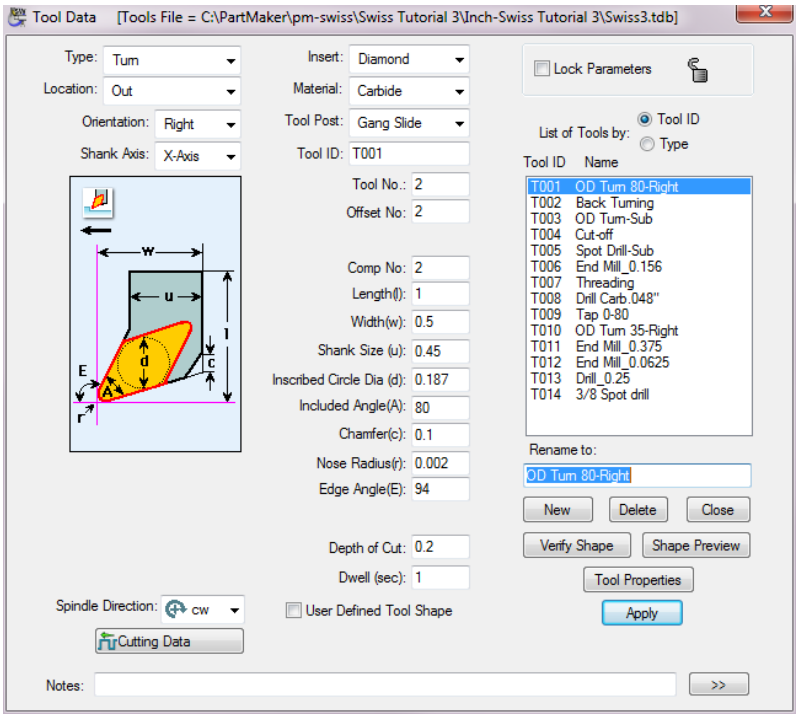
1 Open the Tools file by choosing **Open Tools File** from the **File** menu.



Click the Swiss Tutorial 3 folder(and go to inch or metric folder) and select **swiss3.tdb** file as shown above and click **<Open>**.

2 Choose **Tools** from the **ToolMinder** menu

The first of the tools to be used in the Swiss tutorial is shown below:



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. See Chapter 2 for more information on creating your own tools.*

3 Click the **<Close>** button to close the **Tools** dialog.

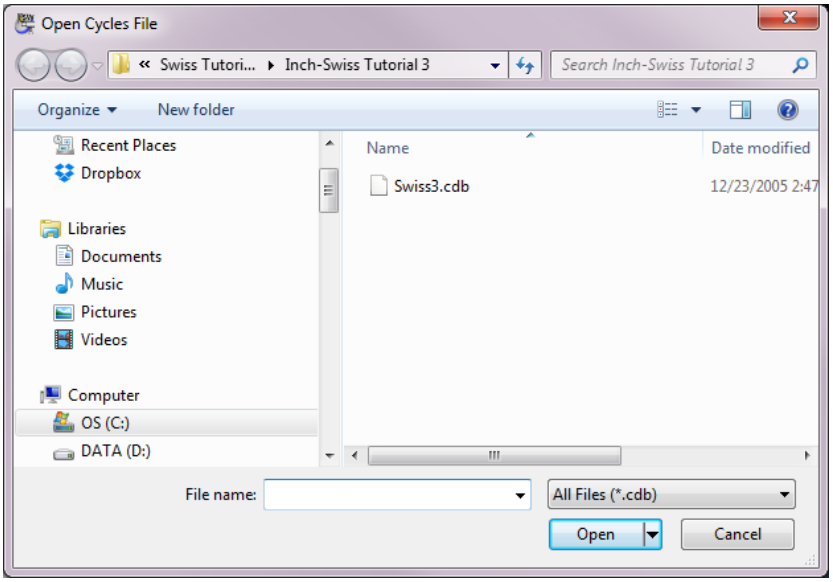
View the Cycles Information

At this point, you should be familiar with the concept of Cycles in **PartMaker**. Cycles are sequences of repetitive operations used to machine a hole

The cycles library for the Swiss tutorial has already been created. To view the cycles:

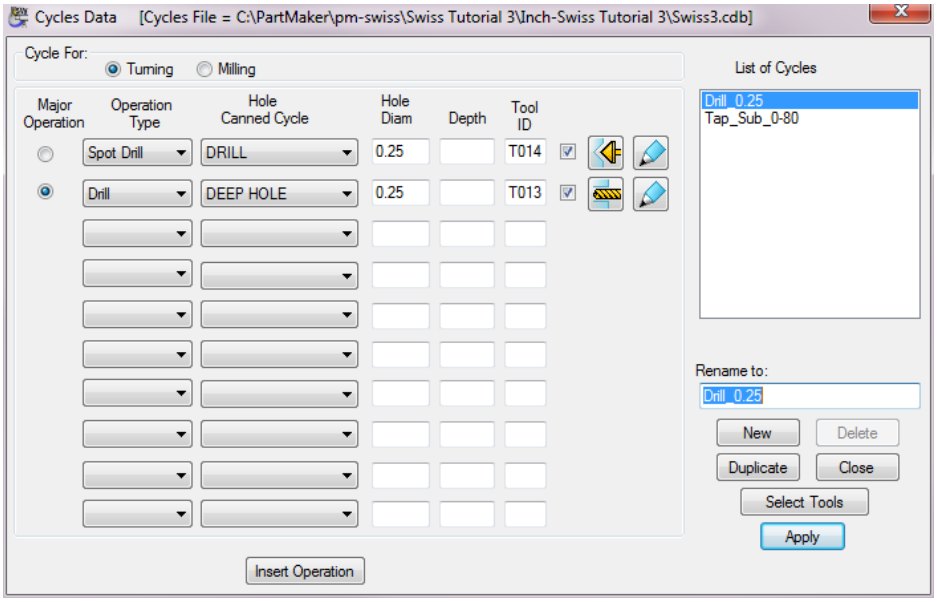
Open the Cycles file by choosing **Open Cycles File** from the **File** menu.

1 Open the **Cycles** File



2 Select **swiss3.cdb** file as shown above and click **<Open>**.

3 Choose **Cycles** from the **ToolMinder** menu

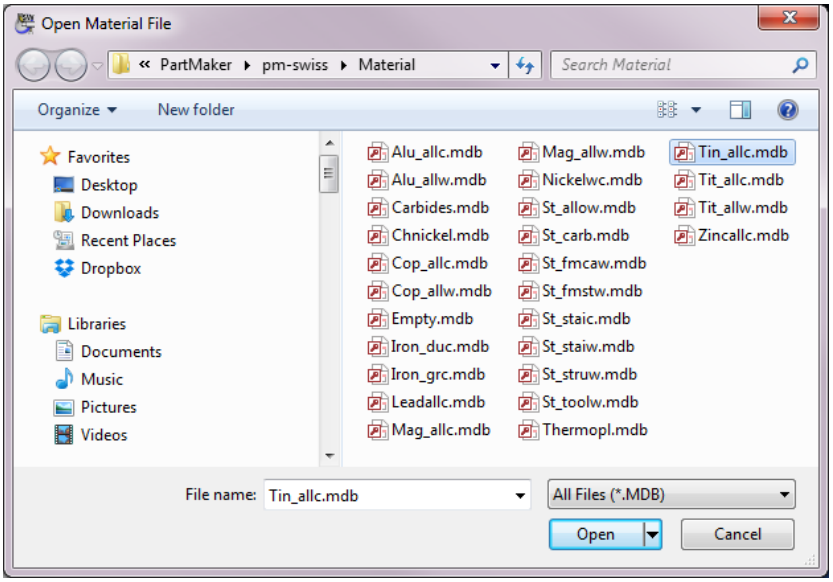


4 Click the **<Close>** button to close the **Cycles** dialog.

Select a Material for the Part

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

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

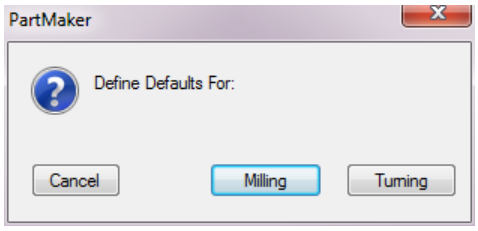


You have now loaded the tools, cycle, and material files. You will be guided through a step-by-step process to develop a job file for this tutorial.

Setting Defaults

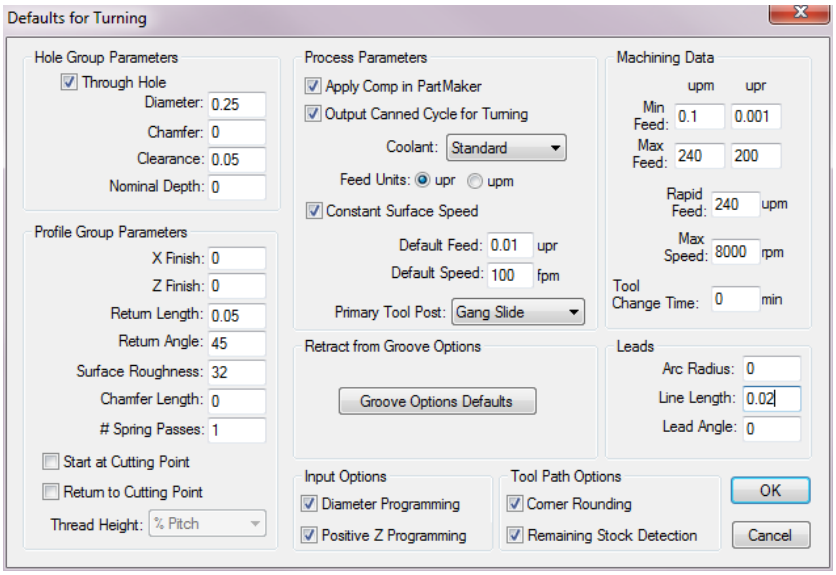
Before beginning the Swiss-tutorial, you will have to set certain turning defaults to facilitate the programming process:

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



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

2. After doing so, you will see the dialog below:



3. In Swiss machining, in order to avoid pulling stock back into the guide bushing it may be desirable not to Start and Return to a Cutting Point when programming in **PartMaker**. When not using Cutting Point to control the start and end point of the tools, you will be able to control tool entry and exit by setting **Leads In** and **Leads Out** in the Profile Group Parameters dialog for each operation.

4. Make sure to check the **Diameter Programming** box. Checking this box will allow you to define all geometry and feature programming in terms of diameter, alleviating the need to divide measurements in half when programming or drawing on the X-axis in Turning Face Windows.

5. Set a default **Line Length** for your programmed **Leads**. For the Swiss tutorial this value should be 0.02 (.1).

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

6. Note that the Positive Z Programming button is checked. This tutorial is done with drawing and programming done in the positive Z.

7 **Thread Height.** The following options are available

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

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

Once you have looked at the tools and cycles databases, selected a material for the job, and setup the defaults for your job as shown above, you can move on to begin programming the tutorial part by clicking on <**OK**> to close out of this dialog.

Turning on the Main Spindle

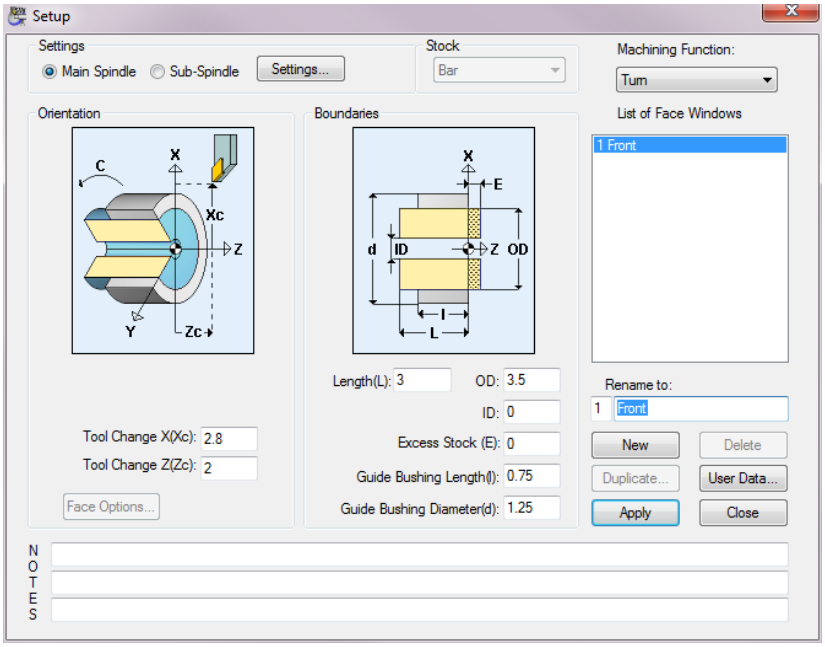
This section of the tutorial will instruct you in the steps necessary to perform the turning operations on the main spindle.

Setting Up the Face Window

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

1 Choose **Setup** from the **View** menu to view the **Setup** dialog box (or click the **Shortcut to the Setup** icon in the upper left hand corner of the screen). In this dialog, you will define the machining methods, part boundaries, and location of the stock with respect to the programming origin.

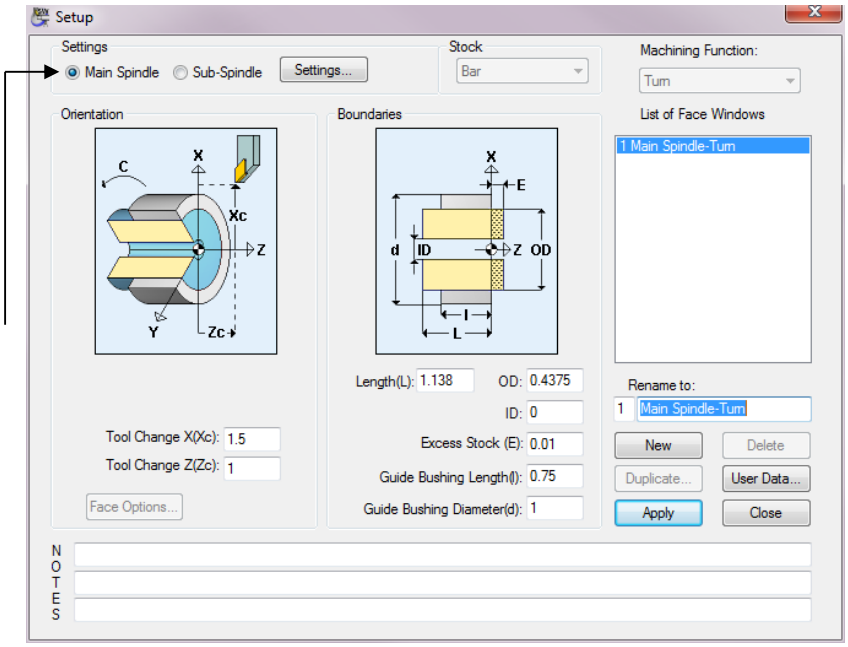
The default **Setup** screen will appear as shown below.



2 Rename the current **Face Window** in the **List of Face Windows** field by typing **Main Spindle-Turn** in the **Rename To:** field and click the **<Apply>** button.

3 Now enter the parameters listed below in the appropriate fields: Press the **<Tab>** key to move from field to field.

Length (L):	1.138	(29.0)
OD:	0.4375	(10.0)
ID:	0	(0)
Excess Stock (E):	0.01	(0.1)
Guide Bushing Length (l):	0.75	(12.0)
Guide Bushing Diameter (d):	1.0	(20.0)
Tool Change X(Xc):	1.5	(75.0)
Tool Change Z (Zc):	1.0	(0)



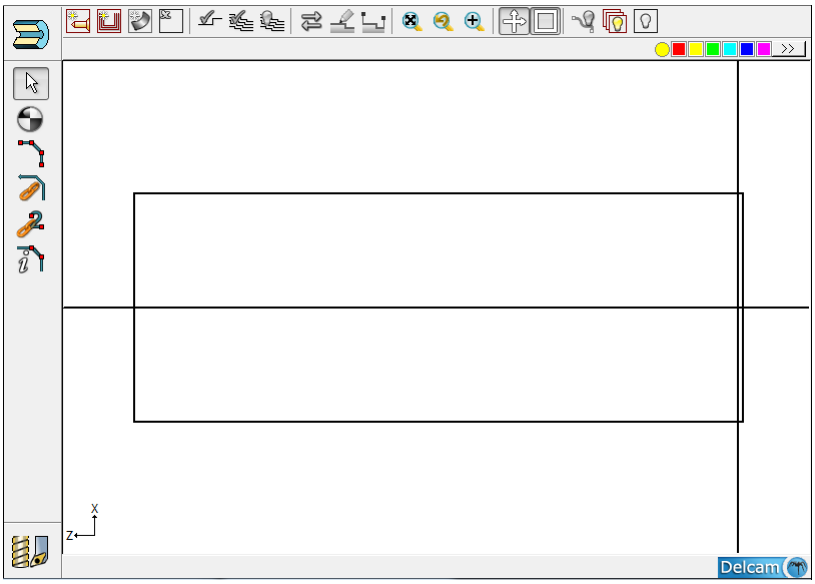
Note: *In the dialog above, the* ***Main Spindle*** *radio button is checked. This designates that this operation will be programmed on the main spindle of the machine.*

4 When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current **Face Window**. Click the **<Close>** button to close the **Setup** dialog.

5 In the CAM **Face Window**, choose **Show Axes** from the **View** menu. This will display the horizontal and vertical axis lines which will help you create the geometry in the next section of the tutorial.

Tip: *If at any time the menu commands become "grayed out" and unusable, this indicates that the* ***Face Window*** *you are working in has become inactive. This will be evident if the top bar of your window (where the name of the window is displayed) has become gray. To reactivate the window, click anywhere within your* ***Face Window****.*

6 Choose **Show Boundaries** and **Show Axis** icons from the main tool bar or the **View** menu. Your screen should appear as shown below:



Creating Geometry in CAD Mode

In this section, you will create the geometry for the turning operations to be performed on the main spindle.

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

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

CAD **CAM**

2 Select the **Connected Lines Icon** from the drawing icons on left side of the **Face Window**. Here, you will draw the part on the print on the second page of this chapter by plotting ZX coordinates, using the **ZX Coordinates Snap Mode** Icon from the **Snap Modes** icon bar at the top of the screen.

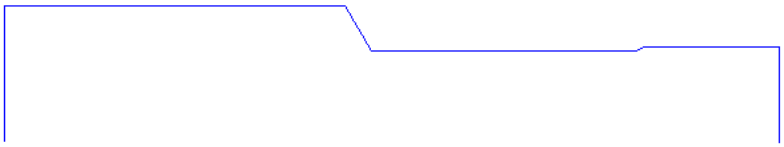
After selecting the **ZX Coordinates Snap Mode** Icon two data entry fields will appear in the lower left hand corner of the **Face Window**.

Note: *When plotting coordinates in* ***PartMaker*** *CAD, you do not need to reenter a coordinate if it is the same as the last one entered. You can simply leave the space blank and* ***PartMaker*** *will default to the last coordinate entered. You can move between data entry fields by pressing the* ***<Tab>*** *button or using your mouse.*

Now enter the ZX coordinates as shown below, using the **<Tab>** button to move between data entry fields and pressing **<Enter>** after each:

Z Coordinate	**X Coordinate**	
0	0	**<Enter>**
<default>	0.28 (7.2)	**<Enter>**
0.2 (5.0)	<default>	**<Enter>**
0.2093 (5.286)	0.267 (6.8)	**<Enter>**
0.5996 (15.586)	<default>	**<Enter>**
0.638 (16.5)	0.4 (10)	**<Enter>**
1.138 (29.0)	<default>	**<Enter>**
<default>	0	**<Enter>**

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

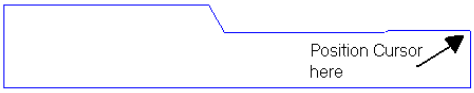


3 You will now place a Radius on the front of the part. To do so, select the **Fillet** icon. A field will then open up at the bottom of the window prompting you to enter the size of the fillet.

4 Enter a value of 0.05(1.0) and press **<Enter>**.

Note: *If your geometry is too far over to the right to see the front of the part where the chamfer needs to be placed, you can pan to the left by holding down* ***<Ctrl>*** *button and tapping the* ***<Left Arrow Key>*** *at the same time. Panning is a very useful feature for Swiss parts which are very long but have small outside diameters (OD).*

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

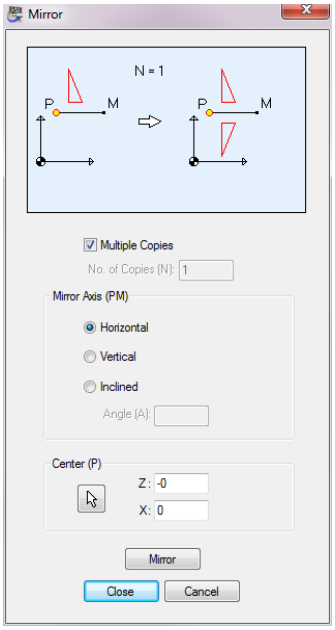


When completed your picture should look like the one below.

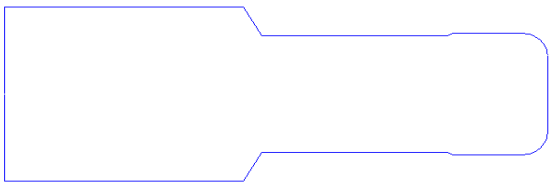
5 Select the geometry you have created by clicking on the selection icon and choosing **Select All** from the **Edit** menu.

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

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



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



Creating the Part Feature for Face Turning

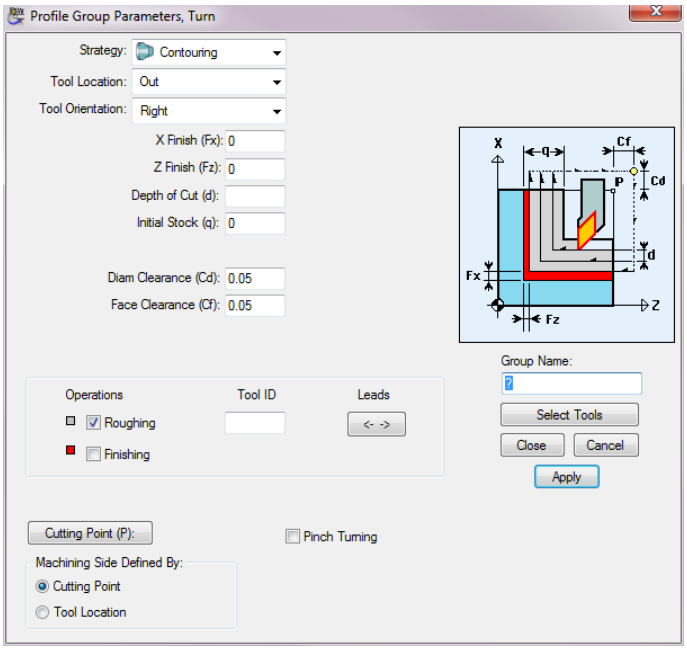
The first turning operation to be performed on the Main Spindle is to face the stock.

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

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

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

4 Choose **New Profile Group** from the **Part Features** menu to display the **Profile Group Parameters** dialog. When selected, you should see a dialog which appears as the one below:

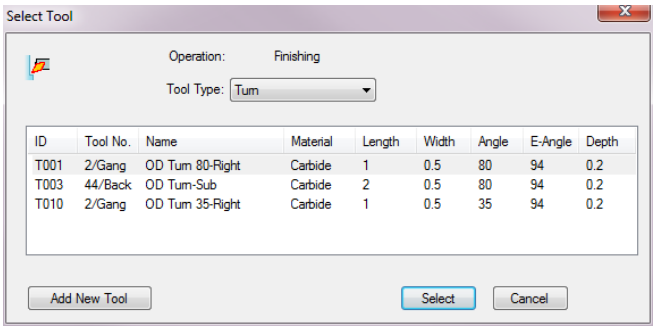


5 From the **Strategy** menu, select **Contouring** (it should be chosen by default).

6 From the **Tool Location** menu, select **Face**.

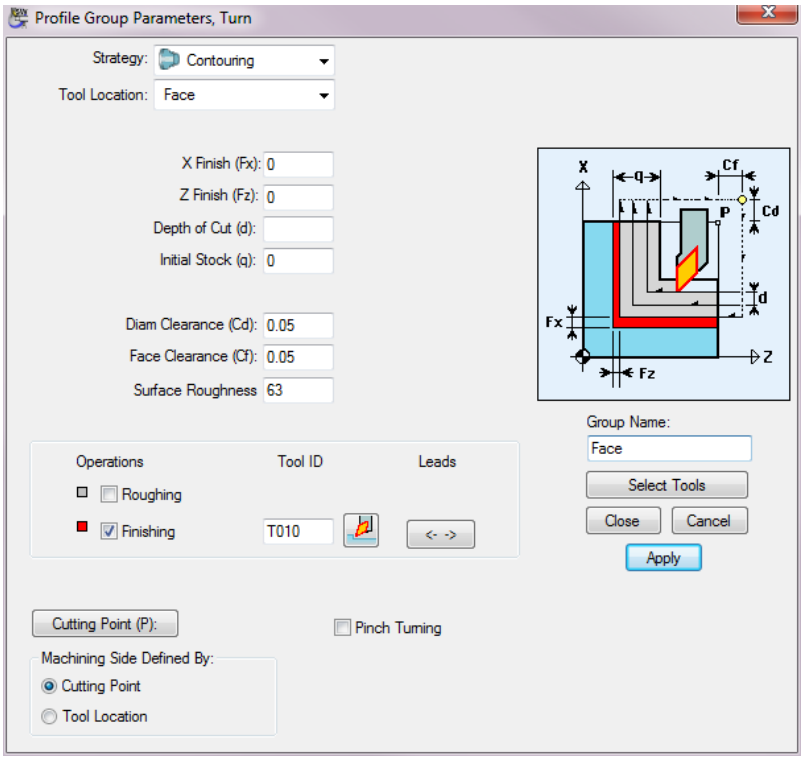
7 Uncheck the box next to Roughing, so that Finishing is checked.

8 Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool. PartMaker will present you a choice of three tools, two are mounted on the gang for work on the main spindle and the other is on the back working slide for work on the sub spindle as shown below:



In this case you are doing main spindle work, so choose **T010** and click **<Select>**. You are choosing **T010** because this tool has been setup up for work on the main spindle and the Tools Database has been designated as **Gang Tool**.

9 In the field called **Group Name:** enter **Face**. Your dialog should now appear as shown below:



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

Leads In/Leads Out

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

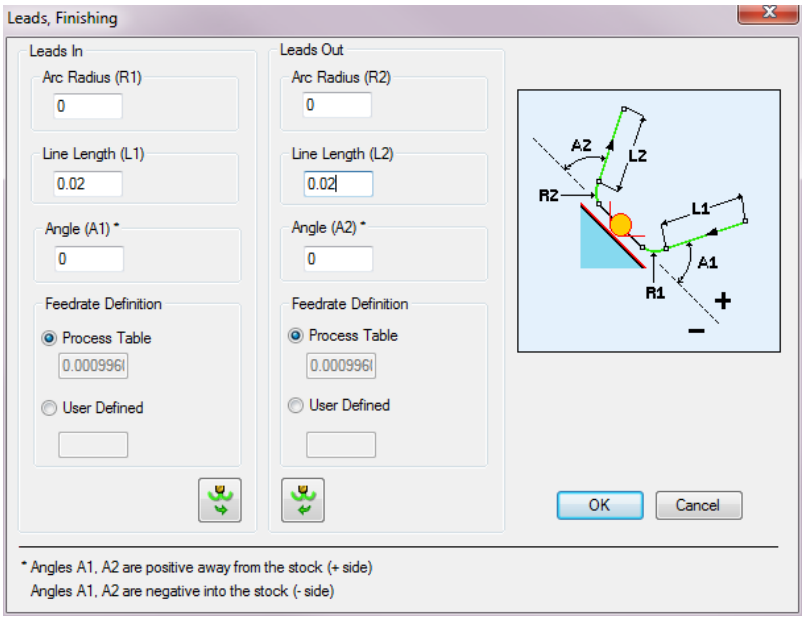
Note: *If you are using the same **Leads In** and **Leads Out** in most of your operations, you can set them as default values in the **Defaults for Turning** dialog accessed under the **Job Optimizer** menu. Setting your own default Leads In and Out values will save time when programming in **PartMaker**.*

<- ->

1 **Leads In** and **Out** are combined the settings for controlling the movement of the tool as it approaches (Leads In) and leaves (Leads Out) the stock into a single **Leads** dialog.

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

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



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

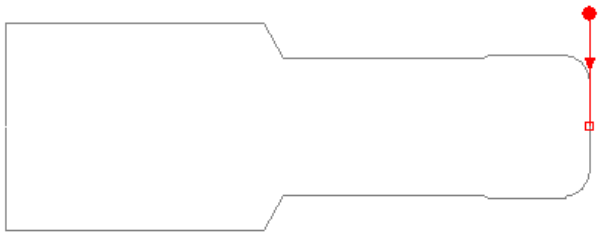
Note: *The **Lead** angle is always set normal to the path of the tool by **PartMaker**. Above, setting the **Leads Out** angle will allow the tool to retract in a manner perpendicular to the part. For example, if turning a 45 degree chamfer, a 0 degree lead angle would allow the tool to exit the stock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

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

Developing the Tool Path

Notice, that by choosing a Tool Location of **Face**, PartMaker automatically applied your facing tool path to the part geometry.

Your completed profile should appear as below:



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

Important! *Before moving on, save your work! To do so:*

1 Choose **Save** from the **File** menu or use the shortcut key **<Ctrl + S>**.

2 In the **Save Job File as**: dialog, give your job file a name. Here we are naming this job **Reamer**.

3 Click the **<Save>** button to save the job.

Note: *When choosing **<Save>**, **PartMaker** also saves your tools, materials, and your cycles file in addition to your job file.*

Tool Path Verification

Once a tool path is defined, you can verify its shape to gauge the accuracy of the cut you have defined. In **PartMaker** SwissCAM, for programming being done on the Main Spindle in a **Face Window** of **Machining Function Turn**, you have the option of performing two types of tool path verification:

- **Standard** verification (default choice)

  Or

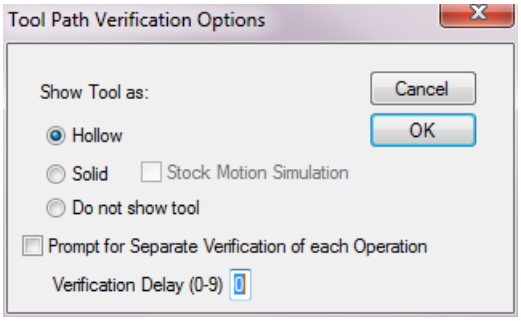
- **Sliding headstock** simulation (Stock Motion Simulation)

Each type of verification has its relative merits. Standard verification simulation is excellent when focusing on the cut being made by a particular tool path. The sliding headstock simulation, referred to as **Stock Motion Simulation**, is excellent for seeing how your part will move through the guide bushing during machining and to assure that the "Tool Shifts" have been set properly and that proper support is being provided by the Guide Bushing during simulation.

Note: *When performing, the Stock Motion Simulation, a guide bushing of the OD and Length described in the* ***Setup*** *dialog will be shown.*

For the facing operation, you will use the Standard verification. To do so:

1 Select **Verify Work Group Tool Path** from the **Part Features** menu to check your work. You will be prompted with the **Tool Verifications Options** dialog as shown below:



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

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

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

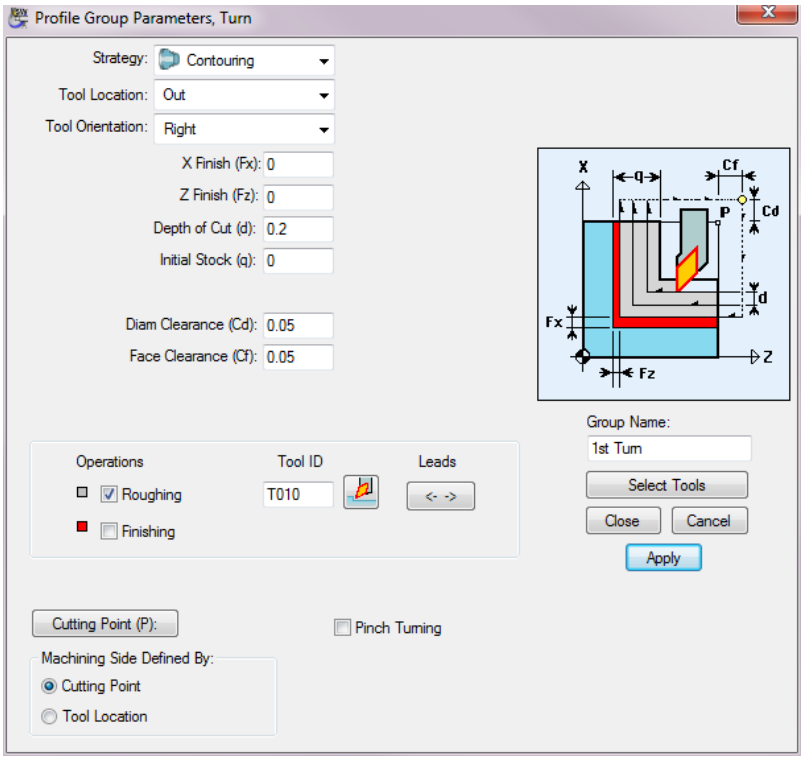
Note: *Whenever you verify a tool path (or every tool path) you MUST choose* ***Hide Work Group Tool Path*** *before moving on and continuing to work in* ***PartMaker****. Not doing so could hamper performance of your system. In order to stop verification before it is complete, hit the* ***<Esc>*** *key.*

Defining the Profile for OD Turning

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

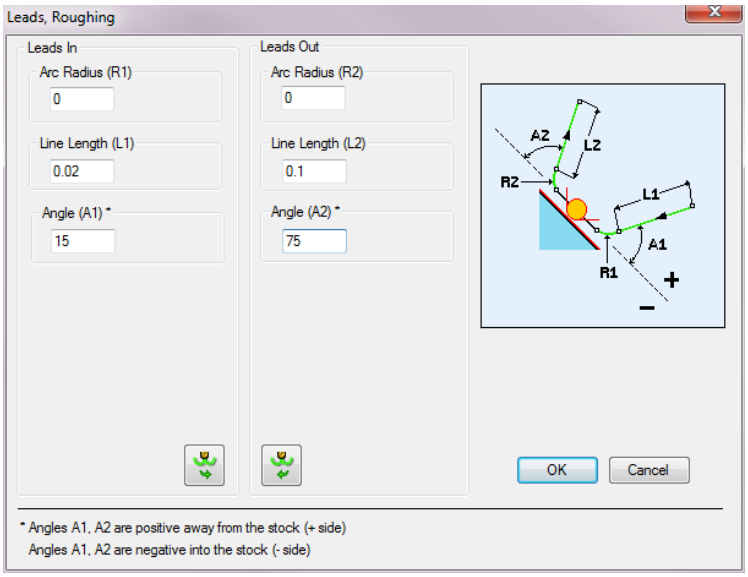
1. Click the blue square in the **Color Bar**.

2. Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
3. From the **Strategy** menu, select **Contouring** (it should be chosen by default).
4. From the **Tool Location** menu, select **Out** (it should be chosen by default).
5. From the **Tool Orientation** menu, select **Right** (it should be chosen by default).
6. Uncheck the Roughing tool and make sure the Finishing tool is check on.
7. Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool. Again, **PartMaker** will present you a choice of three tools, Two are mounted on the main spindle and the other is on the sub spindle. In this case you are going to choose the 35 degree tool doing the main spindle work, so choose **T010** and click **<Select>**.
8. In the field called **Group Name:** type **1st Turn**. Your dialog should now appear as the one below.



Leads In/ Leads Out

9 Now you will set the lead in and lead out for your profile. Click the **Leads** button and the **Leads** dialog will appear. Set the line Lengths as they appear below:



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

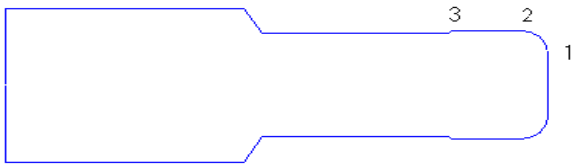
10 Click the <**Close>** button to return to the **Face Window**.

Developing the Tool Path

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

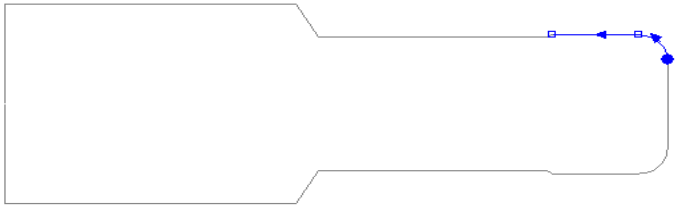
1 From the left-hand side of the screen select the **Define Profile Icon**.

2 From the top of the screen, select the **End of an Element Snap Mode** icon. To define the Front Turn path, click your left mouse on positions 1 through 3 as shown below:



Note: *You may find this exercise easier by zooming in on the area you are working on. (**Zoom In** from the Main tool bar, or use the mouse wheel to zoom in or out).*

When completed, you will have the tool path shown below:



3 Click the **Selection** icon to deselect.

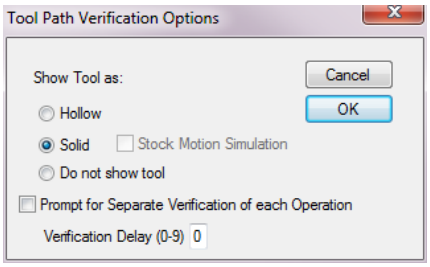
Tool Path Verification – Stock Motion Simulation (Sliding Headstock)

With both the Facing and Front Turning tool paths defined, you can now perform stock motion simulation when verifying the motion of the tool. Stock Motion Simulation allows you to see the headstock moving through the guide bushing while cutting.

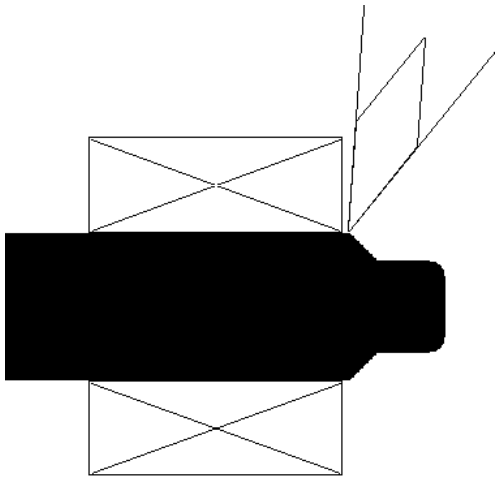
Note: *Before performing* ***Stock Motion Simulation*** *it is generally advisable to move the part to the left on the screen using the panning function, i.e. hold down the* ***<Ctrl>*** *key and* ***Left Arrow*** *key at the same time. It may also be advisable to decrease the size of the part on screen by either choosing rolling the mouse wheel away from you, or* ***Zoom Out*** *from the* ***View*** *menu or by depressing the* ***<F4>*** *key to continuously zoom out.*

To do so:

1. Select **Verify Every Tool Path** from the **Part Features** menu to check your work.
2. You will be prompted with the **Tool Path Verification Options** dialog. Check the **Solid** button and the box marked **Stock Motion Simulation** as shown below. Also enter the desired **Verification Delay**.



When you are satisfied your dialog appears as the one above, click **<OK>**. You should see the part cutting with stock sliding through the guide bushing as shown below.



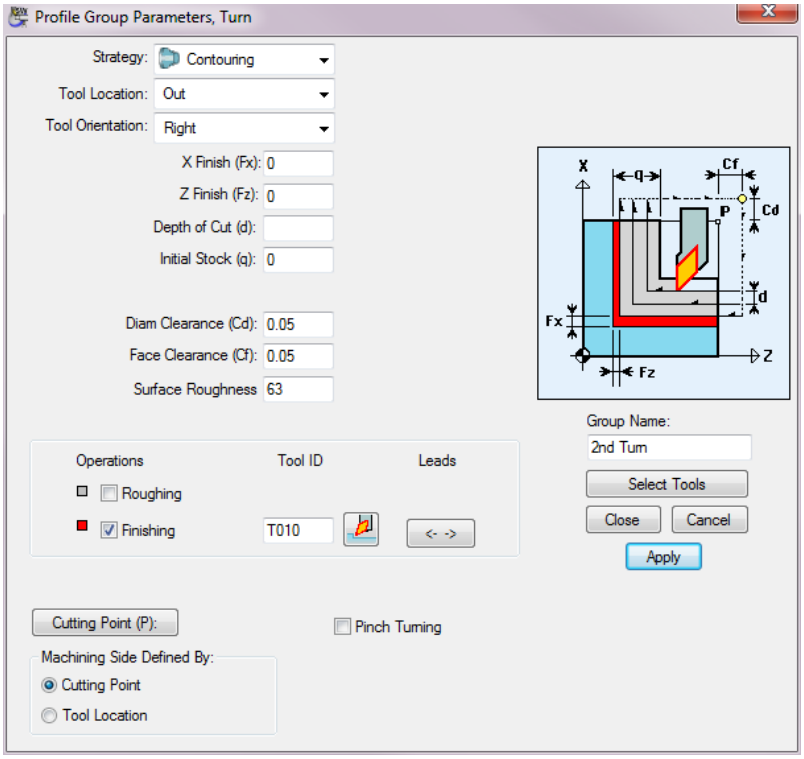
3. Once complete, choose **Hide Every Tool Path** from the **Part Features** menu, or press **<Ctrl +H>,** to hide the verification shown above.

Defining the Profile for the Second OD Turning Operation

The next operation to be performed will be OD turning, starting from a Z length of 0.200 (5.0) to the top of the 120-degree angle.

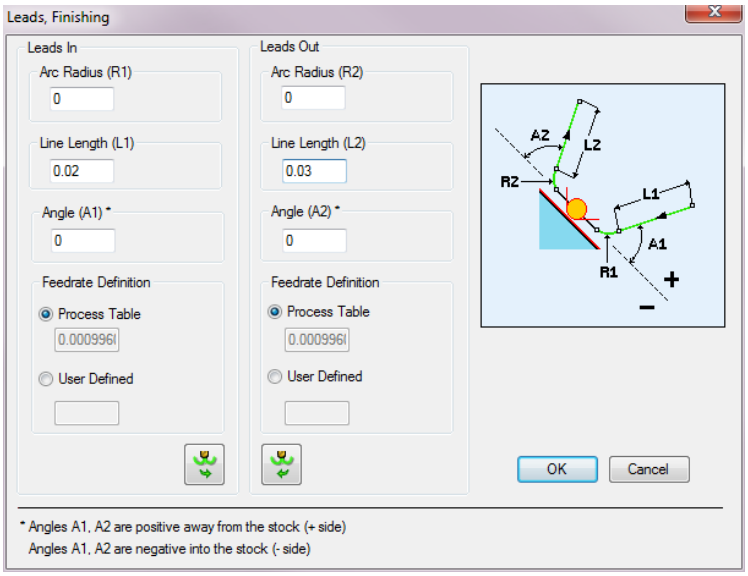
1. Click the green square in the **Color Bar**.

2. Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
3. From the **Strategy** menu, select **Contouring** (it should be chosen by default).
4. From the **Tool Location** menu, select **Out** (it should be chosen by default).
5. From the **Tool Orientation** menu, select **Right** (it should be chosen by default).
6. Uncheck the Roughing tool and make sure the Finishing tool is check on.
7. Click the **Select Tools** button and allow **PartMaker** to select the appropriate tool. Again, **PartMaker** will present you a choice of three tools, Two will be mounted on the main spindle and the other is on the sub spindle. In this case you are going to choose the 35 degree doing the main spindle work, so choose **T010** and click **<Select>**.
8. In the field called **Group Name**, type **2nd Turn**.
9. Click **<Apply>** and your dialog should now appear as the one below.



Leads In/ Leads Out

10 Now you will set the lead in and lead out for your profile. Click the **Leads** button and the **Leads** dialog will appear. Set the line Lengths as they appear below:



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

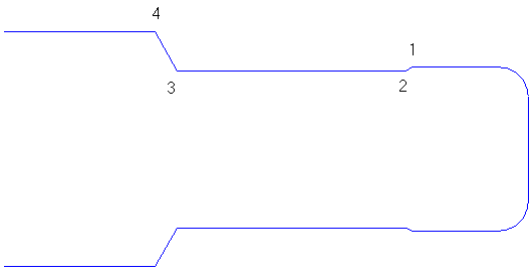
11 Click the <**Close**> button to return to the **Face Window**.

Developing the Tool Path

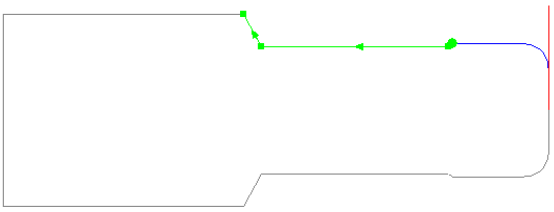
Now that you have defined the cycle to be used, you must tell **PartMaker** the location of the tool path. You can now assign the **2nd Turn** cycle to the geometry you created.

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

2 From the top of the screen, select the **End of an Element** snap mode. To define the rest of the **2nd OD Turn** path, click your left mouse on positions 1 through 4 in the diagram below:



When completed, you will have the tool path shown below:



3 Click the **Selection** icon to deselect.

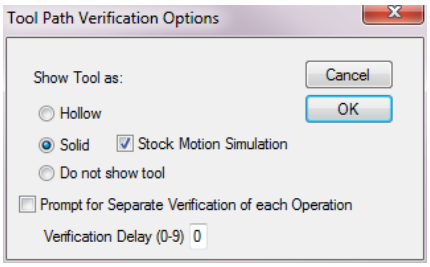
Tool Path Verification – Stock Motion Simulation (Sliding Headstock)

With the Facing, Front Turning and 2nd OD Turn tool paths defined, you can again perform stock motion simulation when verifying the motion of the tool. Stock Motion Simulation allows you to see the headstock moving through the guide bushing while cutting.

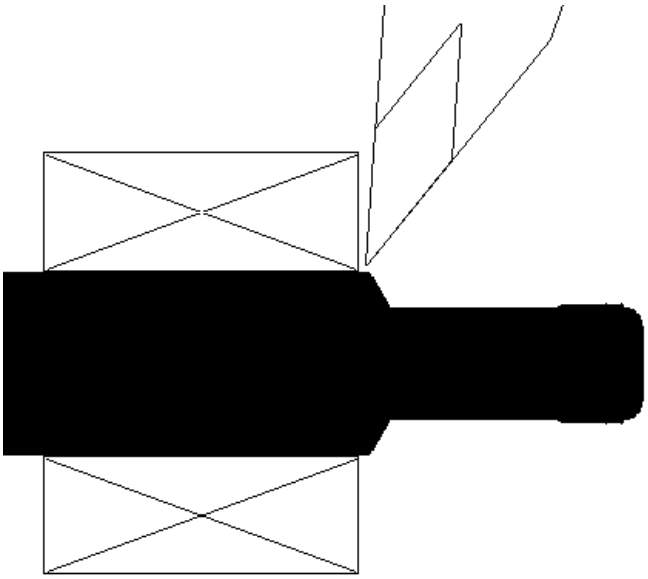
Note: *Before performing* ***Stock Motion Simulation*** *it is generally advisable to move the part to the left on the screen using the panning function, i.e. hold down the* ***<Ctrl>*** *key and* ***Left Arrow*** *key at the same time. It may also be advisable to decrease the size of the part on screen by either choosing* ***Zoom Out*** *from the* ***View*** *menu or by depressing the* ***<F4>*** *key to continuously zoom out.*

To do so:

1. Select **Verify Every Tool Path** from the **Part Features** menu to check your work.
2. You will be prompted with the **Tool Path Verification Options** dialog. Check the **Solid** button and the box marked **Stock Motion Simulation** as shown below. Also enter the desired **Verification Delay**.



When you are satisfied your dialog appears as the one above, click **<OK>**. You should see the part cutting with stock sliding through the guide bushing as shown on the following page.



Once complete, choose **Hide Every Tool Path** from the **Part Features** menu to hide the verification shown above.

Defining the Profile for the Third OD Turning Operation

In this operation you will perform OD turning operations starting from the end of the 120 angle to the overall length.

1 In CAM mode, select light blue from the **color bar**. 

2 Select New Profile Group from the Part Features menu.

3 From the **Strategy** menu, select **Contouring** (Default value).

4 From the **Tool Location** menu, select **Out** (Default value).

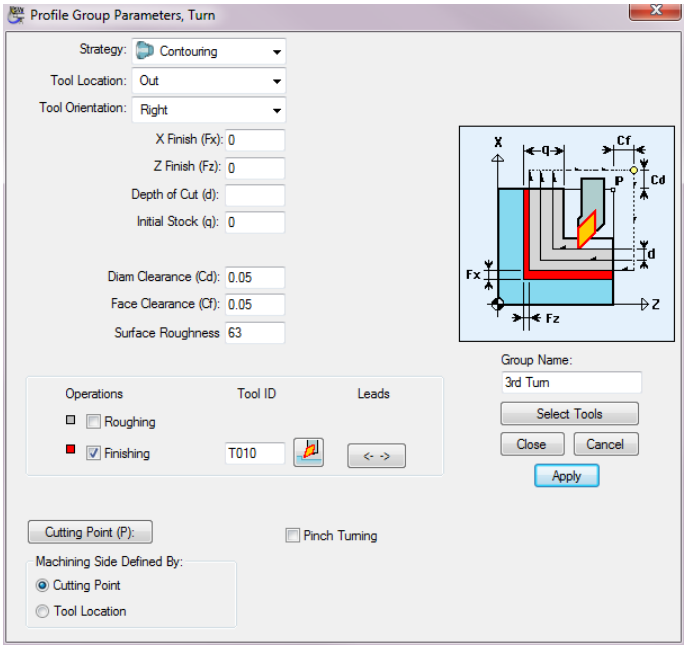
5 From the **Tool Orientation** menu, select **Right** (Default value).

6 Uncheck the Roughing tool and make sure the Finishing tool is check on.

7 Type 3rd Turn next to Cycle Name.

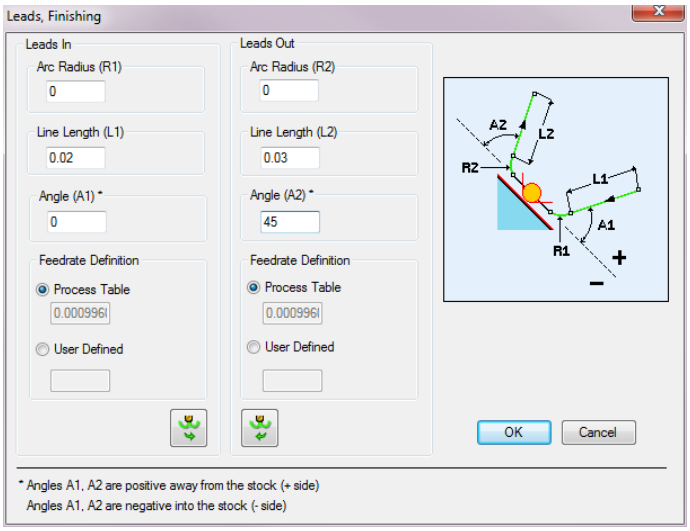
8 After you have made these changes, click the <**Select Tools>** button and allow **PartMaker** to select the appropriate tool. As before **PartMaker** gives you a choice of two tools. Again, choose **T010**.

9 Click **<Apply>.** The completed **Profile Group Parameters** dialog should appear as shown below:



10 Set the **Leads** for the **3rd Turn** by clicking the **Leads** button.

Complete the dialog as shown below:



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

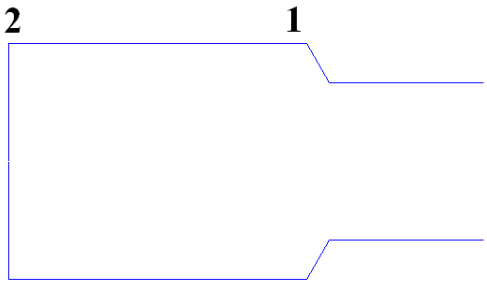
11 Close the **Profile Group Parameters** dialog for **3rd Turn** by clicking the <**Close>** button.

Developing the Tool Path

Now that you have defined the cycle to be used, you must tell **PartMaker** the location of the tool path. You can now assign the **3rd Turn** cycle to the geometry you created.

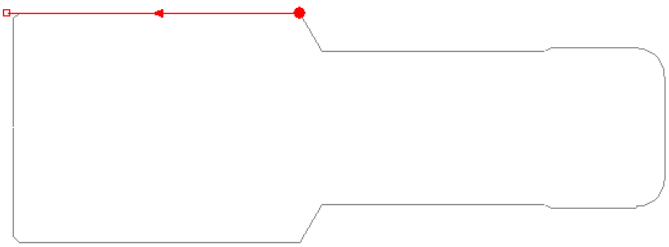
1 From the left-hand side of the screen select the **Profile** icon.

2 From the top of the screen, select the **End of an Element** snap mode. To define the rest of the **3rd Turn** path, click your left mouse on positions 1 and 2 in the diagram below:



Note: *You may find this exercise easier by zooming in on the area you are working on (**Zoom In** from the **View** menu) and panning across to each point (**<Ctrl> + Right Arrow** key).*

When completed, you will have the tool path shown below:



3 Click the **Selection** icon to deselect.

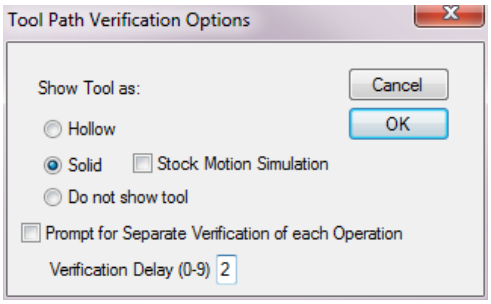
Tool Path Verification – Stock Motion Simulation (Sliding Headstock)

With the Facing, Front Turning, 2nd Turn and 3rd Turn tool path defined, you can again perform stock motion simulation when verifying the motion of the tool. Stock Motion Simulation allows you to see the headstock moving through the guide bushing while cutting.

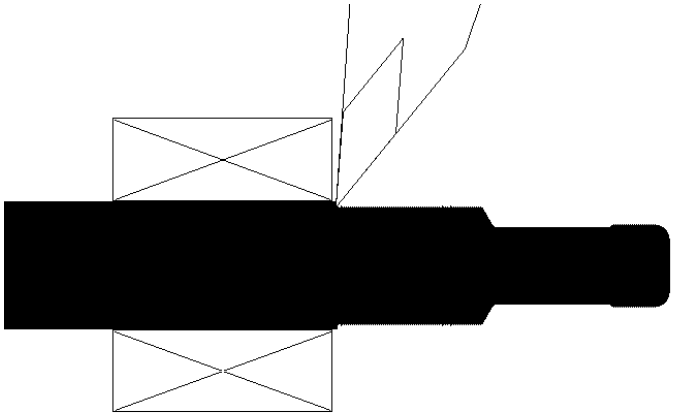
Note: *Before performing* ***Stock Motion Simulation*** *it is generally advisable to move the part to the left on the screen using the panning function, i.e. hold down the* ***<Ctrl>*** *key and* ***Left Arrow*** *key at the same time. It may also be advisable to decrease the size of the part on screen by either choosing* ***Zoom Out*** *from the* ***View*** *menu or by depressing the* ***<F4>*** *key to continuously zoom out.*

To do so:

1. Select **Verify Every Tool Path** from the **Part Features** menu to check your work.
2. You will be prompted with the **Tool Path Verification Options** dialog. Check the **Solid** button and the box marked **Stock Motion Simulation** as shown below. Also enter the desired **Verification Delay**.



When you are satisfied your dialog appears as the one above, click **<OK>**. You should see the part cutting with stock sliding through the guide bushing as shown below.

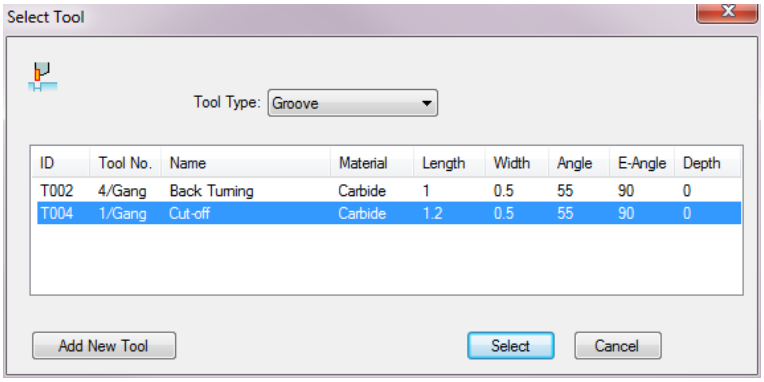


Once completed, choose **Hide Every Tool Path** from the **Part Features** menu to hide the verification shown above.

Defining the Profile for Cut-Off

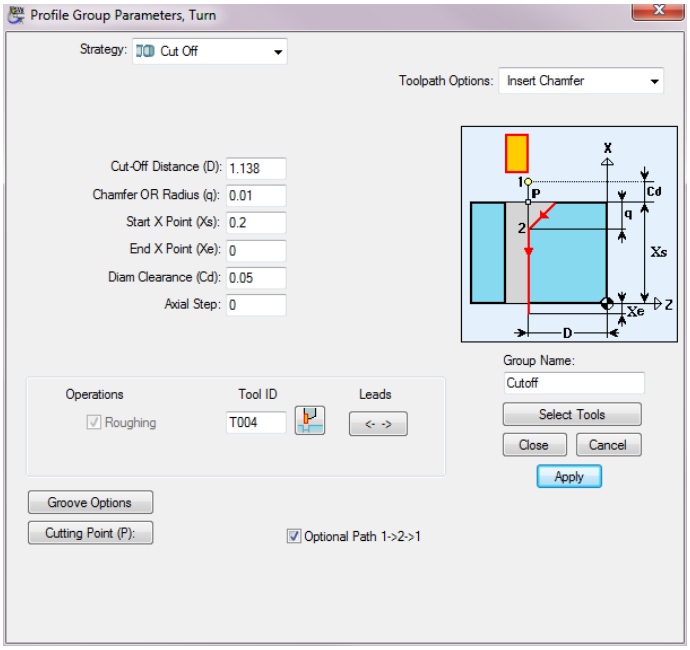
In this operation you will perform a cut-off operation on the main spindle with a grooving tool.

1. In CAM mode, select purple from the **color bar**.
2. Select New Profile Group from the Part Features menu.
3. From the **Strategy:** menu, select **Cut-Off**.
4. Click **Select Tools**. Choose the tool titled **Cut-Off** when the dialog below appears and click **<Select>**.



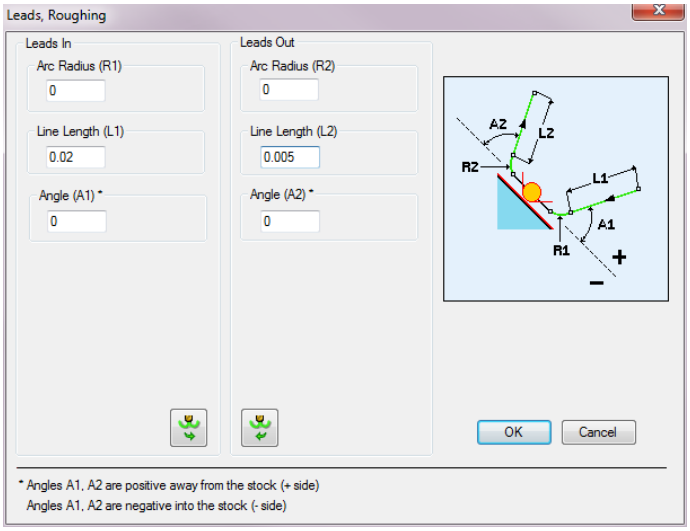
5. Enter a **Cut-Off Z-Distance (D)** of **1.138 (29.0)**. (It should be entered by default.)
6. Enter **0.01** (.3) for Chamfer or Radius (q).
7. Enter **0.2** (5.0) for **Start X Point**.
8. Type **Cutoff** in the **Group Name** field.
9. Check the <Optional Path 1->2->1> checkbox.

Your completed **Profile Group Parameters** dialog should appear as the one below:



10 Set the **Leads** for the **Cutoff** by clicking the **Leads** button.

Complete the dialog as shown below:



11 Close the **Profile Group Parameters** dialog for the cutoff operation by clicking the <**Close**> button.

Notice when using the cut-off cycle, **PartMaker** automatically creates the tool path for you.

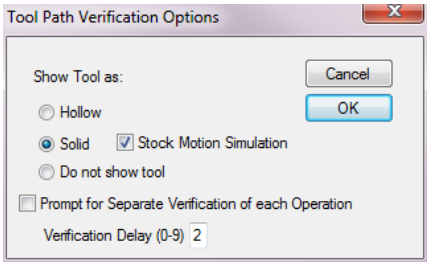
Your completed profile should appear as below:

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

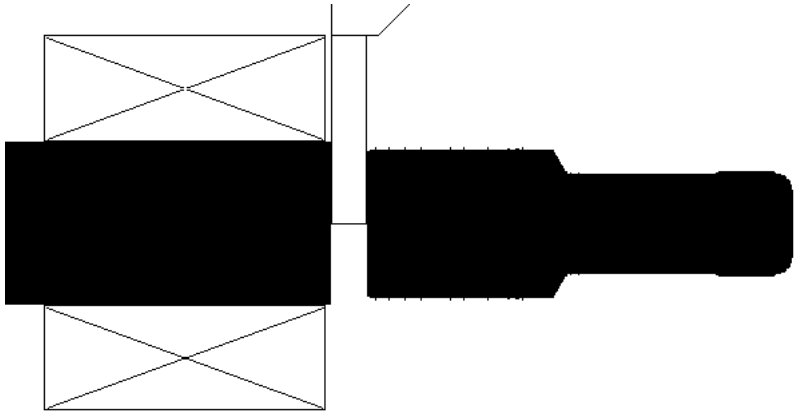
Tool Path Verification – Stock Motion Simulation (Sliding Headstock)

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

1 Choose Verify Every Tool Path from the Part Features menu. In the Verification Options dialog, choose Solid and mark the Stock Motion Simulation box as shown below:



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



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

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

Performing Milling Operations - Main Spindle using the Mill ZY Plane

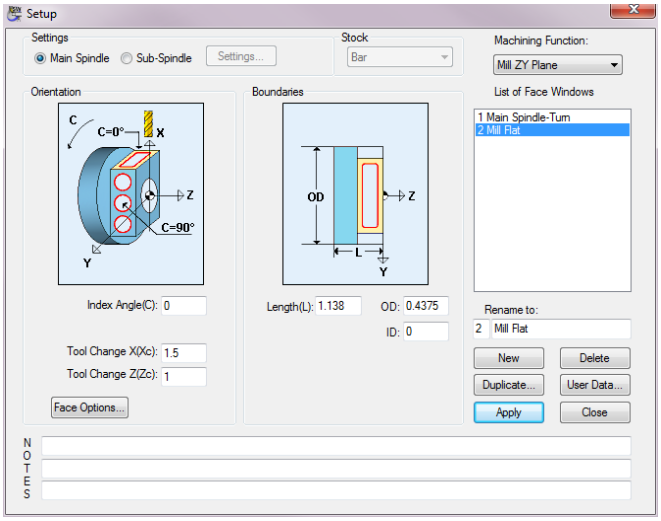
This section of the tutorial will instruct you in the steps necessary to perform milling on the main spindle using the Mill ZY Plane Face Window.

Setting Up the Face Window

The first step you will need to take is to set up the **Face Window** needed to perform the milling operations mentioned above. To do so:

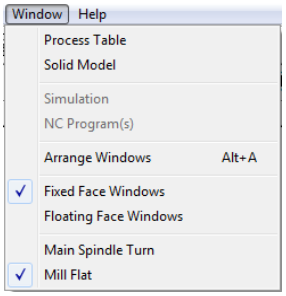
1. Choose **Setup** from the **View** menu to view the **Setup** dialog box or use the **Shortcut to the Setup** icon located above the profile icons on the left-hand side of the **Face Window**.
2. Click the **<New>** button to create a new **Face Window**.
3. From the **Machining Function** drop down menu select **Mill ZY Plane**.
4. Type **Mill Flat** in the **Rename To:** field and press **<Enter>**.

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



Note: *In the dialog above, the* ***Main Spindle*** *box is checked. This designates that this operation will be programmed on the main spindle of the machine.*

5. When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current **Face Window**.
6. Click the **<Close>** button to close the **Setup** dialog.
7. To switch to the **Mill Flat Face Window**, choose **Mill Flat** from the **Window** menu.



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

9 Choose **Show Boundaries** from the **View** menu. Your screen should appear as shown below:

Note: If the image in the **Face Window** appears distorted or oblong, choose **Full View** from the **View** menu to refresh the screen for it to appear as shown above.

Creating Geometry in CAD Mode

In this section, you will create the geometry needed to assist you in defining the milling tool paths you will create in this section by copying the geometry from the **Main Spindle Turn** window to the **Mill Flat** window. To do so:

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

2 Select **Main Spindle Turn** from the **Window** menu.

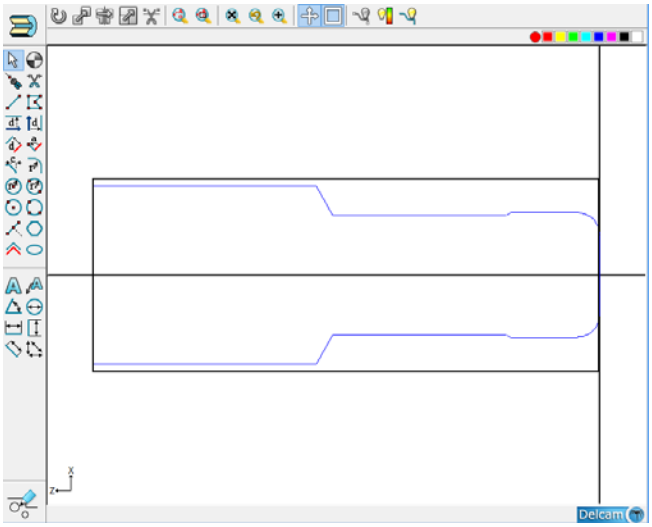
3 Choose **Select All** from the **Edit** menu

4 Choose **Copy** from the **Edit** Menu

5 Select **Mill Flat** from the **Window** Menu

6 Choose **Paste** from the **Edit** Menu. Click anywhere in the Face Window to deselect your geometry.

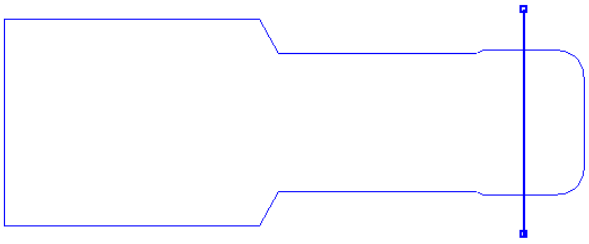
Your screen should appear as shown below:



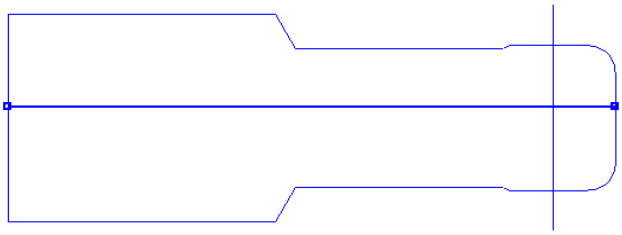
Next you will create the geometry for the milled notch on the front of the part.

1 Click the Line Parallel to Vertical axis Icon

2 Enter .118 (2.5) in the field in the lower left hand of the screen. And press the **<Enter>** button. Your view should look like the one below.

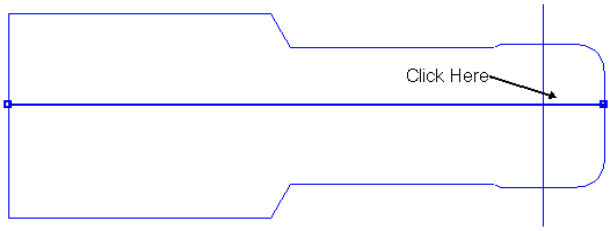


3 Click the **Line Parallel to the Horizontal Axis.** Enter a value of -.02 (-.5)

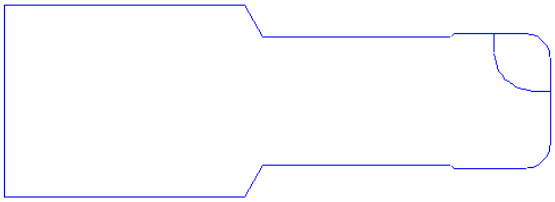


4 Click the **Fillet** icon and enter a value of 0.08 (2.0)

5 Click the intersection of the .118 (.5)Vertical Line and the -0.02 (-.5) you have just created.



6 Click on the **Remove** icon and click on the line segment sticking out above the outside diameter of the drawing so it looks like the one shown below.



Next you will create geometry for the milled slots. To do so

1 Click on a Circle with a Known Radius and Center.

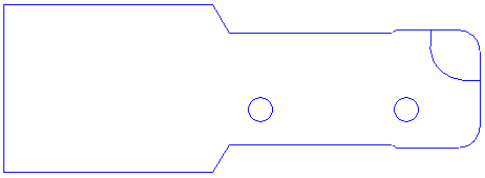
2 Enter a value of the radius of the Circle of 0.0625/2 (.8).

3 Click on **ZY** Snap mode.

4 Enter a value of .174 (4.4) for Z and 0.05 (1.3) for Y and press **<Enter>**.

5 Press **<Enter>** again to confirm the same radius value.

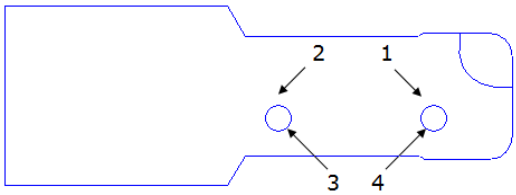
6 Enter a value of .524 (13.3) for Z and 0.05 (1.3)for Y and press **<Enter>**. Your drawing should look like the one shown below.



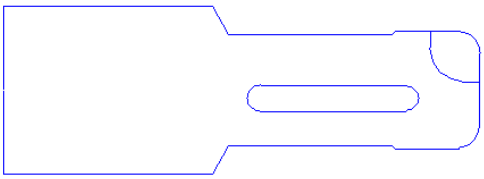
7 Click on the Line through Two Points Icon.

8 Select **Tangency** Snap mode

9 Click on the points shown in the view below.



10 Click on the **Remove** Icon and trim the interior radii .



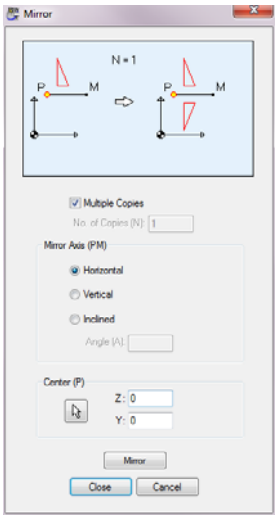
You can now duplicate the Slot using the **Mirror** Function. To do so:

1 Choose the **Selection** icon

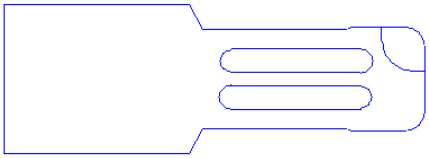
2 Drag a window around just the slot so that it is selected.

3 Choose **Mirror** from the **Edit > Transform** menu

4 Enter in the data as shown below.



Your picture should look like the one show below.



5 Click anywhere in the face window to deselect.

Creating a Mill Flat Profile

In this section, you will define the profile to machine a flat on the work piece.

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

2 In CAM mode, select **Red** from the **color bar**.

3 Select New Profile Group from the Part Features menu.

4 From the **Strategy** drop down menu select **Contour Mill**.

5 From the **Tool Position:** drop down menu choose **Left**.

6 Enter 0.140 (5.0) for **X_Surf(S)**, representing the .280 (10.0) you turned in the previous section divided by 2. **X_Surf**: The distance from the center of the cylinder to the edge of the turned diameter.

7 Enter 0.080 (3.5) for **X_Depth(D)**, representing distance traveled from the **X_Surf (S)**

8 **X_Rapid (R)** will be set to 0.150 (1.0) the tool will rapid to this value above the **X Surf(S)** value before it will perform its work

9 **X_Clear (Cl)** will be set to .050 (1.0) the tool will rapid to this value above the **X_Surf** after it has completed its work.

10 **Bottom Finish (b)** will be set to 0.

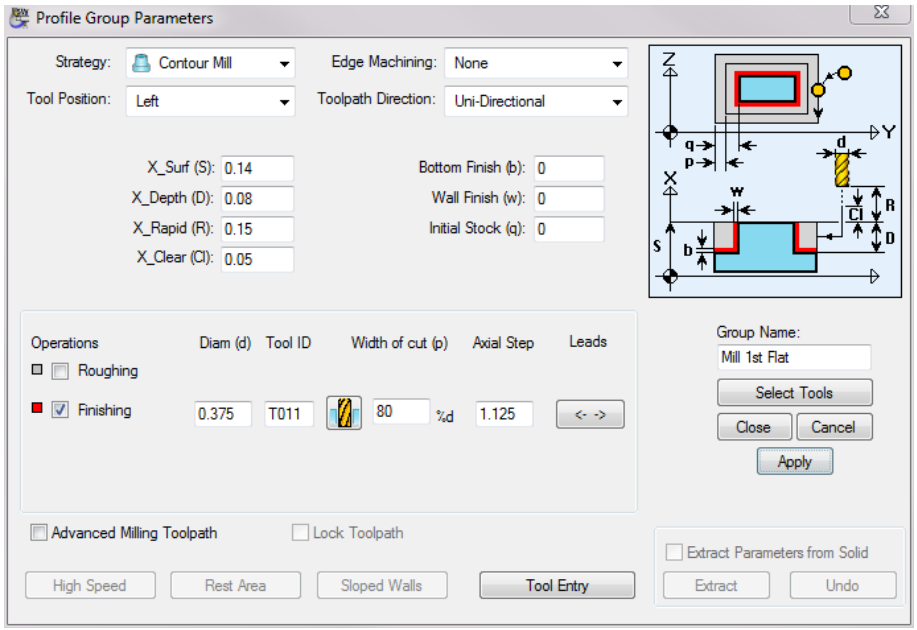
11 **Wall Finish (w)** will be set to 0

12 Uncheck the box next to **Roughing**

13 Make sure that **Finishing** is checked on and enter a value of 0.375 (10) for **Diam (d)**.

14 Click on the **<Select Tools>** button and select the .375 (10) end mill.

15 The dialog box should look like the one below.



16 Change Group Name field to **Mill 1st Flat**

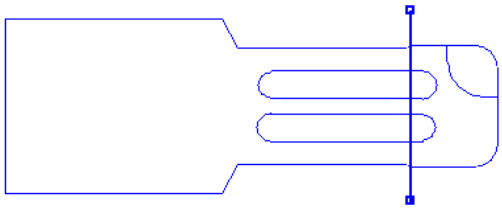
17 Click the <**Close**> button to close the **Profile Group Parameters** dialog box.

The next step is to develop the tool path, to do this you are going to have to add some geometry. To do so:

1. Switch to CAD mode by clicking on the CAD/CAM icon.

2. Draw a vertical line by clicking on the **Line Parallel to the Vertical Axis** icon.
3. Enter a value of 0.200 (5.0). This will represent the path of the endmill.

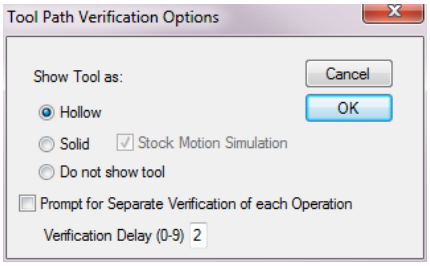
   The screen should appear as shown below:



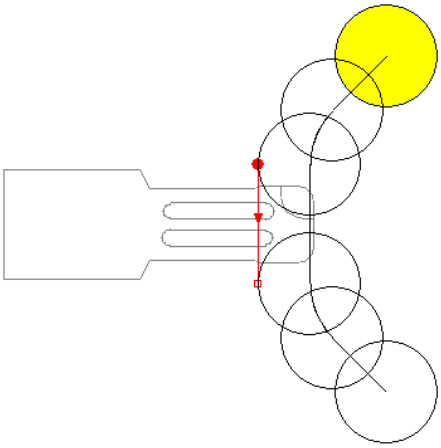
4. Switch back to **CAM** mode.
5. Click on **Chain Geometry** Icon
6. Click your cursor at the top of the vertical line that was drawn in the previous steps.

Tool Path Verification

In the **Setup** dialog for **Machining Function: Turn** that was used for the Facing, Front Turning, 2nd and 3rd toolpath, you were able to perform stock motion simulation when verifying the motion of the tool. In **Machining Function: Mill ZY** Stock Motion Simulation is not an option, only the milling path will be shown.



1. Choose Verify Work Group Tool Path from Part Features.
2. Your path should look like the one shown below



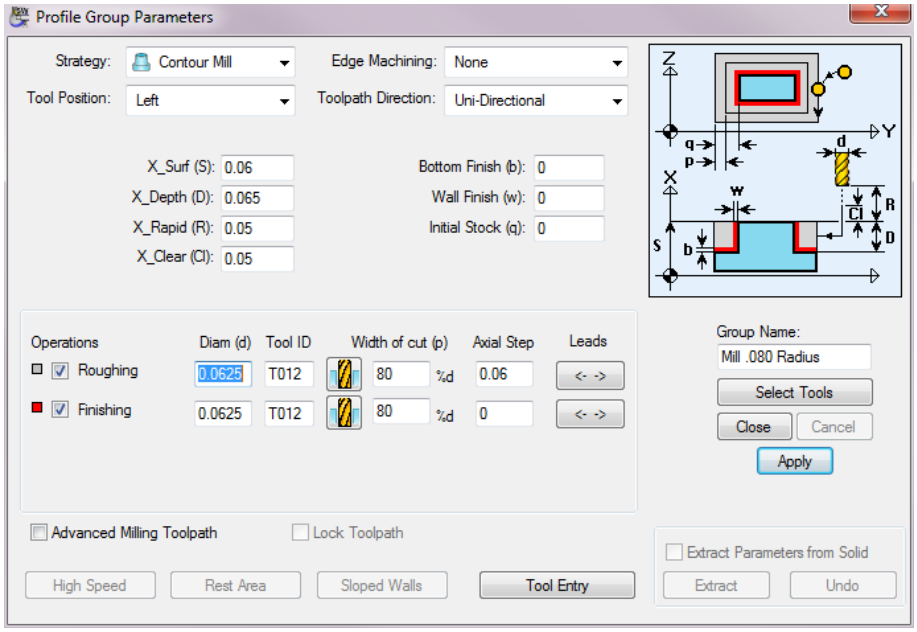
3. Choose **Hide Work Group Tool Path** from the Part Features menu

Creating the 0.08 (2.0) Radius Profile

In this section you will create the tool path for milling the notch on the front of the part.

1. In CAM mode, select Yellow from the **color bar**.
2. Select New Profile Group from the Part Features menu.
3. From the **Strategy** box Select **Contour Mill**.
4. From the **Tool Position:** drop down menu, choose **Left**.
5. Enter 0.06 (1.5) for **X_Surf(S**), representing the distance from the center of the cylinder to the edge of the flat milled in the previous step.
6. Enter 0.065 (1.2) for **X_Depth(D)**, representing distance traveled from the **X_Surf (S)**
7. **Bottom Finish (b)** will be set to 0.
8. **Wall Finish (w)** will be set to 0.
9. Check the <Roughing> and <Finishing> box.
10. Enter a value of 0.0625 (1.5) in the Diam (d) fields for the roughing and finishing operations.
11. Type **Mill .080 Radius** in the box under **Group Name**:
12. Click the **<Select Tools>** button.
13. Change the **Axial Step** value for roughing to .060 (1.1)

    Your completed dialog should appear as shown below.

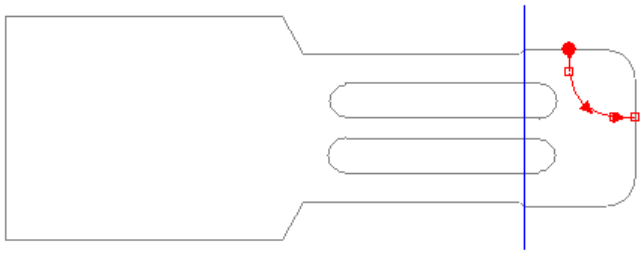


14. Click the **<Close>** button.

Define the Tool Path

1 Click the **Chain Geometry** Icon.

2 Select the .08 (2.0) radius as shown in the picture below by clicking on the upper left hand corner of radius.

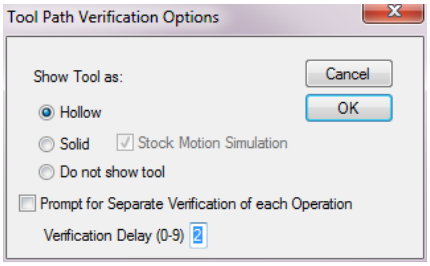


3 Click the **Selection** icon to deselect the tool path.

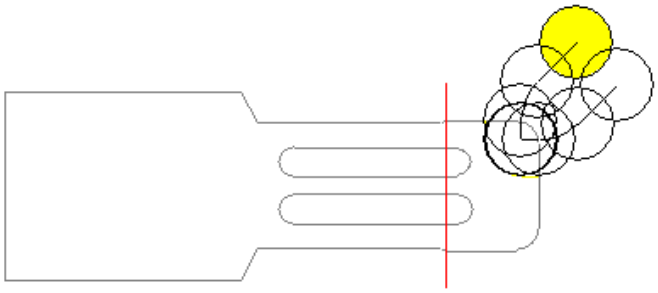
Tool Path Verification

In the **Setup** dialog for **Machining Function: Turn** that was used for the Facing, Front Turning, 2nd and 3rd toolpath, you were able to perform stock motion simulation when verifying the motion of the tool. In **Machining Function: Mill ZY** Stock Motion Simulation is not an option, only the milling path will be shown.

1 Choose Verify Work Group Tool Path from Part Features



2 Your path should look like the one shown below:



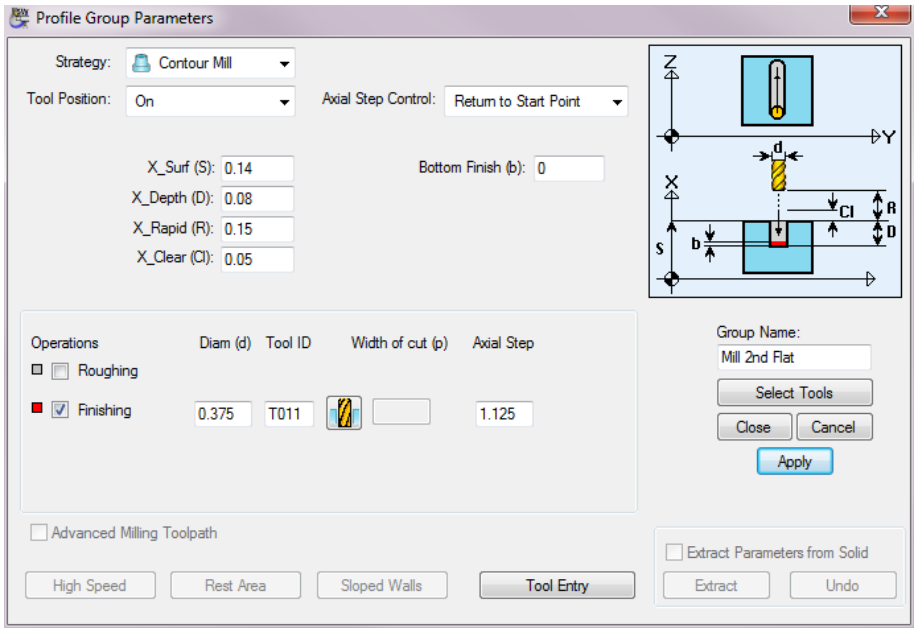
3 Choose **Hide Work Group Tool Path** from the Part Features menu.

Creating a Second Mill Flat Profile

In this section, you will define the profile to machine the second set of flats on the work piece.

1. In CAM mode, select Green from the **Color Bar**. 
2. Select New Profile Group from the Part Features menu.
3. In the **Strategy** box select **Contour Mill**.
4. Choose **On** from the **Tool Position:** drop down menu.
5. Enter 0.140 (5.0) for **X_Surf(S)**, representing the .280 (10.0) you turned in the previous section divided by 2. **X_Surf**: The distance from the center of the cylinder to the edge of the turned diameter.
6. Enter 0.080 (3.5) for **X_Depth(D)**, representing distance traveled from the **X_Surf (S)**
7. **X_Rapid (R)** will be set to 0.150 (1.0) the tool will rapid to this value above the **X Surf(S)** value before it will perform its work
8. **X_Clear (Cl)** will be set to .050 (1.0) the tool will rapid to this value above the **X_Surf** after it has completed its work.
9. **Bottom Finish (b)** will be set to 0.
10. Uncheck the box next to **Roughing**
11. Make sure that **Finishing** is checked on and enter a value of 0.375 (10) for **Diam (d)**.
12. Click on the **<Select Tools>** button and select the .375 (10) end mill.
13. Type **Mill 2nd Flat** in the **Group Name**: field.
14. Click on the **<Select Tools>** button.

    Your dialog should look like the one shown below:



15. Click on the **<Close>** button when completed.

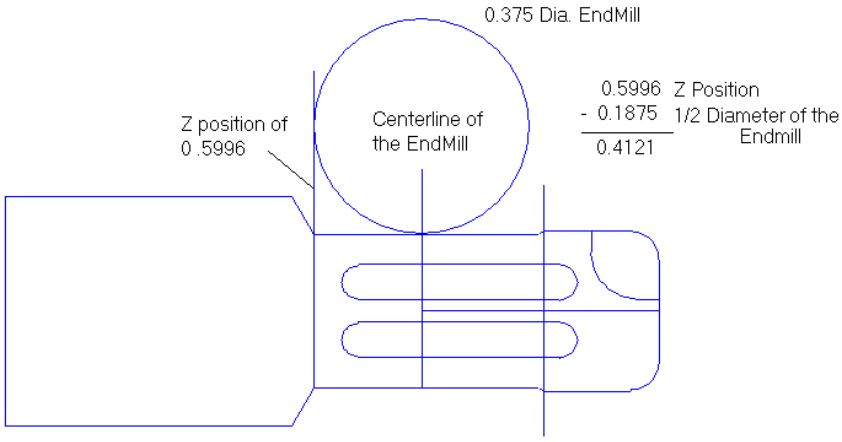
The next step is to develop the tool path. To do so you are going to have to add some geometry.


***Note**: To make the construction of the milling path clearer, choose **Hide Axis** and **Hide Boundaries** from the **View** menu*

To add geometry:

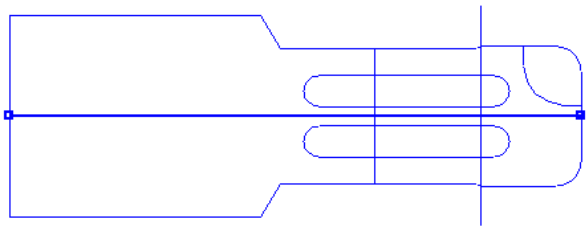
1. Switch to CAD mode by clicking on the lower left icon.

2. Draw a vertical line by clicking on the **Line Parallel to the Vertical Axis** icon.
3. Subtract 1/2 the diameter of the Endmill .1875 (5.0) from the Z position .5996 (15.58). Enter a value of 0.4121 (10.58). This will draw a line representing the center line of the endmill path.

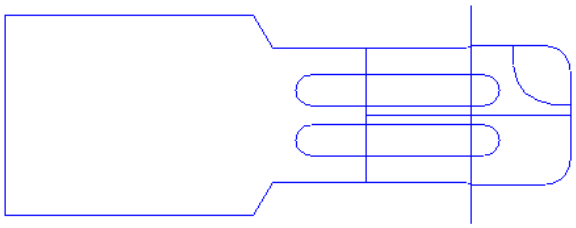


4. Select Line Parallel to the Horizontal Axis icon.
5. Enter a value of 0.

   The screen should appear as shown below:

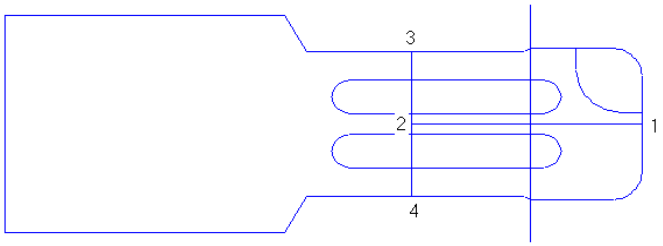


6. Select the **Remove** icon and trim the drawing so it look like the picture below.

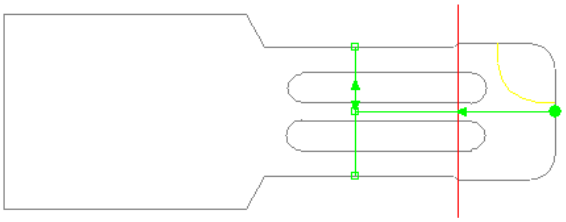


7. Switch back to CAM mode
8. Click Define Profile Icon.
9. Select Closest Intersection Snap Mode Icon.

10 Click on the points outlined in the picture below.



11 Your Path should look like the one shown below.

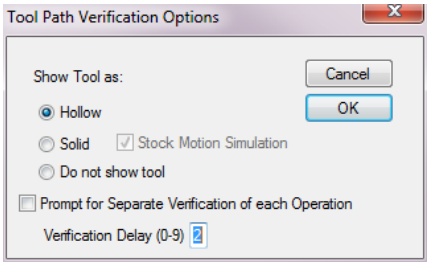


12 Click the seletion icon to deselect.

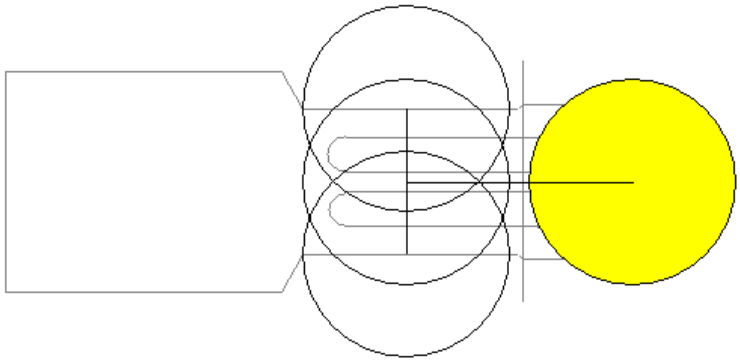
Tool Path Verification

In the **Setup** dialog for **Machining Function: Turn** that was used for the Facing, Front Turning, 2nd and 3rd toolpath, you were able to perform stock motion simulation when verifying the motion of the tool. In **Machining Function: Mill ZY** Stock Motion Simulation is not an option, only the milling path will be shown.

1 Choose Verify Work Group Tool Path from Part Features.



2 Your Path should look like the one shown below.



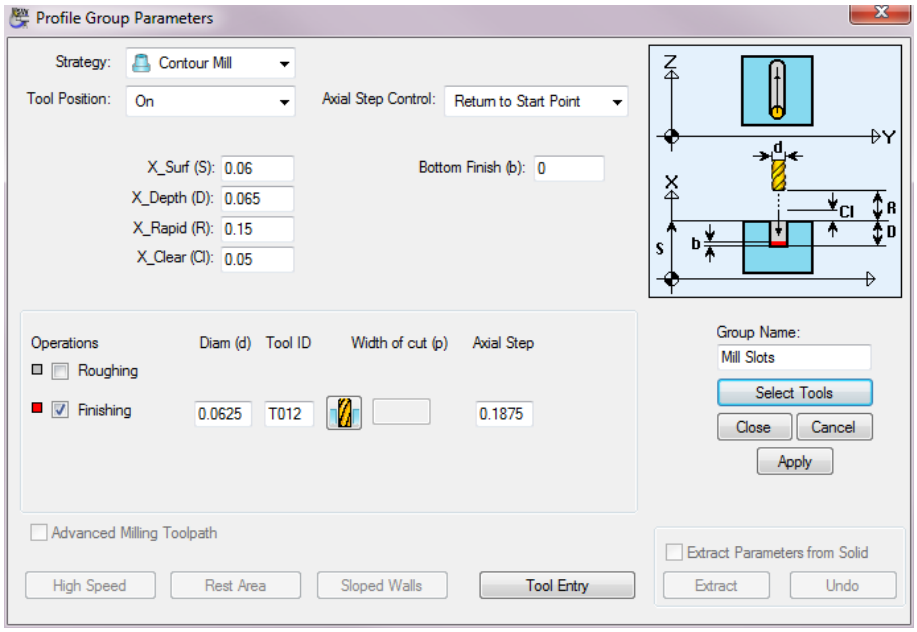
3 Choose **Hide Work Group Tool Path** from the Part Features menu.

Creating the Mill Slots Profile

In this section, you will define the profile to machine the slots on the work piece.

1. In the CAM mode, select blue from the **Color Bar.** 
2. Select New Profile Group from the Part Features menu.
3. In the **Strategy** box select **Contour Mill**.
4. Choose **On** from the **Tool Position:** drop down menu.
5. Enter 0.060(1.5) for **X_Surf(S)**
6. Enter 0.065 (1.6) for **X_Depth(D)**, representing distance traveled from the **X_Surf (S)**
7. **X_Rapid (R)** will be set to 0.150 (5.0) the tool will rapid to this value above the **X Surf (S**) value when making rapid moves.
8. **X_Clear (Cl)** will be set to .050 (1.0) the tool will rapid to this value above the **X_Surf** then start to feed into the part.
9. **Bottom Finish (b)** will be set to 0.
10. Uncheck the box next to **Roughing** so that **Finishing** is checked.
11. Enter a value of 0.0625 (1.5) for **Diam (d)** for finishing.
12. Type **Mill Slots** in the **Group Name**: field.
13. Click on the **<Select Tools>** button. Choose tool **T012**.

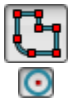
    Your dialog should look like the one shown below



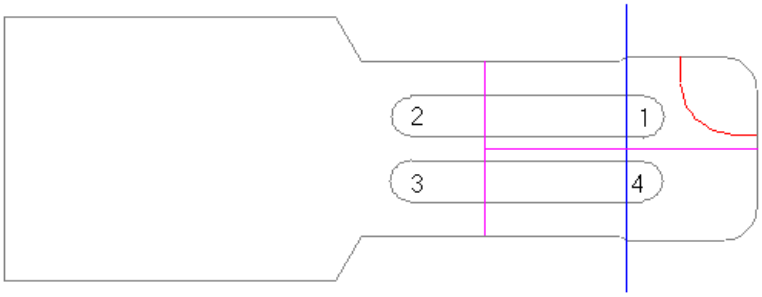
14. Click on **<Close>**.

Developing the Tool Path

Next you will apply this part feature to the geometry to define the tool path.

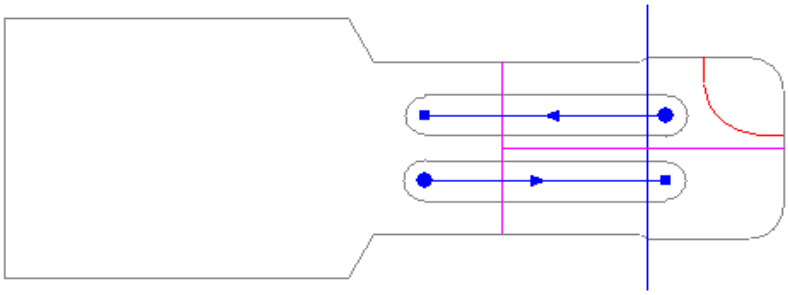


1. Click on Define Profile Icon.
2. Select Circle Center Snap Mode icon.
3. Start by clicking on points 1and 2 outlined in the picture below.



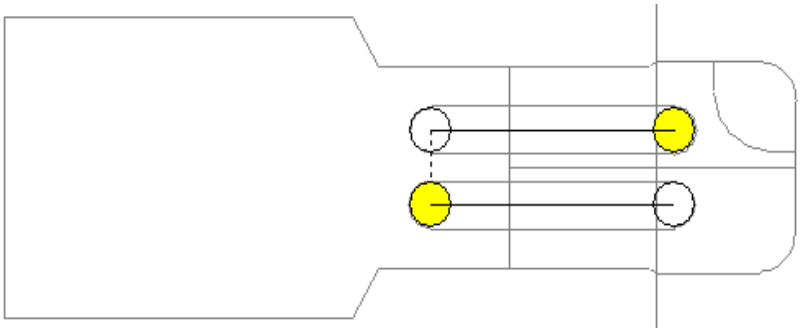
4. Click on the **Define Profile** icon again for the second time to break away from the original path.
5. Click on points 3 and 4 as indicated above.

6. Click on your selection icon when finished.
7. Your Path should look like the one shown below



Tool Path Verification

1. Choose Verify Work Group Tool Path from Part Features.
2. Your tool path should look like the one shown below.

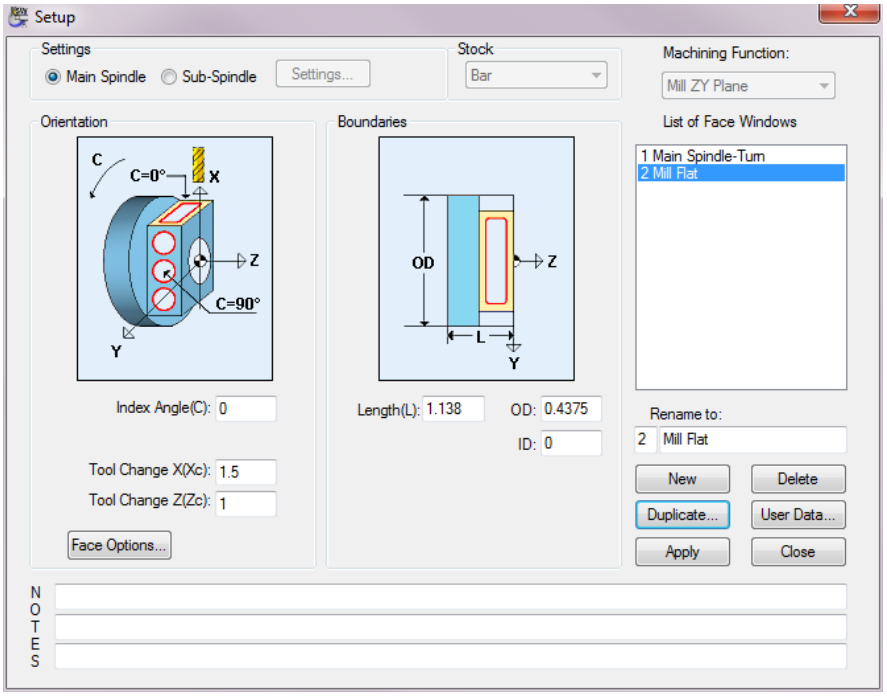


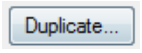
3. Choose **Hide Work Group Tool Path** from the Part Features menu.

Duplicating Identical Face Windows

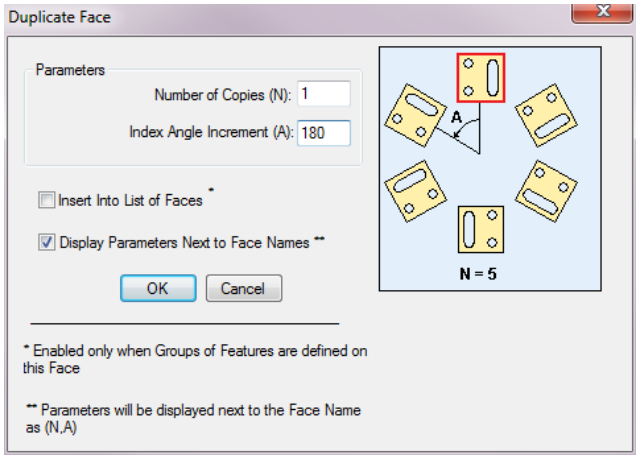
Having programmed the Milling operations on one side of the outside diameter of the part, you now need to create the same feature on the other side of the part, i.e. 180 degrees apart. Instead of having to go through the process of recreating the feature in a new window, you can automatically duplicate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

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



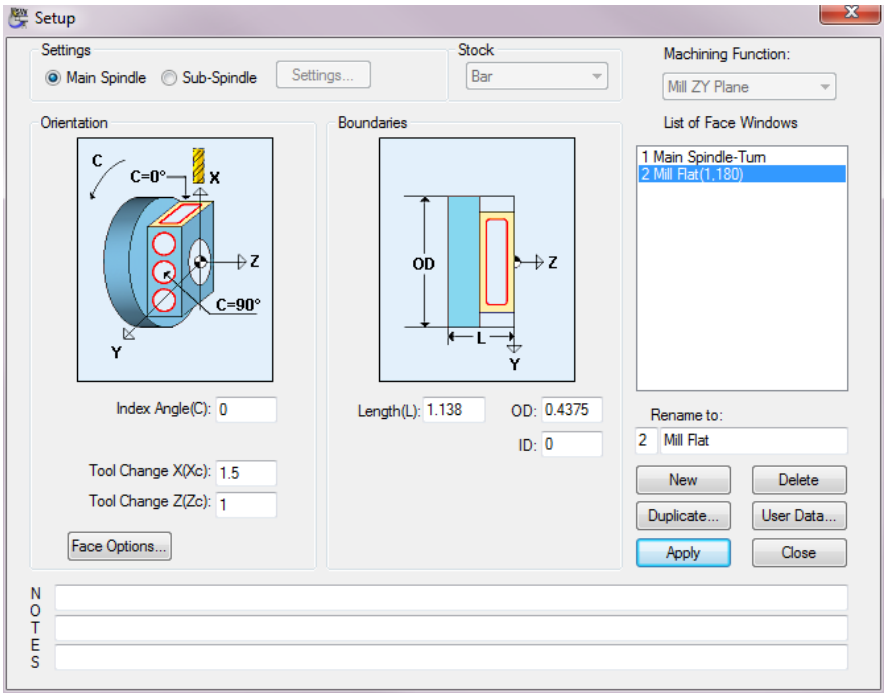


3. Click the **<Duplicate...>** button.
4. In the **Duplicate Face** dialog, enter 1 in the **Number of Copies (N)** field and 180 in the **Index Angle Increment (A)** field. The completed dialog should appear as shown below:



5. Leave the box labeled "**Insert Into List of Faces**" unchecked.
6. Click the **<OK>** button in the **Duplicate Face** dialog.

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



Notice the name of the original **Mill Flat** Face Window has had a **(1,180)** appended to it. All of the processes included in this window will be duplicated at the Process Table. You will see a yellow block on the processes that have been duplicated. The advantage of duplicating the window with the "**Insert Into List of Faces**" unchecked is that any changes that need to be made to the milling is automatically done to all of the duplicated windows.

If you check the box labeled "**Insert Into List of Faces**" the Mill Flat Face Window will have a **_0** appended to it and the duplicated Face Window having been named **Mill Flat_180**. At the Process Table there will be a separate process for each duplicated face window. Notice the **Angle(C)** field in the newly created window will be populated with a value of 180.

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

Turning on the Sub-Spindle

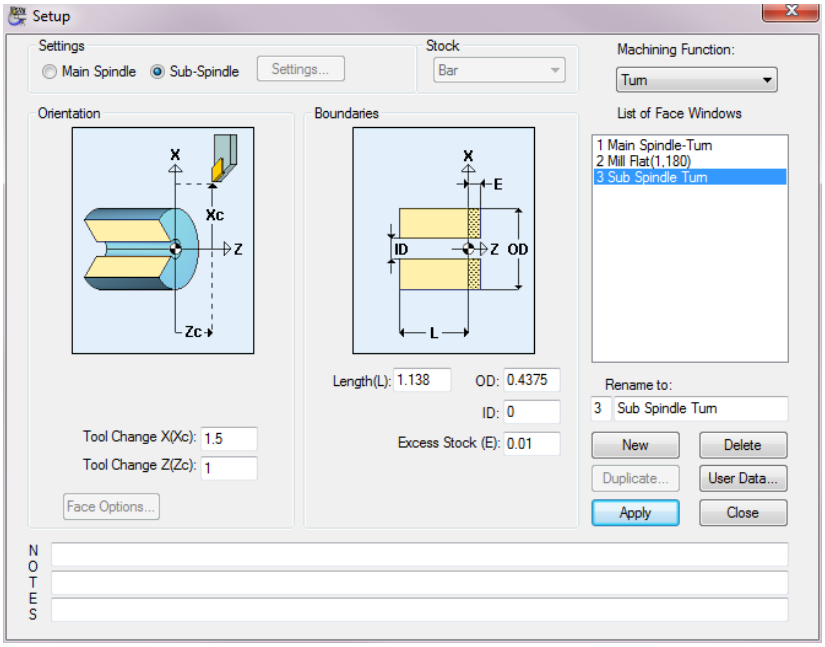
This section of the tutorial will instruct you in the steps necessary to perform the drilling operations on the sub-spindle on the centerline.

Setting Up the Face Window

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

1. Choose **Setup** from the **View** menu to view the **Setup** dialog box or use the **Shortcut to Setup** icon located above the profile icons on the left-hand side of the **Face Window**.
2. Click the **<New>** button to create a new **Face Window**.
3. From the **Machining Function** drop down menu select **Turn**.
4. Check the radio button marked **Sub Spindle**. Notice how there is no guide bushing representation on the sub-spindle, as the stock is fixed when being held in the sub-spindle collet.
5. Type **Sub Spindle Turn** in the **Rename To:** field and press <Enter>.

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



Note: *In the dialog above, the* ***Sub Spindle*** *radio button must be checked. This designates that this operation will be programmed on the sub-spindle of the machine.*

6. When you are satisfied that your **Setup** dialog appears as the one above, click the **<Apply>** button. This will apply the parameters to the current **Face Window**.
7. Click the **<Close>** button to close the **Setup** dialog.
8. Verify that the **Sub Spindle Turn** Face Window is selected by clicking it in the tree view to the left.

9. 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.

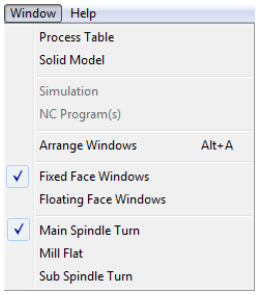
10. Choose **Show Boundaries** from the **View** menu.

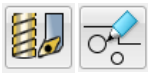
Creating Geometry in the CAD Mode

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

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

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





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

3 With the **Selection** icon highlighted, select the geometry you have created by choosing **Select All** from the **Edit** menu.(It may still be selected from the previous selection)

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

5 Return to the **Sub Spindle Turn** Face Window by choosing **Sub Spindle Turn** from the **Window** menu.

6 In the **Sub Spindle Turn** Face Window, choose **Paste** from the **Edit** menu.

Remembering the **PartMaker** convention that all programming is carried out from right to left, you will now mirror the geometry you just pasted. To do so:

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

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

3 With the geometry selected, choose **Mirror** from the **Edit > Transform** menu.

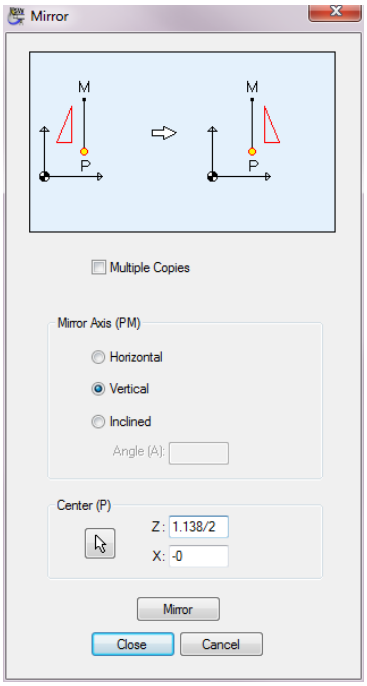
Note: *To flip the part in order to perform operations on the back, you will need to mirror with respect to the middle of the part.*

4 Uncheck Multiple Copies.

5 Click the **Vertical** radio button.

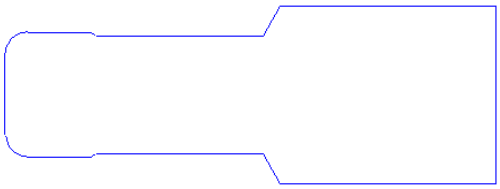
6 Enter 1.138/2 (29.0/2) in **Center Z** field (Part length 1.138 divided by two).

Your dialog box should appear as:



7 Click <**Mirror**> and <**Close**>.

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



8 Click the **Selection** icon to deselect your geometry.

Defining the Drilled Hole on the Sub-Spindle

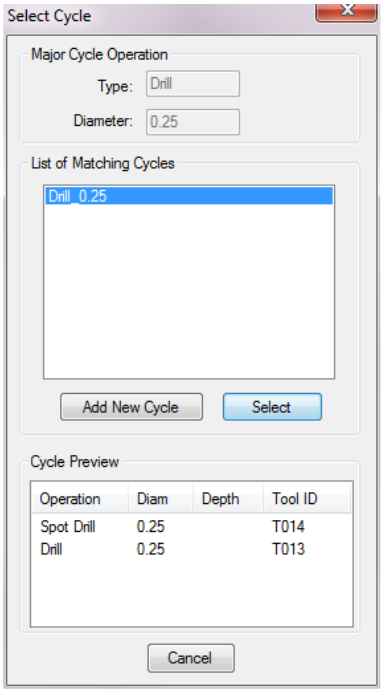
Here you will define the part feature for the drilled hole to be machined on the sub spindle. To do so (Make sure you are in CAM mode):

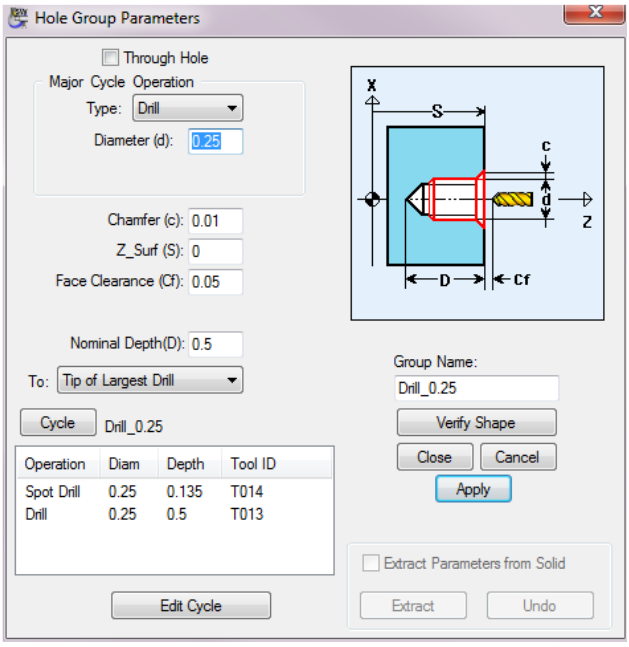
1. Click the green square in the **Color Bar.**

2. Choose **New Hole Group** from the **Part Features** menu to display the **Hole Group** Parameters dialog.
3. Uncheck the **Through Hole** box.
4. In the **Major Cycle Operation** drop down menu select **Drill**.
5. Enter a **Diameter** of 0.25 (6.4)
6. Other parameters should be set as follows:

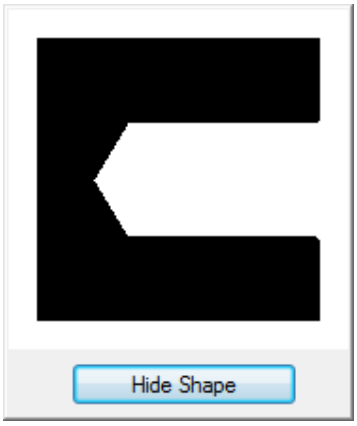
Chamfer (c):	0.01	(0.5)
Z-Surf (S):	0.0	(0)
Face Clearance (Cf):	0.05	(2.0)
Nominal Depth (D):	0.5	(12.0)

7. In the **To:** drop downmenu, choose **Tip of Largest Drill**.
8. Click the **Cycle** button and **PartMaker** will search for a cycle that matches the Major Cycle Operation and size of tool. Select the Drill_0.25(Drill 6.4) cycle. Your **Hole Group Parameters** should appear as shown below:



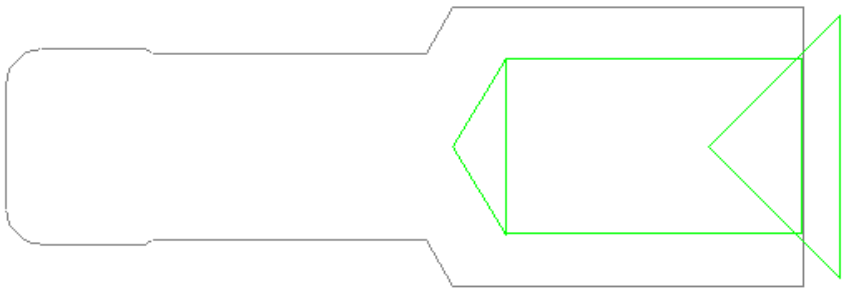


9 You can verify the shape of the drilled hole by clicking the **<Verify Shape>** button. If you entered the parameters correctly you should see the following:



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

Your **Face Window** should now appear as shown below:



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

Creating an NC Program

At this point, you have completed creating the four faces for this tutorial part. You are now prepared to take the final steps in developing your part program.

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

For purposes of this tutorial it is assumed that we are using a machine with the architecture on page 3 on this tutorial.

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

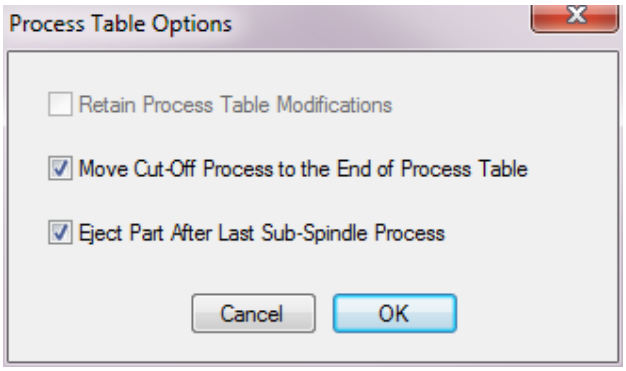
Generating a Process Table

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

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

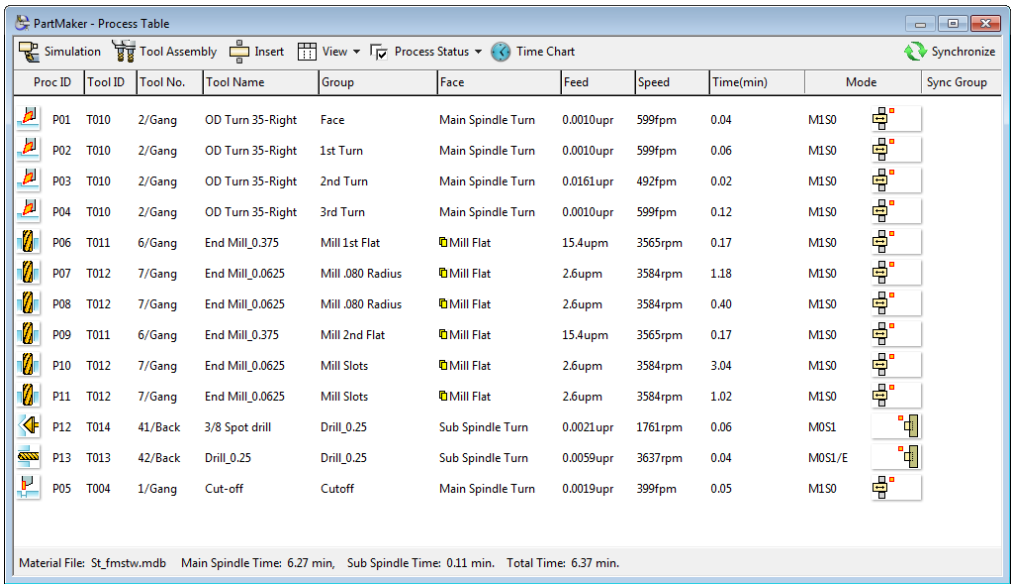
To create the Process Table:

1 Choose **Generate Process Table** from the **Job Optimizer** menu. Upon doing so, you will be prompted with the dialog below:



2 Click on **<OK>**.

The following Process Table will appear:



Visual Process Synchronization and PartMaker Specific Modes

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

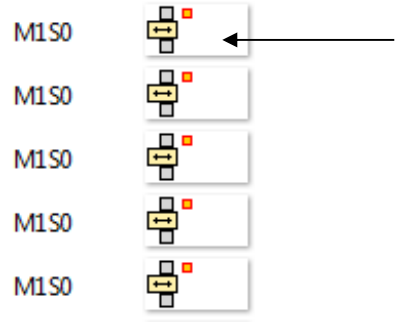
Of course, in Swiss machining, to conserve cycle time, many processes may be happening simultaneously, rather than in the sequential order in which you have created your tool paths in **PartMaker**. The Process Table accommodates this unique feature of Swiss machines allowing you to define the true order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

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

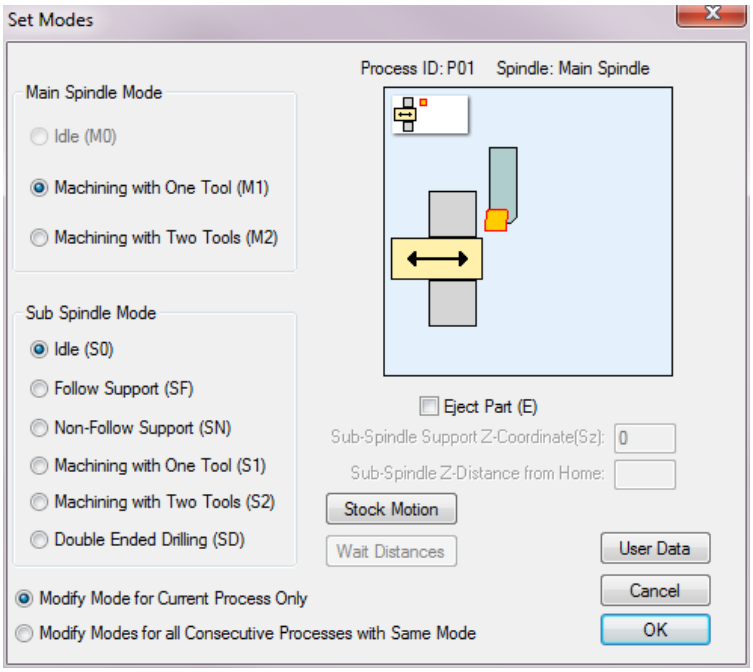
Synchronizing Processes on the Main Spindle

To synchronize the Main Spindle operations on the Process Table for this part:

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

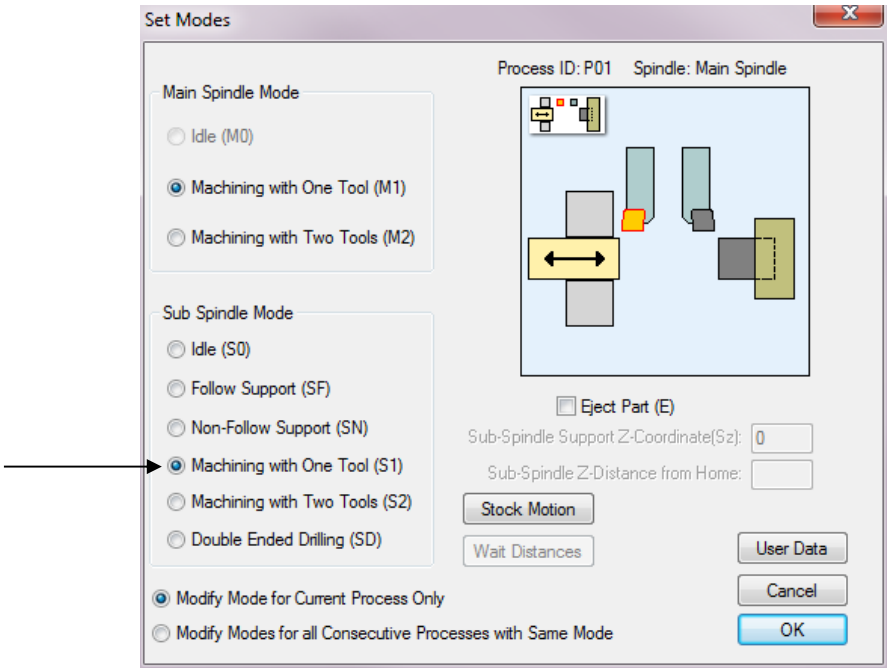


2 After doing so, you will see the **Set Modes** dialog appear as shown below:



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

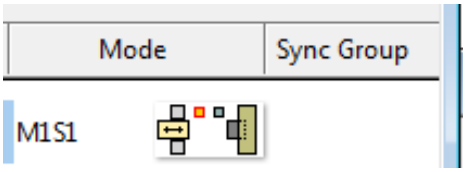
3 In this case, you will be synchronizing a number of Main Spindle operations with all of the Sub Spindle operations. To indicate that process 1 will be running simultaneously as a process on the Sub Spindle, click the **Machining with One Tool (S1)** radio button in the **Sub-Spindle Mode** area of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



Note: *In the picture above, the double headed arrow indicates that the stock is moving in the guide bushing for the given process. The yellow colored tool represents the tool doing the cutting in the selected process, while the gray one is the that is being syncronized with.*

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

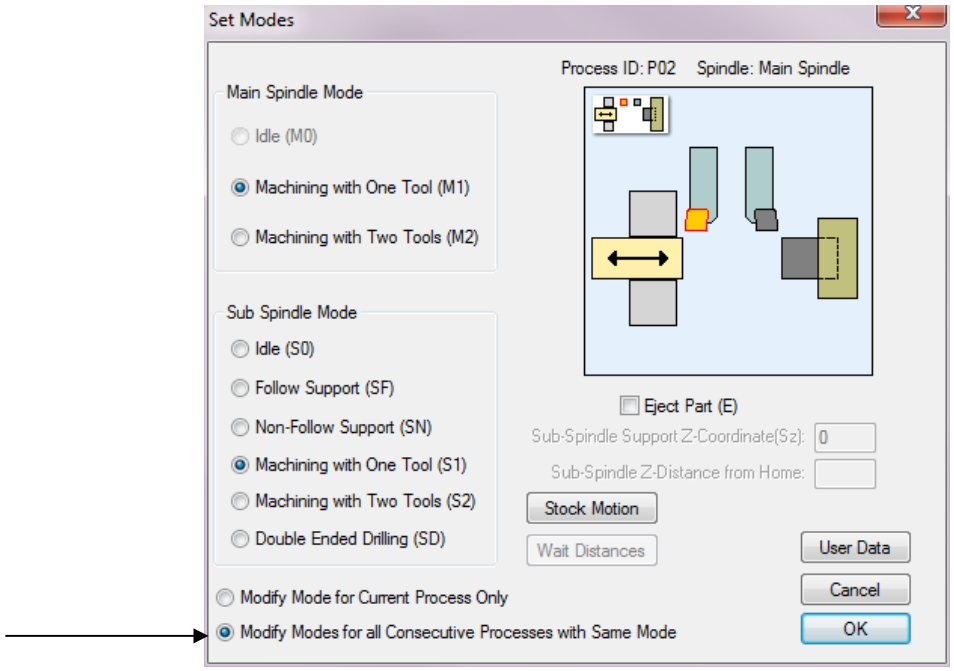
Notice how the **Set Modes** icon now depicts two tools cutting, one on the Main Spindle and one on the Sub Spindle. The Mode column is now filled in with M1S1, indicating that during this process one tool is cutting on the main spindle and one is cutting on the sub spindle at the same time.



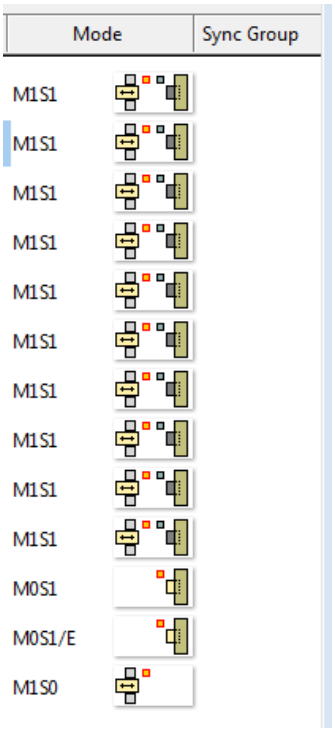
4 In order to quickly set the remaining Main Spindle Processes to **M1S1** (i.e. simultaneous cutting on Main and Sub Spindle) click the **Set Modes** dialog associated with process 2. Again, click the **Machining with One Tool (S1)** radio button in the **Sub-Spindle Mode** area of the **Set Modes**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

5 Next, click the Modify Modes for all Consecutive Processes with Same Mode radio button.

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



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



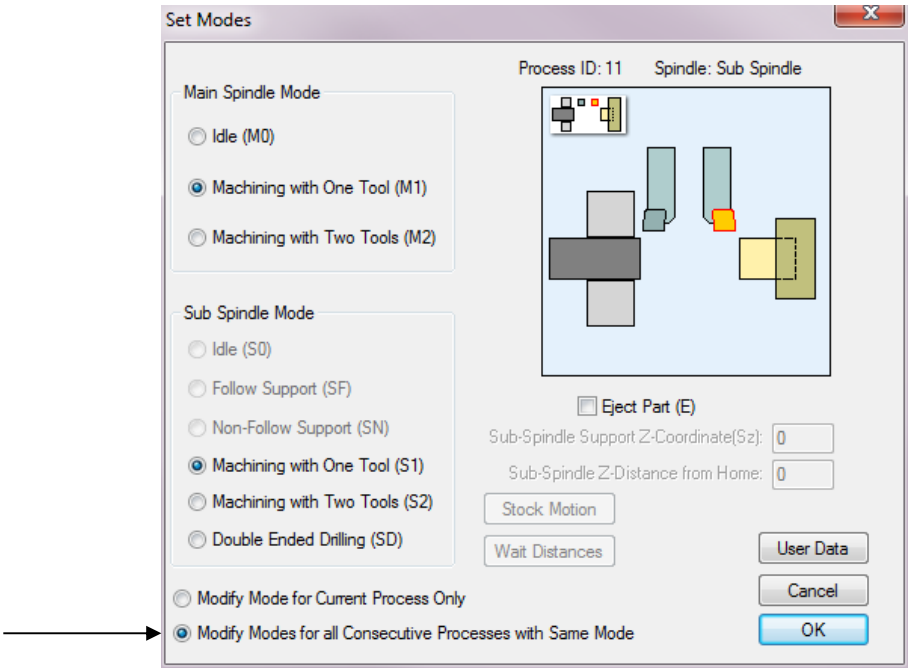
Synchronizing Processes on the Sub Spindle

To synchronize the Sub Spindle operations with the Main Spindle operations discussed above on the Process Table for this part.

1. Click on the set modes icon for Process 12, the first operation programmed on the Sub Spindle. With Process 12 selected (it should have a rectangle around it) click the **Set Modes** icon.

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

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



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

Setting the Cut-Off Operation

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

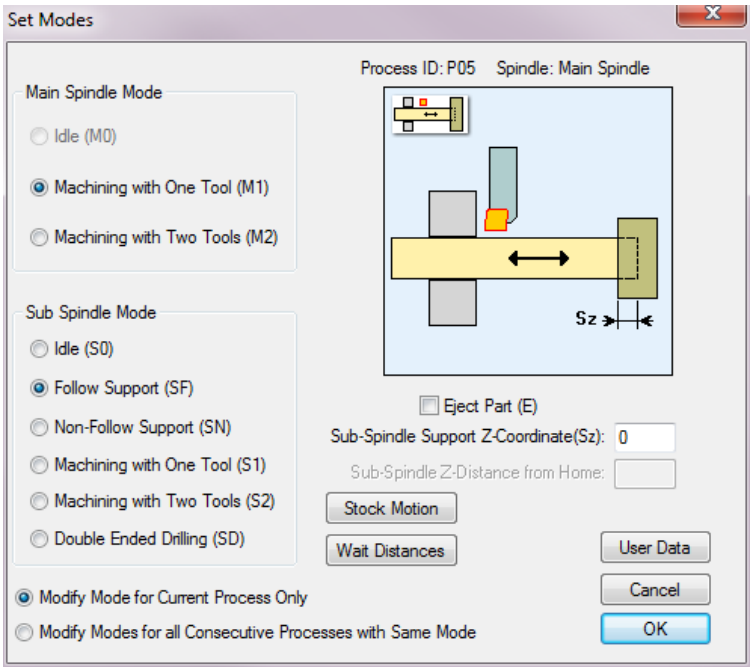
To properly set-up the cut-off and transfer from the main spindle to the sub spindle:

1 Click the **Set Modes** icon associated with process 5 (the Cut-off operation) to enter the **Set Modes** dialog. The proper mode for the Cut-Off operation is **Cutting with One Tool (M1)** on the Main Spindle while providing **Follow Support (SF)** with the sub-spindle, meaning the sub-spindle is moving in sync with the main spindle as the part is being cut.

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

When setting this mode, enter a value of 0.9 (25) for the **Sub-Spindle Support Z-Coordinate (Sz)**, the position at which the sub-spindle is gripping the part during transfer.

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



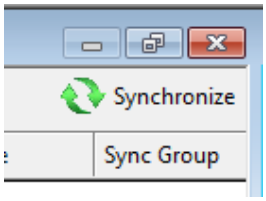
2 Click the **<OK>** button to return to the **Process Table**. Notice the appearance of the Cut-Off process on the Process Table:

Creating Sync Groups

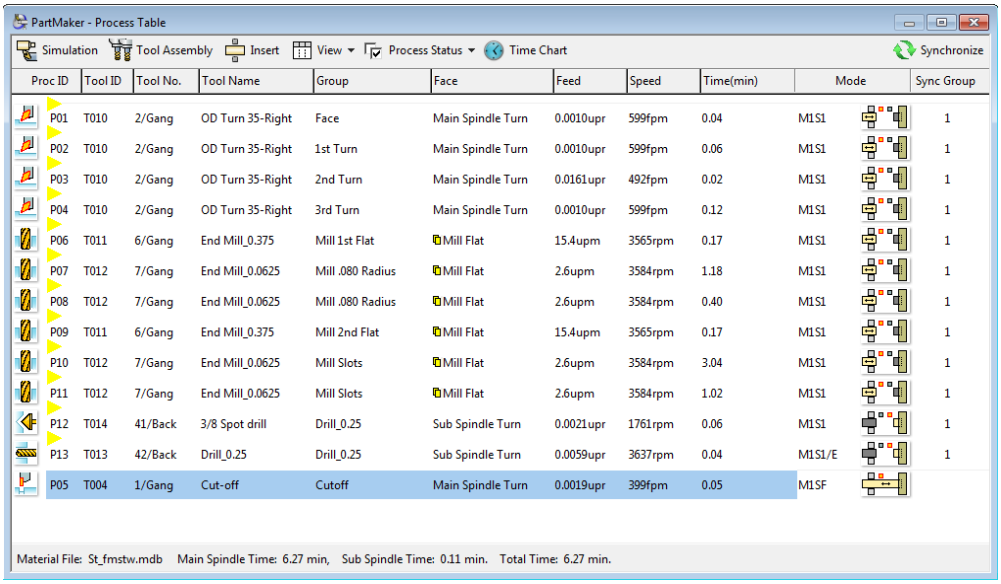
With the Process Table generated and sequence of operations established, you can now verify that your synchronization approach is correct by creating Sync Groups, groups of processes being executed simultaneously.

To do so:

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



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

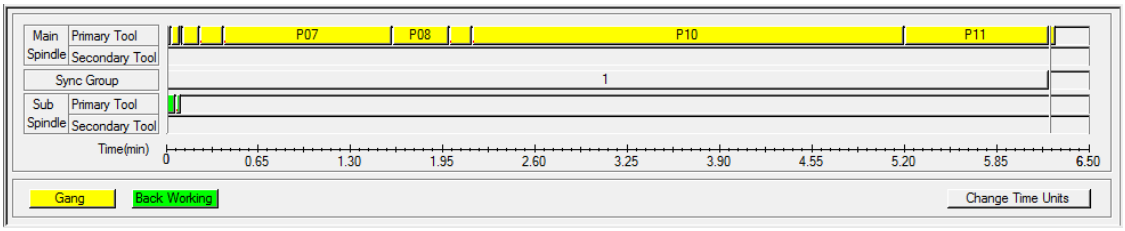


Proc ID	Tool ID	Tool No.	Tool Name	Group	Face	Feed	Speed	Time(min)	Mode	Sync Group
P01	T010	2/Gang	OD Turn 35-Right	Face	Main Spindle Turn	0.0010upr	599fpm	0.04	M1S1	1
P02	T010	2/Gang	OD Turn 35-Right	1st Turn	Main Spindle Turn	0.0010upr	599fpm	0.06	M1S1	1
P03	T010	2/Gang	OD Turn 35-Right	2nd Turn	Main Spindle Turn	0.0161upr	492fpm	0.02	M1S1	1
P04	T010	2/Gang	OD Turn 35-Right	3rd Turn	Main Spindle Turn	0.0010upr	599fpm	0.12	M1S1	1
P06	T011	6/Gang	End Mill_0.375	Mill 1st Flat	Mill Flat	15.4upm	3565rpm	0.17	M1S1	1
P07	T012	7/Gang	End Mill_0.0625	Mill .080 Radius	Mill Flat	2.6upm	3584rpm	1.18	M1S1	1
P08	T012	7/Gang	End Mill_0.0625	Mill .080 Radius	Mill Flat	2.6upm	3584rpm	0.40	M1S1	1
P09	T011	6/Gang	End Mill_0.375	Mill 2nd Flat	Mill Flat	15.4upm	3565rpm	0.17	M1S1	1
P10	T012	7/Gang	End Mill_0.0625	Mill Slots	Mill Flat	2.6upm	3584rpm	3.04	M1S1	1
P11	T012	7/Gang	End Mill_0.0625	Mill Slots	Mill Flat	2.6upm	3584rpm	1.02	M1S1	1
P12	T014	41/Back	3/8 Spot drill	Drill_0.25	Sub Spindle Turn	0.0021upr	1761rpm	0.06	M1S1	1
P13	T013	42/Back	Drill_0.25	Drill_0.25	Sub Spindle Turn	0.0059upr	3637rpm	0.04	M1S1/E	1
P05	T004	1/Gang	Cut-off	Cutoff	Main Spindle Turn	0.0019upr	399fpm	0.05	M1SF	

On the Process Table above, processes 1 through 12 are all being executed at the same time and as a result are in the same **Sync Group**.

Spindle Balancing and Time Chart

Once you have syncronized your operations, you can view the Time Chart by clicking the Show Time Chart button in the lower right hand corner of the process table.

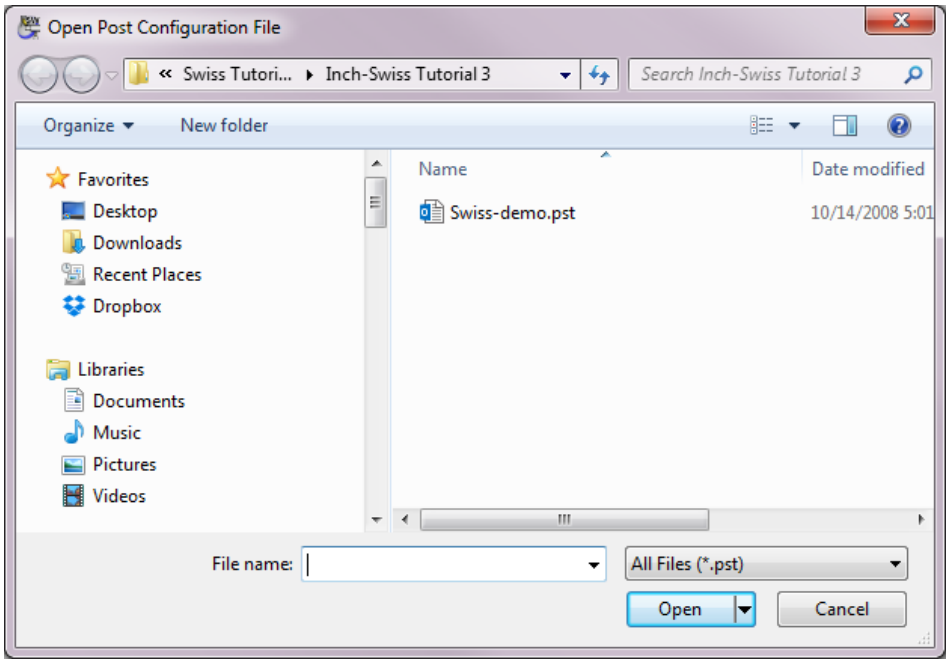


After clicking the Sync Group button, your time chart will show you graphically which operations are over lapping and where additional opportunities for saving cycle time are by combining them with other operations.

Selecting a Post Processor

Having finished generating the Process Table and synchronizing operations, you must now select a Post Configuration File before you can output an NC program. This post configuration file is stored in the PM-Swiss\Swiss tutorial 3 directory. If you do not have any other post configuration files loaded in this directory the **Swiss_demo.pst** will be loaded automatically. If you have other post configuration files in this directory:

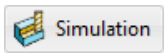
1. Choose Post Config File = ? from the Job Optimizer menu.
2. Select **Swiss-demo.pst** post as shown below. **PartMaker** SwissCAM comes standard with a post configuration for a Fanuc 18t control which supports live tooling, y-axis, c-axis and subspindle operations.



3. Click **<Open>** to load the post processor.

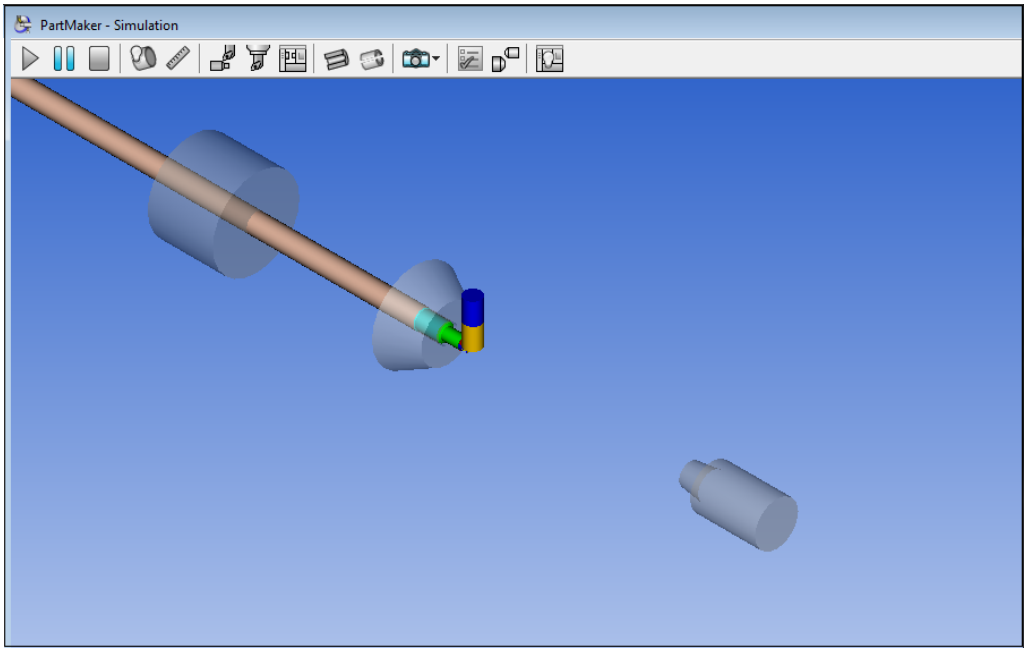
Simulating the Process

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



1 Click the <**Simulation>** button at the bottom left hand portion of the Process Table.

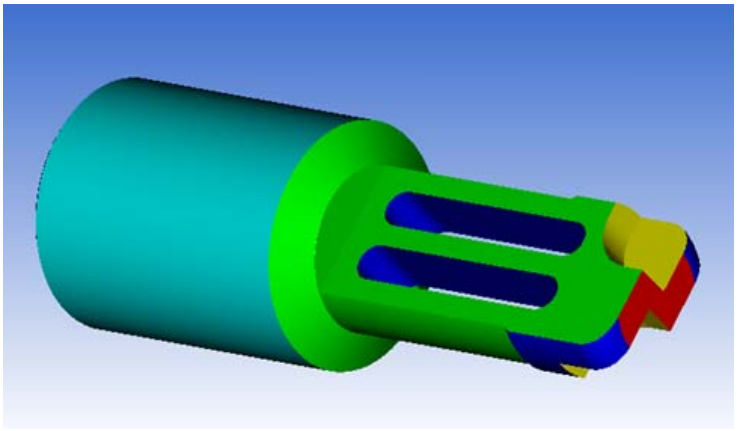
2 Press the Space Bar or choose **Start Simulation** from the **Simulation** menu to begin the simulation.



3 The simulation will be completed when the completed part is resting in the sub-spindle.

4 Choose **Show Finished Part** from the **Simulation** menu, or right click on the Simulation window, choose **Show Finished Part.**

5 Use the left mouse button to rotate the view of the finished part.



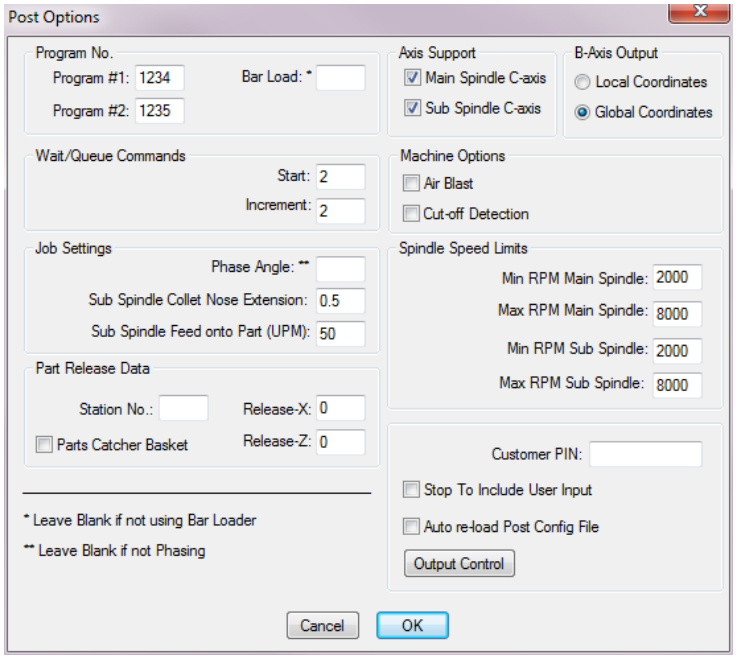
Note: *The simulation being shown here is part of PartMaker's Basic Simulation function. If you have licensed Advanced Simulation, please proceed chapter 6 for more information on using Advanced simulation.*

Generating an NC Program

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

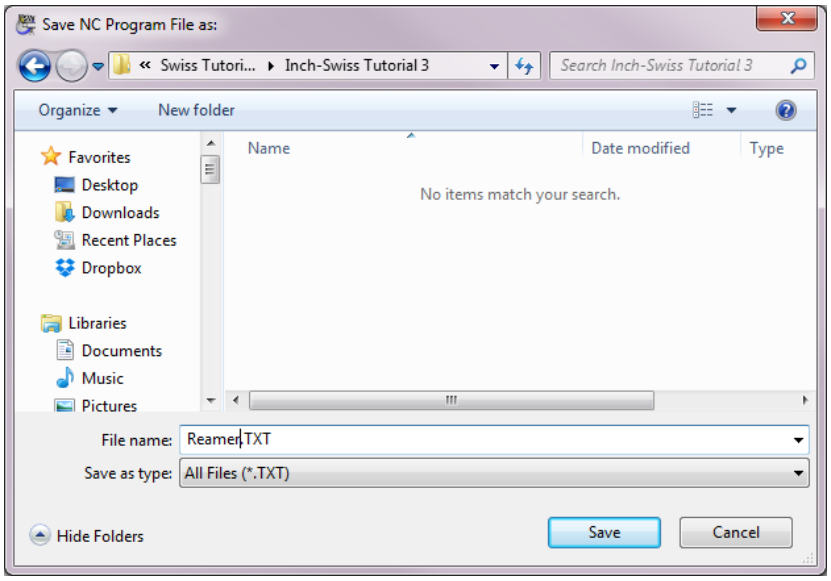
1 Choose Generate NC Program from the Job Optimizer menu.

2 You will next see the **Post Options** dialog. Set the **Post Options** dialog as shown below:

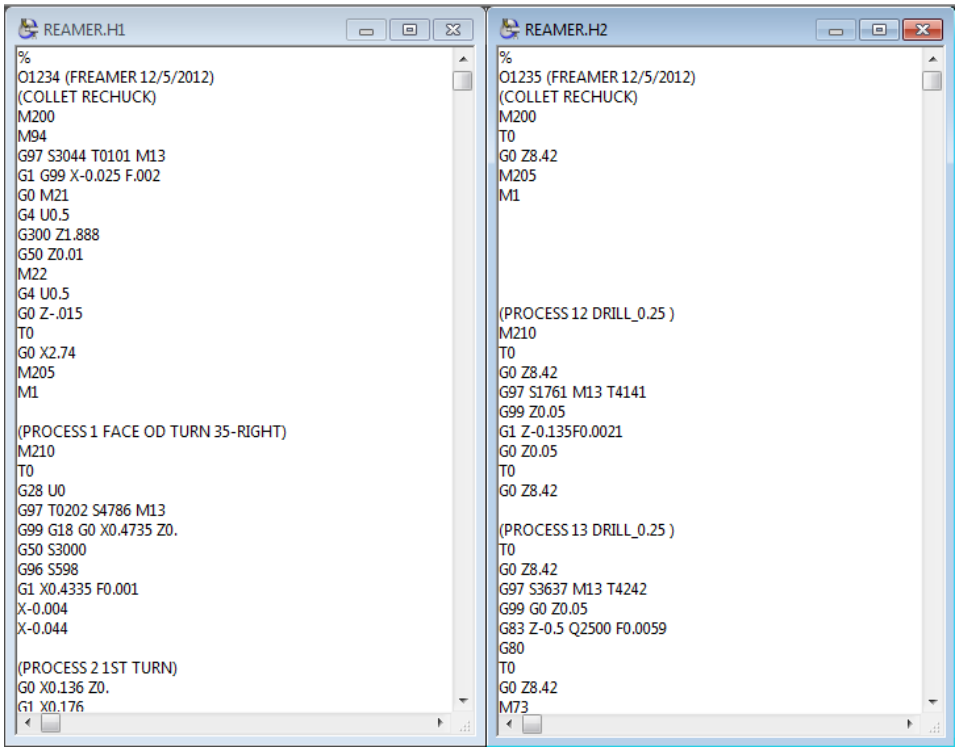


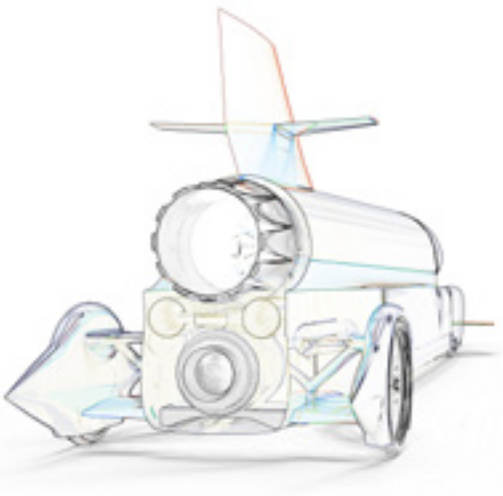
In the **Post Options** dialog you set certain parameters for assignment during post processing. Please refer to your machine specific addendum for more information on how each of the fields in this dialog pertain to your Swiss turning equipment.

3 Click **<OK>** in the **Post Options Dialog** to proceed to the **Save NC Program** dialog. In this dialog, enter a name for your program. Here you should enter **"Reamer"**.



4 Click **<Save>** to generate the NC program. You will see your programs as shown below:

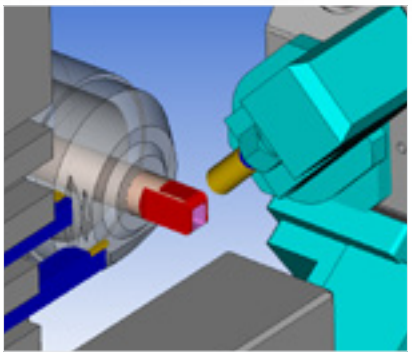




PartMaker 2015

World-leading CAM software for production machining

www.partmaker.com

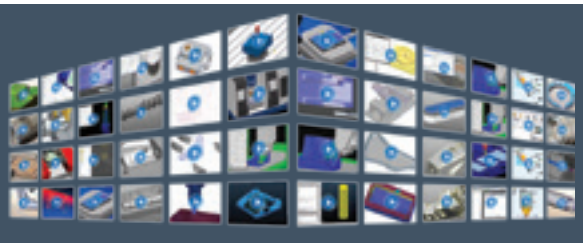


PartMaker 2015

www.partmaker.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

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

Delcam Advanced Manufacturing Solutions

Powering your productivity

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

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

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

AUTODESK.