Practical Applications of PartMaker SwissCAM Machining Function Face Windows for Live Tooling Operations

Introduction

This appendix to the PartMaker SwissCAM User Manual is intended to give you, the user, a more in-depth and practical understanding of applying PartMaker's Machining Function Windows for use with Live Tooling Operations.
This document will explain the following:

What types of Part Features Should be created in each Machining Function Face Window
Which Hole Group and Profile Groups Parameters options are available in each Machining Function Face Window
Which Graphics Icons are available in each Machining Function Face Window
How Part Features should be created in each Machining Function Face Window

Many of the part examples provided in the pages to follow can be found in the various sub-directories labeled with machine names in the Sample Directory found in the Pm-Swiss directory in which PartMaker SwissCAM is installed. The file name and path for each file and specific *.job, *.tdb and *.cdb are provided where relevant for each example.

Machining Function Selection

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

Milling capabilities of a CNC Swiss lathe vary significantly and can include:

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

PartMaker CAM Software from IMCS, Inc. applies a Patented 'divide and conquer' programming strategy to simplify programming of Swiss Type Lathes.

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

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

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

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

Getting Started: The Setup Dialog

PartMaker employs a programming methodology whereby each machine motion type can be separated into a different window dedicated to a specific machining function. Every one of these windows is known as a **Face Window**. Once created, each **Face Window** can be accessed from the Window menu.
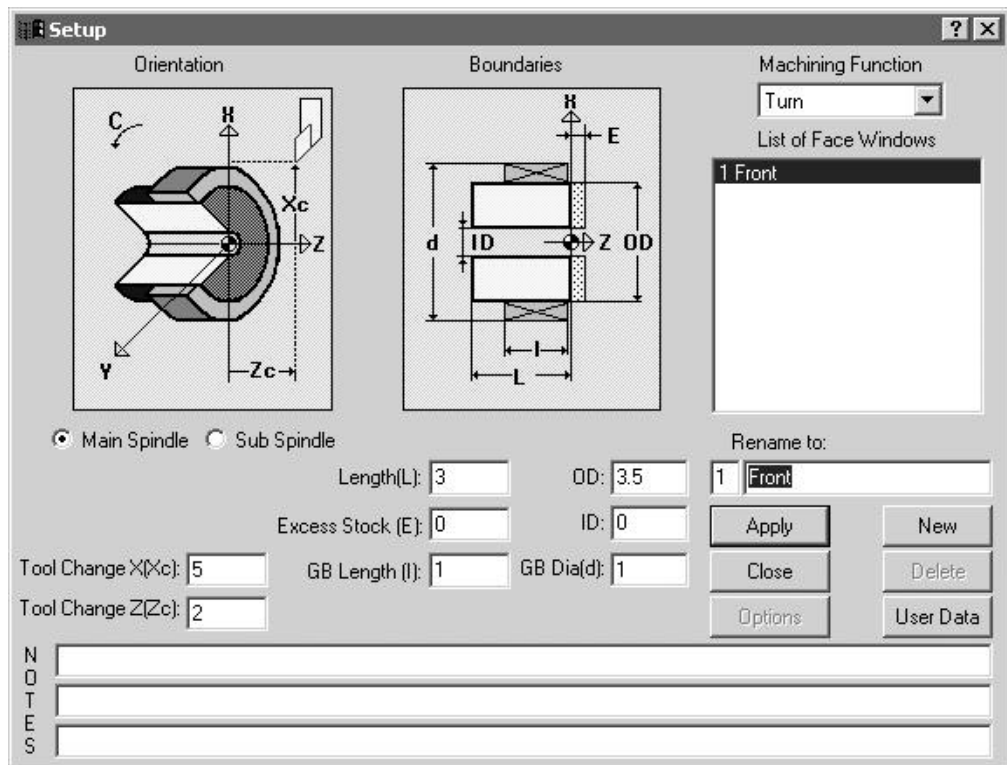
Once divided into much smaller, similar motion components, programming of a multi-axis Swiss-type lathe becomes a much easier task. To best explain how **PartMaker** handles this task, it is necessary to first explain how each of these individual-machining functions is defined.

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

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

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

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

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

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

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

|	
<Apply>	Accepts or applies any new parameters or data entered into the various fields in the **Setup** dialog, and the dialog remains open.
<New>	Creates new face windows of different machining function types or on different spindles, adding to the List of Face Windows
<Close>	Accepts or applies any new parameters or data entered into the various fields in the **Setup** dialog, and closes the dialog.
<Delete>	Deletes the current selection from the List of Face Windows

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

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

Machining Function: Turning

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

The picture below shows the appearance of the **Setup** dialog when performing Turning on the Main Spindle.
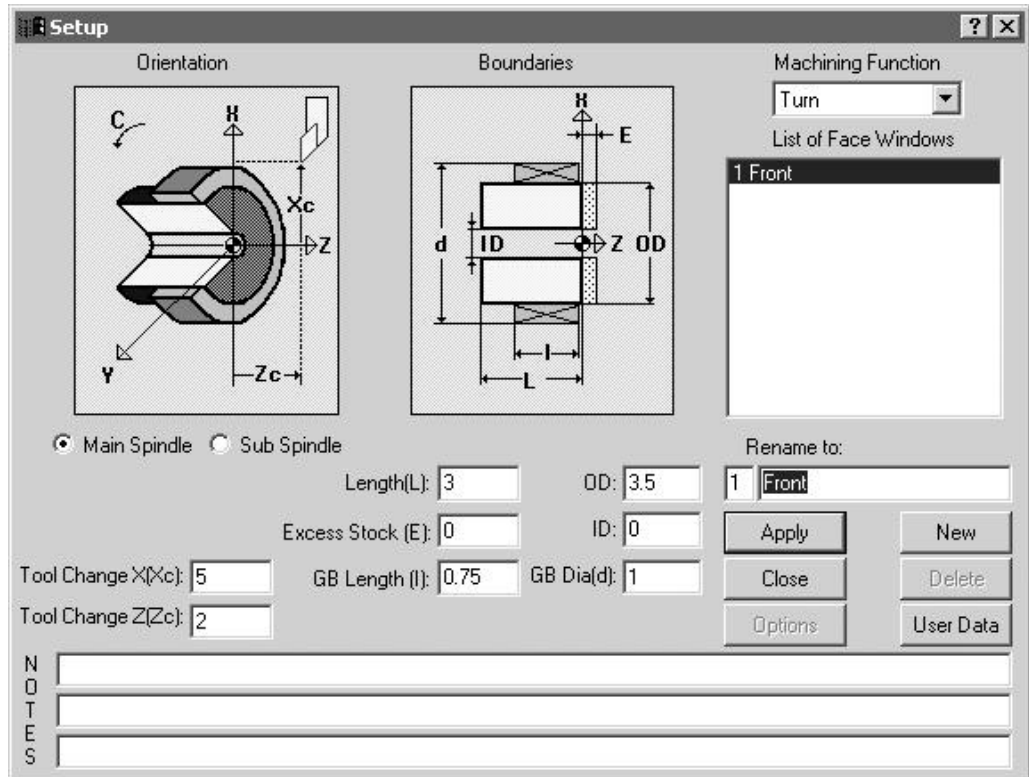
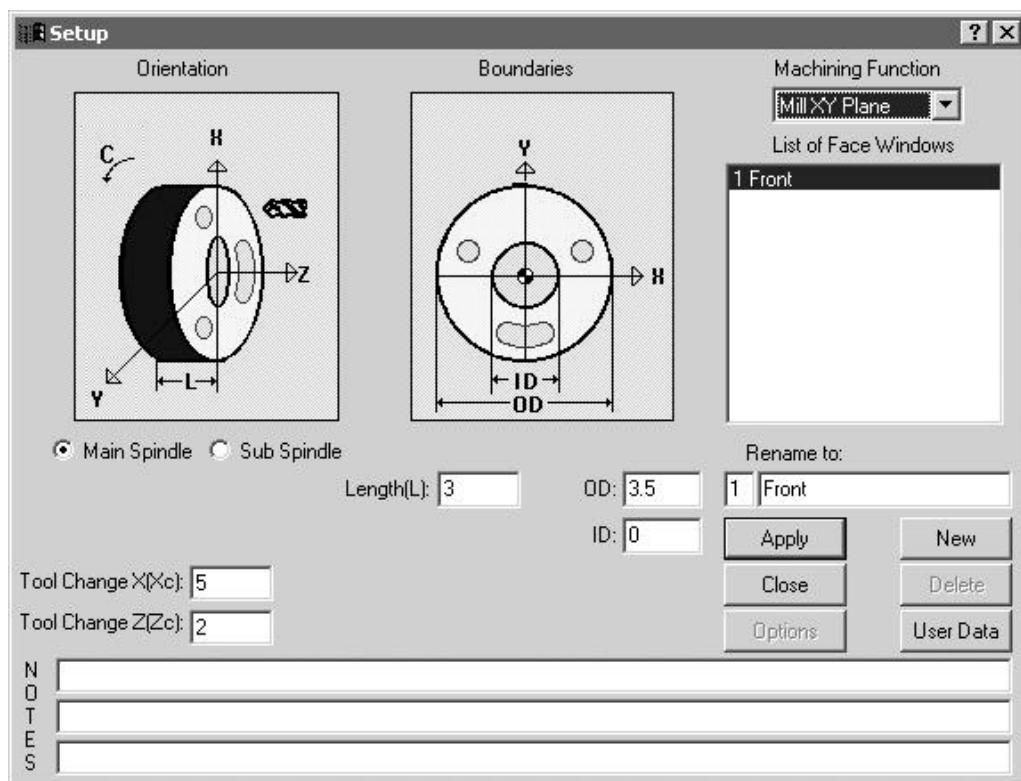|	
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
GB Length (l)	Length of the "land area" of the guide bushing mechanism controlling the motion of the headstock through the main spindle
ID	Starting inside diameter of bar stock before machining
Excess Stock (E)	Rough stock left for facing on the front of the part
GB Dia (d)	Outside diameter of the guide bushing mechanism controlling the motion of the headstock through the main spindle
Length (L)	Length of the finished part

Machining Function: Mill XY Plane

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

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

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

Types of Features Created in Mill XY Machining Function Window

The Mill XY window uses a Z-oriented "live" tool. There is no polar C-axis movement (the C-axis indexes and locks).

Note: Your machine must have y-axis capability and also needs to have ample travel to machine in the negative quadrant of the part.

The features shown on the parts below are representative examples of features that can be created in the Mill XY Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Samples directory which was installed when you initially installed PartMaker SwissCAM. Where relevant, the locations of these files have been provided.
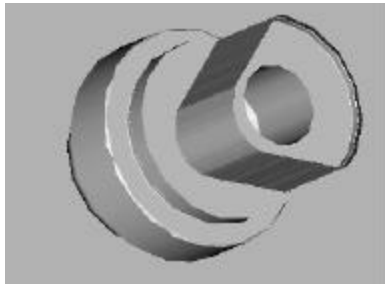
Figure 1: Milled Cam
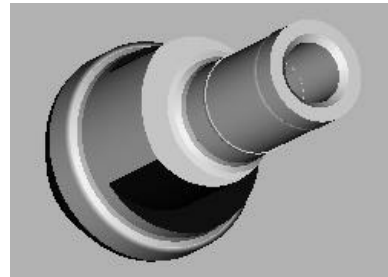
Figure 2.: Milled Cam
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_M\CitizenM.job, CitizenM.cdb, CitizenM.tdb)

Figure 3: Off-Center Holes
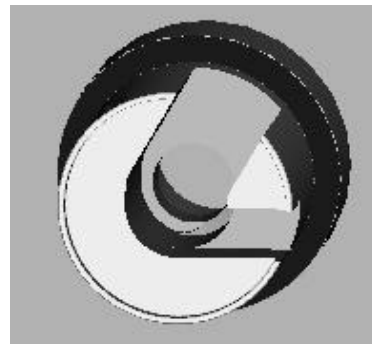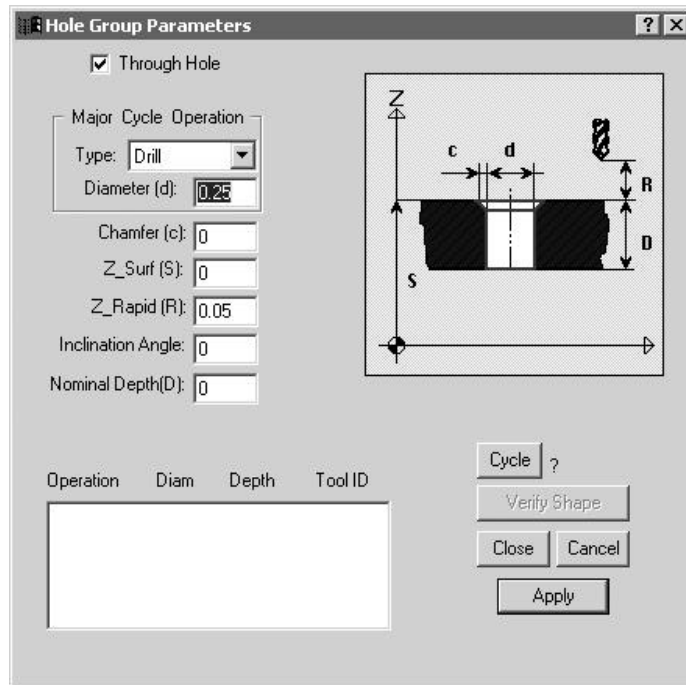(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Star_SV\StarSV.job, StarSV.cdb, StarSV.tdb)

Figure 4: Milled Cam
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Star_SV\StarSV.job, StarSV.cdb, StarSV.tdb)

Part Features Dialogs Available in Mill XY Machining Function

Both the **New Hole Group** and **New Profile Group** commands from the **Part Features** menu can be used in the Mill XY Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in Z.

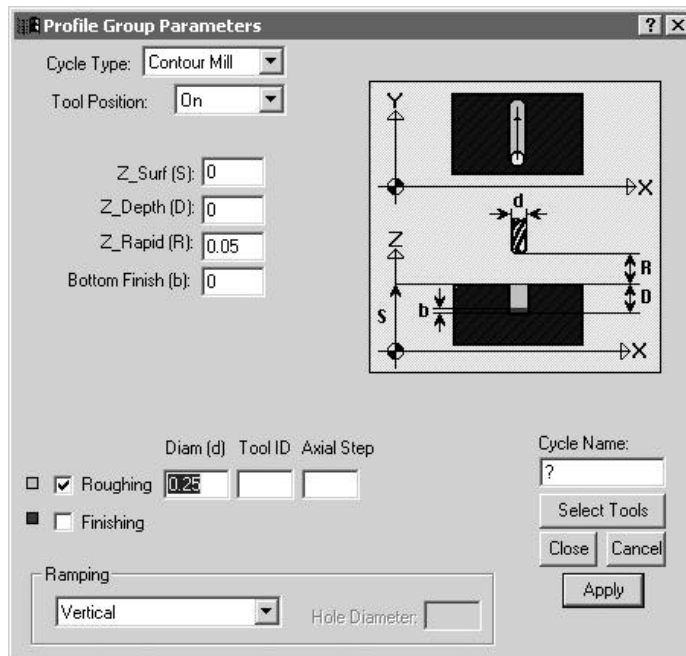| Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycles database
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
Z_Surf (S)	Establishes your Z-zero work plane
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf
Inclination Angle	Specifies the angle of the tool when making inclined holes (you must have a "B" axis or an angled attachment.)
Nominal Depth (D)	The depth of the operation to be performed

The Mill XY machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with **Tool Position** On centerline is shown. To create a pocket, choose **Pocket Mill** from Cycle Type drop down menu.

Cycle Type	Indicates the type of feature to be milled, either a **Contour Mill** or **Pocket Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling)
Z_Surf (S)	Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part
Z_Depth (D)	The depth of the operation to be performed
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Indicates the manner of entry of end mill into a feature, either **Vertical, Helical, Sloped** or **Sloped Down and Return**

Graphics Icons Available in the Mill XY Plane Machining Function

The Mill XY Plane Machining Function Face Window uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.

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

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

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

Programming Example using the Mill XY Plane Machining Function: Square and Off-Center Hole on Face

A practical programming application of the Mill XY Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a ½ Inch Square with a ¼ inch End Mill and create a .1 diameter off-center hole. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill XY Plane Face Window should have an orientation of **Z Tool**)
1.   .25 Inch Diameter End Mill
2.   .1 Inch Diameter Drill
3.   Spot Drill with a diameter larger than .1 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .1 diameter drill and a spot drill.
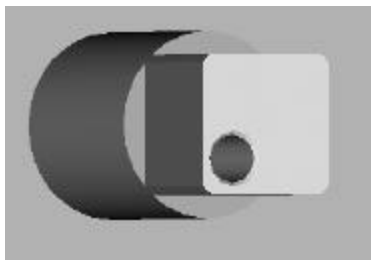
Features to be created:

1.   *Mill a .5 inch square with a .25 dia end mill*
2.   *Spot Drill and Drill an off center hole with a .1 diameter drill*

Step 1. Create a Mill XY Plane Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill XY Plane Machining Function Face Window. To do so:
1.   Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2.   In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3.   From the **Machining Function** drop down menu, choose **Mill XY Plane**
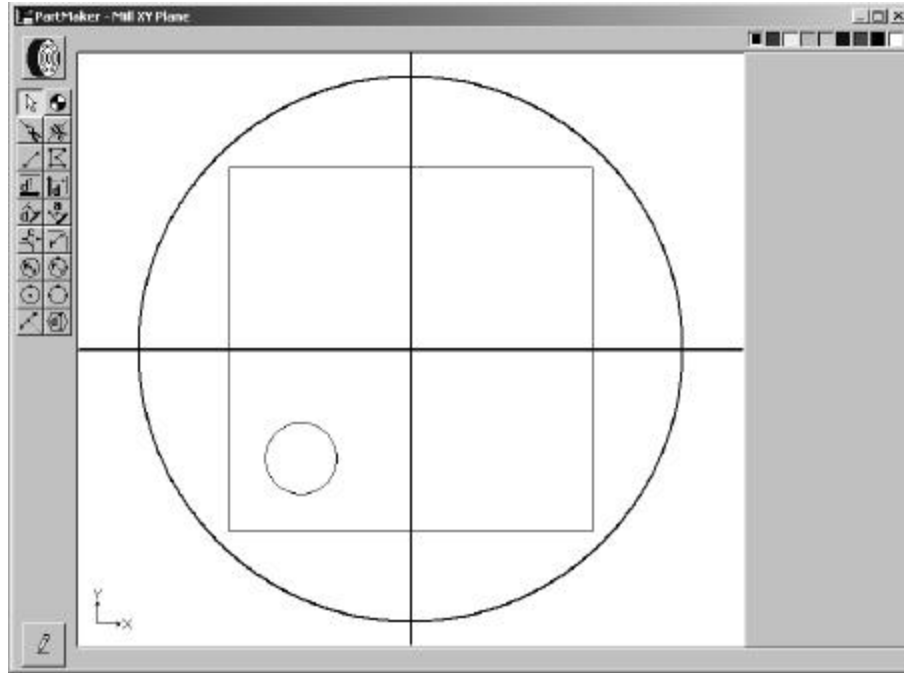4.   Enter the setup parameters as shown below:
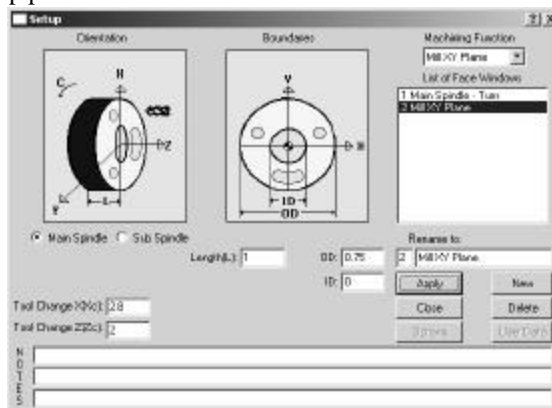
Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill XY Plane Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a ½ inch square
4. Draw a .05 inch radius circle with a center location at X<-.15> and Y<-.15>

Your completed geometry and CAD Face Window should appear as shown below:


Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

Step 3. Create Part Features and assign tool paths

Start by entering the CAM mode in the Mill XY Plane Window. First you will create the tool path to mill the square. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Contour Mill** under **Cycle Type**
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter a Z_Depth of .5
5. Enter a Tool Diameter of .25 and click **Select Tools**
6. In the **Cycle Name** field, type **Mill Square**

Your completed **Profile Group Parameters** dialog should appear as shown below:
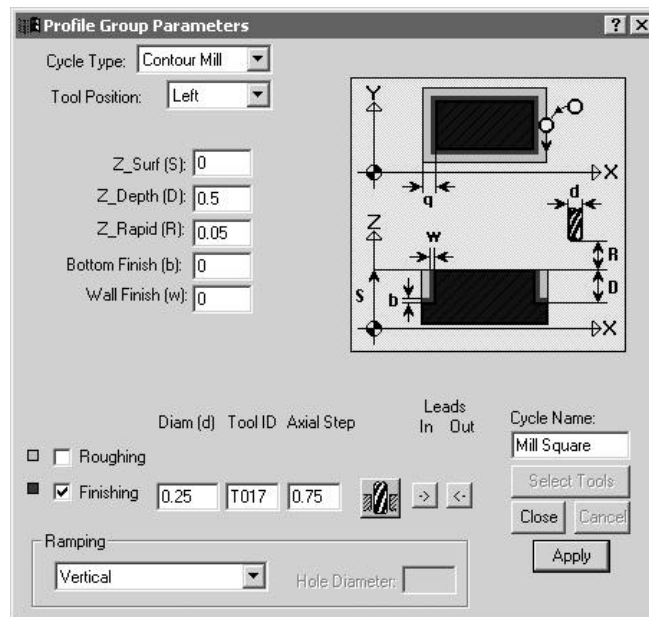
7. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
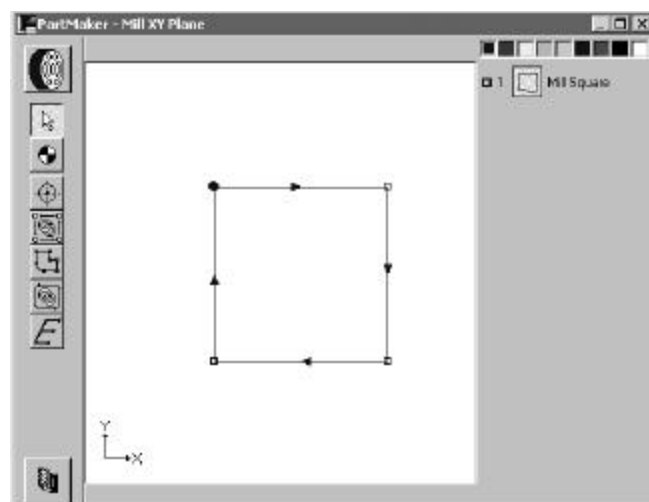
8. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
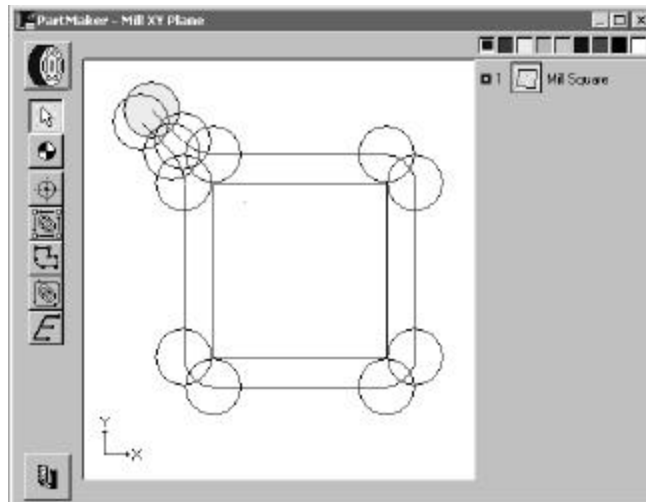
Choose **Hide Work Tool Path** to hide the tool path.

Next, you will create the off-center hole. To do so:

1. From the **Part Features** menu, choose **New Hole Group**
2. Uncheck the **Through Hole** Box
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .1
5. Enter a **Chamfer** of .01
6. Enter a **Nominal Depth (D)** of 0.5
7. Click the **Cycle** button.

Your completed **Hole Group Parameters** dialog should appear as shown below:
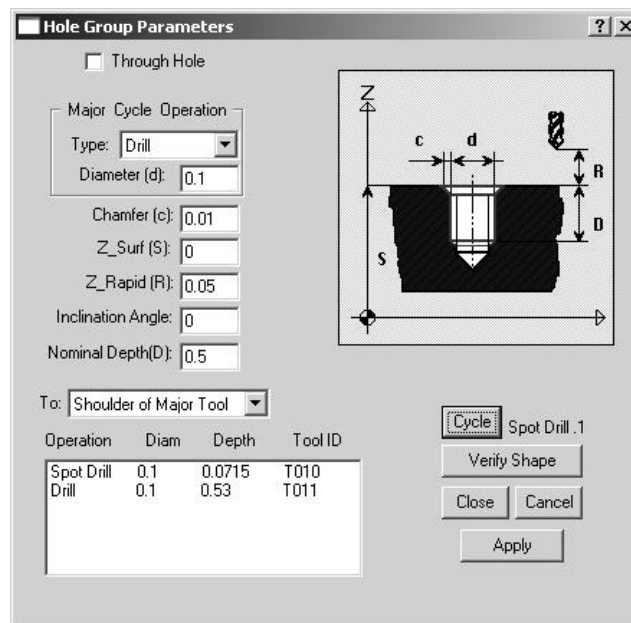
1. Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circle representing the hole to create the tool path. It should appear as shown below:
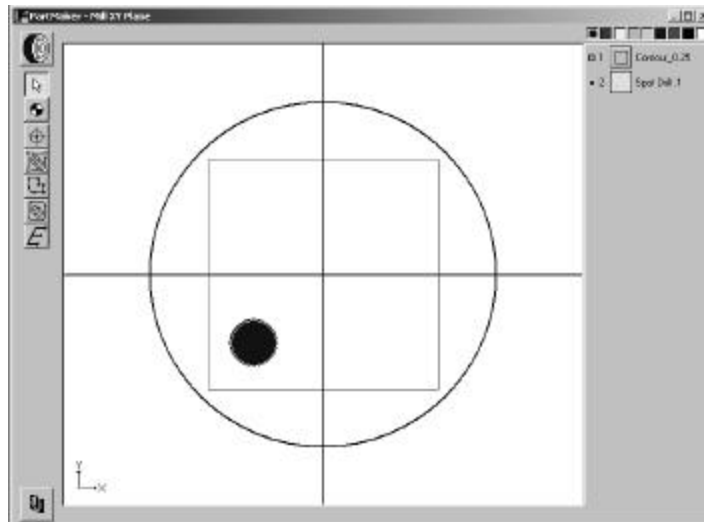
Machining Function: Mill ZY Plane

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

Programming of this machining function requires that your CNC Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.
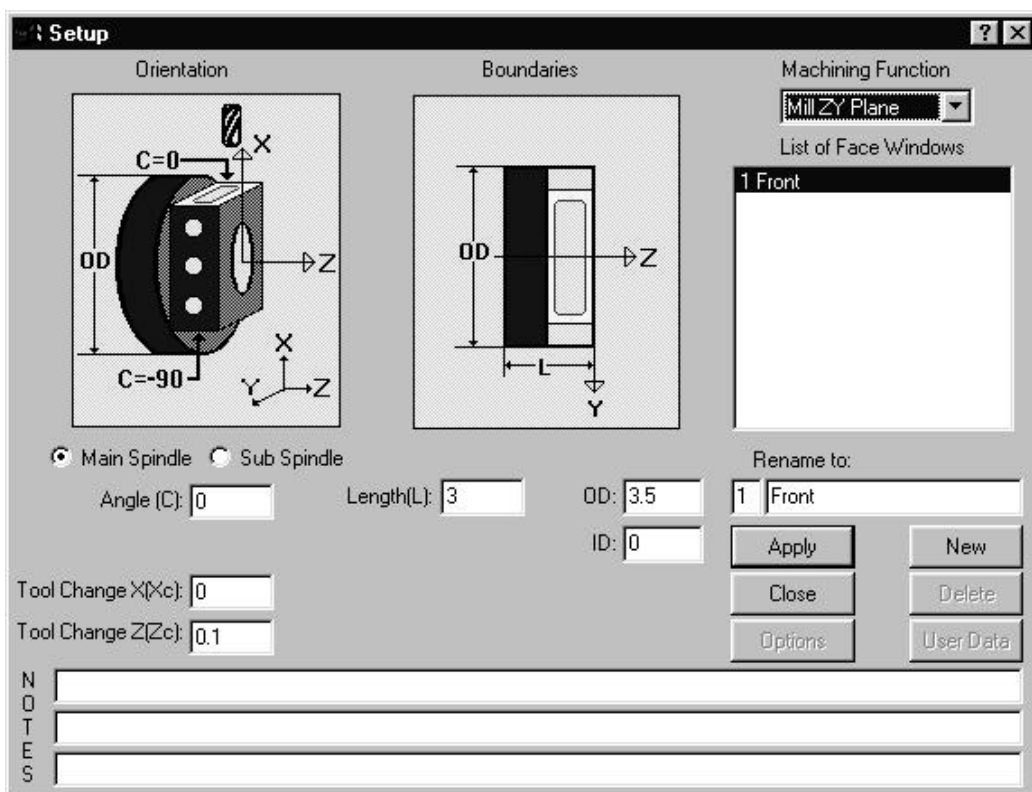
Angle (C)	Orientation angle of the stock to achieve the desired face position
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill ZY Machining Function Window

The Mill ZY window uses an X-oriented "live" tool. There is no polar c-axis movement (the c-axis indexes and locks). This window can be used for creating off-center cross-holes, flats on the diameter of the part and interpolated features. Saw Slots should also be created in this Face Window.

The features shown on the parts below are representative exa mples of features that can be created in the Mill ZY Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker SwissCAM. Where relevant, the locations of these files have been provided.
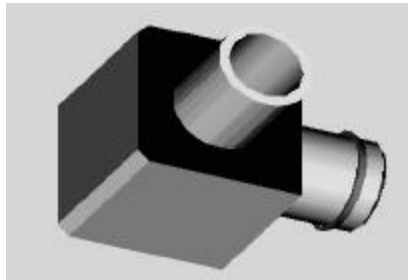
Figure 1.: Milled Cylinder
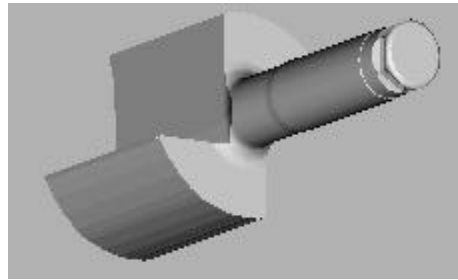(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_L\CitizenL-Yaxis.job, CitizenL-Yaxis.cdb, CitizenL-Yaxis.tdb)

Figure 2.: Milled Flats
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_L\CitizenL.job, CitizenL.cdb, CitizenL.tdb)
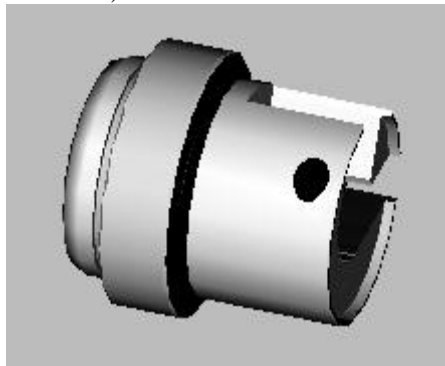

Figure 3: Off-Center Cross Hole
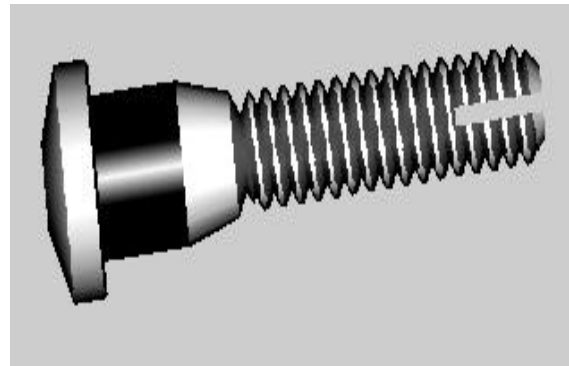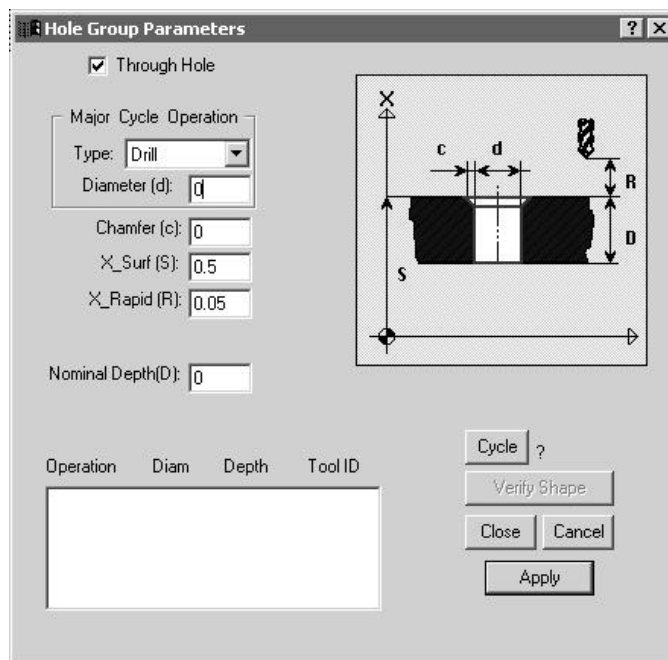(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Star_SV\StarSV.job, StarSV.cdb, StarSV.tdb)

Figure 4: Milled slot

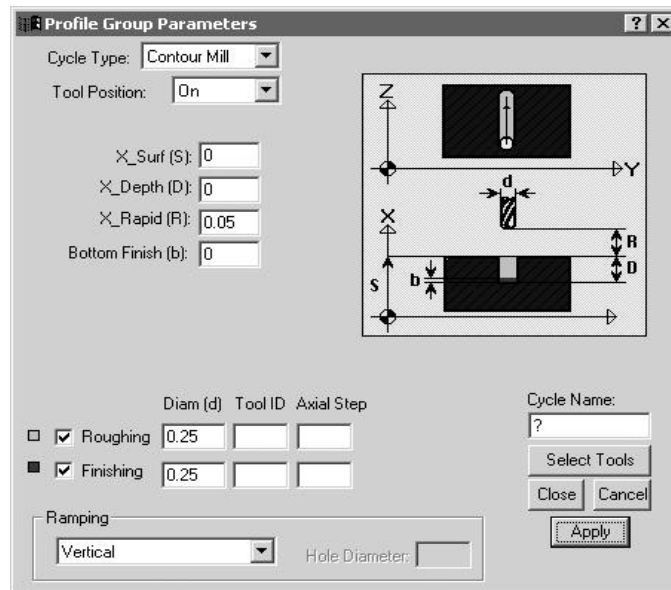### *Part Features Dialogs Available in Mill ZY Machining Function*

Both the **New Hole Group** and **New Profile Group** commands from the Part Features menu can be used in the Mill ZY Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in X.

Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycles database
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
X_Surf (S)	Establishes your X-zero work plane on the OD of the part
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf
Nominal Depth (D)	The depth of the operation to be performed

The Mill ZY machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.

Cycle Type	Indicates the type of feature to be milled, either a **Contour Mill** or **Pocket Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above of your X_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Indicates the manner of entry of end mill into a feature, either **Vertical, Helical, Sloped** or **Sloped Down and Return**

Graphics Icons – Machining Functions for Mill ZY Plane

The Machining Functions Mill ZY Plane use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.

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

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

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

Programming Example using the Mill ZY Plane Machining Function:

Practical programming applications of the Mill ZY Plane machining function have been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

Example A: Creating a single flat with 2 off-center holes

In the example given below, you will mill a flat with a ½ inch End Mill and create two .1 diameter off-center holes. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZY Plane Face Window should have an orientation of **X Tool**)
1. .500 Inch Diameter End Mill
2. .1 Diameter Drill
3. Spot Drill with a diameter larger than .1 inch

Also make sure you have a cycle in PartMaker's Cycles Database created to include both a .1-diameter drill and a spot drill.
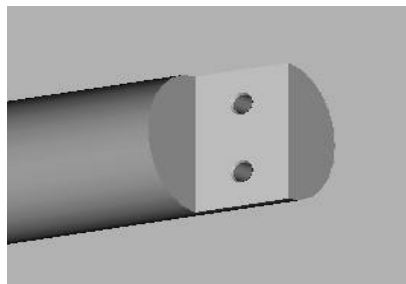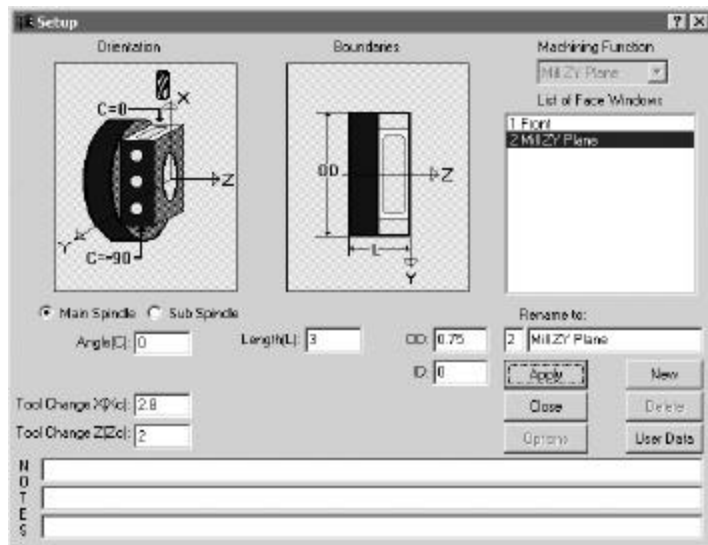

Features to be created:

1. *Mill a flat with a .500 dia end mill*

2. *Spot Drill and Drill two off center holes with a .1 diameter drill*

Step 1. Create a Mill ZY Plane Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill ZY Plane Machining Function Face Window. To do so:
1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill ZY Plane**
4. Enter the setup parameters as shown below:

Once the **Setup** dialog appears as above, clic k on **Close.**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZY Plane Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw one line at Z<. 5> to represent the end of the flat.
4. Draw two .05-inch radius circles with a center location at Z<. 25> and Y<-.1875> and at Z<. 25> and Y <. 1875>.

Your completed geometry and CAD Face Window should appear as shown below:
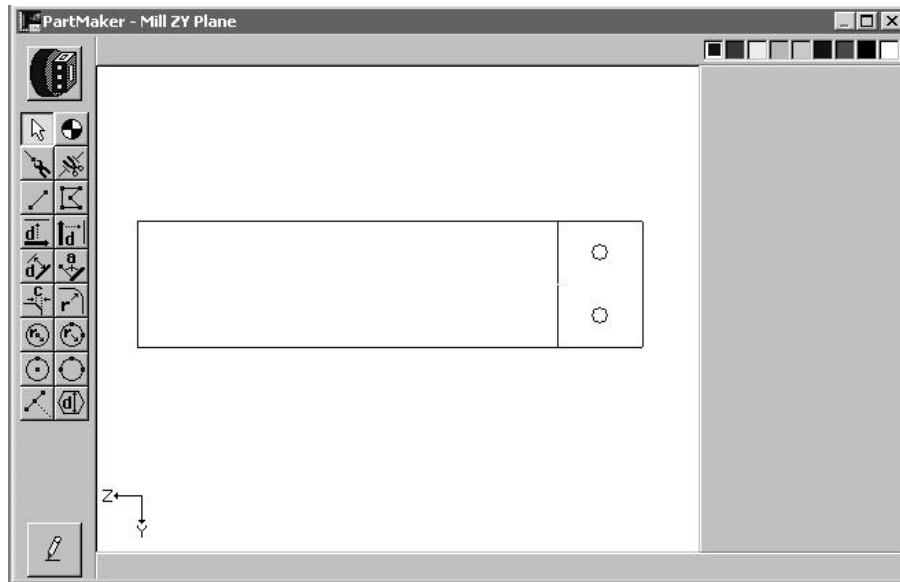
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

Step 3. Create Part Features and Assign Tool Paths

Start by entering the CAM mode in the Mill ZY Plane Window. First you will create the tool path to mill the flat. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Contour Mill** under **Cycle Type**
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter an X_Surf of .375
5. Enter an X_Depth of .375 (we want the tool to go to center)
6. Enter a Tool Diameter of .5 and click on **Select Tools**.
7. In the **Cycle Name** field, type **Mill Half.**

Your completed **Profile Group Parameters** dialog should appear as shown below:
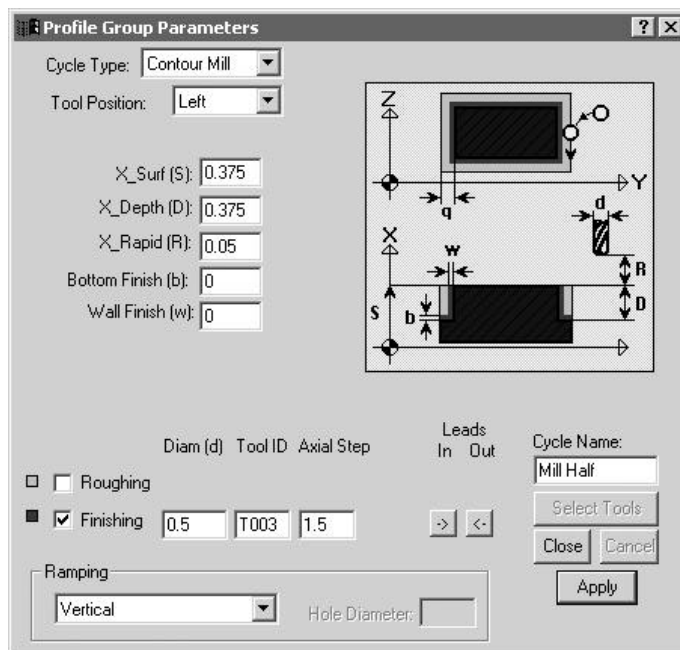
8. Assign the part feature to the flat drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
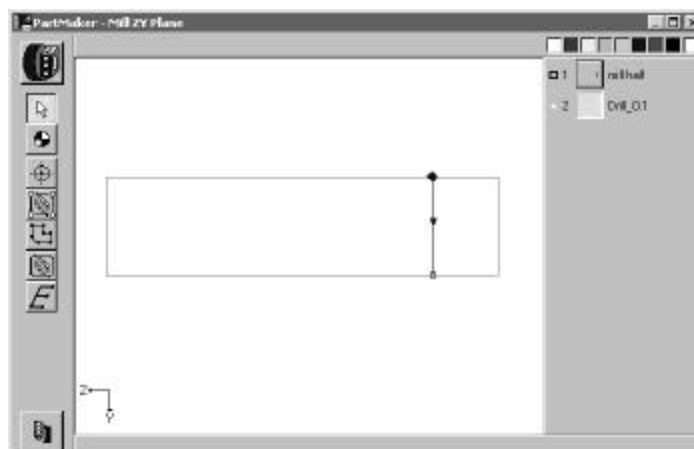
9. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
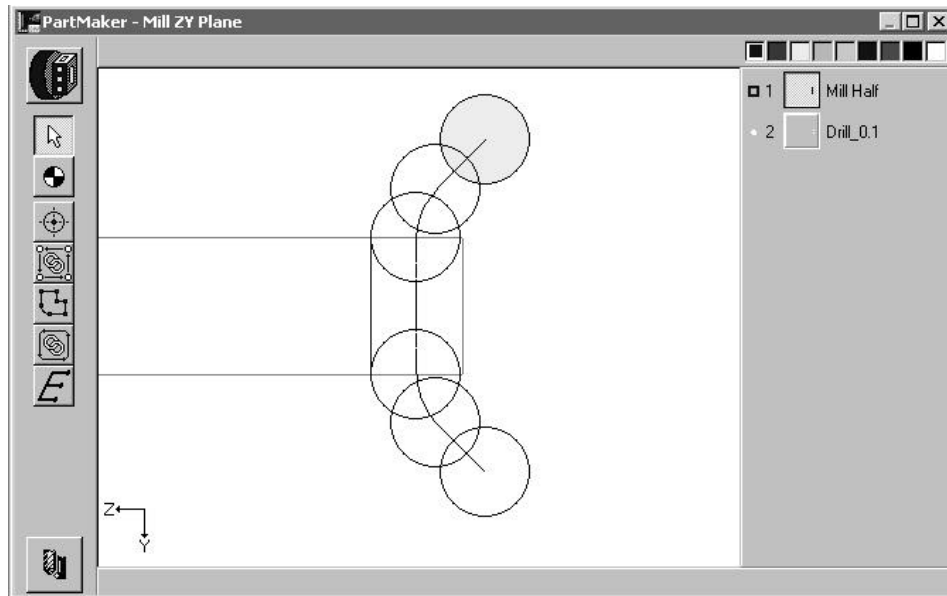

Choose **Hide Work Tool Path** to hide the tool path.

Next, you will create the off-center holes. To do so:

1. From the **Part Features** menu, choose **New Hole Group**
2. Uncheck the **Through Hole** Box
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .1
5. Enter a **Chamfer** of .01
6. Enter a **Nominal Depth (D)** of 0.2
7. Click the **Cycle** button.

Your completed **Hole Group Parameters** dialog should appear as shown below:
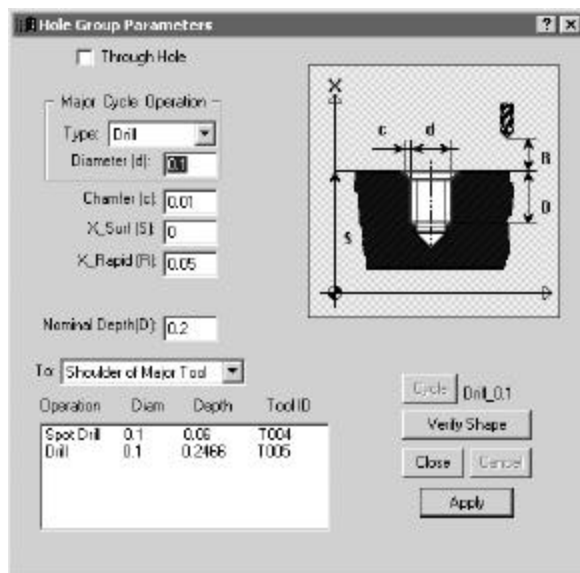
1. Assign the part feature to the circles by clicking the **Chain Circle** icon and clicking on the circle representing the holes to create the tool path. It should appear as shown below:
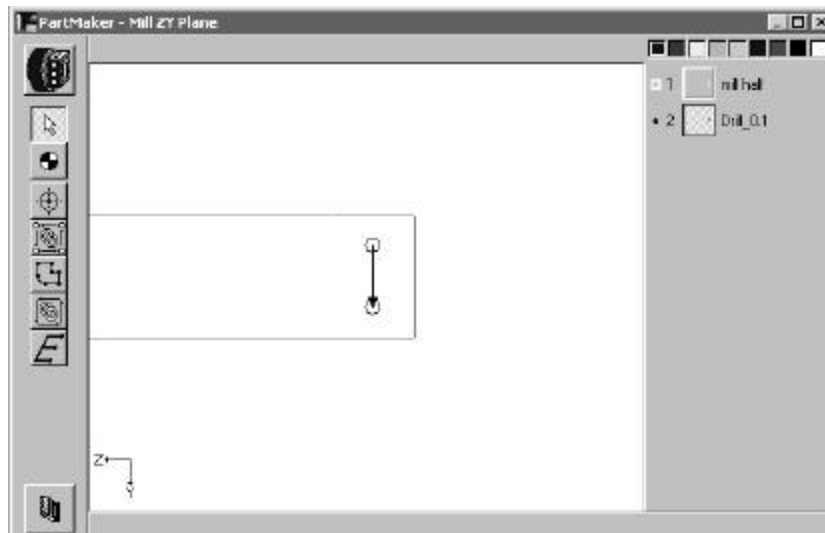

Example B: Creating a Saw Slot

In the example given below, you will mill a .050 wide slot with a 1-inch Diameter Slot Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZY Plane Window should have an orientation of **X Tool**)

Note: You will have needed to purchase the advanced tools option to create a Slotting Saw in your tools directory.

1. 1 Inch Diameter Slot Mill
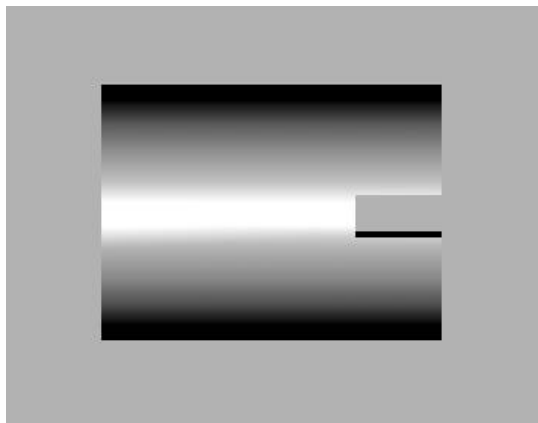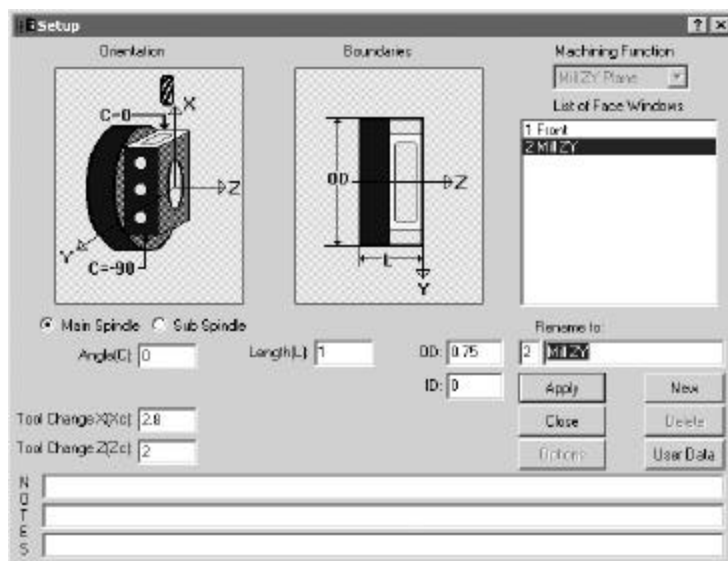

Features to be created:

Mill a .125 thick slot x ¼ deep along the face of the part.

Step 1. Create a Mill ZY Plane Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill ZY Plane Machining Function Face Window. To do so:
1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill ZY Plane**
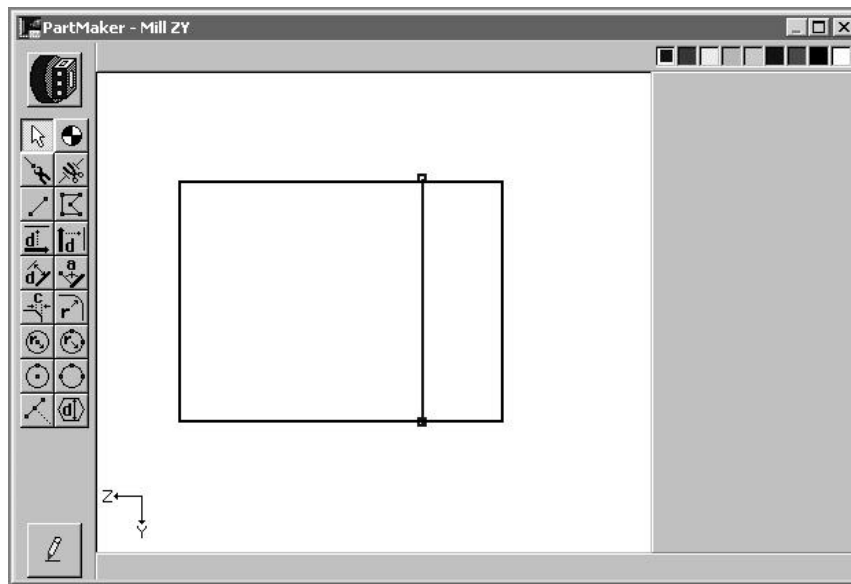4. Enter the setup parameters as shown below:

Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZY Plane Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a line that represents the .250 depth in Z for the end of the mill slot.

Your completed geometry and CAD Face Window should appear as shown below:

Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

Step 3. Create Part Features and assign tool paths

Start by entering the CAM mode in the Mill ZY Plane Window. First you will create the tool path to mill the square. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Contour Mill** under **Cycle Type**
3. Choose **Left** from **Tool Position** to indicate that climb milling will be performed
4. Enter an X_Depth of .4375 (X_Surf + ½ thickness of the Slotting Saw.)
5. Enter the Tool ID number of the Slotting Saw in the **Tool ID** box and then hit **Apply**.
6. In the **Cycle Name** field, type **Slotting Saw.**
7. Click on the **Close** Button

Your completed **Profile Group Parameters** dialog should appear as shown below:
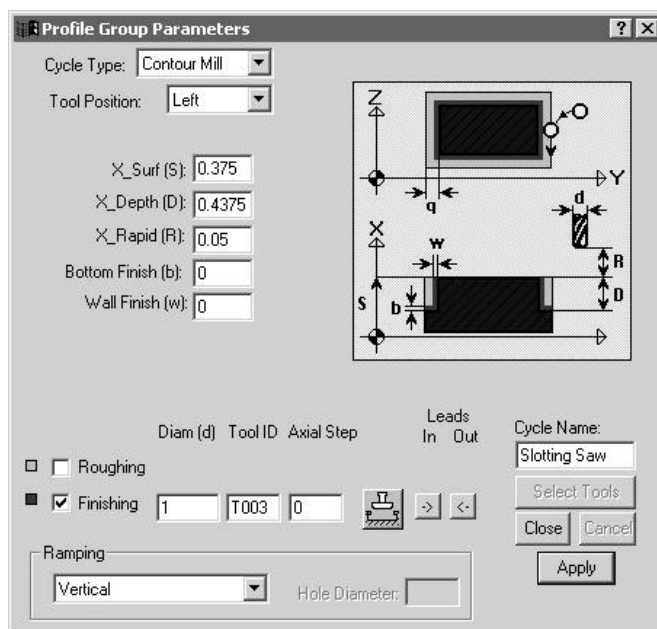
8. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
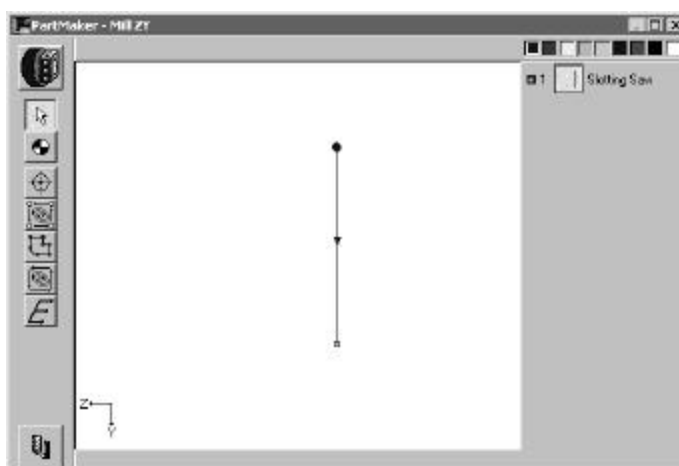
9. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
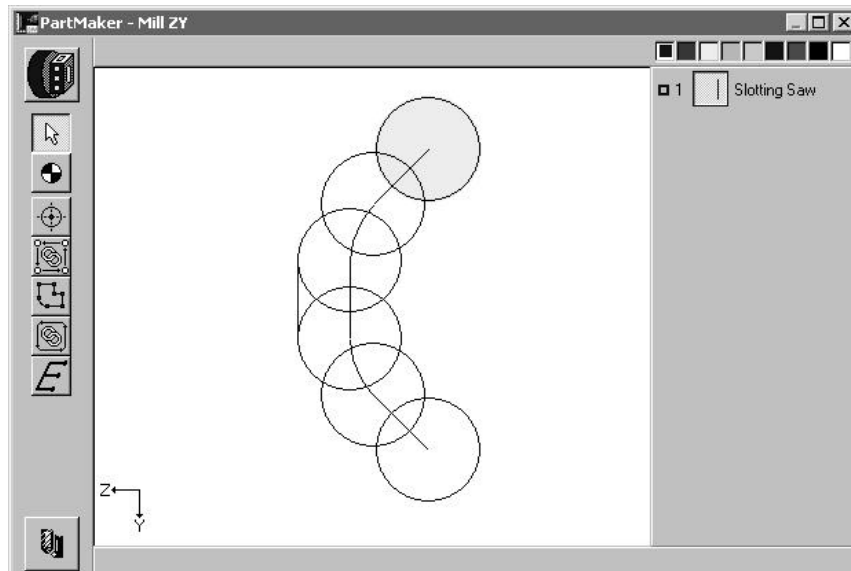
Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill ZX Plane

The Mill ZX Plane machining function is used to perform milling in the Z and X planes only. Such operations typically involve following a turning profile with an end mill (often ball nose) with the machine spindle in a locked position.

Programming of this machining function requires that your CNC Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.

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

Types of Features Created in Mill ZX Machining Function Window

The Mill ZX window uses an X-oriented "live" tool. There is no polar c-axis movement (the C-axis indexes and locks). This window is mainly used for doing key slots and operations that require simultaneous Z-axis and X-axis moves.

The features shown on the parts below are representative examples of features that can be created in the Mill ZX Window.
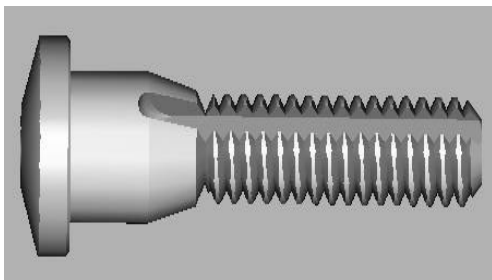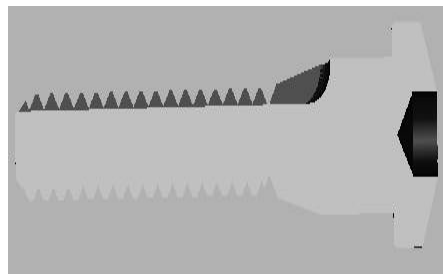

Figure 1. Milled Slot
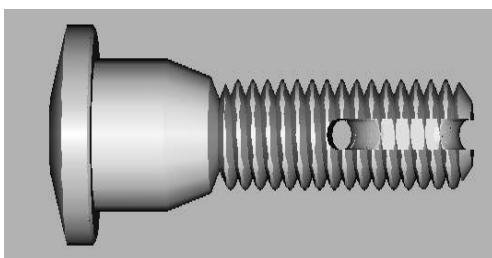

Figure 1a. (Cross section of milled slot)
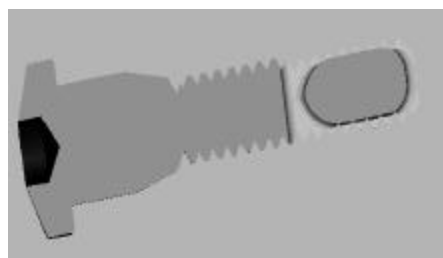

Figure 2. Milled slot

Figure2a. (Cross section of milled slot)

Part Features Dialogs Available in Mill ZX Machining Function

The Mill ZX machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing Left or Right from the Tool Position drop down menu.

Note: In the Mill ZX window you are only allowed to use a New Profile Group. A New Hole group and Pocket Milling are not accessible.
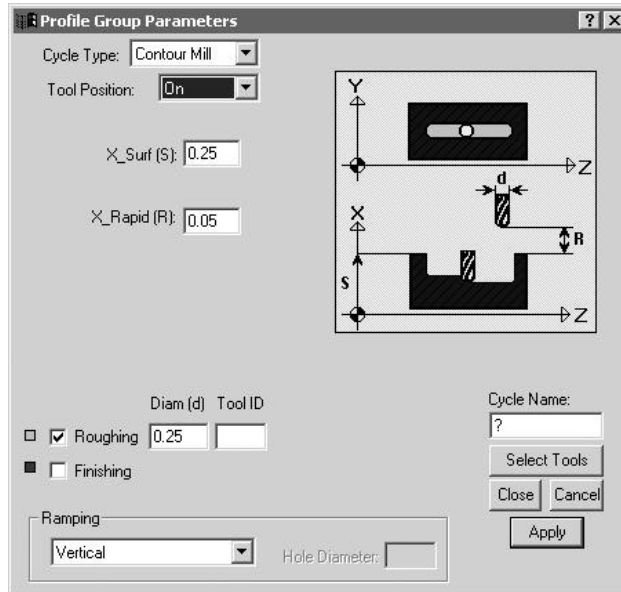
Cycle Type	Indicates the type of feature to be milled. In the Mill ZX Face Window this must always be set to Contour Mill.
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling).
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf

Graphics Icons – Machining Function Mill ZX Plane
The Machining Function Mill ZX Plane uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

Single Hole Icon Not applicable

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

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

Programming Example using the Mill ZX Plane Machining Function:

A practical programming application of the Mill ZX Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a .125 wide slot down the length of the part and radius out. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill ZX Plane Face Window should have an orientation of **X Tool**)

1.   .125 Inch Diameter End Mill
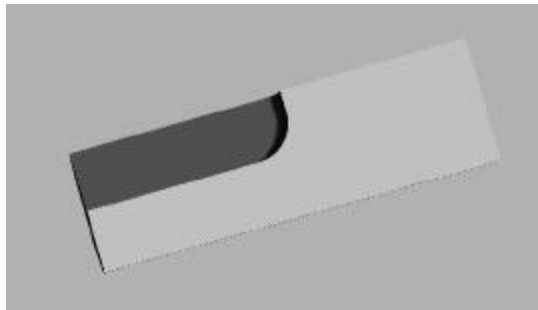
Features to be created:

Slot running down the length of the part then radius out in X.

Step 1. Create a Mill ZX Plane Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill ZX Plane Machining Function Face Window. To do so:
1.   Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2.   In the **Setup** dialog, click the **New** button to create a new Machining Function face Window
3.   From the **Machining Function** drop down menu, choose **Mill ZX Plane**
4.   Enter the setup parameters as shown below:
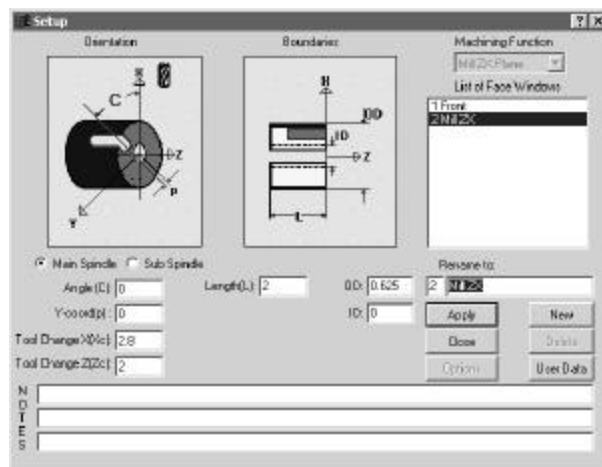
Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill ZX Plane Window. Draw the following features:

1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a horizontal line down the center of the part. Draw a vertical line 1 inch parallel to the face of the part.
4. Choose the **Fillet Icon** and enter a value of .2 and click on the inside of the intersection you just created to generate a radius.

Your completed geometry and CAD Face Window should appear as shown below:
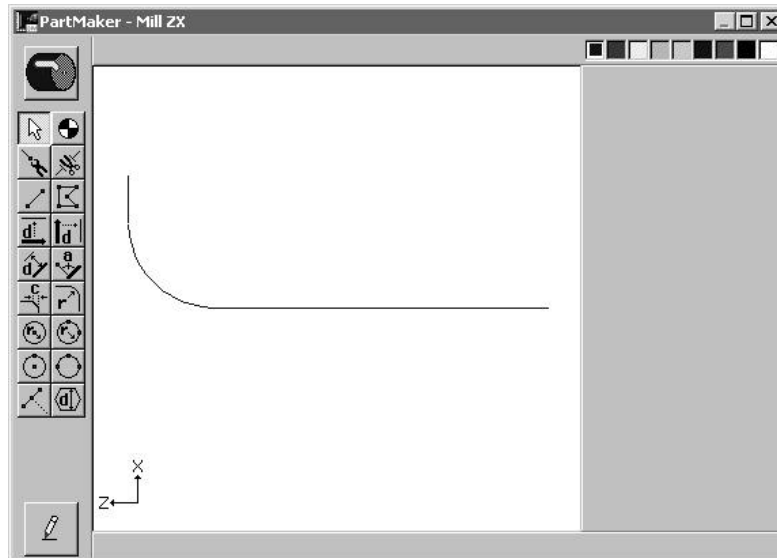
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

### *Step 3. Create Part Features and assign tool paths*

Start by entering the CAM mode in the Mill ZX Plane Window. First you will create the tool path to mill the slot. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Contour Mill** under **Cycle Type**
3. Choose **On** from **Tool Position** to indicate that we will be milling with the centerline of the tool.
4. Enter a **Z_Surf** of 0.3125
5. Enter a Tool Diameter of 0.125 and click **Select Tools**
6. In the **Cycle Name** field, type **Mill Slot**

Your completed **Profile Group Parameters** dialog should appear as shown below:
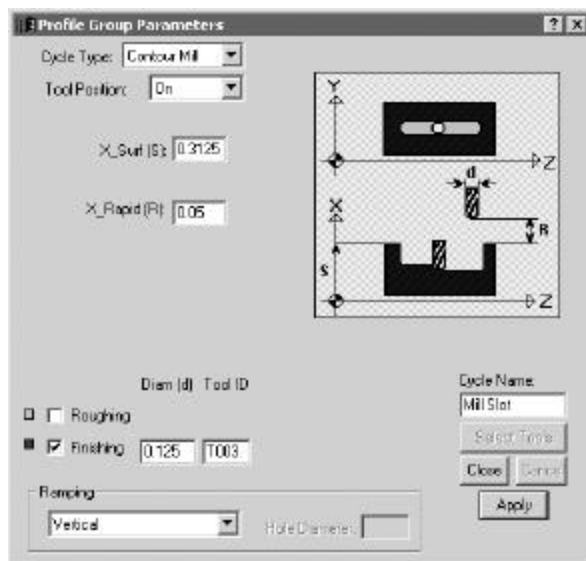
7. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
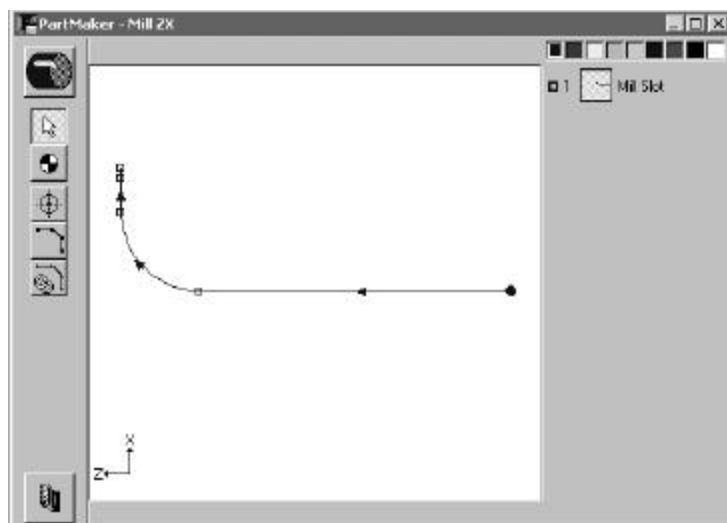
8. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
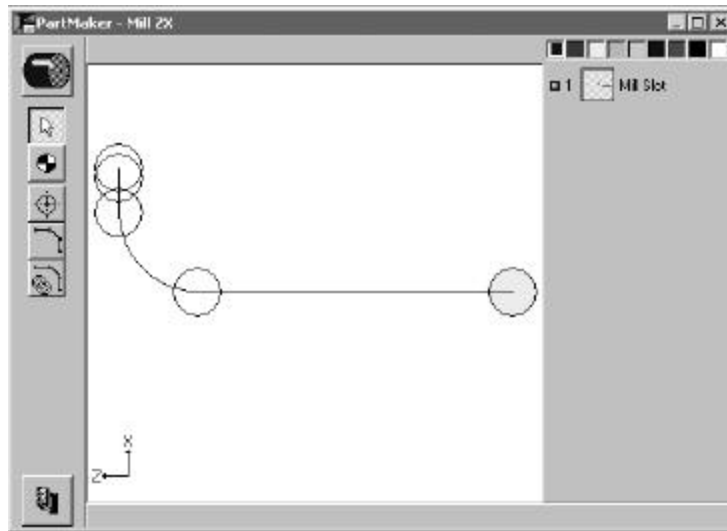

Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill Polygon

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

Programming of this machining function requires that your CNC Swiss-type lathe have a programmable Y-axis and that you have purchased the **PartMaker** Y-axis milling option.
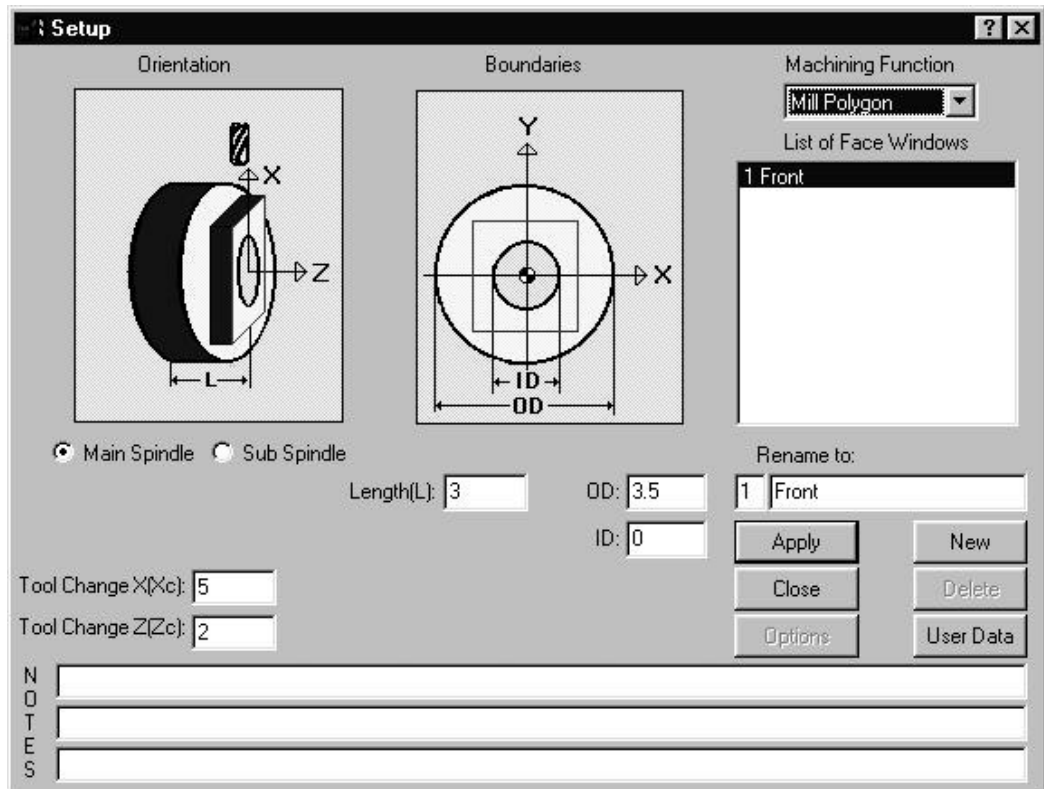
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill Polygon Machining Function Window

The Mill Polygon window uses an X-oriented "live" tool. There is no polar C-axis movement (the C-axis indexes and locks). This window is mainly used when creating multiple flats like a polygon, square or a hex. This operation does use bi-directional cutting therefore it cuts in one direction and indexes then cut in the opposite direction.

The features shown on the parts below are representative examples of features that can be created in the Mill Polygon Window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker SwissCAM. Where relevant, the locations of these files have been provided.
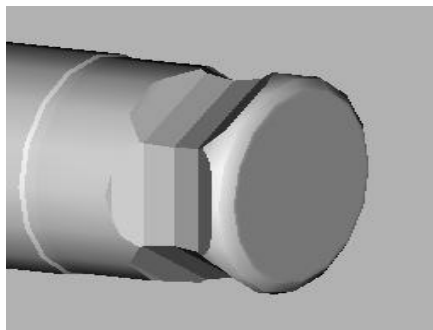

Figure 1.: Milled Hex
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_L\CitizenL.job, CitizenL.cdb, CitizenL.tdb)
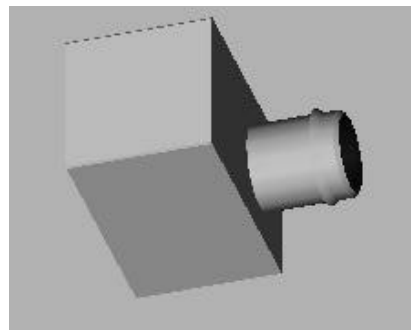

Figure 2.: Milled square
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_L\CitizenL-Yaxis.job, CitizenL-Yaxis.cdb, CitizenL-Yaxis.tdb)
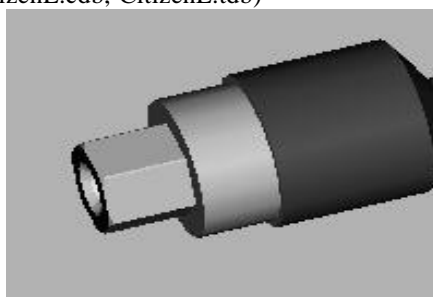

Figure 3.: Milled Hex
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_L\Citizen_L_Swiss.job, Citizen_L_Swiss.cdb, Citizen_L_Swiss.tdb)
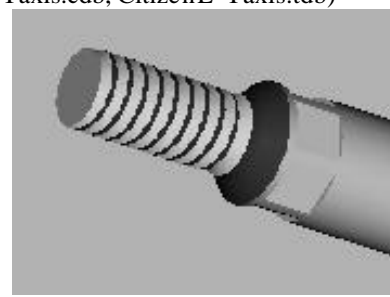

Figure 4.: Milled Polygon
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Hardinge ST\Hardinge-Caxis.job, Hardinge-Caxis.cdb, Hardinge-Caxis.tdb)

Part Features Dialogs Available in Mill Polygon Machining Function

The Mill Polygon machining function can be used to create flats only. Below, a Profile Group parameter window is shown. In the drop down menu you pick the tool edge you are going to use with respect to the part.

Note: In the Mill Polygon window you are only allowed to use New Profile Group. A New Hole Group is not accessible.
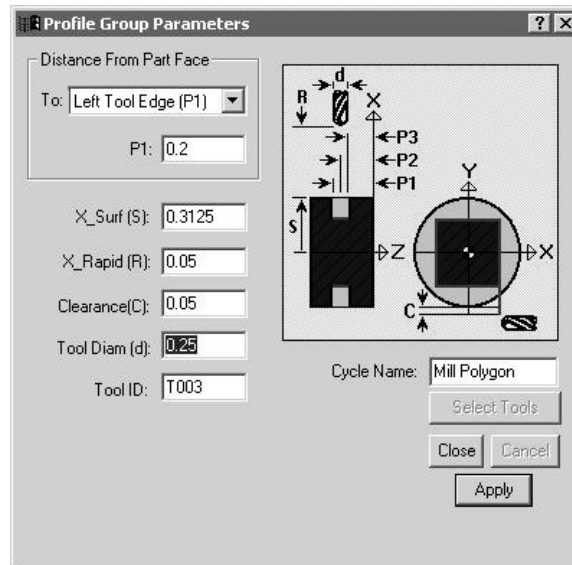
Distance From Part Face, To:	Indicates placement of tool with respect to the part face. The tool can be positioned with respect to the **Left Tool Edge (P1),** the **Tool Center (P2)** or the **Right Tool Edge (P3)**.
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf
Clearance(C)	Specifies the clearance amount the tool will position to past the barstock diameter.

Graphics Icons – Machining Function Mill Polygon

The machining functions for Mill Polygon uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

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

Programming Example using the Mill Polygon Machining Function: Milling a Hexagon

A practical programming application of the Mill Polygon Plane machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database.

In the example given below, you will mill a hexagon with a ¼ inch End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Polygon Face Window should have an orientation of **X Tool**)

1.   .25 Inch Diameter End Mill
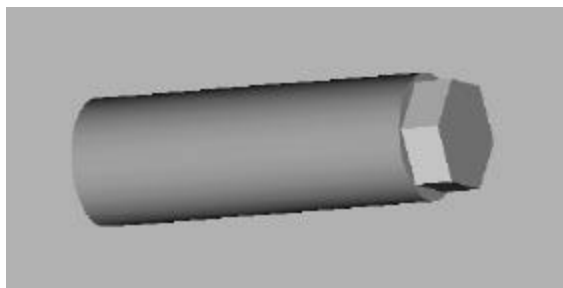
Features to be created:

Mill a hexagon with a .25 dia end mill

Step 1. Create a Mill Polygon Plane Machining Function Face Window

Note: When creating a milled feature in PartMaker Swis sCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill Polygon Plane Machining Function Face Window. To do so:
1.   Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2.   In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3.   From the **Machining Function** drop down menu, choose **Mill Polygon**
4.   Enter the setup parameters as shown below:
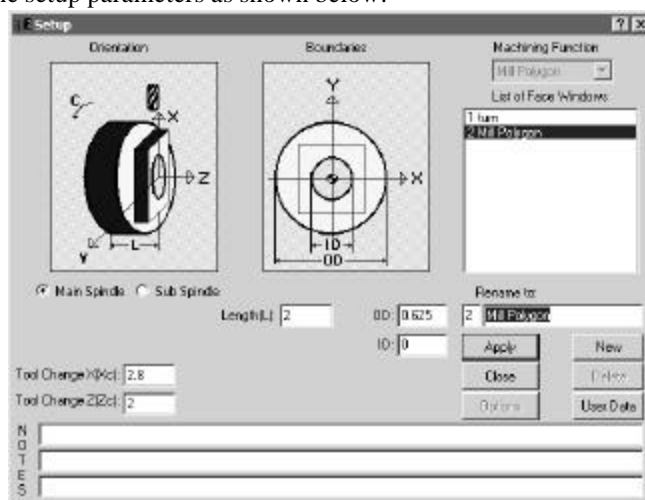
Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mo de in the Mill Polygon Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Choose the **Hexagon Icon** and enter .25 for the **Flat to Flat Distance**

Your completed geometry and CAD Face Window should appear as shown below:
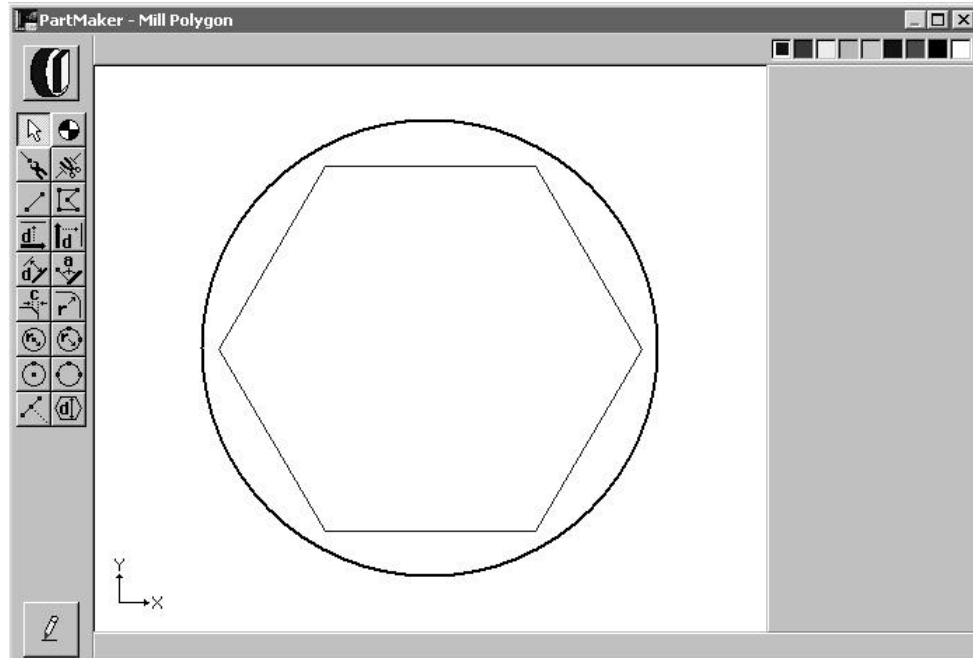
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

### *Step 3. Create Part Features and assign tool paths*

Start by entering the CAM mode in the Mill Polygon Window. First you will create the tool path to mill the hexagon. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Left Tool Edge (P1)** under **Distance From Part Face**
3. Enter 0.2 For the **P1** value
4. Enter a Tool Diameter of .25 and click **Select Tools**
5. In the **Cycle Name** field, type **Mill Polygon**

Your completed **Profile Group Parameters** dialog should appear as shown below:
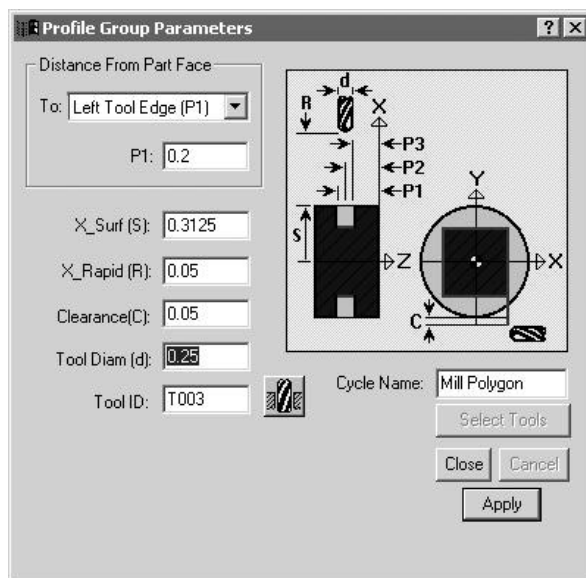
6. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
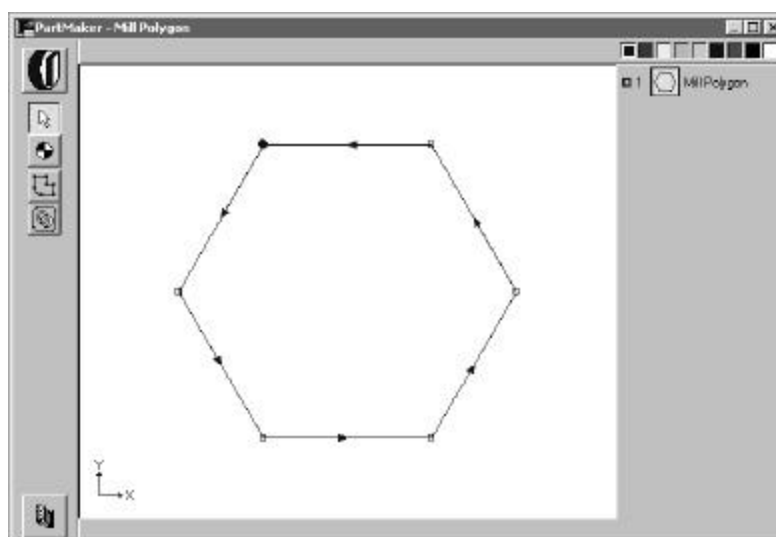
7. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
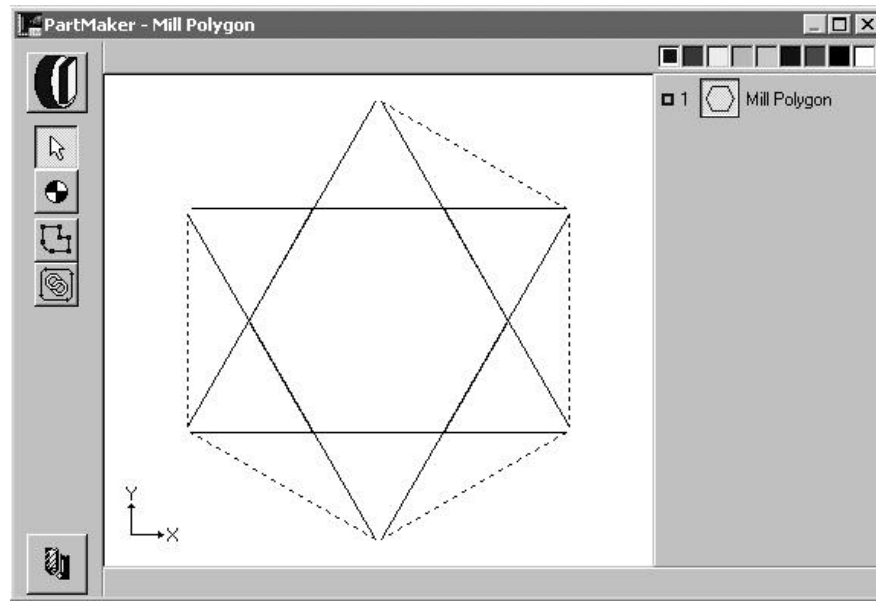
Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill End, Index

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

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

?? Holes in the end of the work created by Z-axis feed. This motion type supports off-center drilling on the end of the part, using C-axis indexing and X-axis positioning to locate the hole position.

?? Radial slots which are milled using C-axis indexing to position and hold the part while X-axis motion is used to cut the part. The tool would be a milling tool oriented along the Z-axis.
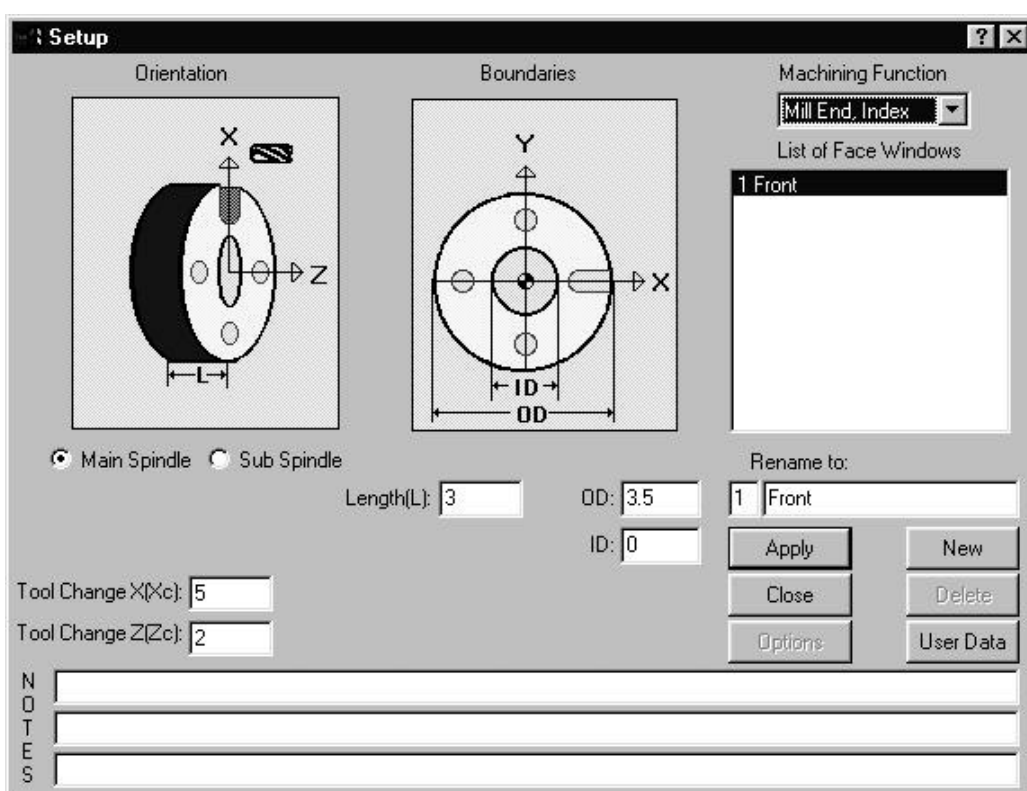
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill End Index Machining Function Window

The MILL END INDEX window uses a Z-oriented "live" tool. There is no polar c-axis movement (the C-axis indexes and locks). This window is mainly used for doing any number of holes or slots using a Z-oriented tool.

The features shown on the parts below are representative examples of features that can be created in the MILL END INDEX window
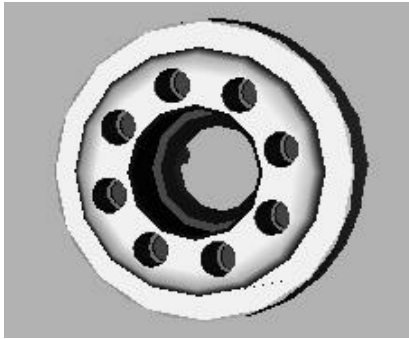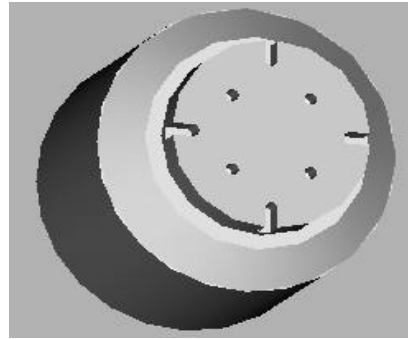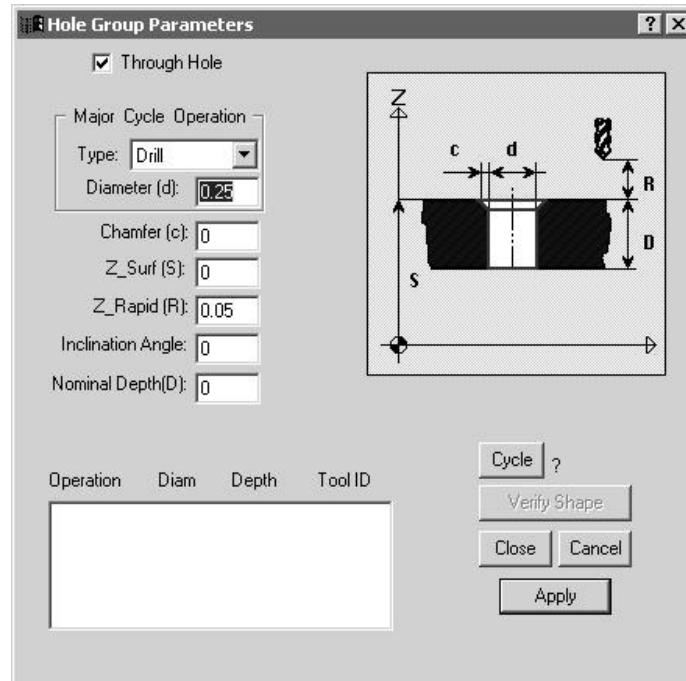

Figure 1. Face holes

Figure 2. Face holes and Slots

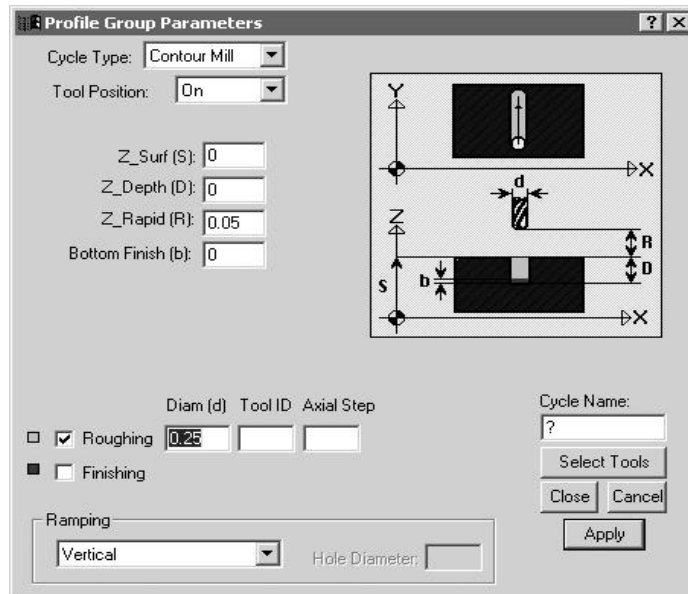### *Part Features Dialogs Available in Mill End Index Machining Function*

Both the **New Hole Group** and **New Profile Group** commands from the **Part** Features menu can be used in the **Mill End Index** Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in Z.

Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycle's database
Diameter (d)	Diameter of the tool, which is the major cycle operation
Chamfer(c)	Specifies how much chamfer per side is needed
Z_Surf (S)	Establishes your Z-zero work plane
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf
Inclination Angle	Specifies the angle of the tool when making inclined holes (you must have a "B" axis or an angled attachment.)
Nominal Depth (D)	The depth of the operation to be performed

The Mill End Index machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown.

Cycle Type	Indicates the type of feature to be milled is a **Contour Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, **On** (on centerline milling)
Z_Surf (S)	Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part
Z_Depth (D)	The depth of the operation to be performed
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Roughing/Fi nishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Does not apply to the **Mill End Index** machining function

Graphics Icons – Machining Functions Mill End Index

The Machining Functions Mill End Index uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.

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

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

Programming Example using the Mill End Index Machining Function: Bolt Hole Pattern on the Face

A practical programming application of the Mill End Index machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will drill six .125 diameter holes on the face of the part. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill End Index Face Window should have an orientation of **Z Tool**)

1. .125 Inch Diameter Drill
2. Spot Drill with a diameter larger than .125 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .125-inch drill and a spot drill.
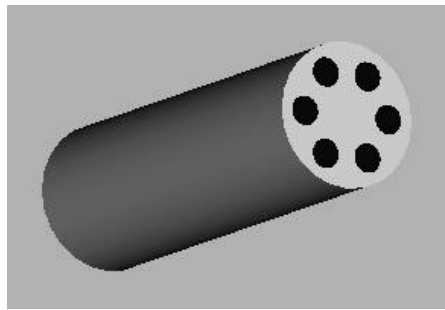
Features to be created:

*Spot Drill and Drill (6) holes with a .125 diameter drill*

Step 1. Create a Mill End Index Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill End Index Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill End Index**

4. Enter the setup parameters as shown below:
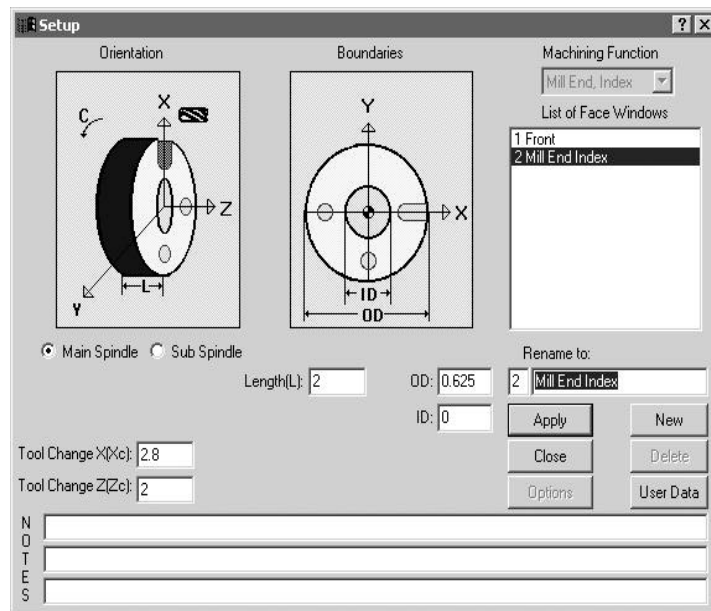
Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill End Index Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a .0625 inch radius circle with a center location at X<-.2> and Y<0>
4. Choose the **Rotate** command and make 5 copies and rotate about the X<0> and Y<0>.

Your completed geometry and CAD Face Window should appear as shown below:
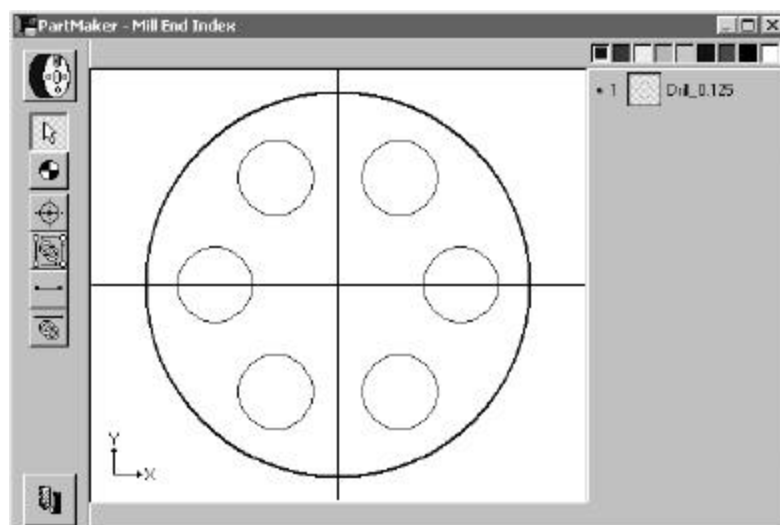
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

Step 3. Create Part Features and assign tool paths

Start by entering the CAM mode in the Mill End Index Window. First you will create the tool path to mill the square. To do so:

Next, you will create the 0.125 Boltholes, to do so:

1. From the **Part Features** menu, choose **New Hole Group**
2. Uncheck the **Through Hole** Box
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .125
5. Enter a **Chamfer** of .01
6. Enter a **Nominal Depth (D)** of 0.450
7. Click the **Cycle** button.

Your completed **Hole Group Parameters** dialog should appear as shown below:
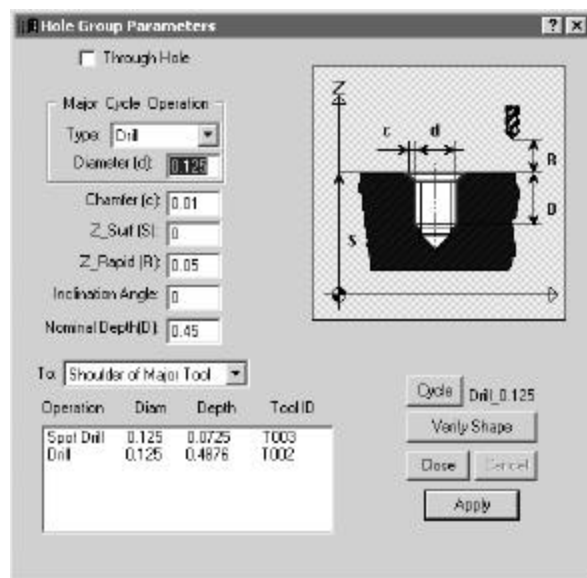
8. Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circle representing the hole to create the tool path. It should appear as shown below:
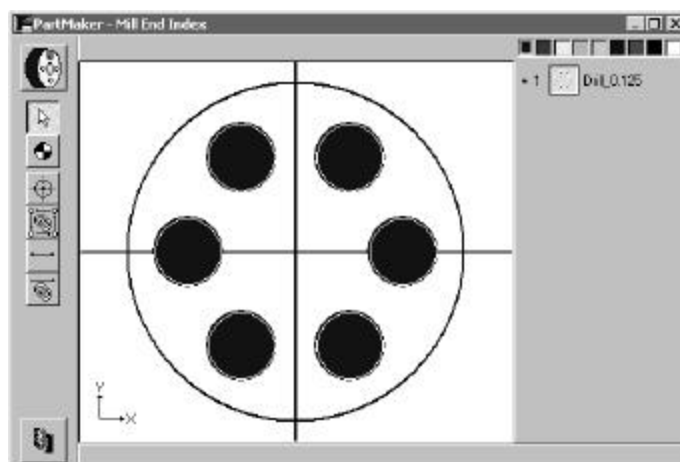
Machining Function: Mill Diam, Index

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

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

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

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

Types of Features Created in Mill Diam Index Machining Function Window

The Mill Diam Index window uses a X-oriented "live" tool. There is no polar c-axis movement (the c-axis indexes and locks). This window is mainly used for doing any number of holes or slots using a X-oriented tool.

The features shown on the parts below are representative examples of features that can be created in the Mill Diam Index window.
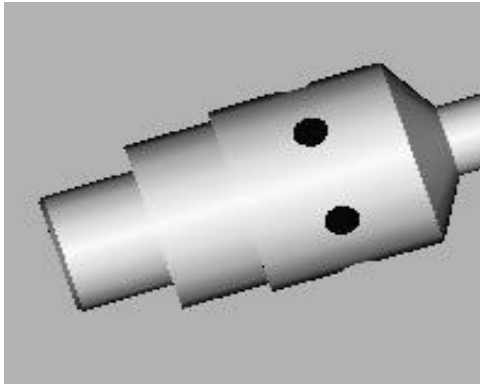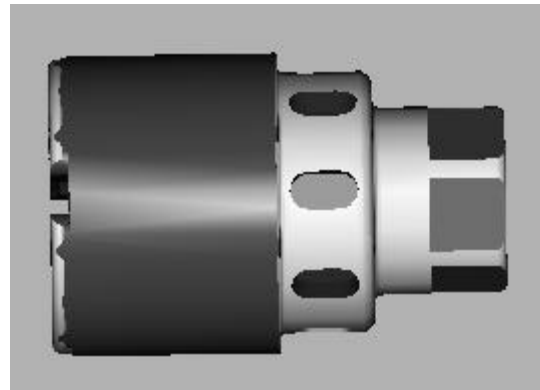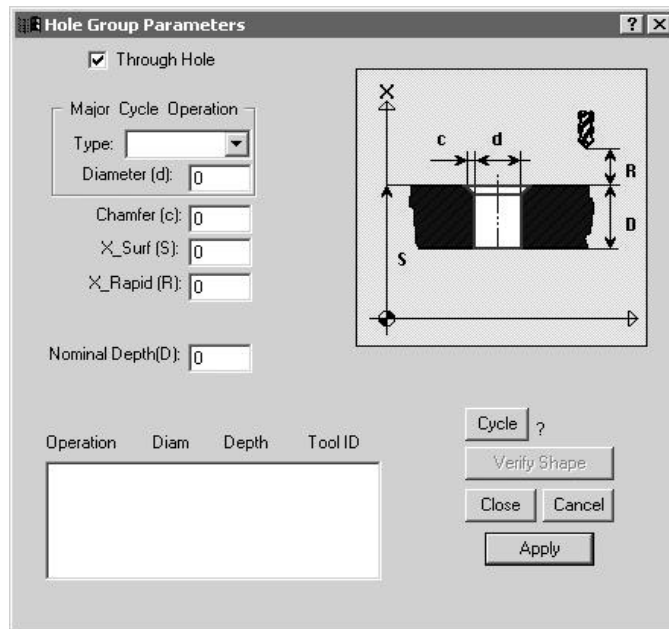

Figure 1. Cross holes

Figure 2. Cross slots

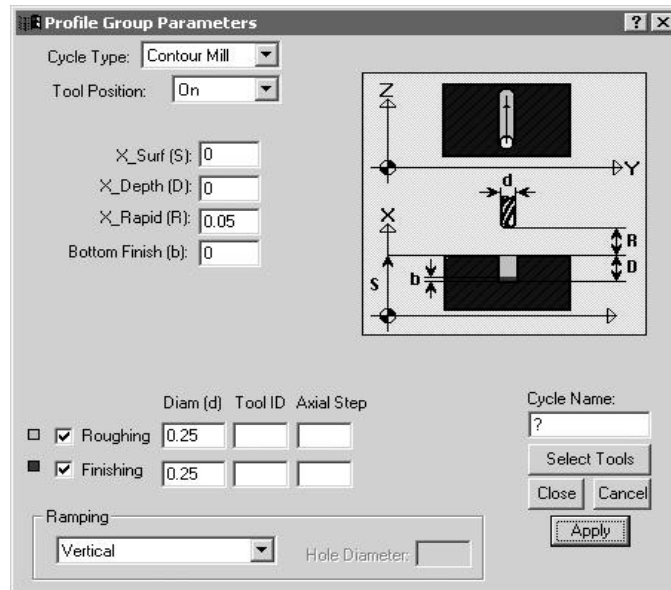### *Part Features Dialogs Available in Mill Diam Index Machining Function*

Both the **New Hole Group** and **New Profile Group** commands from the **Part** Features menu can be used in the Mill Diam Index Machining Function Face Window.

The Hole Group Parameters dialog will appear as shown below. Notice the orientation of the tool is in X.

Through Hole	Check this box if you are drilling through the part
Major Cycle Operation	Indicates the major cycle operation as specified in the cycle's database.
Diameter (d)	Diameter of the tool, which is the major cycle operation.
Chamfer(c)	Specifies how much chamfer per side is needed
X_Surf (S)	Establishes your X-zero work plane.
X_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your X_Surf.
Nominal Depth (D)	The depth of the operation to be performed

The Mill Diam,Index machining function can be used to create Contours only. Below, a Contour Mill cycle with tool motion on centerline is shown.

Cycle Type	Indicates the type of feature to be milled is a **Contour Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, **On** (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above your X_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Does not apply to the **Mill Diam, Index** machining function

Graphics Icons – Machining Functions for Mill Diam, Index

The Machining Functions for Mill Diam, Indexuses the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

Chain Circle Icon enables you to click an enclosed circle on the geometry and specify that the currently active Hole Group Symbol is also to be automatically applied at the centers of all the other circles (holes) of the same size that appear on the geometry.

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

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

Programming Example using the Mill Diameter Index Machining Function: Creating Cross holes

A practical programming application of the Mill Diameter Index machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will Drill six holes with a .125 diameter drill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Diameter Index Face Window should have an orientation of **X Tool**).
1. .125 Inch Diameter Drill
2. Spot Drill with a diameter larger than .125 inch

Also make sure you have cycle in PartMaker's Cycles Database created to include both a .125-inch drill and a spot drill.
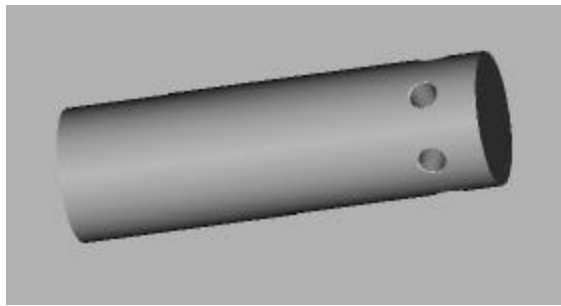
Features to be created:

Spot Drill and Drill six holes on diameter with a .125 diameter drill

Step 1. Create a Mill Diameter Index Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill Diam, Index Machining Function Face Window. To do so:
1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill Diam, Index**
4. Enter the setup parameters as shown below:
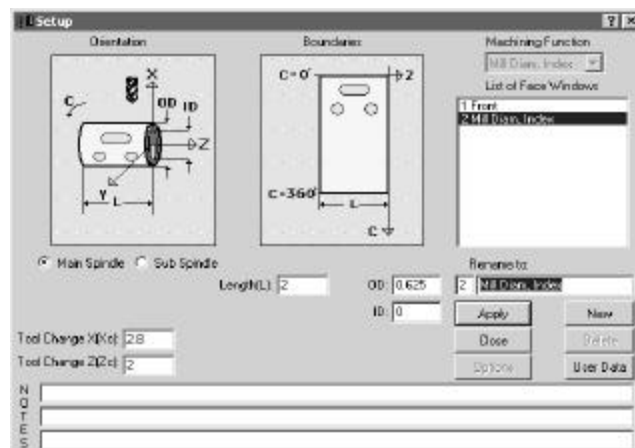
Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill Diam, Index Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a .0625-inch radius circle with a center location at Z<. 25. > and C<0>.
4. Click on the **Selection Icon** and click on the circle you just created.
5. Choose **Translate** from the **Edit Menu** and make 5 more copies, Z distance is 0 and C Distance of 60 degrees.

Your completed geometry and CAD Face Window should appear as shown below:
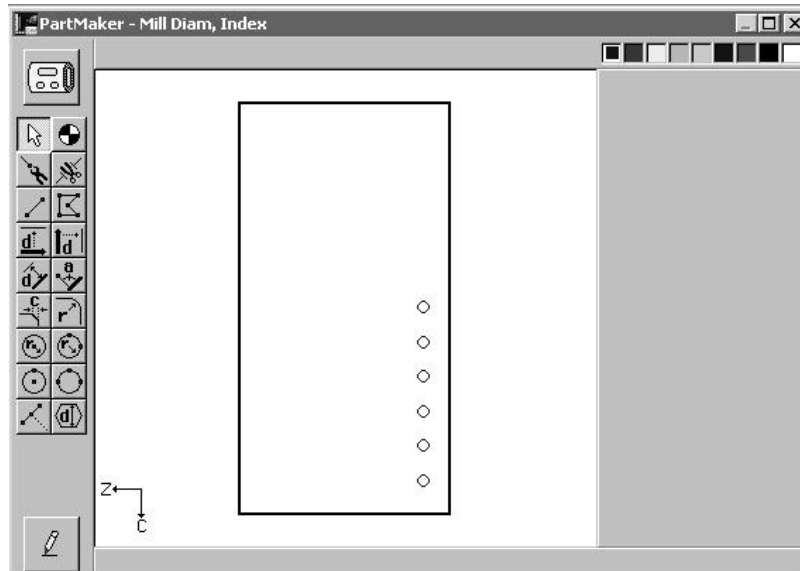
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

### *Step 3. Create Part Features and assign tool paths*

Start by entering the CAM mode in the Mill Diam, Index Window. First you will create the tool path to drill the cross holes. To do so:

1. From the **Part Features** menu, choose **New Hole Group**
2. Uncheck the **Through Hole** Box
3. Choose a Major Cycle Operation of **Type: Drill**
4. Enter a Major Cycle Operation **Diameter** of .125
5. Enter a **Chamfer** of .01
6. Enter a **Nominal Depth (D)** of 0.3125
7. Click the **Cycle** button.

Your completed **Hole Group Parameters** dialog should appear as shown below:
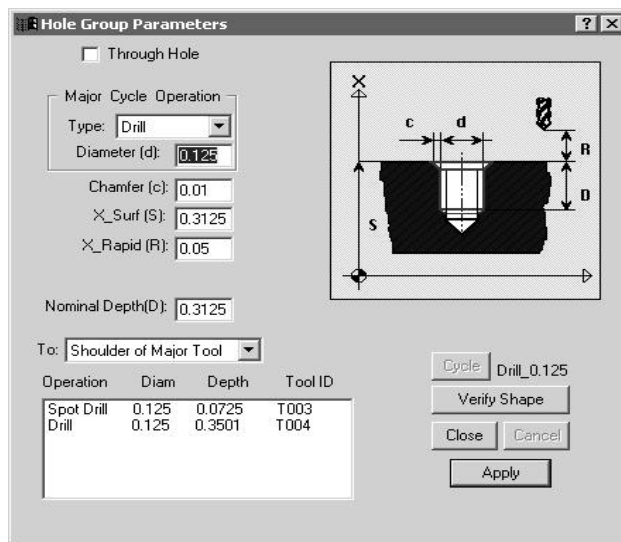
8. Assign the part feature to circle by clicking the **Chain Circle** icon and clicking on the circles representing the holes to create the tool path. It should appear as shown below:
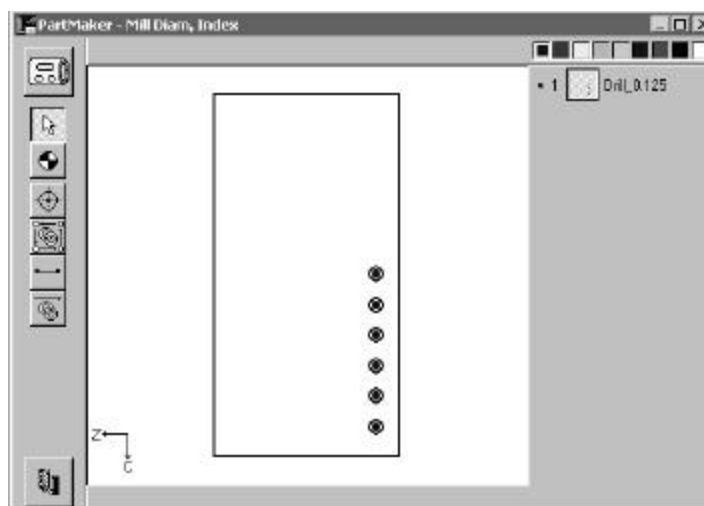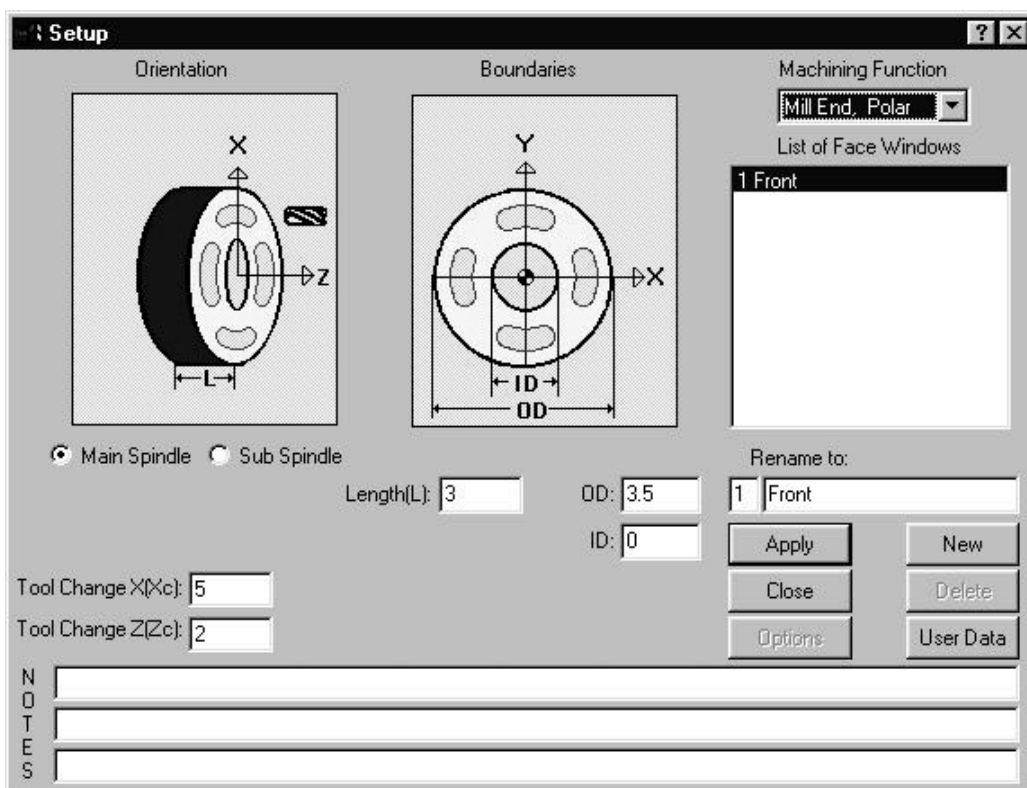
Machining Function: Mill End, Polar

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

The following rules apply to Mill End, Polar:

??   Tools must be oriented along the machine's Z-axis.

??   Shapes created must not pass, or if extended, pass through the center of rotation of the stock (X=0). For example, the radial slots that are programmable in Mill End, Index may NOT be programmed using Mill End, Polar. Also, any arc move that intersects the X=0 center is **NOT** allowed to be programmed using this machining function.

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

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

Types of Features Created in Mill End Polar Machining Function Window

The MILL END POLAR window uses a Z-oriented "live" tool. There is polar C-Axis movement (the C-Axis does not lock). This window is mainly used for milling profiles and pockets on the face of the part.

The features shown on the parts below are representative examples of features that can be created in the MILL END POLAR window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker SwissCAM. Where relevant, the locations of these files have been provided.
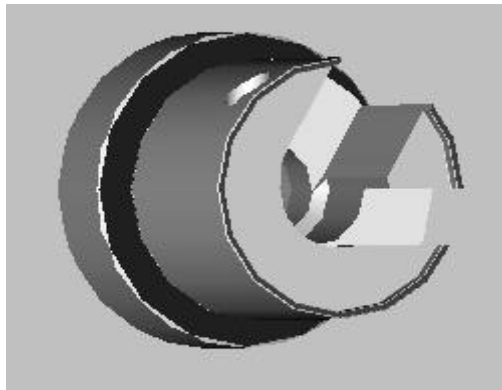

Figure 1. Milled slot
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Star_SV\StarSV.job, StarSV.cdb, StarSV.tdb)
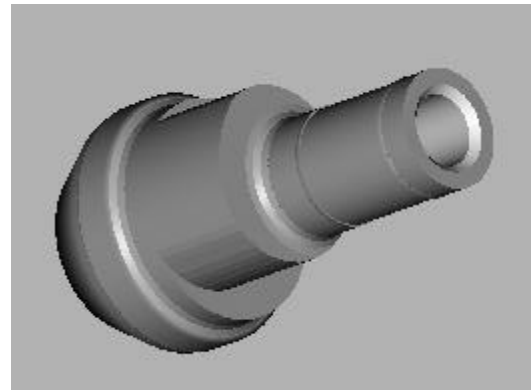

Figure 2. Milled Cam
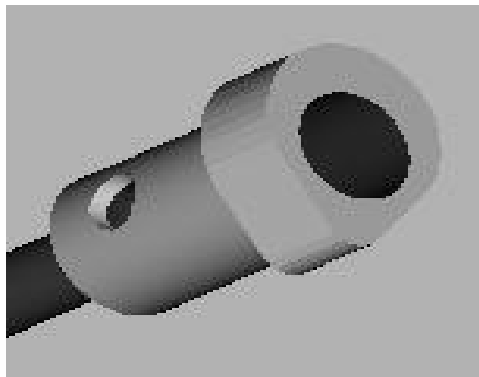(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Citizen_M\CitizenM.job, CitizenM.cdb, CitizenM.tdb)

Figure 3. Milled Cam
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Hardinge_ST\Hardinge-Caxis.job, Hardinge-Caxis.cdb, Hardinge-Caxis.tdb)
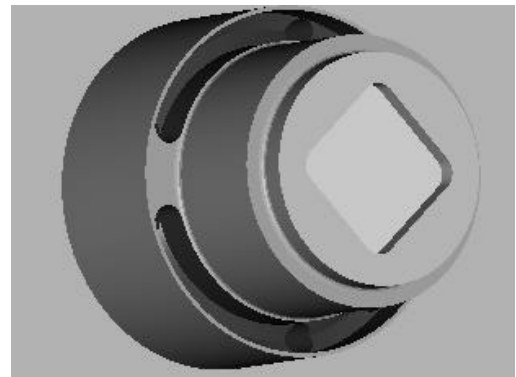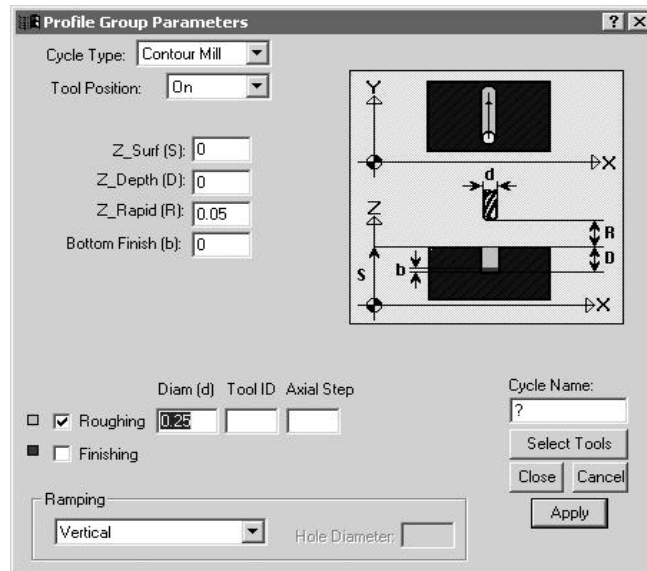

Figure 4. Milled Pocket on the face

Part Features Dialogs Available in Mill End Polar Machining Function

The Mill End Polar machining function can be used to create both Contours and Pockets. Below, a Contour Mill cycle with tool motion on centerline is shown. If you wish to create a pocket, choose Pocket Mill from Cycle Type drop down menu. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.

Note: In the Mill End Polar window you are only allowed to use a New Profile Group. New Hole group is not accessible

Cycle Type	Indicates the type of feature to be milled, either a **Contour Mill** or **Pocket Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling)
Z_Surf (S)	Establishes the Z-zero work plane i.e. a value 0 indicates the face of the part, a negative value indicates a Z-zero along the length of the part
Z_Depth (D)	The depth of the operation to be performed
Z_Rapid (R)	Specifies how much clearance the tool will rapid to in front of your Z_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Indicates the manner of entry of end mill into a feature, either **Vertical, Helical, Sloped** or **Sloped Down and Return**

Graphics Icons Available in the Mill End Polar Machining Function

The Machining Functions Mill End Polar, uses the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

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

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

Programming Example using the Mill End Polar Machining Function: Creating a Pocket on the Face of the Part

A practical programming application of the Mill End Polar machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

In the example given below, you will mill a Square Pocket .400 from side to side with a .08 diameter End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill End Polar Face Window should have an orientation of **Z Tool**)
1. .080 Diameter End Mill


Features to be created:

Mill a square pocket with a .080 dia. end mill

Step 1. Create a Mill End Polar Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill End Polar Machining Function Face Window. To do so:
1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill End, Polar**
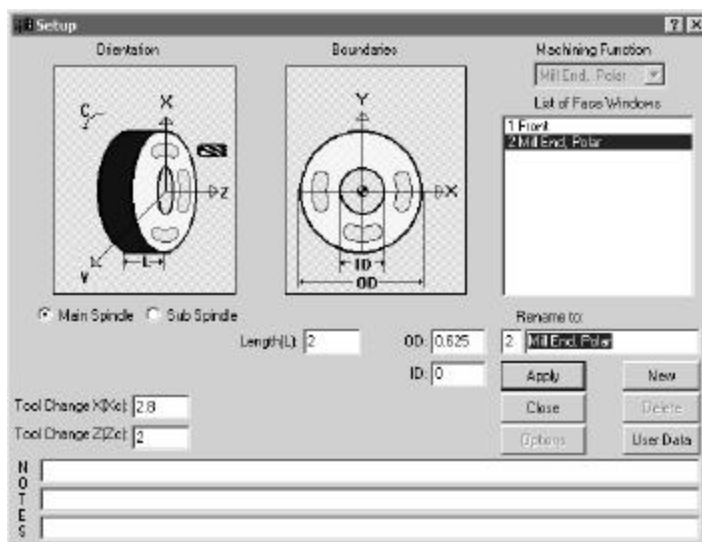
Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill End Polar Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a square that is .400 from flat to flat.
4. Draw a .0625 inch radius using the **Fillet Icon** to round off each corner

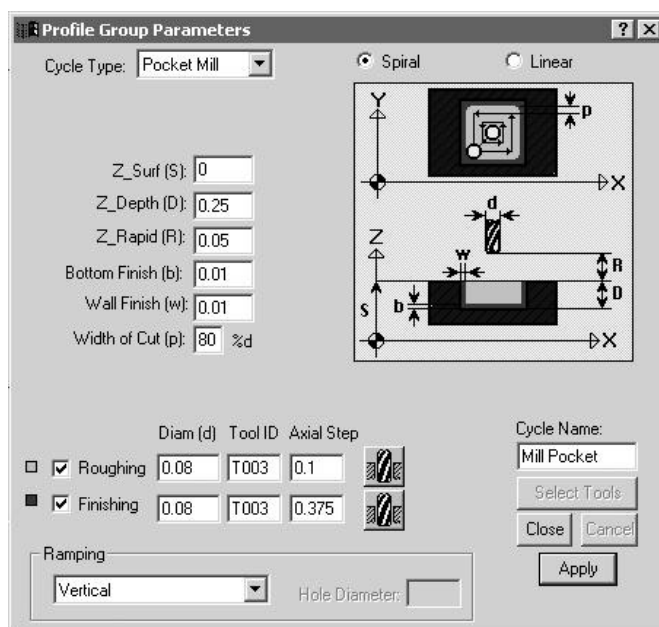Your completed geometry and CAD Face Window should appear as shown below:
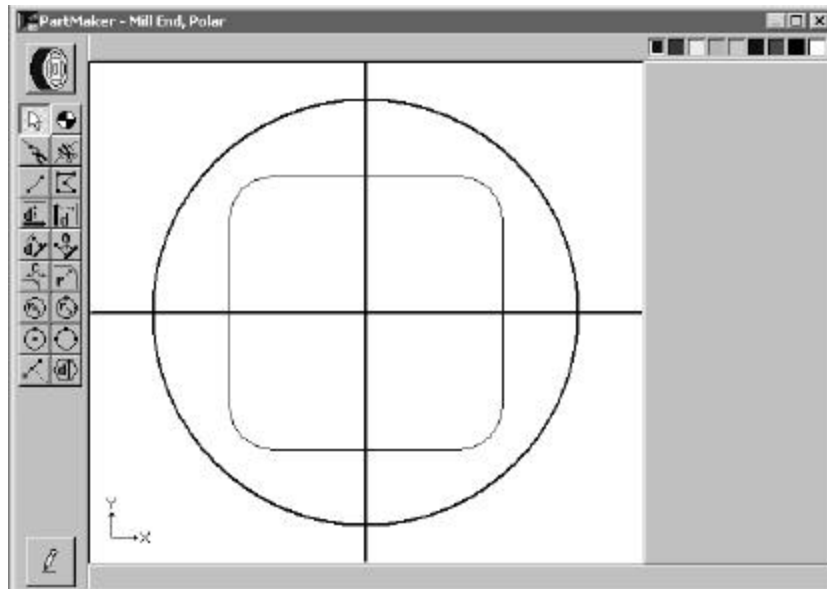
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active face Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

### *Step 3. Create Part Features and assign tool paths*

Start by entering the CAM mode in the Mill End, Polar Window. First you will create the tool path to mill the square. To do so:

1. From the **Part Features** menu, choose **New Profile Group**
2. Choose **Pocket Mill** under **Cycle Type**
3. Choose **Spiral** for the type of pocket to be performed
4. Enter a Z_Depth of 0.25
5. Enter a Tool Diameter of .08 for the Roughing and Finishing and click **Select Tools**
6. In the **Cycle Name** field, type **Mill Pocket**

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

7. Assign the part feature to the square drawn earlier by clicking the **Chain Geometry** icon and clicking on the geometry to create the tool path. It should appear as shown below:
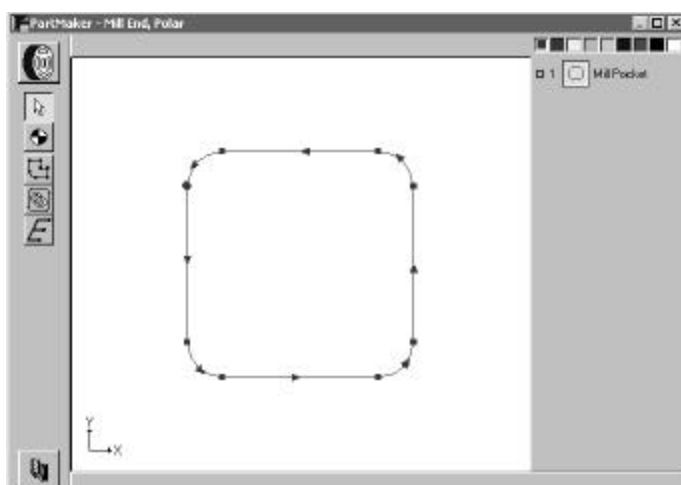
8. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:
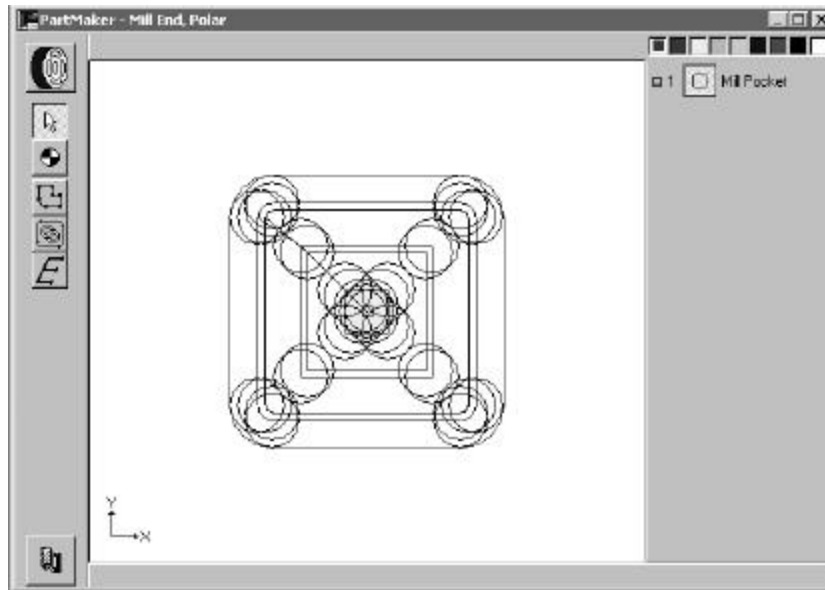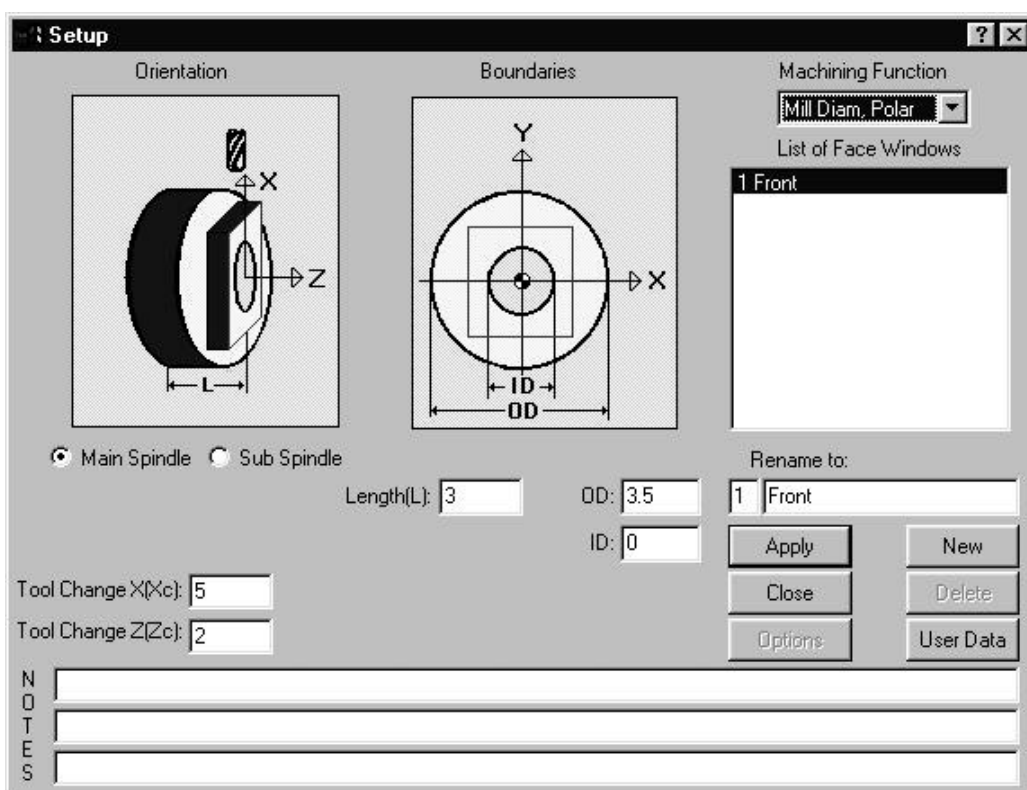

Choose **Hide Work Tool Path** to hide the tool path.

Machining Function: Mill Diam, Polar

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

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

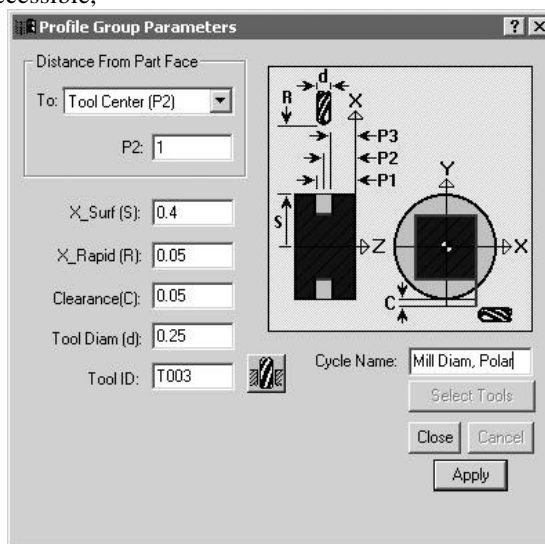|	
Length (L)	Length of the finished part
OD	Starting outside diameter of bar stock before machining
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill Diam Polar Machining Function Window

The Mill Diam Polar window uses an X-oriented "live" tool. There is polar C-axis movement (the C-axis does not lock). This window is most commonly used to machine flats when a Y-axis motion is not available. Because many Swiss Machines come equipped with a Y-axis and such tool motion is less precise, the Mill Diam Polar window is not a commonly used window because you can achieve better and more efficient results using the Mill Polygon Machining function.

Part Features Dialogs Available in Mill Diam Polar Machining Function

The Mill Diam Polar machining function can be used to create flats only. Below, a Profile Group parameter window is shown.

Note: In the Mill Diam Polar window you are only allowed to use New Profile Group. A New Hole Group is not accessible,

Distance From Part Face, To:	Indicates placement of tool with respect to the part face. The tool can be positioned with respect to the **Left Tool Edge (P1),** the **Tool Center (P2)** or the **Right Tool Edge (P3)**.
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Rapid (R)	Specifies how much clearance the tool will rapid above your X_Surf
Clearance(C)	Specifies the clearance amount the tool will position past the barstock diameter.

Graphics Icons – Machining Functions for Mill Diam Polar

The Machining Functions, Mill Diam Polar use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

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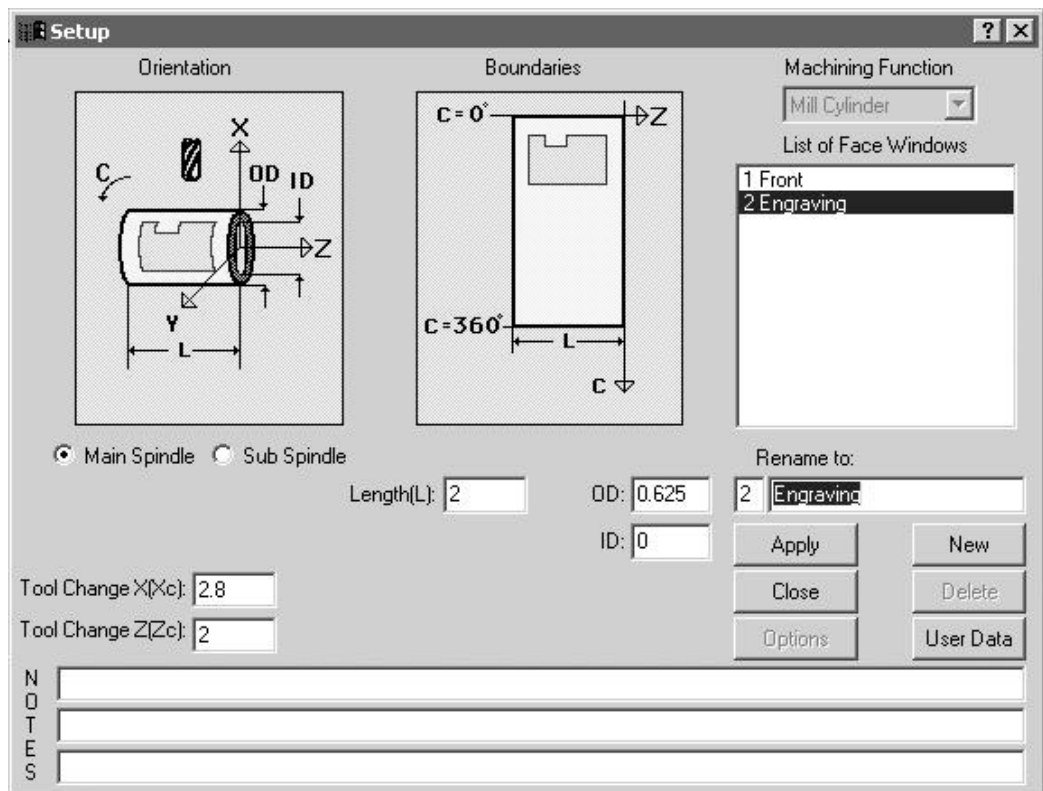
Machining Function: Mill Cylinder

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

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

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

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

Length (L)	Length of the finished part
OD	Diameter of the surface to be machined
ID	Starting inside diameter of bar stock before machining

Types of Features Created in Mill Cylinder Machining Function Window

The Mill Cylinder window uses an X-oriented "live" tool. There is polar c-axis movement (the c-axis does not lock). This window is mainly used for doing contoured pockets and or slots as well as engraving.

The features shown on the parts below are representative examples of features that can be created in the Mill Cylinder window. The *.job, *.cdb and *.tdb files used to create some of these parts can be found on in the in the Sample directory which was installed when you initially installed PartMaker SwissCAM. Where relevant, the locations of these files have been provided.

Figure 1. J-Slot
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Hardinge_ST\HardingeST.job, HardingeST.cdb, HardingeST.tdb)
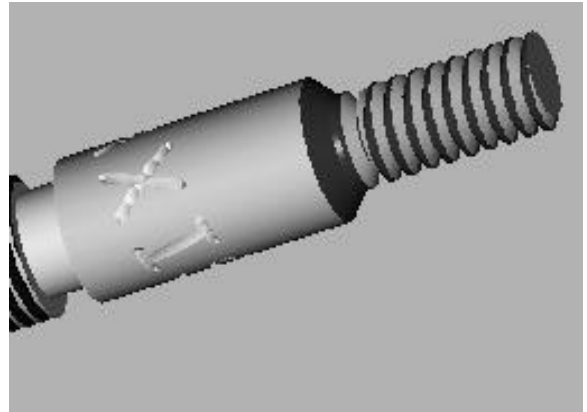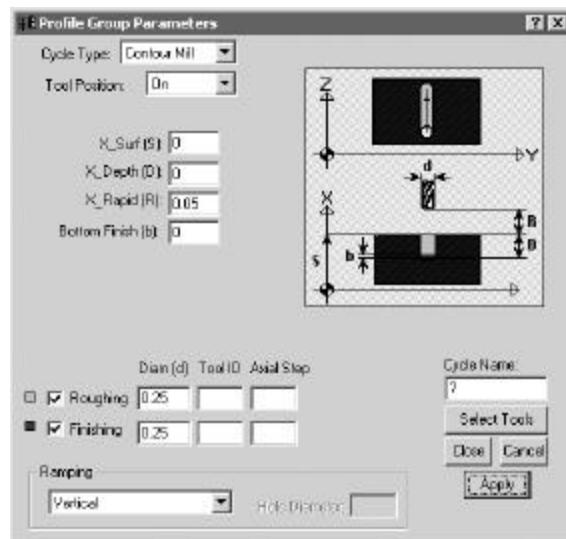

Figure 2. Engraving
(Relevant files: C:\PartMaker\Pm-Swiss\Samples\Hardinge_ST\Hardinge-Caxis.job, Hardinge-Caxis.cdb, Hardinge-Caxis.tdb)

Part Features Dialogs Available in Mill Cylinder Machining Function

The Mill Cylinder machining function can be used to create Contours, Pockets and Lettering on the outside diameter. Below, a Contour Mill cycle with tool motion on centerline is shown. Cutter compensation can also be used by choosing **Left** or **Right** from the Tool Position drop down menu.

Cycle Type	Indicates the type of feature to be milled, either a **Contour Mill** or **Pocket Mill**
Tool Position	For **Contour Mill** indicates the tool position during cutting, such as **Left** (climb milling), **Right** (conventional milling) or **On** (on centerline milling)
X_Surf (S)	Establishes the X-zero work plane, i.e. a value 0.25 indicates the X-zero work plane is .25 from the center of the part.
X_Depth (D)	The depth of the operation to be performed
X_Rapid (R)	Specifies how much clearance the tool will rapid to above your X_Surf.
Bottom Finish (b)	Amount of material left on the bottom of the feature for the finishing end mill to remove
Wall Finish (w)	Amount of material left on the side (or wall) of the feature for the finishing end mill to remove
Roughing/Finishing	Indicates whether a Roughing and/or Finishing Tool will be used
Ramping	Indicates the manner of entry of end mill into a feature, either **Vertical, Helical, Sloped** or **Sloped Down and Return**

Graphics Icons-Machining Functions for Mill Cylinder

The Machining Functions for Mill Cylinder use some or all of the following icons to allow you to assign part features to created geometry in order to create a tool path.

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

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

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

Programming Examples using the Mill Cylinder Machining Function:

A practical programming application of the Mill Cylinder machining function has been provided below. This example assumes you have a knowledge of creating part geometry in the PartMaker CAD mode as well as creating tools in PartMaker's Tool's Database and creating cycles in PartMaker's Cycles Database.

Example A: Engraving on the Outside Diameter

In the example given below, you will engrave the words "PartMaker" with a ¼ inch Spot Drill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Cylinder Face Window should have an orientation of **X Tool**)

1. .25 Inch Spot Drill

Features to be created:

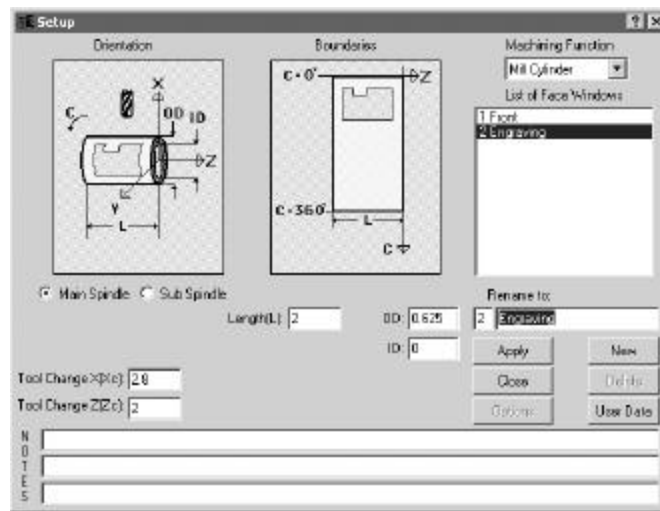*Engraving with the words "PARTMAKER" on the Outside Diameter*

Step 1. Create a Mill Cylinder Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill Cylinder Machining Function Face Window. To do so:

1. Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen
2. In the **Setup** dialog, click the **New** button create a new Machining Function face Window
3. From the **Machining Function** drop down menu, choose **Mill Cylinder**

4.   Enter the setup parameters as shown below:

Once the **Setup** dialog appears as above, click on **Close**

Step 2. Create Part Features

Note: When performing engraving it is not necessary to create part geometry.

Start by entering the CAM mode in the Mill Cylinder Window.
1.   Choose **Show Axes** from the View Menu
2.   Choose **Show Boundaries** from the View Menu
3.   From the **Part Features** menu, choose the **New Profile Group**.
4.   Choose **Contour Mill** under **Cycle Type**
5.   Choose **ON** from **Tool Position** to indicate that we will be working on centerline with the tool.
6.   Enter an X_Surf of 0.3125
7.   Enter an X_Depth of 0.025
8.   Enter the Tool ID number of the 0.25 diameter Spot Drill created and click on **Select Tools**
9.   When all of the necessary data has been entered click on **Close**
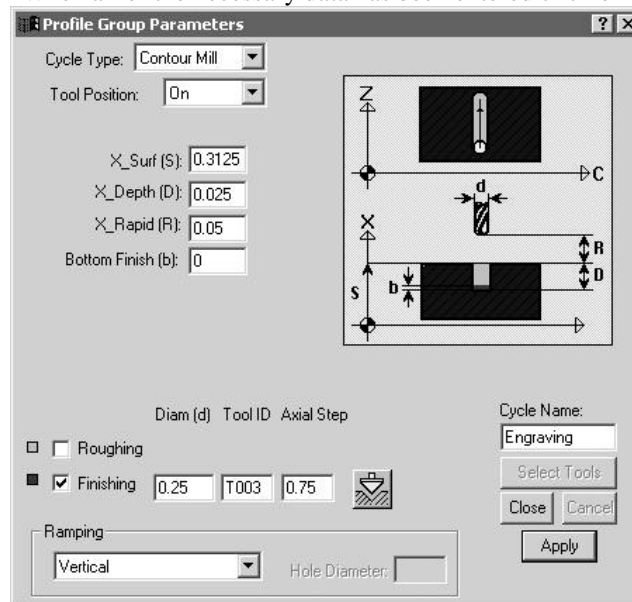
Step 3. Create Text

1. Click on the **Engraving Icon** and you will get the following dialog box shown below.
2. Type the word "Partmaker" in the box shown below then click on **OK**.
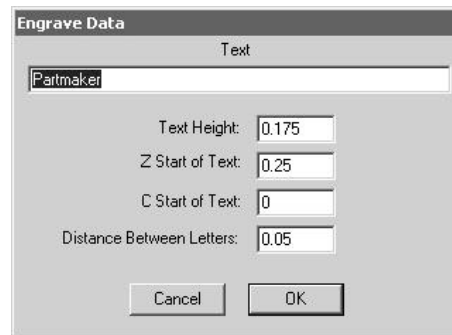
3. Drag a window around the text "PartMaker" and choose the **Rotate** command under **Edit** menu you will get the following dialog box shown below.
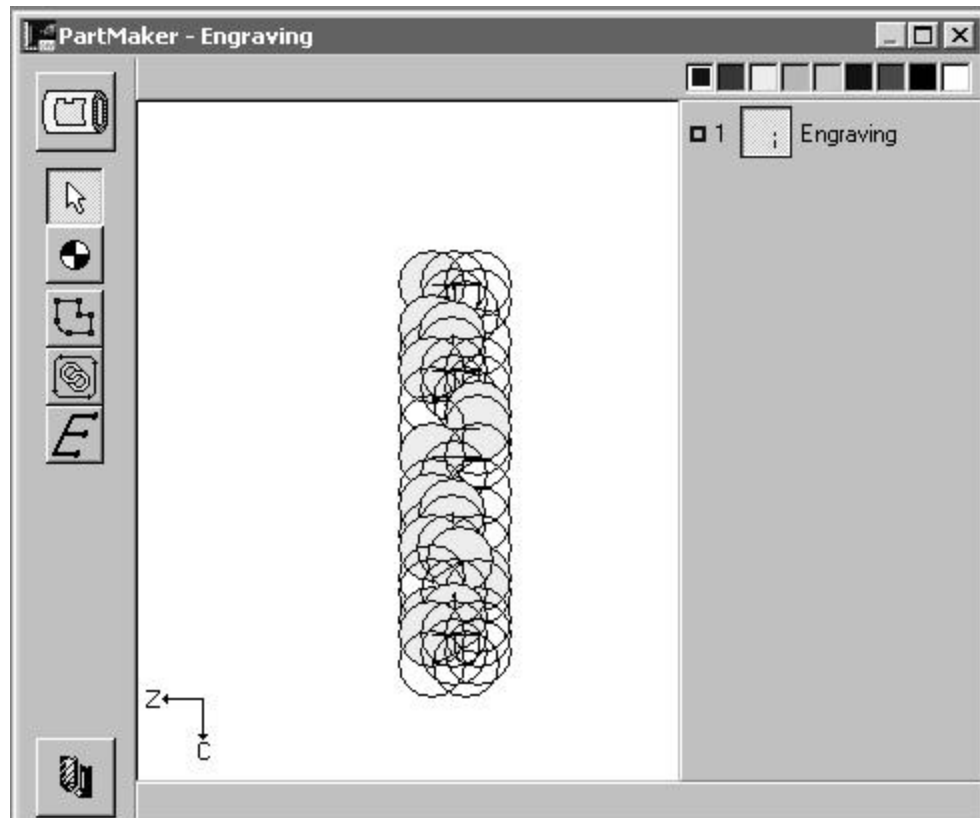4. Fill in all the necessary information from the dialog box below.
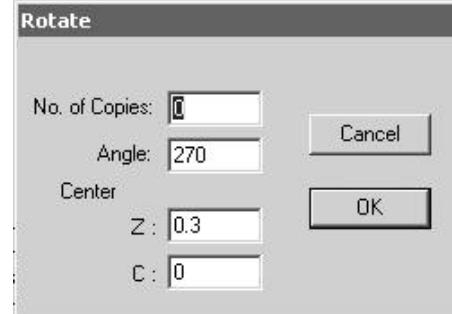
5. When all of the necessary data has been entered click on **OK**

Your completed geometry should appear as shown below:
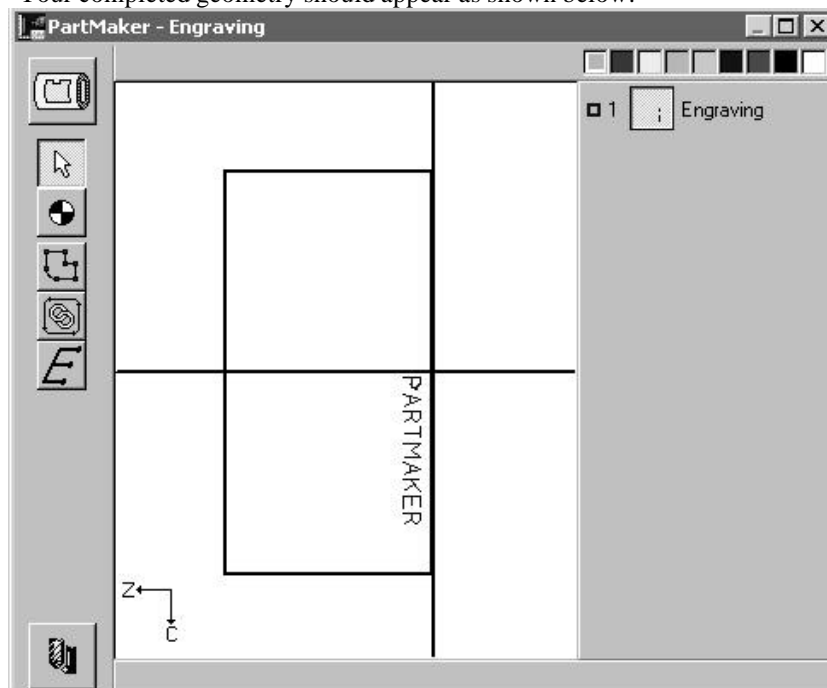
1. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:


Choose **Hide Work Tool Path** to hide the tool path.

Example B: Milling a Contoured Pocket

In the example given below, you will mill a contoured pocket on the diameter with a 1/8-inch End Mill. If you wish to complete this exercise on your own, make sure you have the following tools created in your PartMaker Tools Database (all tools used in the Mill Cylinder Face Window should have an orientation of **X Tool**)

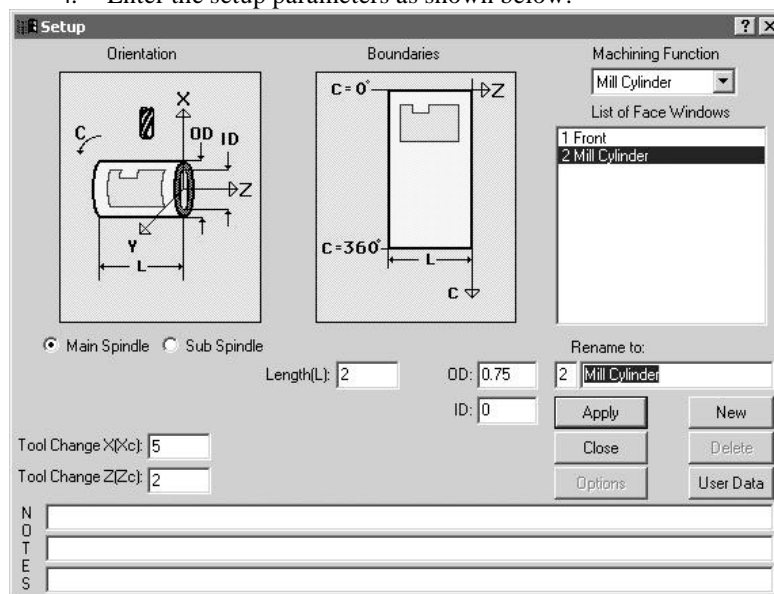   1.   .125 Inch Diameter End Mill
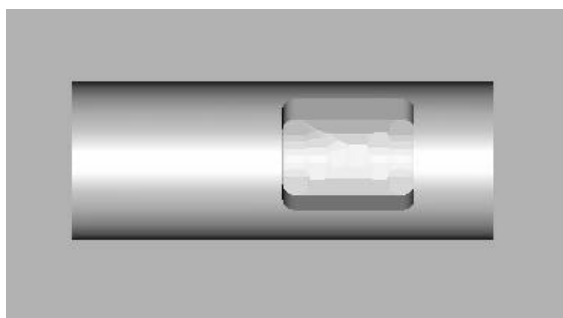
Features to be created:

Mill a Contoured Pocket with a .125 diameter End Mill.

Step 1. Create a Mill Cylinder Machining Function Face Window

Note: When creating a milled feature in PartMaker SwissCAM, you must always start with a turning operation. Thus you must have a face window of machining function Turn created and at least one Turning part feature and tool path created (such as a facing operation).

Start by creating a Mill Cylinder Machining Function Face Window. To do so:

      1.   Enter the **Setup** dialog by choosing **Setup** from the **View** menu or by clicking on the **Setup** dialog icon in the upper left hand corner of the screen

      2.   In the **Setup** dialog, click the **New** button create a new Machining Function face Window

      3.   From the **Machining Function** drop down menu, choose **Mill Cylinder**

   4.   Enter the setup parameters as shown below:

Step 2. Create Part Geometry

Start by entering the CAD mode in the Mill Cylinder Window. Draw the following features:
1. Choose **Show Axes** from the View Menu
2. Choose **Show Boundaries** from the View Menu
3. Draw a ½ square with .125 radius corners anywhere in the boundaries.

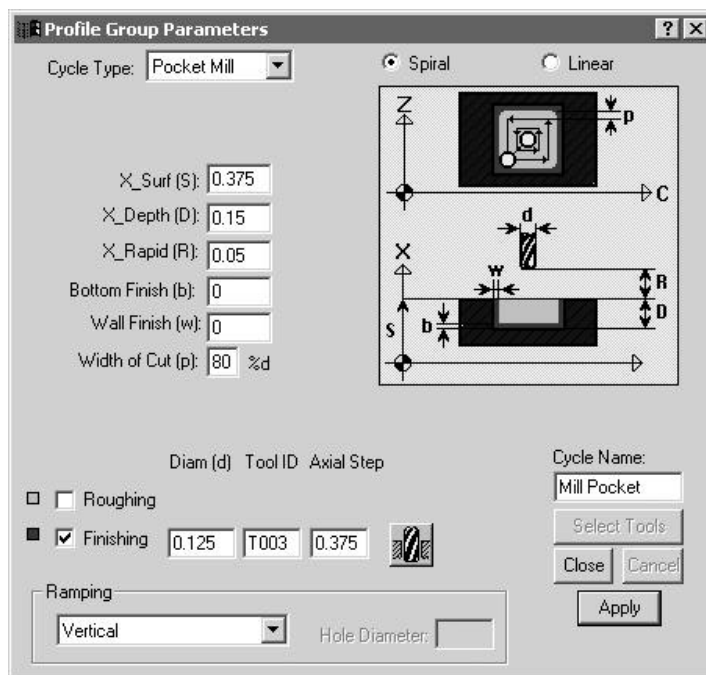Your completed geometry and CAD Face Window should appear as shown below:
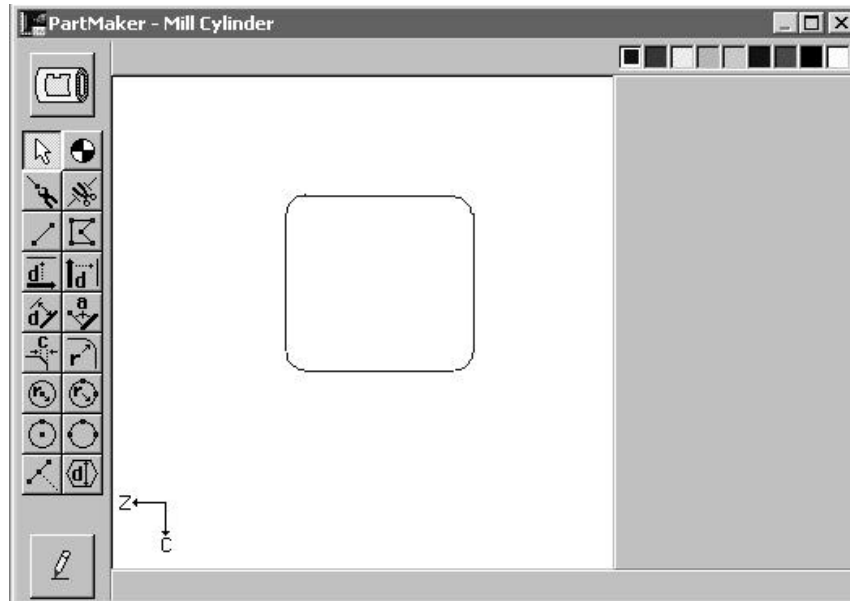
Note: You can always go to a different Face Window in your *.job file by selecting from the **Window** menu. Additionally, you can toggle between different Face Windows within the active Window by choosing **Show Windows Icons** from the **View** menu and double clicking on the icon representing the Window you want to view.

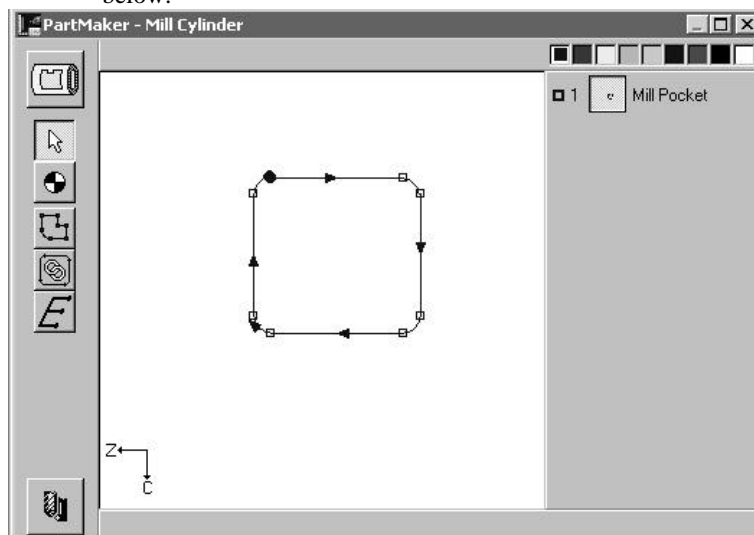### *Step 3. Create Part Features and assign tool paths*

Start by entering the CAM mode in the Mill Cylinder Window. First you will create the tool path
to mill the square. To do so:

1.   From the **Part Features** menu, choose **New Profile Group**
2.   Choose **Pocket Mill** under **Cycle Type**
3.   Enter a X_Depth of 0.15
4.   Enter a Tool Diameter of 0.125 and click **Select Tools**
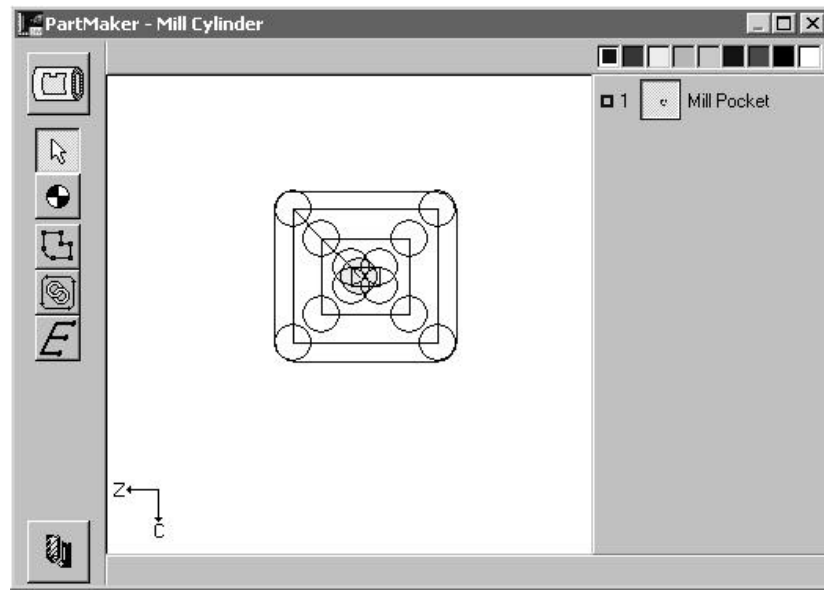5.   In the **Cycle Name** field, type **Mill Pocket**

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

6.   Assign the part feature to the square drawn earlier by clicking the **Chain Geometry**
     icon and clicking on the geometry to create the tool path. It should appear as shown
     below:

7. From the **Part** Features menu, choose **Verify Tool Path** to see the tool path as shown below:

Choose **Hide Work Tool Path** to hide the tool path.