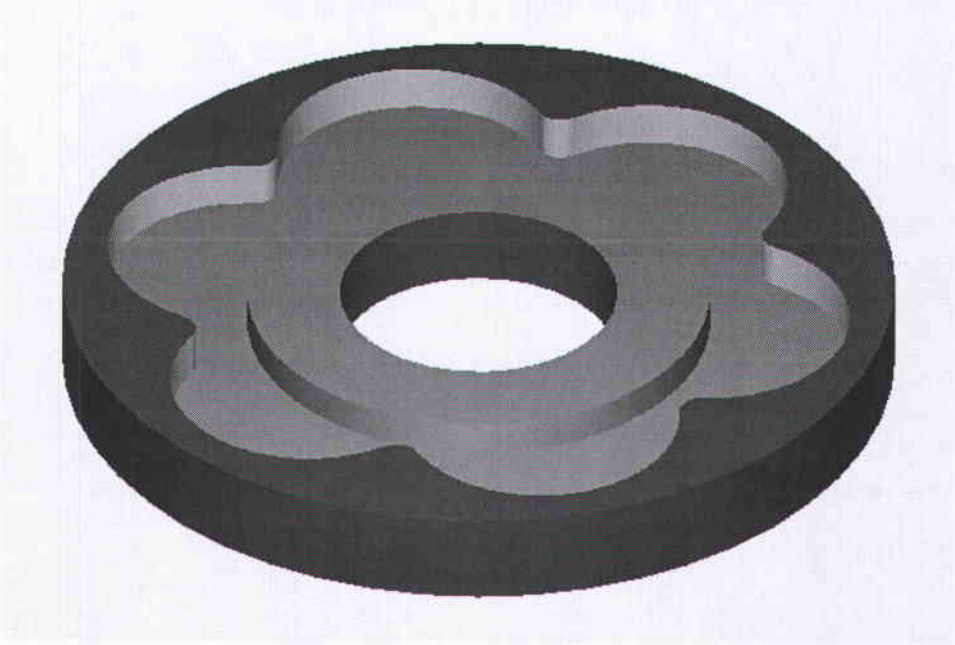
TUTORIAL SERIES FOR

Mastercam X²

TUTORIAL 5

LEVEL 1 – 2D POCKET WITH ISLAND AT A DIFFERENT DEPTH, CIRCLE MILL AND CYLINDRICAL STOCK.



Objectives:

The Student will design a 3-dimensional wireframe drawing by:

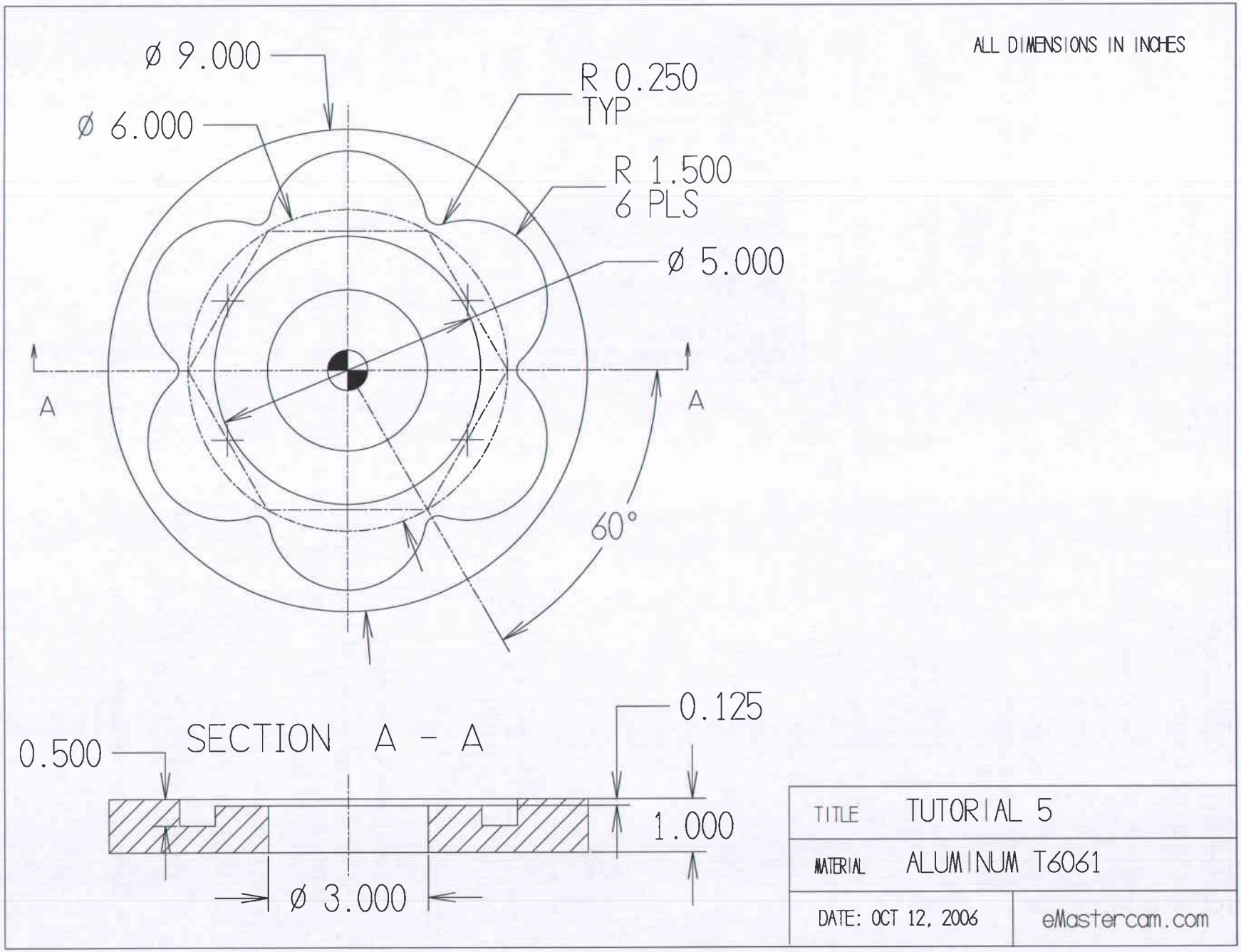
- Creating arcs knowing the center points and the diameters.
- Creating a polygon.
- Creating an arc knowing the endpoints and the radius.
- Rotating the arc to complete the geometry.
- Creating fillet arcs.

The Student will create a 2-dimensional milling toolpath consisting of:

- Cutting a pocket with island at a different depth.
- Circle milling the 3" diameter hole.

The Student will check the toolpath using Mastercam's Verify module by:

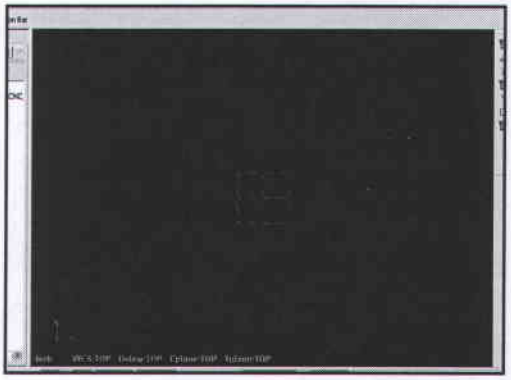
- Defining a 3-dimensional cylindrical block the size of the workpiece.
- Running the Verify function to machine the part on the screen.


ALL DIMENSIONS IN INCHES
ø 9.000
ø 6.000
R 0.250
TYP
R 1.500
6 PLS
ø 5.000
A
A
60°
SECTION A - A
0.500
0.125
1.000
ø 3.000
TITLE TUTORIAL 5
MATERIAL ALUMINUM T6061
DATE: OCT 12, 2006
eMastercam.com

GEOMETRY CREATION
Setting the toolbar states

To start a new file from Mastercam:
File
➢ **New**

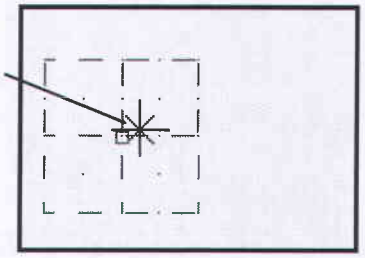
- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-5 in the **User Notes**.
  - **Toolpaths/Solids manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.
  - Before starting the geometry make sure that the **Grid** is enabled. It will show you at each moment where the part origin is. See **Getting started** page A-5 for details.



STEP 1:
CREATE THE OUTSIDE & THE INSIDE ARCS KNOWING THE CENTER POINT AND THE DIAMETER.

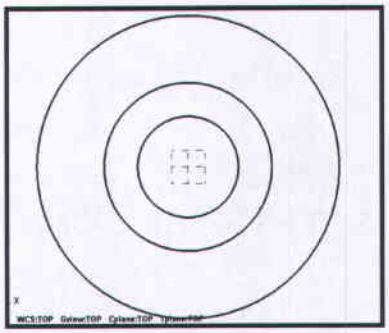
Create
➢ **Arc**
➢ **Create Circle Center Point**
➢ Enter the **Diameter** value 9.0 (Enter).
➢ [Enter the center point]: Move the cursor to the center of the grid to select the **Origin.**

Select Origin



➢ Select the **Apply** button to remain in the same command.
➢ Select the **Fit** button to fit the geometry to the screen.
➢ Enter the **Diameter** value 5.0 (Enter).
➢ [Enter the center point]: Move the cursor to the center of the grid to select the **Origin**.
➢ Select the **Apply** button to remain in the same command.

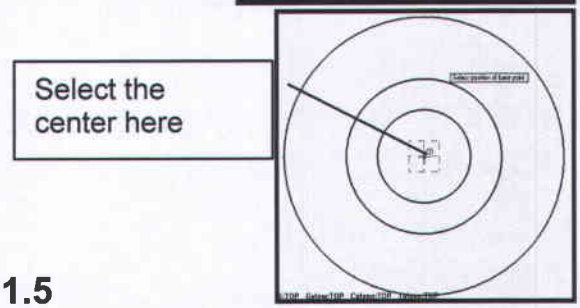
➢ Enter the **Diameter** value 3.0.
➢ [Enter the center point]: Move the cursor to the center of the grid to select the **Origin**.
➢ Select the **OK** button to exit the command.
➢ The drawing should look as shown to the right.



- During the geometry creation of this tutorial, if you make a mistake you can undo the last step using the **Undo** icon. You can undo as many steps as needed.
- If you delete or undo a step by mistake, just use the **Redo** icon.

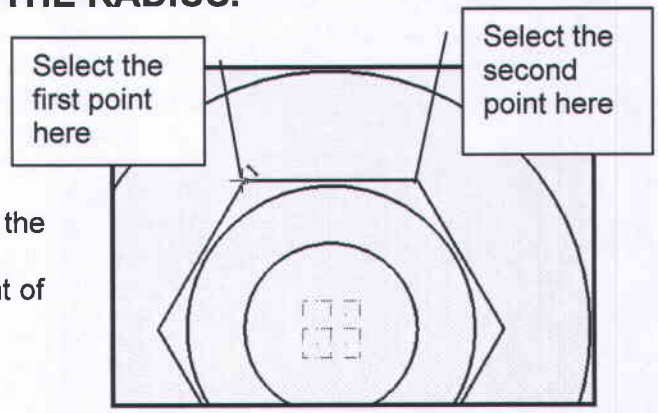
STEP 2:
CREATE THE POLYGON.

Create
➢ **Create Polygon**
➢ Change the number of sides (**#**) to 6.
➢ Enter the **Radius** value.
➢ Enable **Corner** for the system to measure the radius to the corner of the polygon.
➢ [Select position of base point]: Select the center of the arcs as shown in the following screenshot.



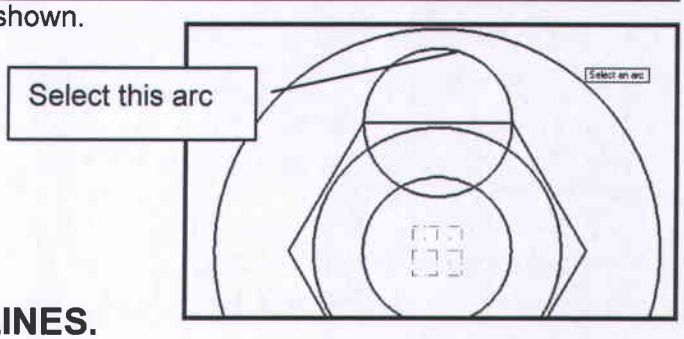
➢ Exit the polygon command.

STEP 3:
CREATE AN ARC WITH A RADIUS OF 1.5 KNOWING THE ENDPOINTS AND THE RADIUS.



Create
➢ **Arc**
➢ **Create Arc Endpoints**
➢ Enter the **Radius** 1.5 (Enter).
➢ [Enter the first point]: Select the endpoint of the line as shown to the right.
➢ [Enter the second point]: Select the endpoint of the line as shown to the right.

➢ [Select an arc]: Select the arc to keep as shown.



➢ Select the **OK** button.

STEP 4:
DELETE THE CONSTRUCTION LINES.

➢ Select the **All** button.

➢ Enable **Entities** and **Lines** as shown to the right.

➢ Select the **OK** button to exit.

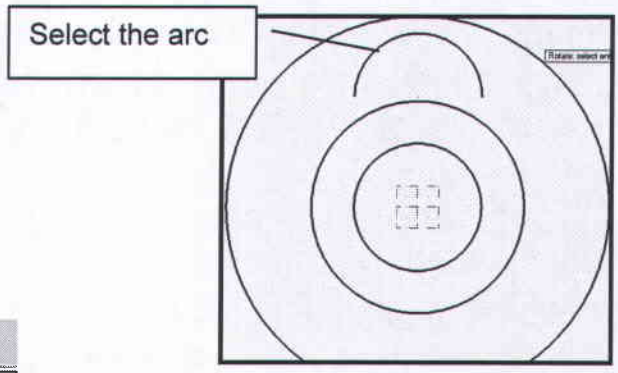
➢ Select the **Delete** entity icon.

STEP 5:
ROTATE THE ARC ABOUT THE ORIGIN TO COMPLETE THE PART.

Xform

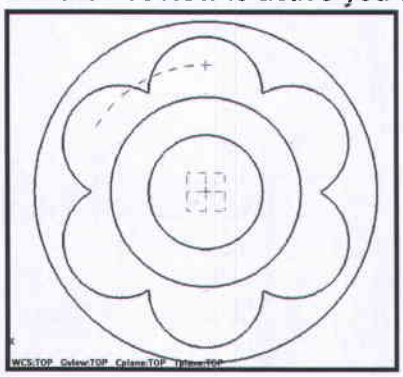
➢ **Xform Rotate**

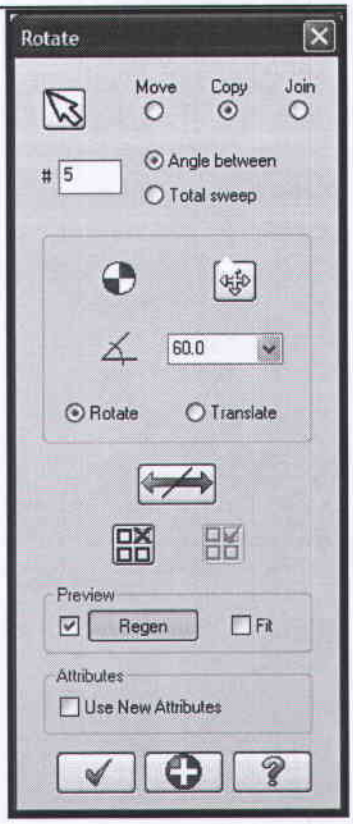
➢ [Rotate: select entities to rotate]: Select the 1.5 radius arc.



➢ Click on the **End Selection** button.

- Make sure that you make all the changes as shown in the screenshot to the right.
- Change the Number (#) to 5 (Enter).
- Enable **Angle between**.
- Change the **Angle** to 60 degrees (Enter).
- If the **Preview** is active you should be able to see the result.





- Select the **OK** button to exit.

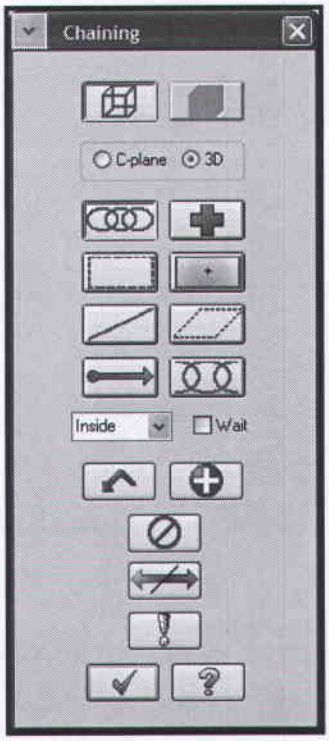
Screen

- **Clear colors**

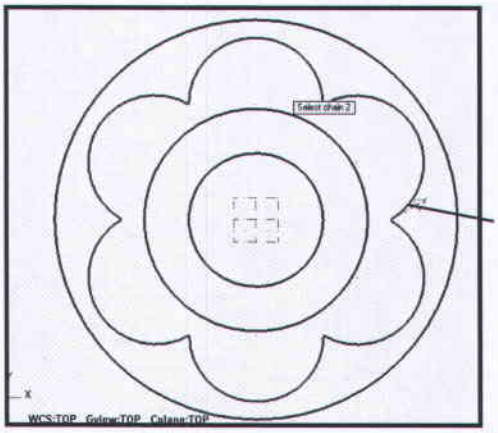
STEP 6:
FILLET THE CORNERS.

Create

- **Fillet**
- **Fillet Chains**



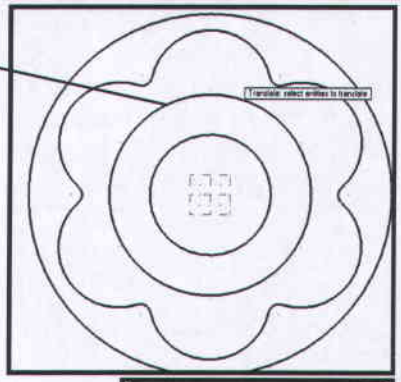
- Enter the fillet **Radius** 0.25.
- [Select chain 1]: Select Entity A.



- Select the **OK** button to exit chaining.
- Select the **OK** button to exit the command.

STEP 7:
TRANSLATE THE 5" DIAMETER ARC DOWN.

Select this arc



Xform
- **Xform Translate**
- [Select entities to translate]: Select the 5" diameter arc.
- Select the **End Selection** button. 

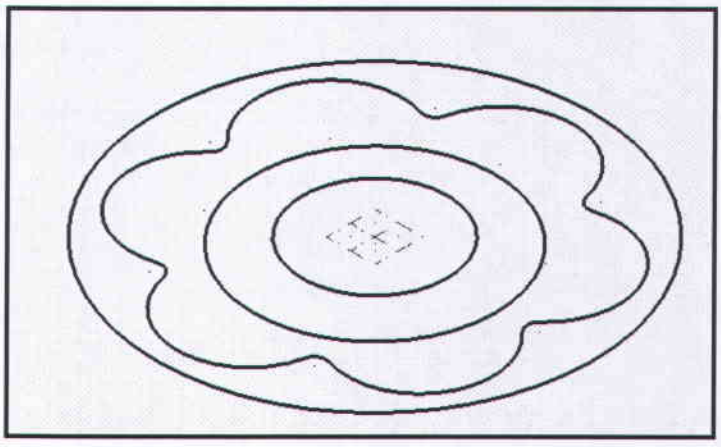
- Enable **Move**.
- Set the number of translations to **#** 1.
- Change the **Delta** value on Z to -0.125.

- Select the **OK** button to exit. 

Screen
- **Clear colors**

- Select the **Isometric View** from the view toolbar to see the stock.

Select the **Top View** from the view toolbar to see the part from the top.



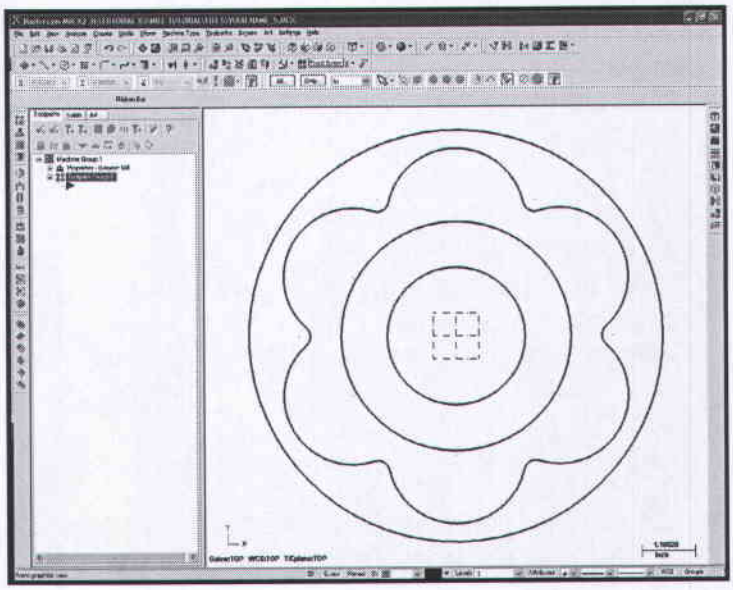
STEP 8:
SAVE THE GEOMETRY

File
- **Save as**
- **File Name**: "Your Name_5"
- Select the **OK** button. 

TOOLPATH CREATION
STEP 9:
SET UP THE STOCK TO BE MACHINED.

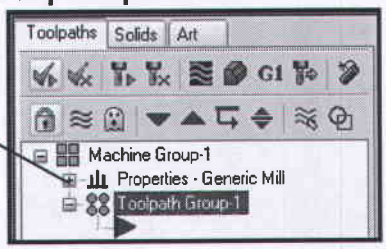
Machine type

- **Mill**
- Select **Default**.
- To display the **Toolpaths Manager** press **Alt + O**.



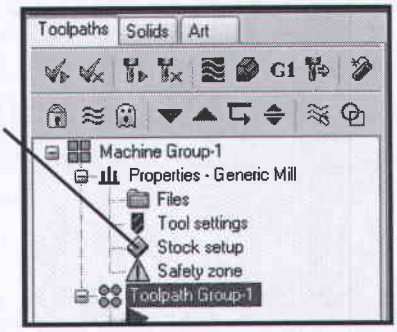
- Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus

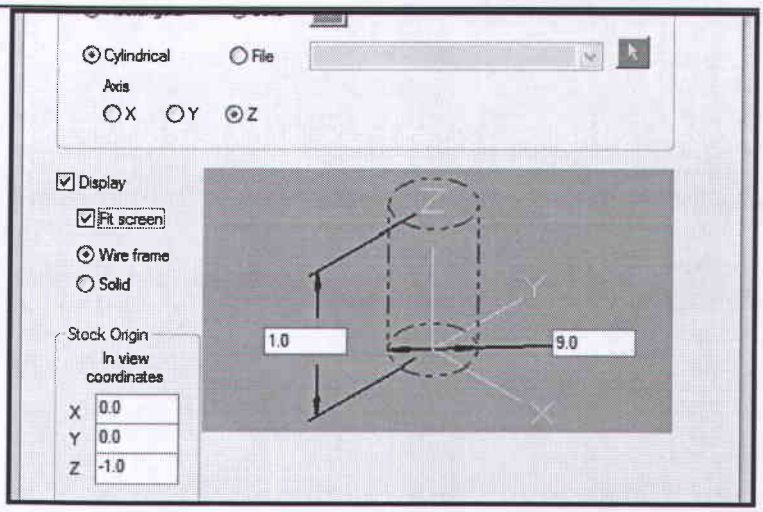


- Select **Stock setup**.

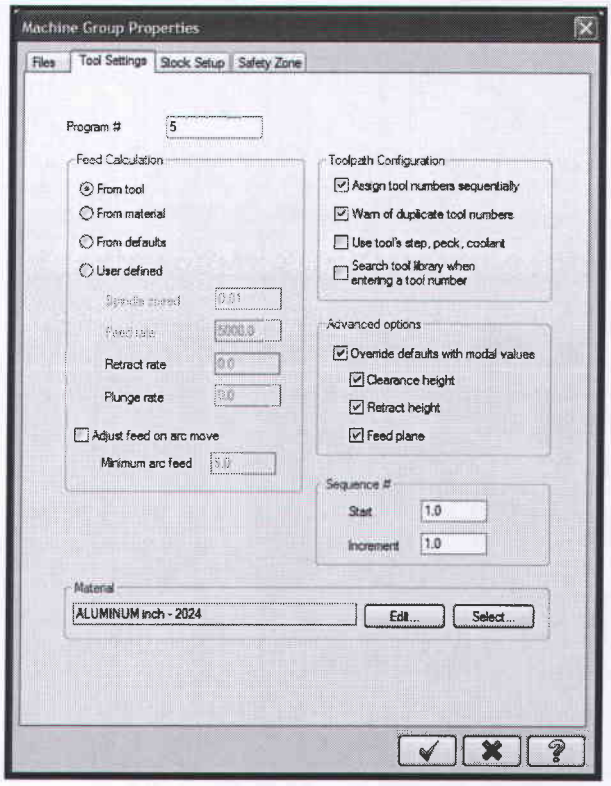
Select Stock setup



- The stock shape should be set to **Cylinder**.
- Enable the **Z**-Axis.
- Enter the **Diameter** and **Z** values of the stock size.
- Enable **Display stock** as **Wireframe** and enable **Fit Screen** to the stock.
- Change the **Z** value in the **Stock origin** to set the datum at the top of the part.



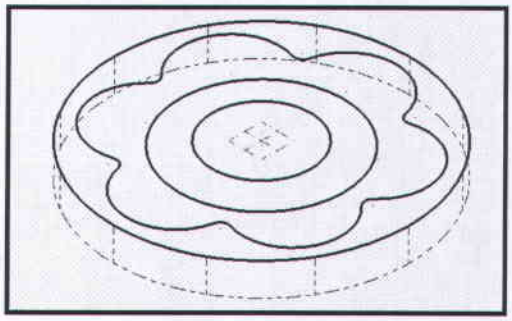
- Select the **Tool Settings** tab to set the tool parameters and the part material.
- Change the parameters to match the following screenshot.



For more information on the **Tool Setting parameters** check **Tutorial #1**.

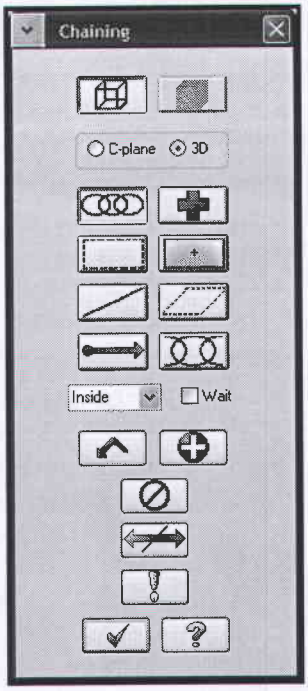
- Select the **OK** button to exit **Toolpath Group Properties**.

- Select the **Isometric View** from the view toolbar to see the stock.



- Select the **Top View** from the view toolbar to see the part from the top.

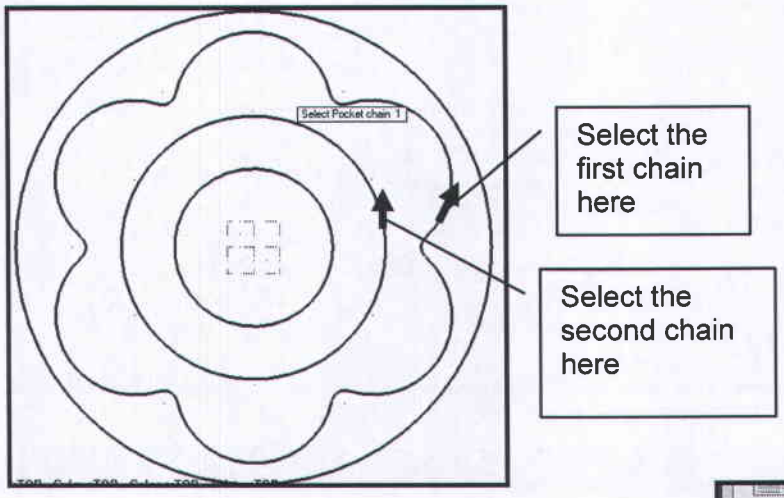
STEP 10:
POCKET AND ISLAND FACING.



Toolpaths
➢ **Pocket Toolpath**

➢ Select the **OK** button to accept the **NC** name.

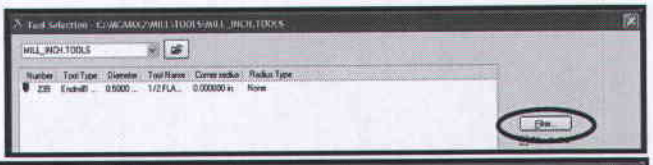
➢ Select the two chains as shown below.



➢ Select the **OK** button to exit **Chaining**.

➢ Click on the **Select library tool** button.

➢ Select the **Filter** button in the **Tool Selection** window.



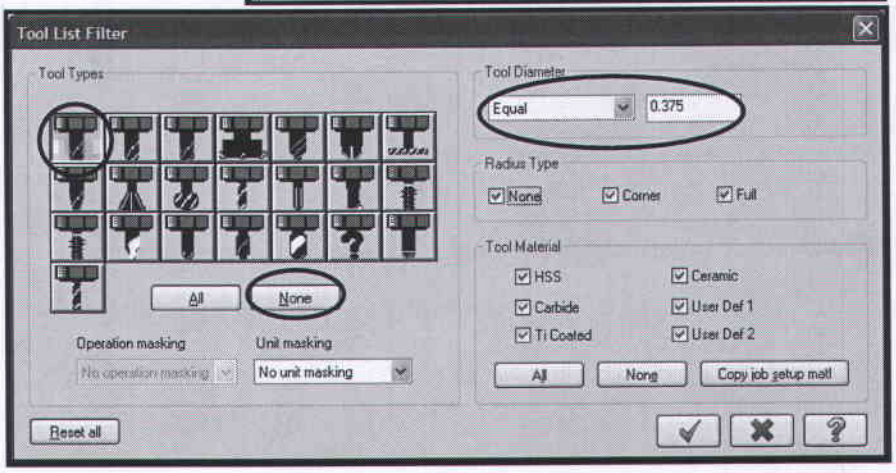
➢ In the **Tool Types** field select the **None** button to disable all tools.

➢ Select the **Flat Endmill** button as shown.

➢ In the **Tool Diameter** field click the pull-down arrow and select **Equal**.

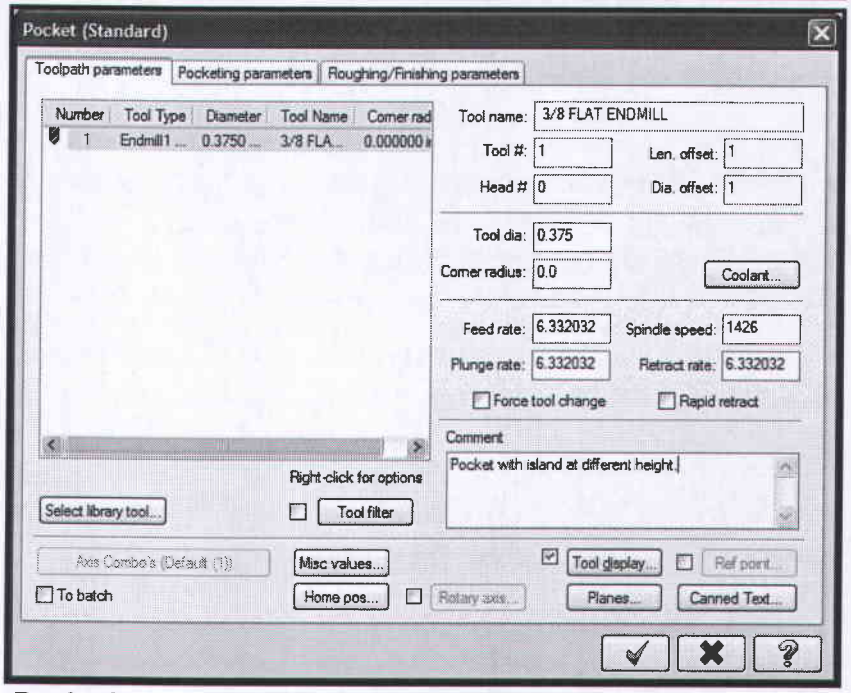
➢ Enter the tool diameter value.

➢ Select the **OK** button to exit **Tool List Filter.**



- Make sure that the tool is selected (highlighted) in the **Tool Selection** window.
- Select the **OK** button to exit the **Tool Selection** window.

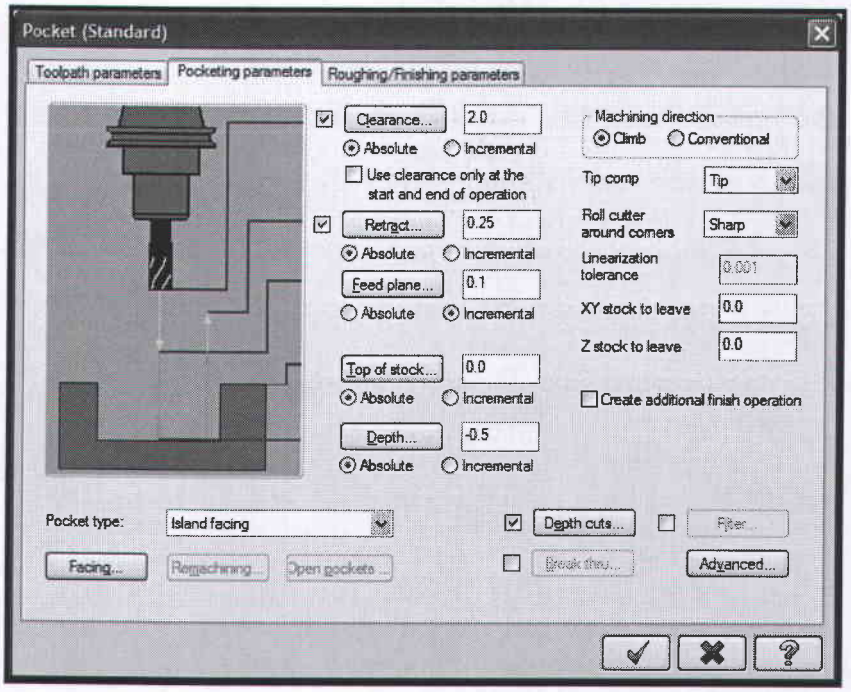
- Make the changes as shown in the screenshot.



- The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition. Change them as desired.

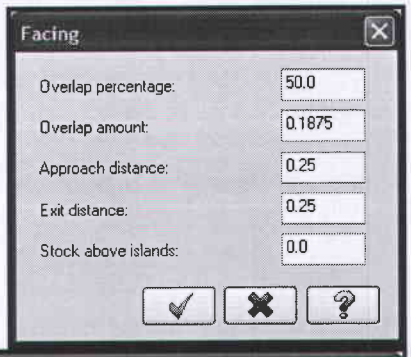
- Change the parameters in the **Pocketing parameter** page as shown.

Clearance value sets the height at which the tool rapids to or from the part.
Retract value sets the height the tool rapids/feed-rates up to, before the next step down.
Feed plane as an incremental value sets the height the tool rapids to before changing to the plunge rate.
Depth value sets the final machining depth for the pocket operation.

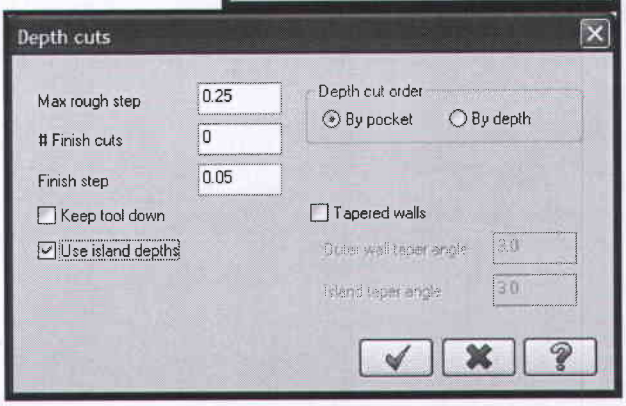


- Select the drop-down arrow in the **Pocket type** and choose **Island facing**.
- Select the **Facing** button and change the parameters as shown.

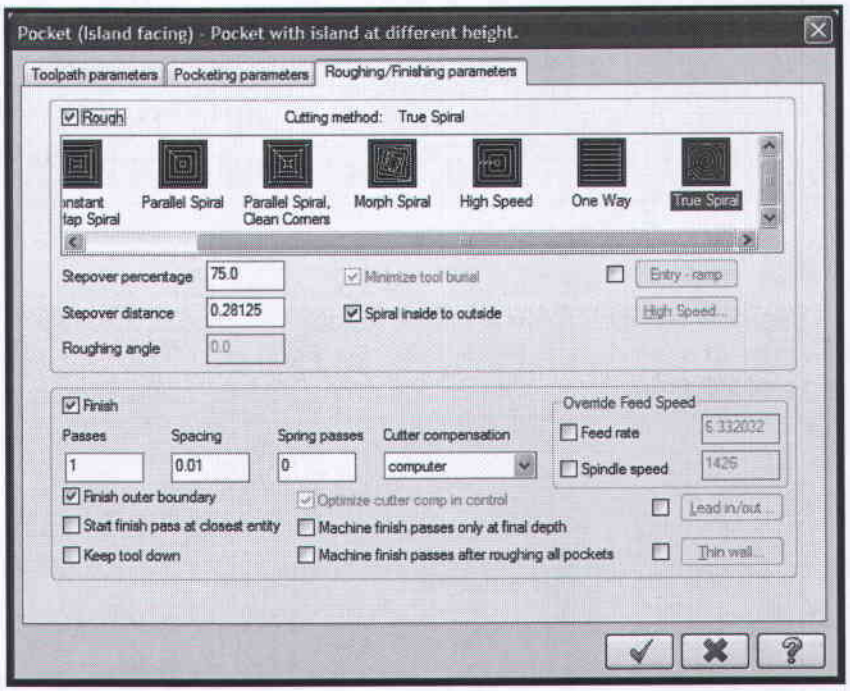
- Set the **Overlap percentage** (percentage of the tool diameter) to set the amount the toolpath overlaps the island.
- The **Overlap amount** will be automatically updated.
- Enter the **Approach & Exit distance** values to set the distance from the island to the starting/ exit point in the facing.



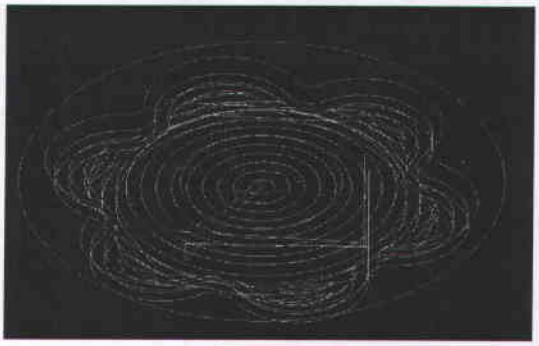
- Select the **OK** button to exit **Facing**.
- Enable and select the **Depth** cuts button to set the cut steps along the **Z**-axis.
- Enable **Use island** depth to set the depth cuts based on the island depth.
- Select the **OK** button to exit **Depth cuts**.



- Select the **Roughing/Finishing parameters** tab.
- Select the **True Spiral** as the **Cutting method**.
- Enable **Spiral inside to outside** to spiral from the center to the pocket wall.



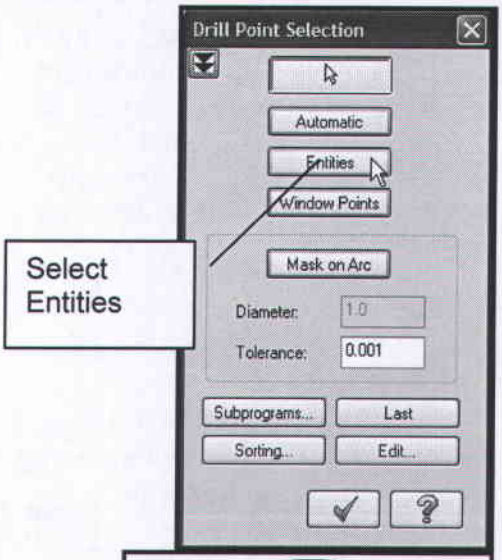
- Select the **OK** button to exit **Pocket parameters**.
- Select the **Isometric View** from the view toolbar to see the toolpath.



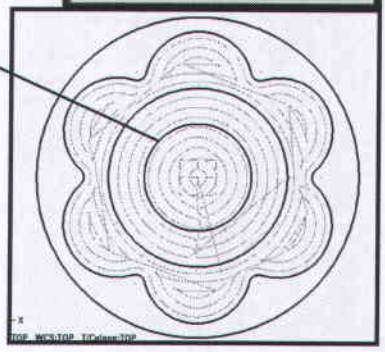
➢ Select the **Top View** from the view toolbar to see the part from the top. 

STEP 11:
CIRCLE MILLING THE INNER BORE.

Tool paths
➢ **Circle Paths**
➢ **Circmill Toolpath**
➢ Select **Entities** in the **Drill Point Selection** dialog box.

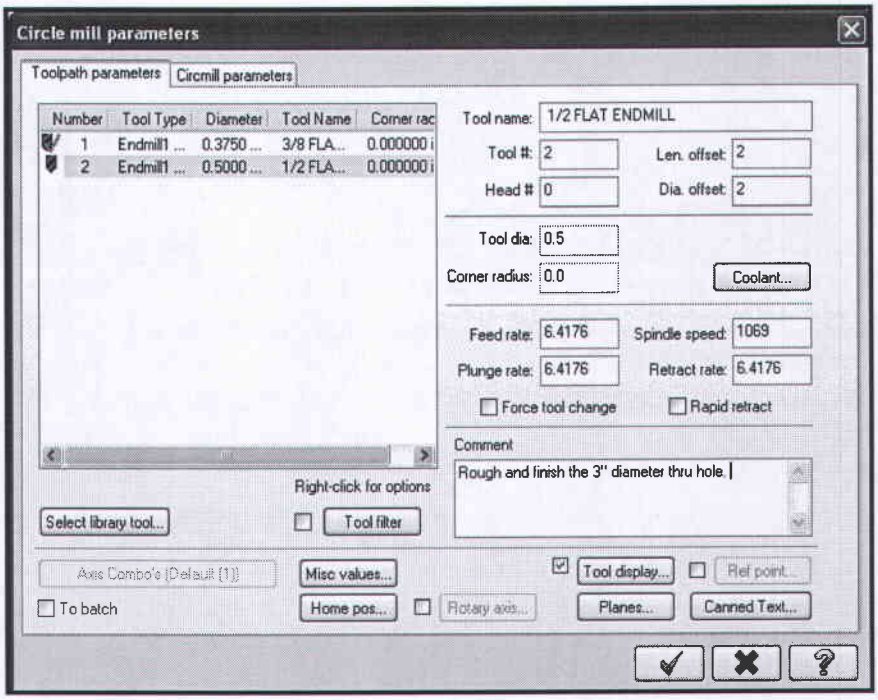


➢ [Select an entity]: Select the 3" diameter arc.



➢ Select the **OK** button to exit **Drill Point Selection**. 
➢ Click on the **Select library tool** button.
➢ Use **Filter** and select the ½" **Flat End Mill** (see **STEP 10** pages **5-10**).

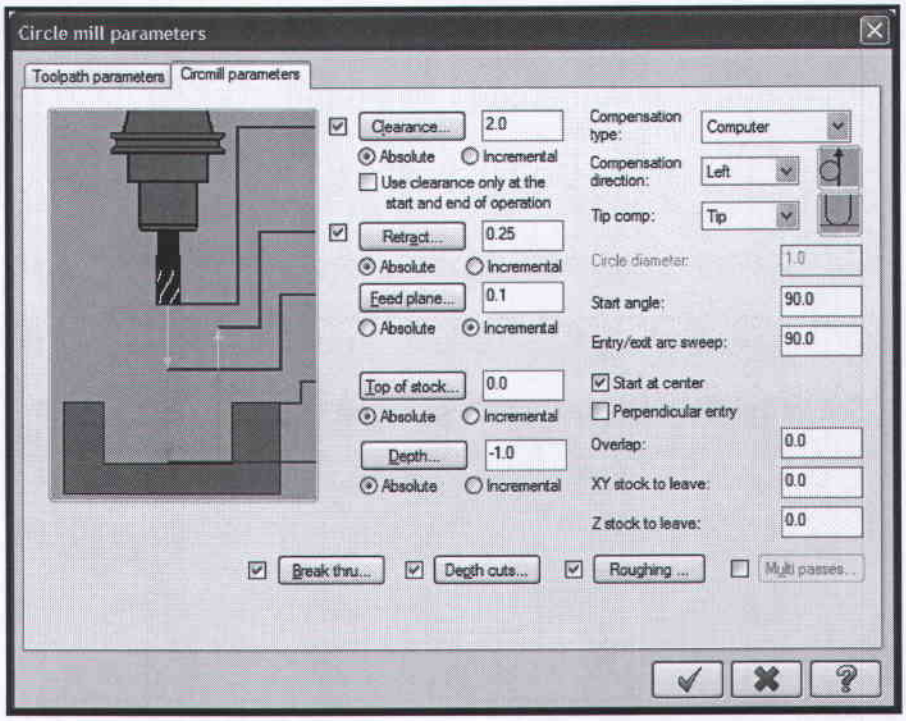
➢ Change the parameters as shown in the following screenshots:



- The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition. Change them as desired.

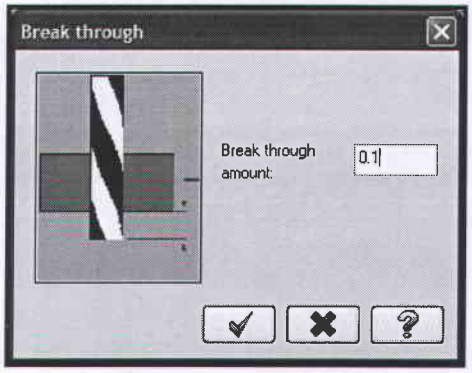
➢ In the **Comment** field type a comment about the toolpath for future reference.

➢ Select the **Circmill parameters** tab and change the parameters as shown.



➢ Enter the **Depth** = -1.0.

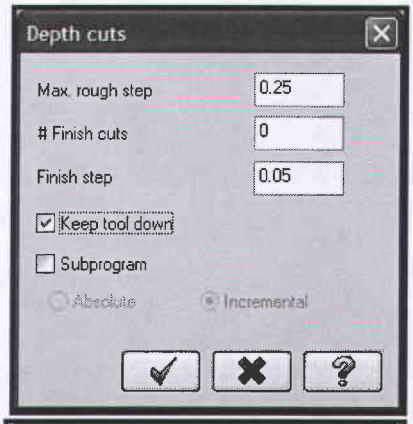
➢ Enable and select the **Break thru** button.



➢ Select the **OK** button to exit the **Break through** dialog box.

➢ Enable and select the **Depth cuts** button to set the cut steps along the **Z**-axis.

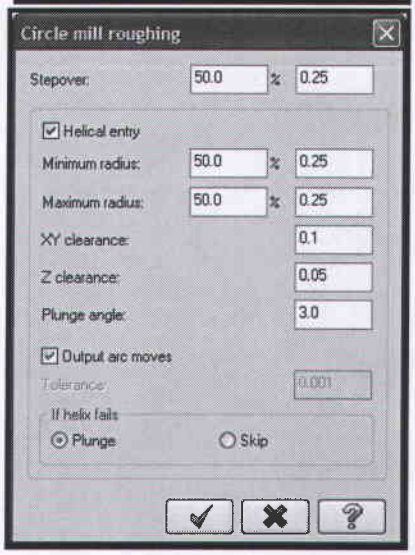
➢ Enable **Keep tool down** to not allow the tool to retract.



➢ Select the **OK** button to exit the **Depth cuts** dialog box.

➢ Check the box in front of the **Roughing** button and select it.

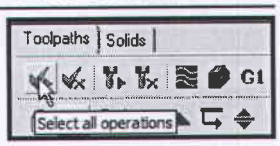
This style creates roughing passes using tangent arcs. The result provides a smooth motion for the tool, a short NC program, and good cleanout.
Stepover is set as a percentage of the tool diameter.
Helical entry parameters create the roughing motion tangent to a helical entry.



➢ Select the **OK** button to exit.

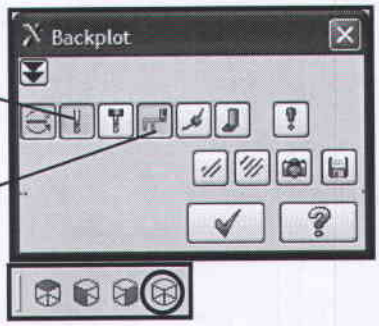
➢ Select the **OK** button from the parameter screen.

STEP 12:
BACKPLOT THE TOOLPATH.

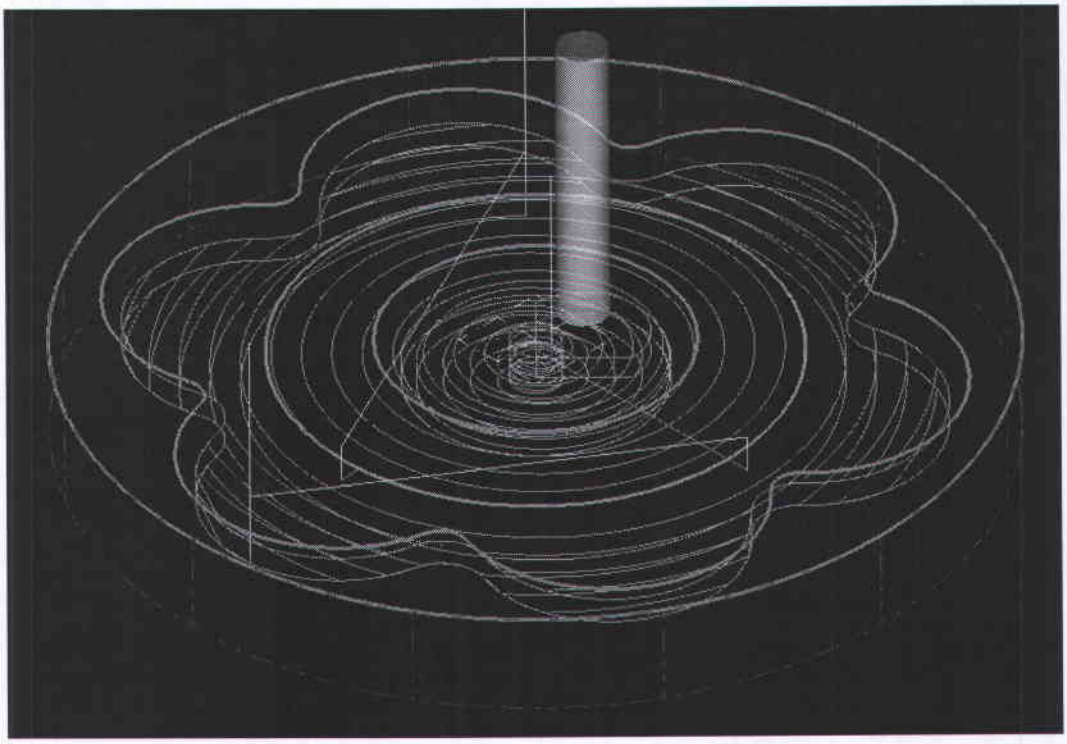


- Click on the **Toolpath Manager tab** to make it active.
- Select the **Select all operations** button.

- Select the **Backplot selected operations** button.
- Make sure that you have the following buttons turned on (they will appear pushed down).
- **Display tool**
- **Display rapid moves**



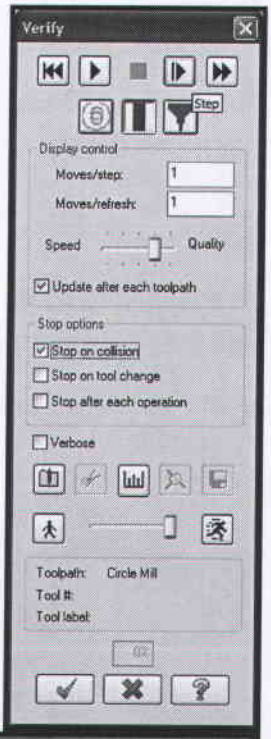
- Select the **Isometric View** from the view toolbar to see the stock.
- Select the **Play** button.



- Select the **OK** button to exit **Backplot**.

TOOLPATH VERIFICATION
STEP 13:
VERIFY.

➢ Select the **Verify selected operations** button.

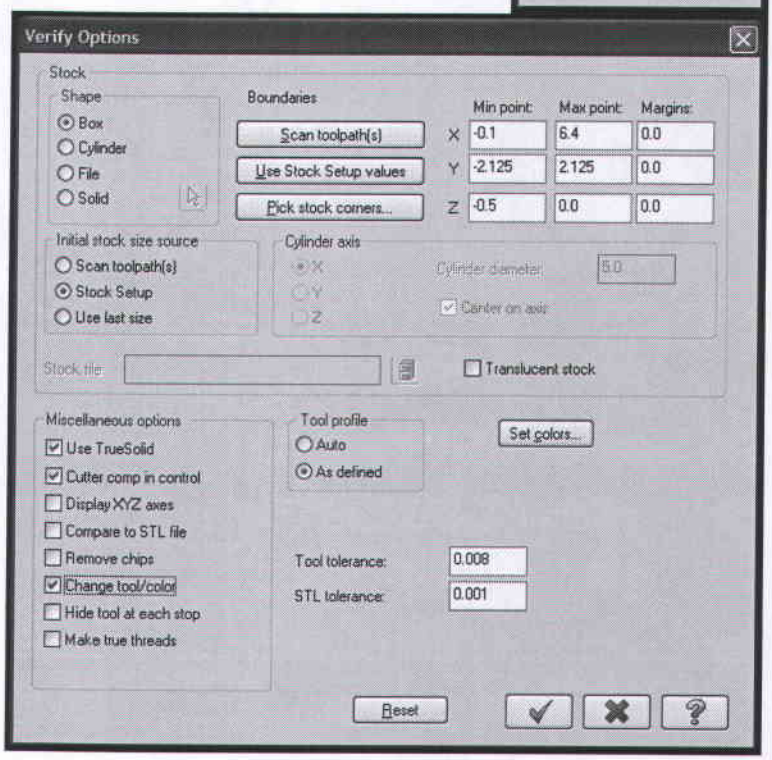


➢ Select the **Configure** button. 

➢ Enable **Use True Solid** to be able, after verifying the part, to rotate and magnify it to more closely to check features, surface finish, or scallops.

➢ Enable **Change tool/color** to change the color of the cut stock to indicated tool changes in the toolpath.

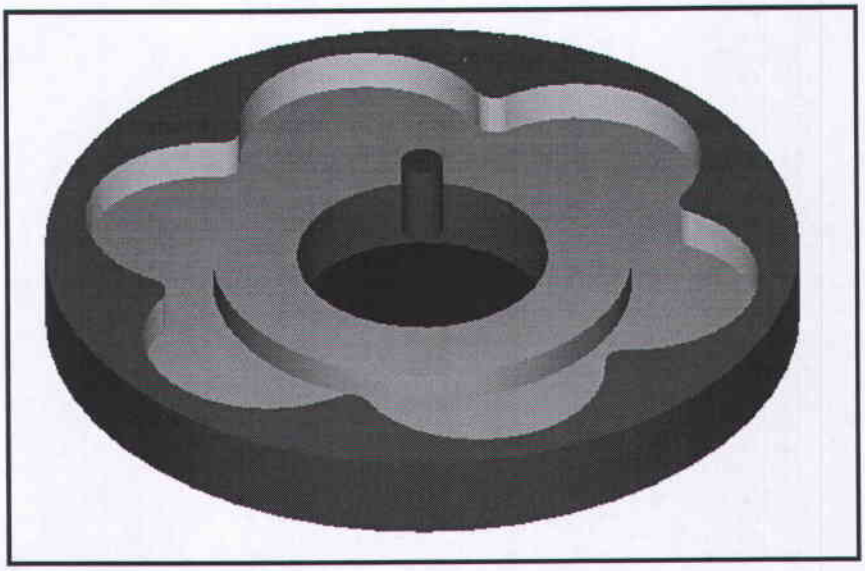
➢ Select the **OK** button to exit **Verify Options**. 



➢ Set the **Verify speed** by moving the slider bar in the speed control bar.

➢ Select the **Machine** button to start simulation.

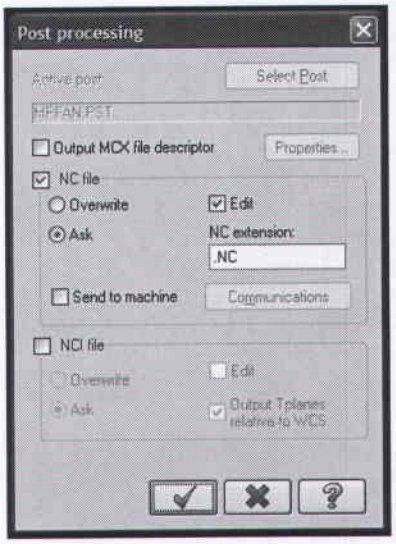
➢ The finished part should appear as shown in the following picture.



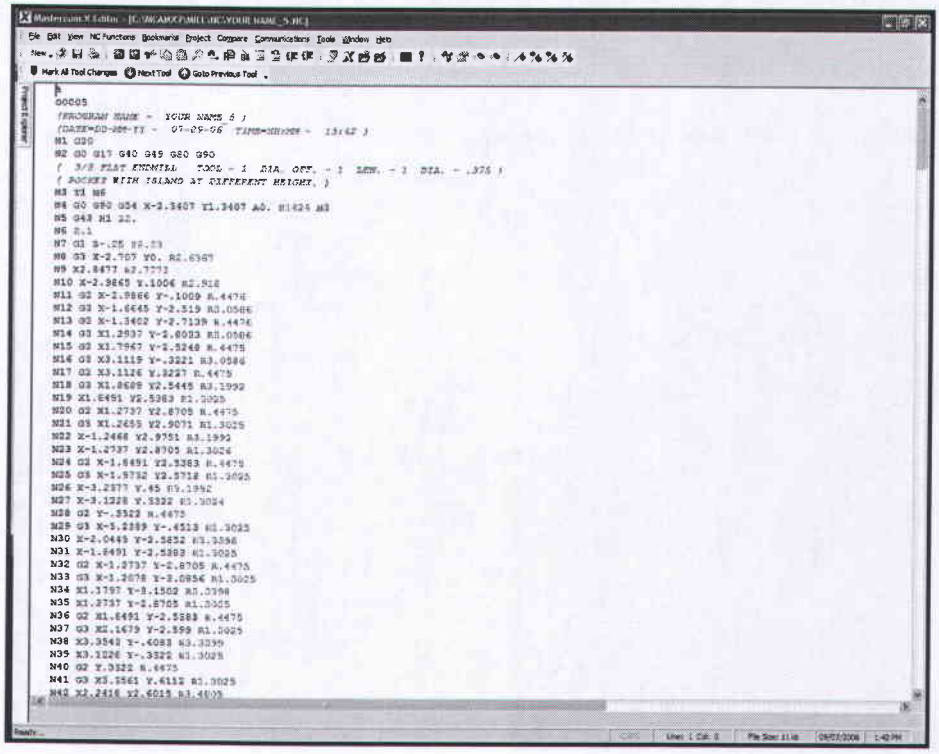
➢ Select the **OK** button to exit **Verify**.

STEP 14:
POST THE FILE.

➢ Make sure that all operations are selected, otherwise:
➢ Select the **Post selected operations** button from **Toolpath Manager**.
➢ In the **Post processing** window, make all the necessary changes as shown below.
➢ Enable **NC file** to keep the NC file assigning the same name as the MCX file.
➢ Enable **Edit** to automatically launch the default editor.
➢ Select the **OK** button to continue.



➢ Select the **Save** button to accept the file name.

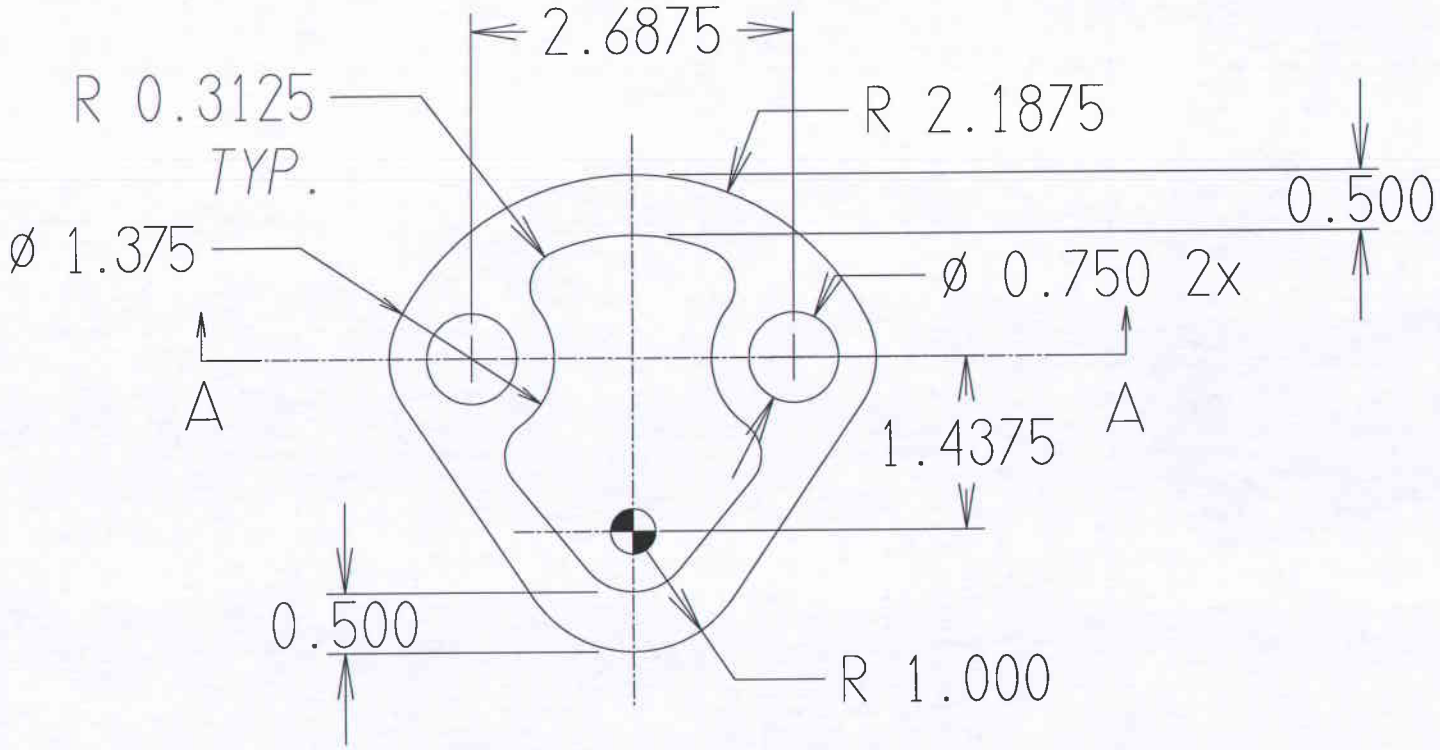


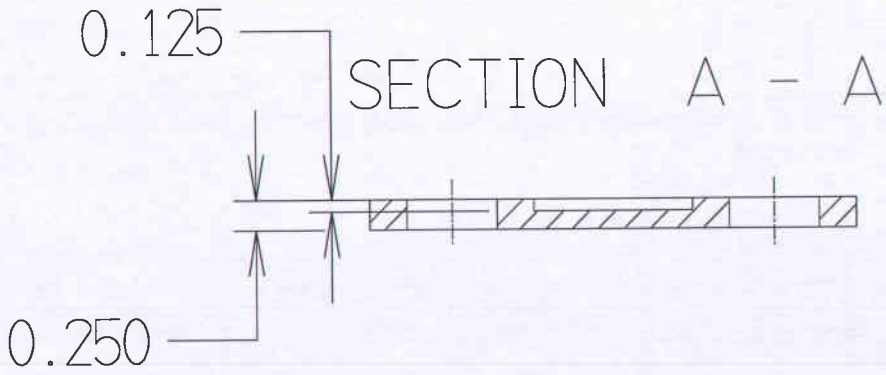
➢ To exit the editor click on the red **X**.

STEP 15:
SAVE THE UPDATED MCX FILE.

➢ Select the **Save** icon.

ALL DIMENSIONS IN INCHES

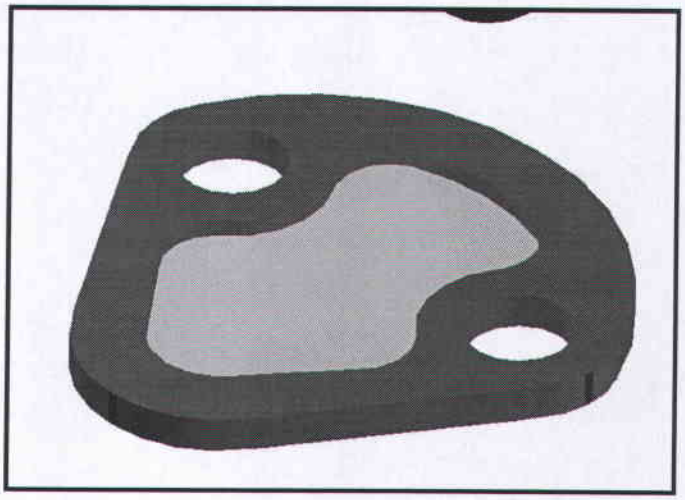




TITLE TUTORIAL 5 - EXERCISE 1	
MATERIAL ALUMINUM T6061	
DATE: JUNE 12, 2000	eMastercam.com

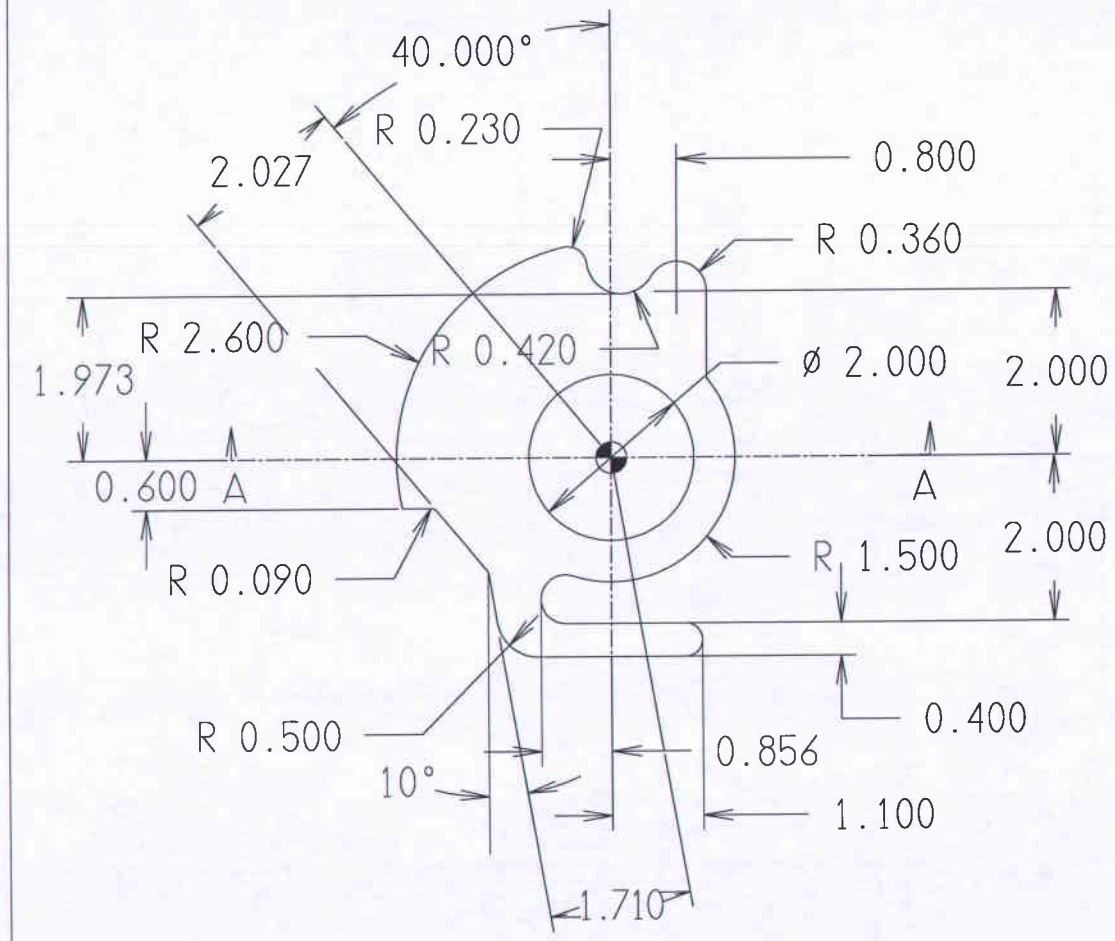
REVIEW EXERCISE 1.

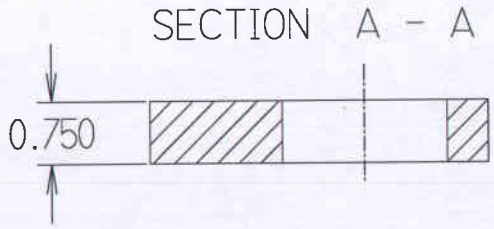
Student practise. Create the Toolpath for Exercise 1-Tutorial 5 as per the instructions below;



☞ **Tips:**

Stock size use **Bounding box** to establish X &Y sizes and give Z = 0.25"
Tool Settings use the same settings as in the tutorial.
Center drill the holes using ½ " Center drill
Drill the thru holes using 3/4" Drill
Pocket the part using 3/4" Flat End Mill
Use Parallel spiral cutting method
Contour the part using 1.5" Flat End Mill
1 roughing passes with spacing =0.5"
1 finish pass 0.05" (Multi passes)
Backplot and **Verify** the toolpaths.
Post process the file.

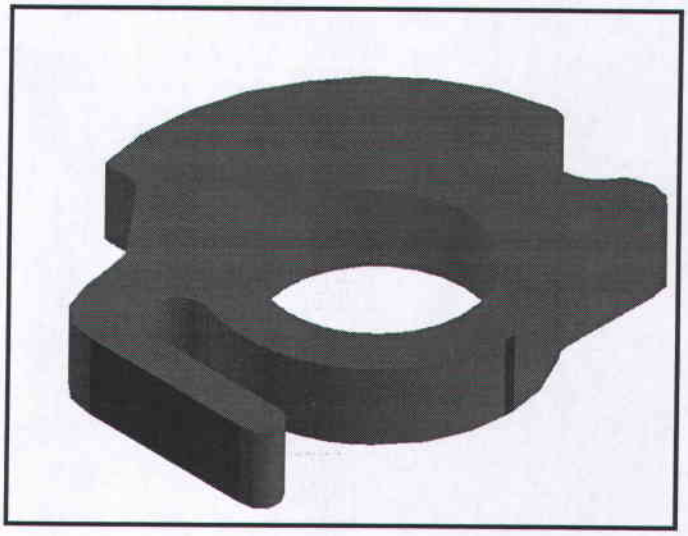




TITLE TUTORIAL 5 - EXERCISE 2	
MATERIAL ALUMINUM T6061	
DATE: JUNE 12, 2000	eMastercam.com

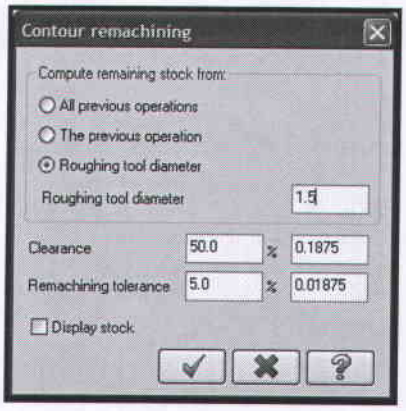
REVIEW EXERCISE 2.

Student practise. Create the Toolpath for Exercise 2-Tutorial 5 as per the instructions below;



☞ **Tips:**

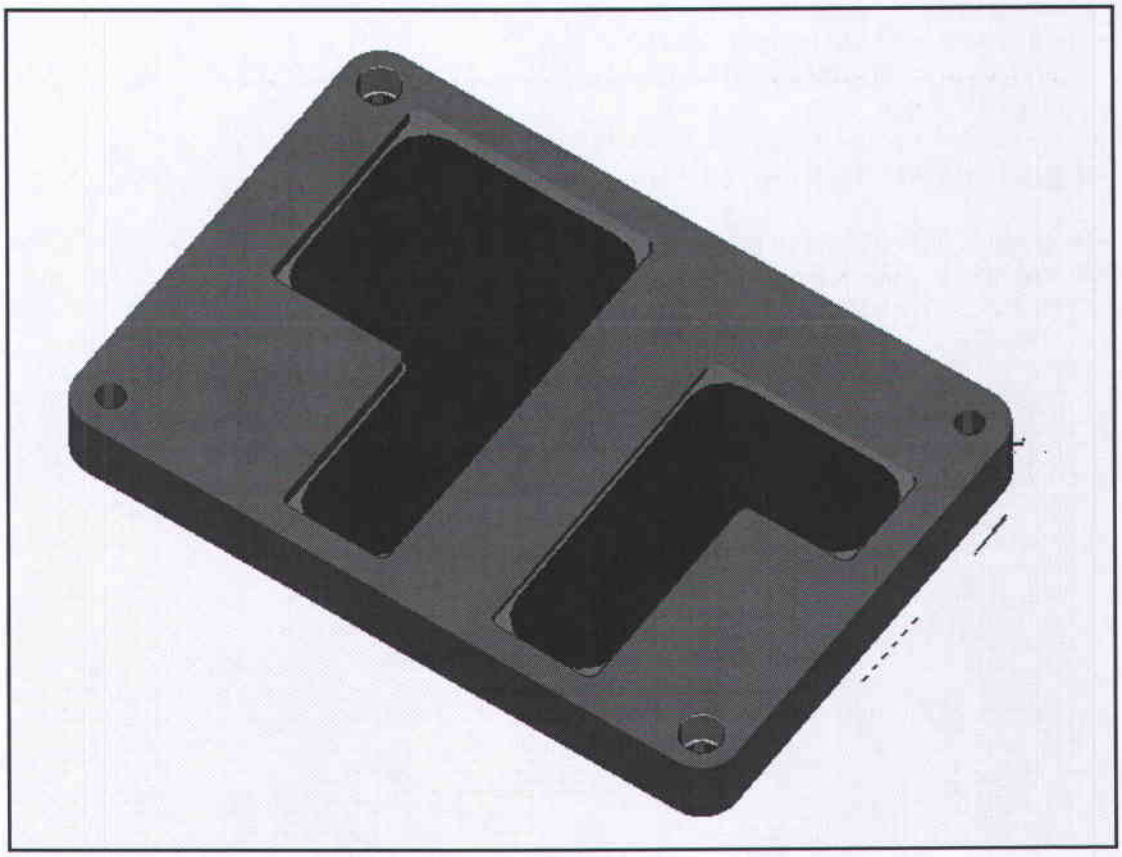
Stock size use **Bounding box** to establish X &Y sizes and give Z = 0.75"
Tool Settings use the same settings as in the tutorial.
Center drill the hole using ½ " Center drill
Drill the thru hole using 3/4" Drill
Circle mill the 2" diameter thru hole using 3/4" Flat End Mill
Use depth cuts; max rough step = 0.375; keep tool down.
Enable roughing
Contour the part using 1.5" Flat End Mill
1 roughing passes with spacing =0.1"
1 finish pass 0.05" (Multi passes)
Contour remachining the part using 0.375" Flat End Mill
Change the contour type to **Remachining** and set the parameters as shown.
Use Depth cuts; max rough step = 0.25
1 roughing passes with spacing =0.1"
1 finish pass 0.05" (Multi passes)
Backplot and **Verify** the toolpaths.
Post process the file.



TUTORIAL SERIES FOR

Mastercam X²

TUTORIAL 6
LEVEL 1 – FACING, CONTOURING, POCKETING, DRILLING, IMPORT FROM LIBRARY.



Objectives:

The Student will design a 3-dimensional wireframe drawing by:

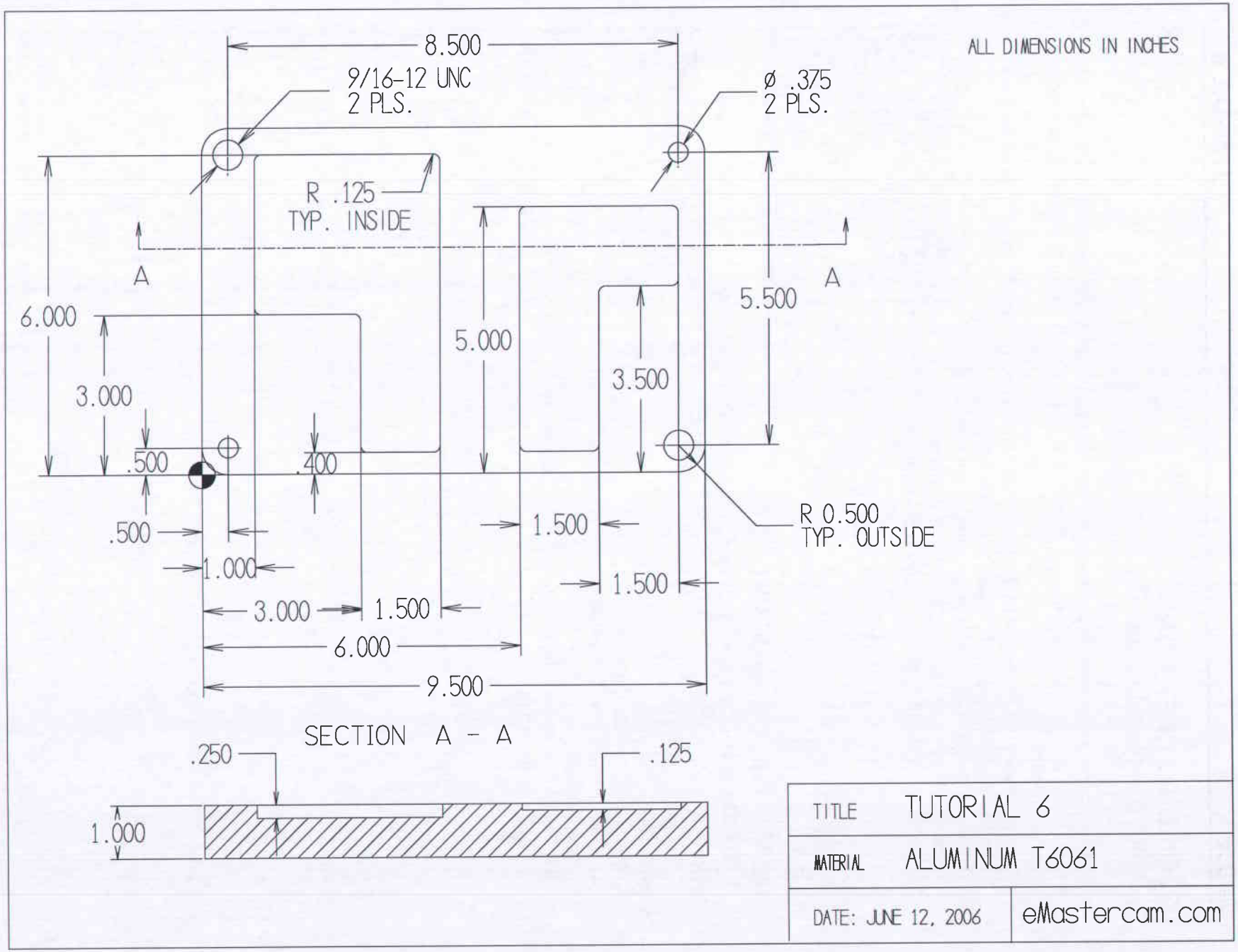
- Creating a rectangle.
- Creating parallel lines.
- Creating fillet radii.
- Creating arcs.
- Using Translate to create a 3-dimensional wireframe.

The Student will create a 2-dimensional milling toolpath consisting of:

- Facing the top of the part.
- Machining 2 simultaneous pockets with different depths.
- Machining a 2D contour.
- Lead in and lead out toolpath entry.
- Importing from the library 9/16 tap holes.
- Drilling 3/8 holes.

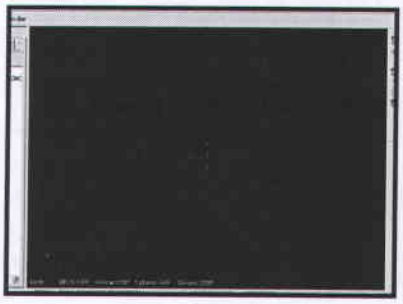
The Student will check the toolpath using Mastercam's Verify module by:

- Defining a 3-dimensional block larger than the size of the workpiece.
- Running the Verify function to machine the part on the screen.


ALL DIMENSIONS IN INCHES
8.500
9/16-12 UNC
2 PLS.
Ø .375
2 PLS.
R .125
TYP. INSIDE
A
A
6.000
3.000
.500
.500
.400
5.000
3.500
5.500
1.500
1.500
R 0.500
TYP. OUTSIDE
1.000
3.000
1.500
6.000
9.500
SECTION A - A
.250
.125
1.000
TITLE TUTORIAL 6
MATERIAL ALUMINUM T6061
DATE: JUNE 12, 2006
eMastercam.com

GEOMETRY CREATION
Setting the toolbar states

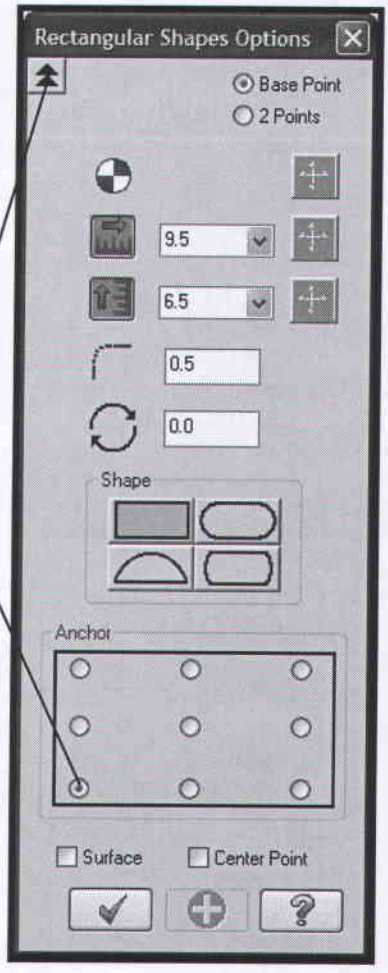
- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-5 in the **User Notes**.
- **Toolpaths/Solids manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.
- Before starting the geometry make sure that the **Grid** is enabled. It will show you at each moment where the part origin is. See **Getting started** page **A-5** for details.



STEP 1:
CREATE THE OUTSIDE PROFILE.
1.1 Create a rectangle knowing the width, the height and the base point.

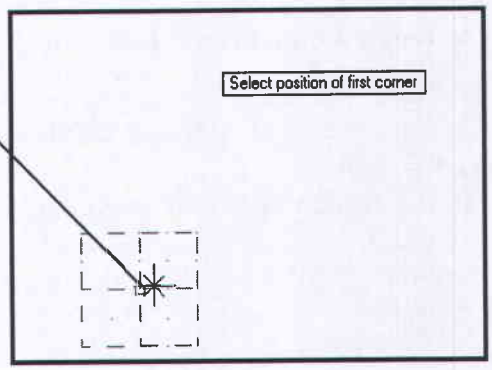
Create

- **Create Rectangular Shapes**
- Select the double arrow to expand the **Rectangle Options** as shown (if needed).
- Type the **Width** the **Height** and the **Fillet Radius** as shown in the picture to the right.



- Select the lower left corner radio button as the anchor.

➢ [Select position for the base point]: Select the center location of the grid (the origin).

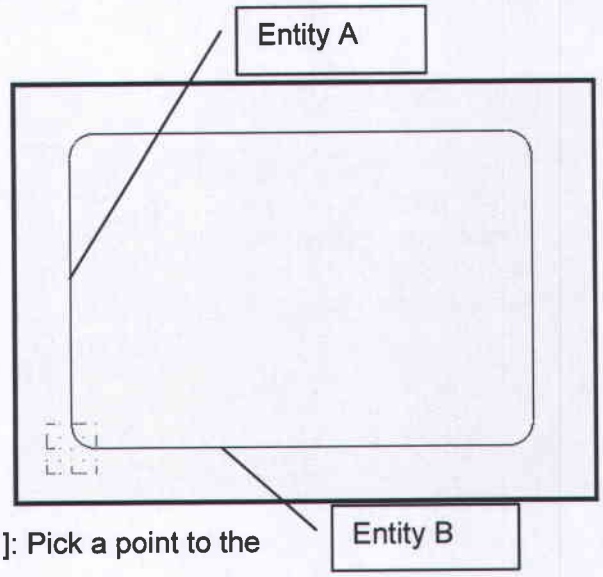


➢ Select the **OK** button to exit the rectangle dialog box.

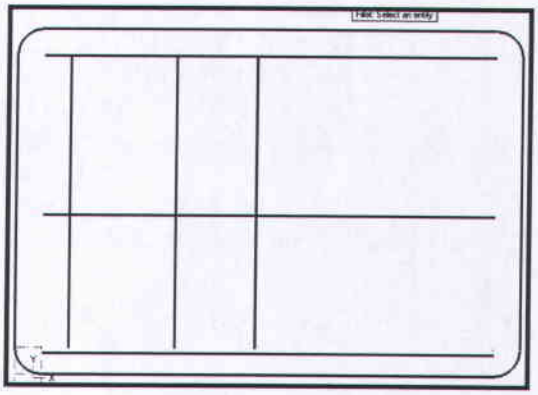
➢ Select the **Fit** button to fit the geometry to the screen.

During the geometry creation of this tutorial, if you make a mistake you can use the **Undo** icon to undo the last step. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.

STEP 2:
CREATE THE FIRST POCKET.
2.1 Create parallel lines.



Create
➢ **Line**
➢ **Create Line Parallel**
➢ [Select a line]: Select Entity A.
➢ [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
➢ Type the **Distance** 1.0 (Enter).
➢ [Select a line]: Select Entity A.
➢ [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
➢ Type the **Distance** 3.0 (Enter).
➢ [Select a line]: Select Entity A.
➢ [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
➢ Type the **Distance** 4.5 (Enter).
➢ [Select a line]: Select Entity B.
➢ [Select the point to place a parallel line through]: Pick a point above the selected line.
➢ Type the **Distance** 0.4 (Enter).
➢ [Select a line]: Select Entity B.
➢ [Select the point to place a parallel line through]: Pick a point above the selected line.
➢ Type the **Distance** 3.0 (Enter).
➢ [Select a line]: Select Entity B.
➢ [Select the point to place a parallel line through]: Pick a point above the selected line.



➢ Type the **Distance** 6.0 (Enter).

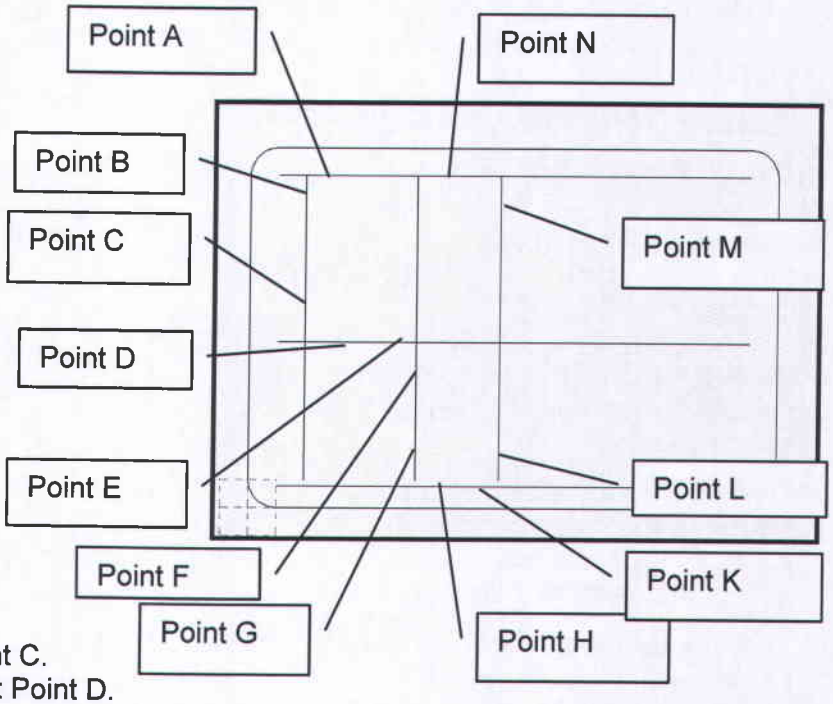
➢ Select the **OK** button to exit the command.

➢ The drawing should look as shown in the picture to the right.

2.2 Create the corner fillets.

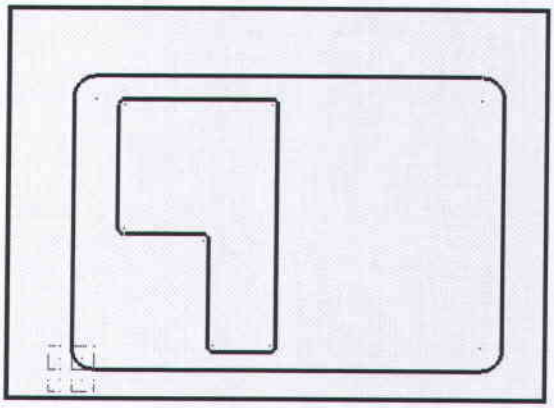
Create
➢ **Fillet**
➢ **Fillet Entities**

➢ Enter the fillet **Radius** 0.125.
➢ [Select an entity]: Select Point A.
➢ [Select another entity]: Select
➢ Point B.



Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities.

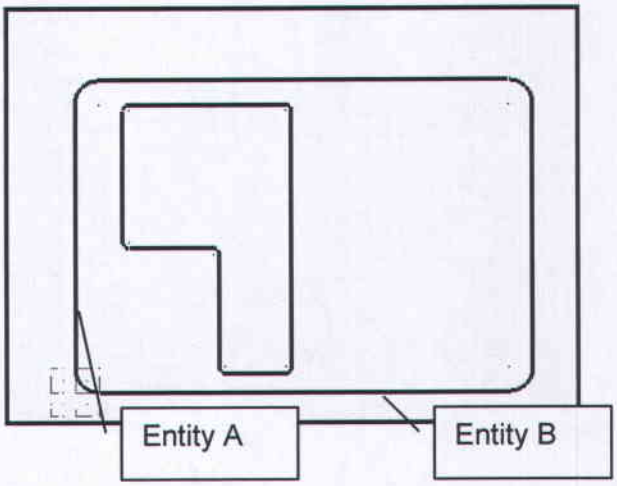
➢ [Select an entity]: Select Point C.
➢ [Select another entity]: Select Point D.
➢ [Select an entity]: Select Point E.
➢ [Select another entity]: Select Point F.
➢ [Select an entity]: Select Point G.
➢ [Select another entity]: Select Point H.
➢ [Select an entity]: Select Point K.
➢ [Select another entity]: Select Point L.
➢ [Select an entity]: Select Point M.
➢ [Select another entity]: Select Point N.
➢ The drawing should look as shown in the picture to the right.



➢ Select the **OK** button.

STEP 3:
CREATE THE SECOND POCKET.
3.1 Create parallel lines.

Create
- **Line**
- **Create Line Parallel**
- [Select a line]: Select Entity A.
- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Enter the **Distance** 6.0.



- [Select a line]: Select Entity A.
- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Enter the **Distance** 7.5

- [Select a line]: Select Entity A.
- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Type the **Distance** 9.0.

- [Select a line]: Select Entity B.
- [Select the point to place a parallel line through]: Pick a point above the selected line.
- Type the **Distance** 0.4.

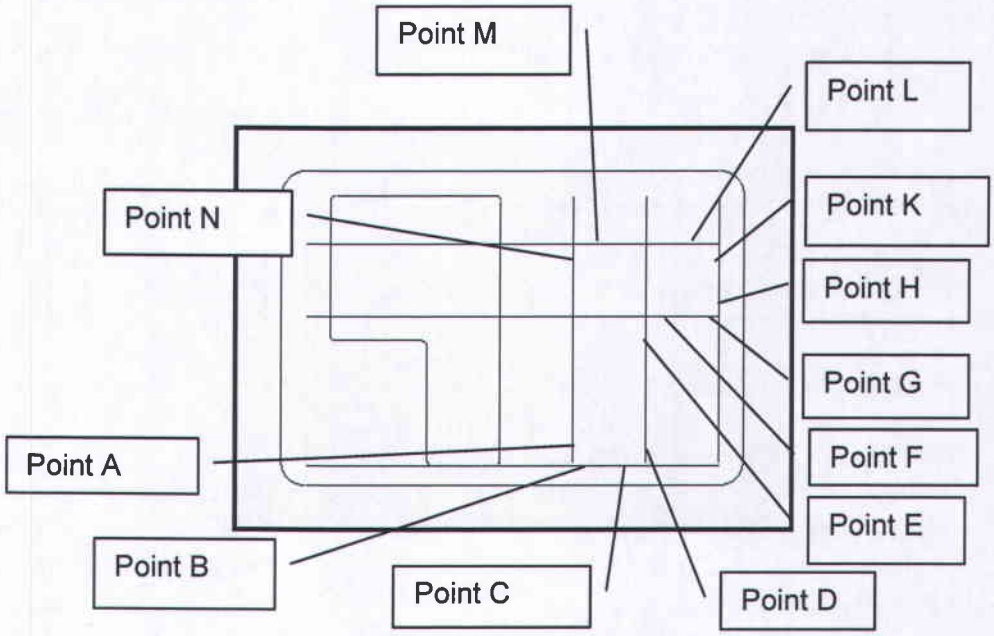
- [Select a line]: Select Entity B.
- [Select the point to place a parallel line through]: Pick a point above the selected line.
- Type the **Distance** 3.5.
- Select the **Apply** button to continue,

- [Select a line]: Select Entity B.
- [Select the point to place a parallel line through]: Pick a point above the selected line.
- Type the **Distance** 5.0.
- Select the **OK** button to exit the command,

3.2 Create the corner fillets.

Create
➢ **Fillet**
➢ **Fillet Entities**

➢ Enter the fillet **Radius** 0.125.
➢ [Select an entity]: Select Point A.
➢ [Select another entity]: Select Point B.



➢ [Select an entity]: Select Point C.
➢ [Select another entity]: Select Point D.

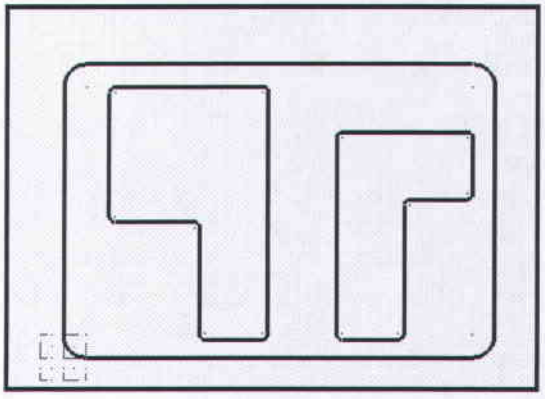
➢ [Select an entity]: Select Point E.
➢ [Select another entity]: Select Point F.

➢ [Select an entity]: Select Point G.
➢ [Select another entity]: Select Point H.

➢ [Select an entity]: Select Point K.
➢ [Select another entity]: Select Point L.

➢ [Select an entity]: Select Point M.
➢ [Select another entity]: Select Point N.

➢ Select the **OK** button to exit the command.
➢ The drawing should look as shown to the right:



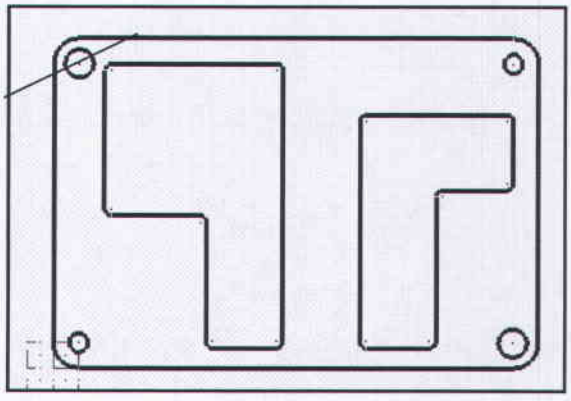
STEP 4:
CREATE ARCS KNOWING THE DIAMETER AND THE CENTER POINT.

Create
➢ **Arc**
➢ **Create Circle Center Point**
➢ Enter the **Diameter** value 9/16 (Enter).
➢ [Enter the center point]: Select the **Fast Point** icon.
➢ Enter the coordinates: 0.5, 6 (Enter).
✹ To create more arcs with the same diameter click on the diameter icon. The diameter and radius values will be highlighted in red. 0.5625
➢ Select the **Apply** button to continue.
➢ [Enter the center point]: Select the **Fast Point** icon.
➢ Enter the coordinates: 9, 0.5.
➢ Select the **Apply** button to continue.
➢ Enter the **Diameter** value 3/8 (Enter).
✹ Note that the diameter and radius values will still be highlighted in red.
➢ [Enter the center point]: Select the **Fast Point** icon.
➢ Enter the coordinates: 0.5, 0.5.
➢ Select the **Apply** button to continue.
➢ [Enter the center point]: Select the **Fast Point** icon.
➢ Enter the coordinates: 9, 6 (Enter).
➢ Select the **OK** button to exit.

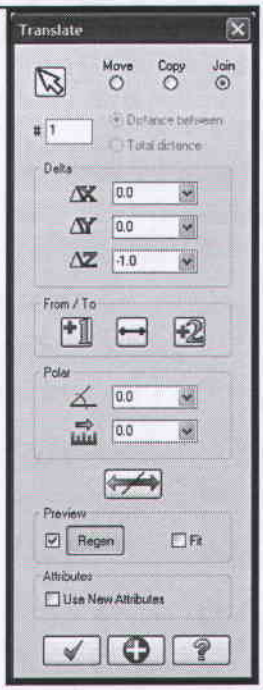
STEP 5:
CREATE THE 3-D DESIGN.

Xform
➢ **Xform Translate**
➢ [Select entities to translate]:
➢ Hold-down the shift key and select Entity A as shown.
✹ Note that the entire rectangle is highlighted.

Entity A



➢ Select the **End Selection** button.

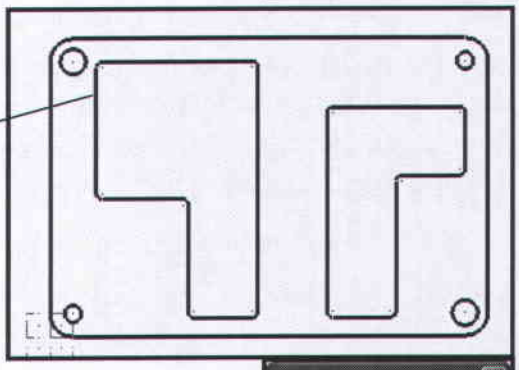


- Enable **Join**.
- Set the number of translations to **#** 1.
- Change the **Delta** value on **Z** to -1.0.

- Select the **Apply** button to continue.

- [Select entities to translate]: Hold-down the Shift key and select Entity B.

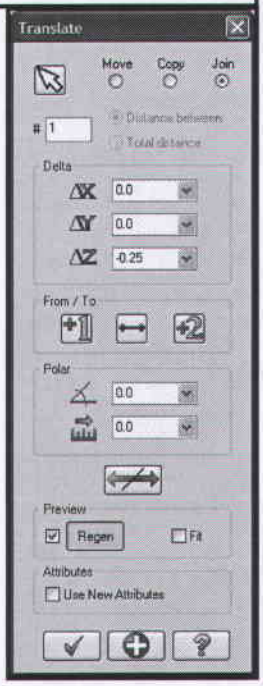
Note that the entire pocket is highlighted.



- Select the **End Selection** button.

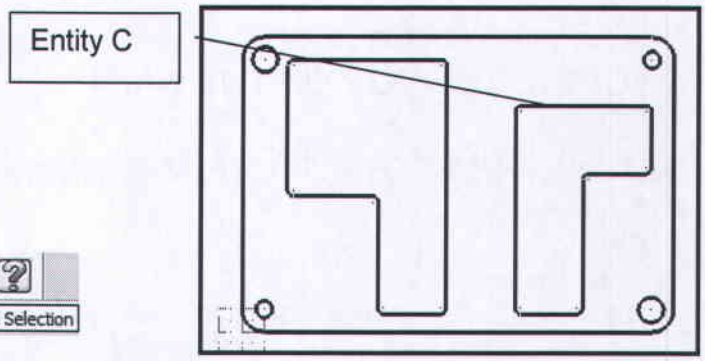
- Enable **Join**.
- Set the number of translations to **#** 1.
- Change the **Delta** value on **Z** to -0.25.

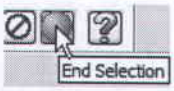
- Select the **Apply** button to continue.



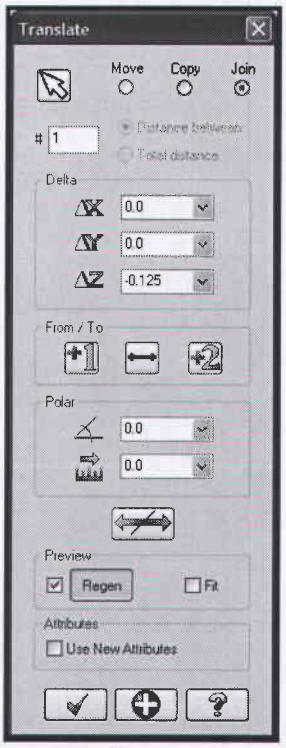
➢ [Select entities to translate]: Hold-down the Shift key and select Entity C.

Note that the entire pocket is highlighted.



➢ Select the **End Selection** button. 

➢ Change the **Delta** value on **Z** to -0.125.



➢ Select the **OK** button to exit.

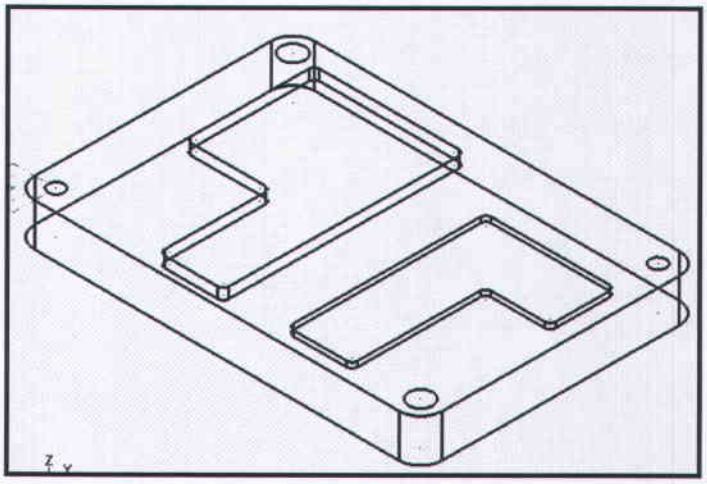
Screen
➢ **Clear colors**

➢ Select the **Isometric View** from the view toolbar to see the stock. 

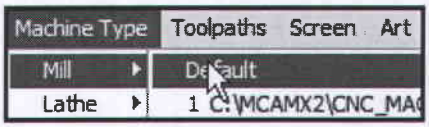
➢ Select **Fit** button.

STEP 6:
SAVE THE GEOMETRY

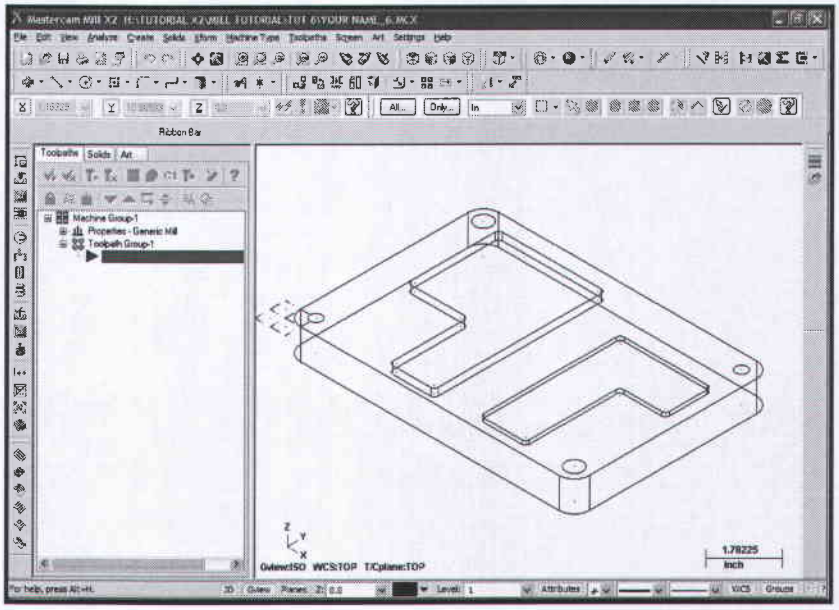
File
➢ **Save as**
➢ **File Name**: "Your Name_6"
➢ Select the **OK** button.



TOOLPATH CREATION
MACHINE THE TOP OF THE PART
STEP 7:
SET UP THE STOCK TO BE MACHINED.



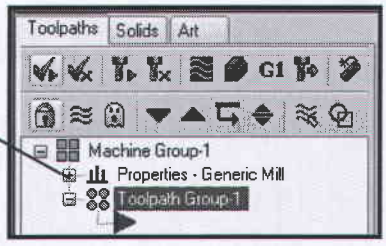
- **Machine type**
- **Mill**
- Select **Default**.
- To display the **Toolpaths Manager** press **Alt + O**.



Make sure that **3D** mode is enabled in the **Status Bar**.

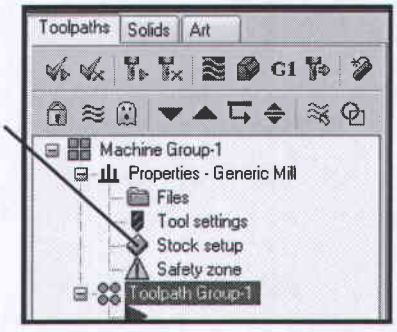
- Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus

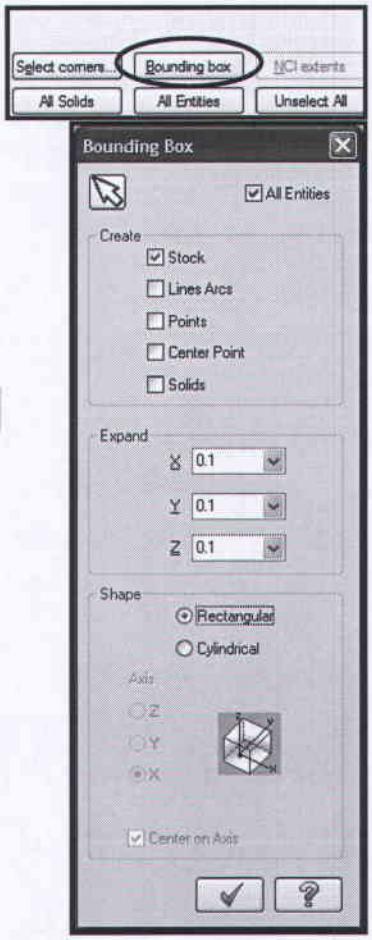


- Select **Stock setup**.

Select Stock setup



➢ Select the **Bounding box** button as shown.



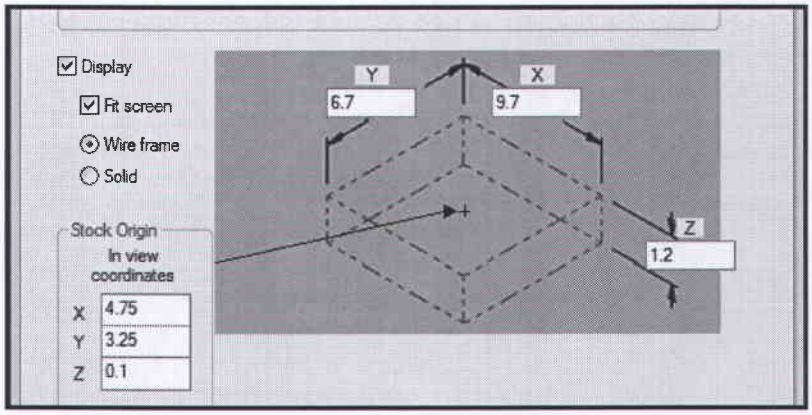
> The **Bounding box** function automatically finds the extents of the part based on the part geometry.

➢ Expand the X, Y and Z sizes with 0.1

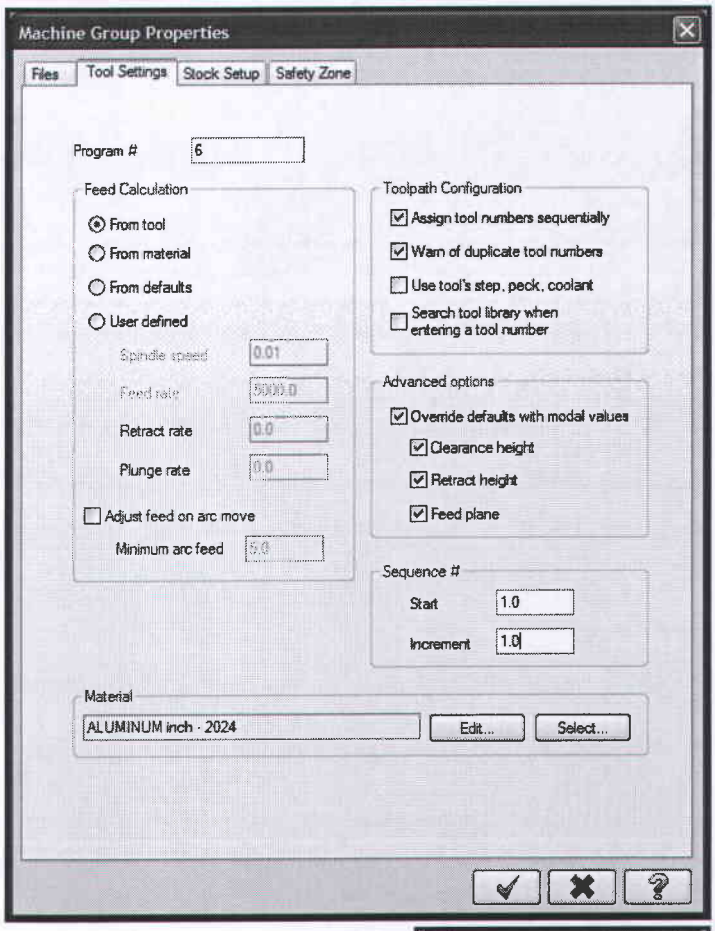
- The size of the blank material will be the finish size plus 0.2" on width, length and on thickness

➢ Select the **OK** button to exit the **Bounding Box** dialog box.

➢ Make sure that you have the same parameters as shown in the following screenshot.

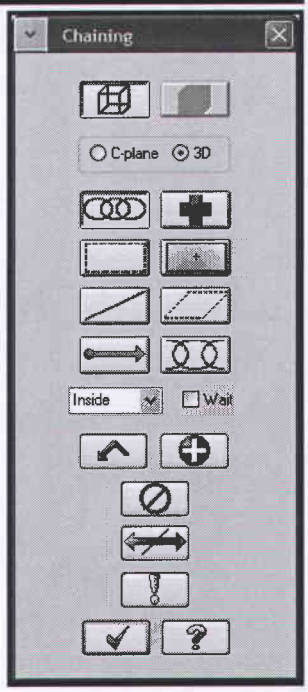


- Select the **Tool Settings** tab and change the parameters to match the screenshot to the right.
- For more info on parameters see Tutorial #1.



- Select the **OK** button to exit **Toolpath Group Properties**.
- Use the **Fit** icon to fit the drawing to the screen.

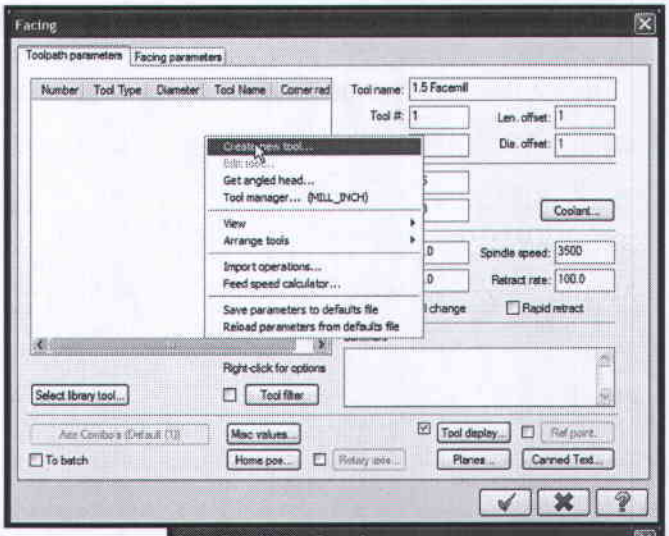
STEP 8:
FACING THE TOP OF THE PART.



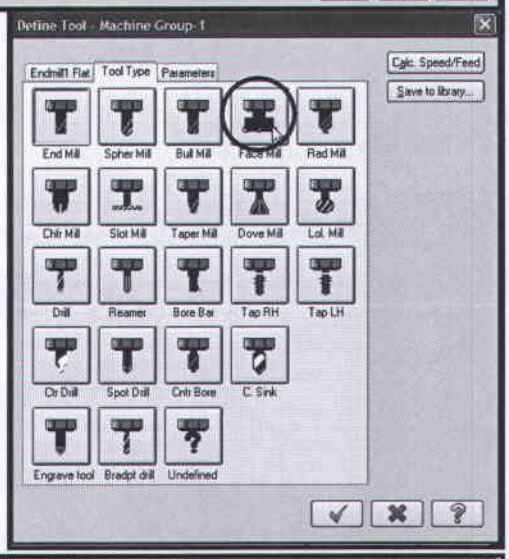
Toolpaths

- **Face Toolpath**
- Select the **OK** button to accept the **NC** file **name.**
- [Select OK to use defined stock or select chain1]:
- Select the **OK** button to use defined stock.

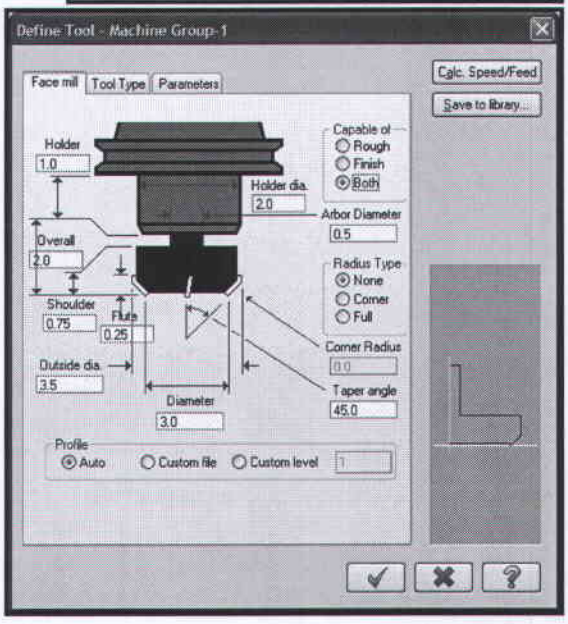
- Right-mouse click in the **Toolpath parameters** and select **Create new tool**.



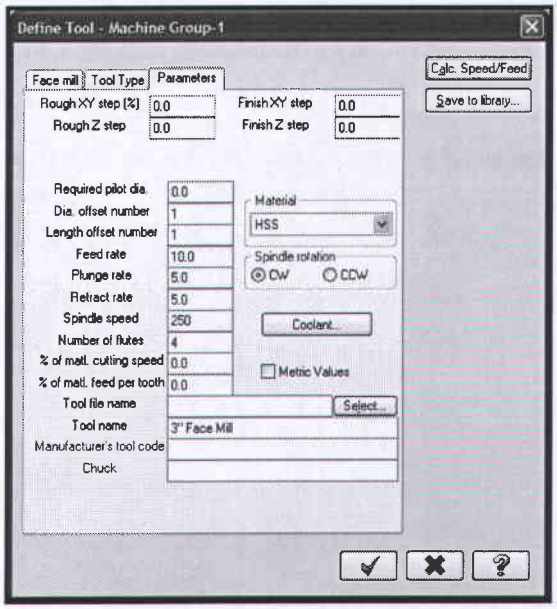
- Select **Face Mill** in the **Tool Type** as shown.



- Change the **Diameter** to 3.0" and the **Outside dia** to 3.5", and press Enter.
- The **Facing** dialog box should look as shown in the screenshot to the right.

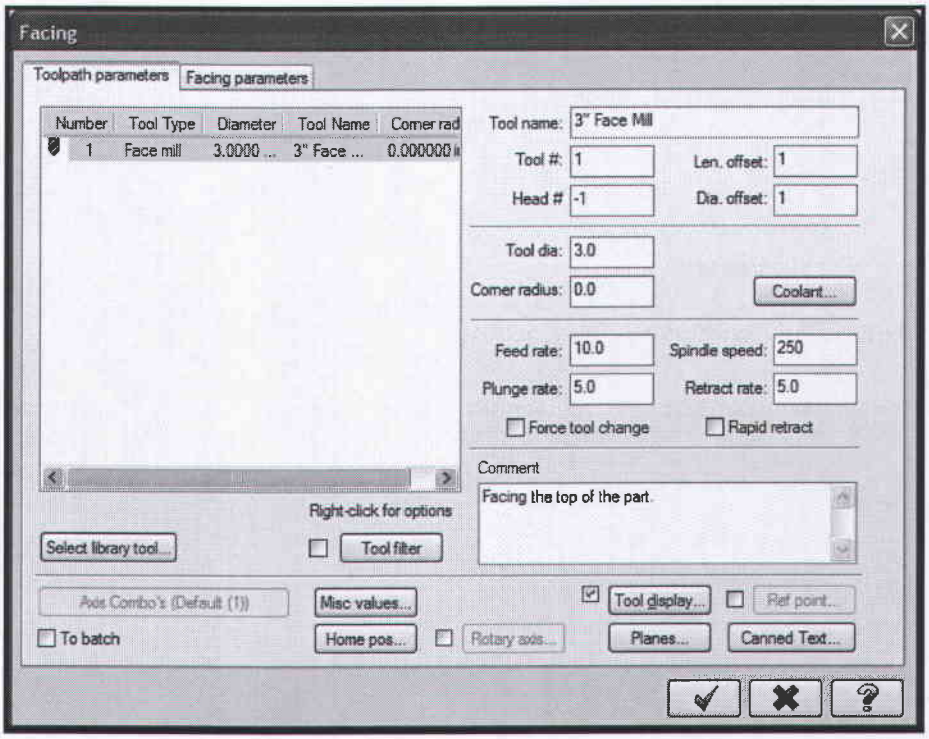


- Select the **Parameters** tab and make the changes as shown to the right.
- Type in the **Tool name** field: **3" Face Mill.**



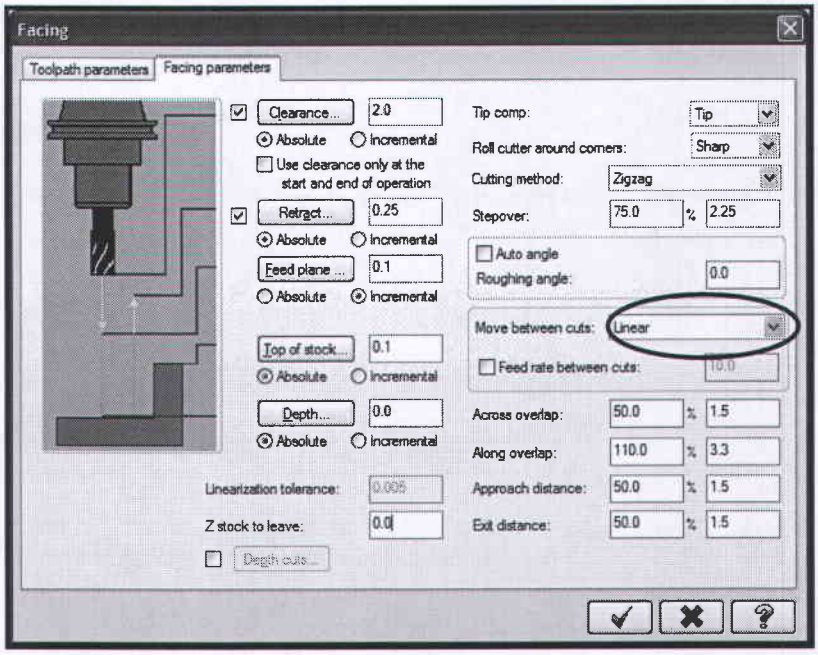
- The tool will be available just for the current job.To save the tool in a library select **Save to library** button .

- Select the **OK** button to close **Define tool** dialog box. 
- Change the parameters in the **Toolpath Parameters** page as shown.



➢ Select the **Facing parameters** page and change the parameters as shown.
➢ Select the drop-down arrow and change the **Move between cuts** to **Linear**.

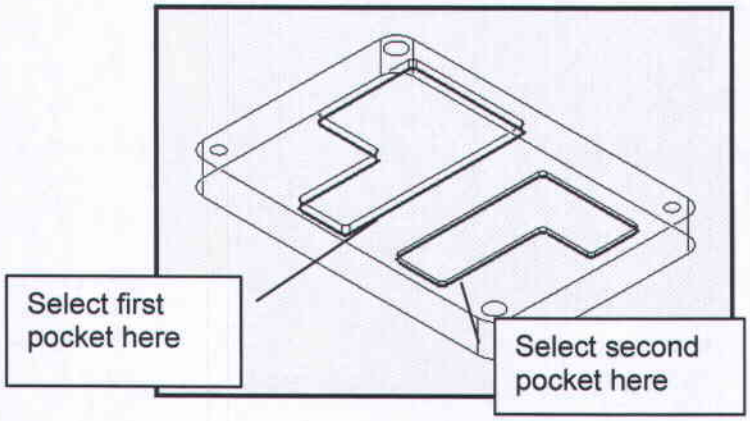
Clearance value sets the height at which the tool rapids to or from the part.
Retract value sets the height the tool rapids/feed-rates up to, before the next step down.
Top of stock value sets the height of the stock in the Z-axis. It is based on the job setup values.
Depth value sets the final machining depth for the facing operation.

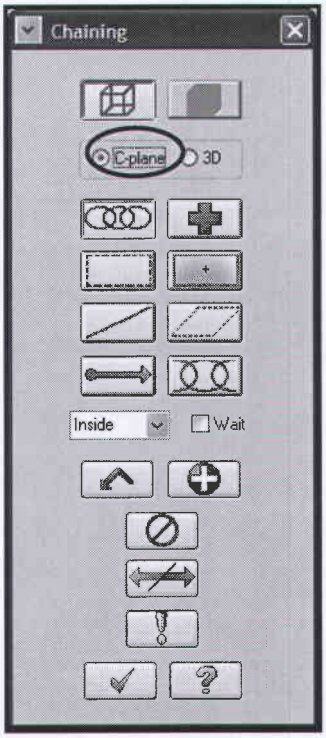


➢ Select the **OK** button to exit.

STEP 9:
ROUGH CUTTING THE TWO POCKETS.

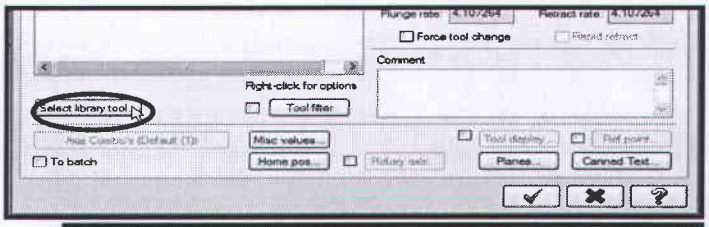
Toolpaths
➢ **Pocket Toolpath**
➢ Enabled **C-plane** in **Chaining** to be able to select the chains without stopping at the branches.
➢ Select the two pockets at the bottom, as shown.



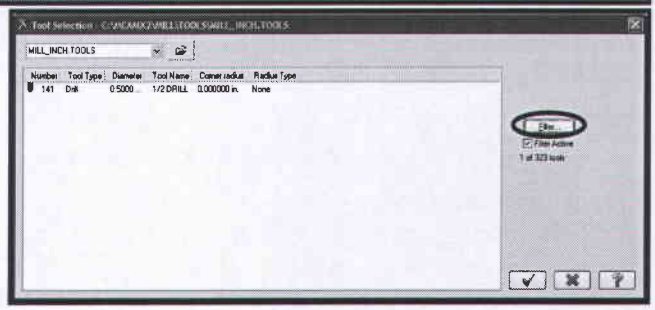


Note that the entire pockets are highlighted.
➢ Select the **OK** button to exit **Chaining**.

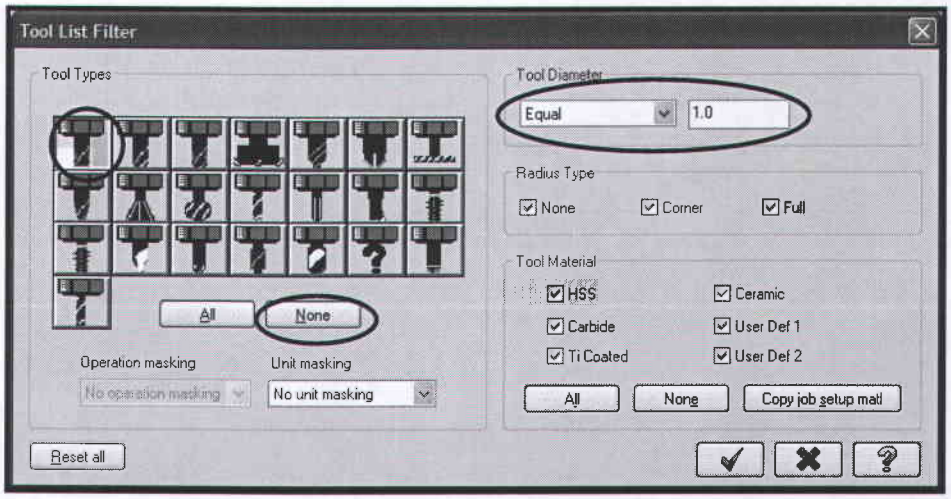
➢ Click on the **Select library tool**.



➢ Select the **Filter** button in the **Tool Selection**.

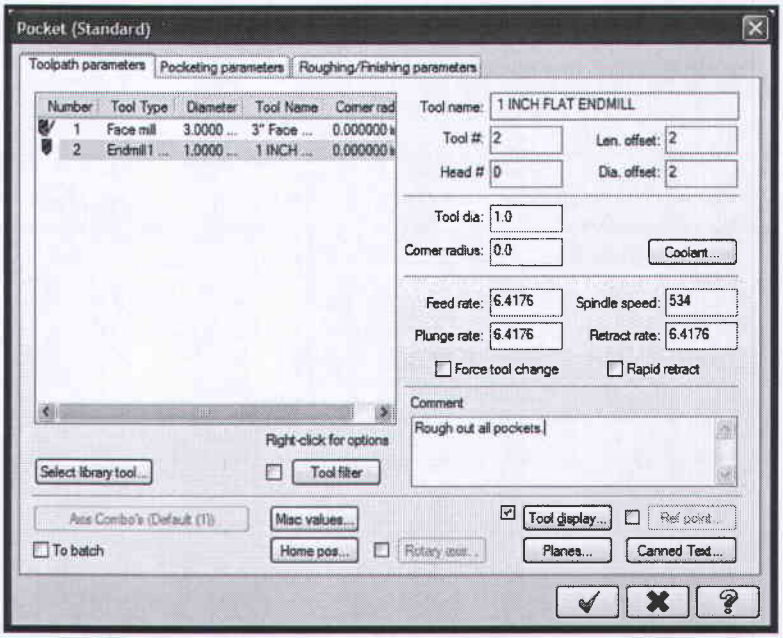


➢ In the **Tool Types** field select the **None** button to disable all tools.

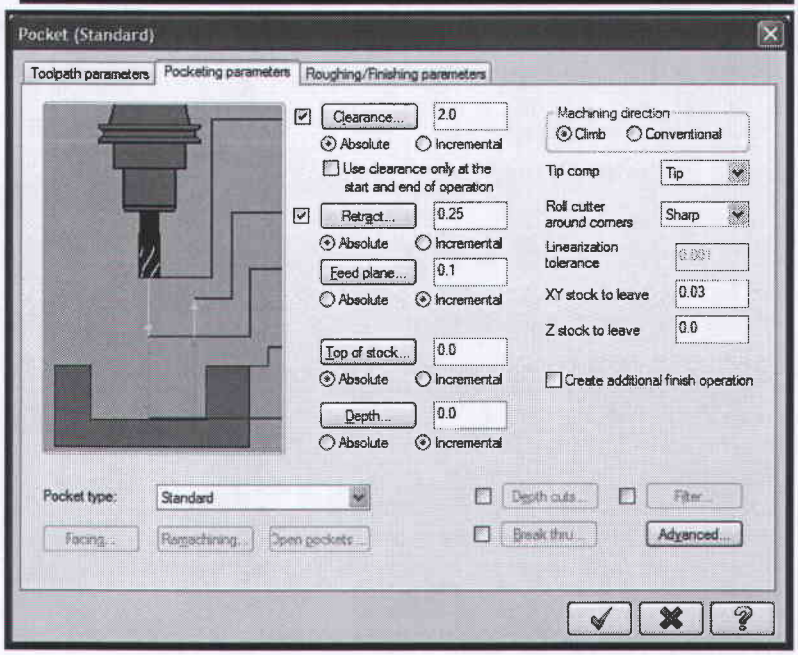


➢ Select the **Flat Endmill** tool type as shown (upper right corner).
➢ In the **Tool Diameter** field click the pull-down arrow and select **Equal**.
➢ Enter the **Tool Diameter** value to 1.0.
➢ Select the **OK** button to exit **Tool List Filter**.
➢ Make sure that the tool is selected (highlighted).
➢ Select the **OK** button to exit the **Tool Selection** dialog box.

➢ Make the necessary changes to match the parameters with the screenshot to the right.



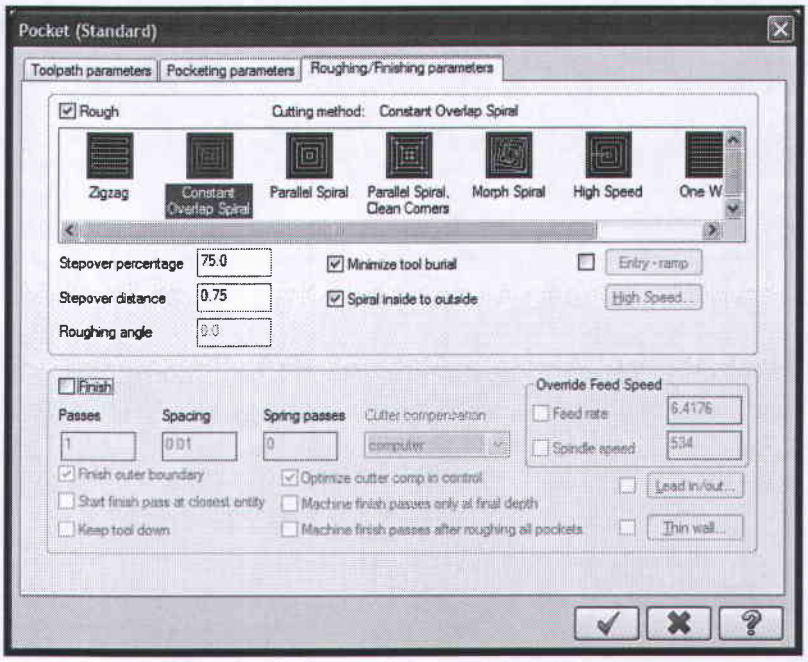
➢ Select the **Pocket parameters** page and change the parameters as shown.



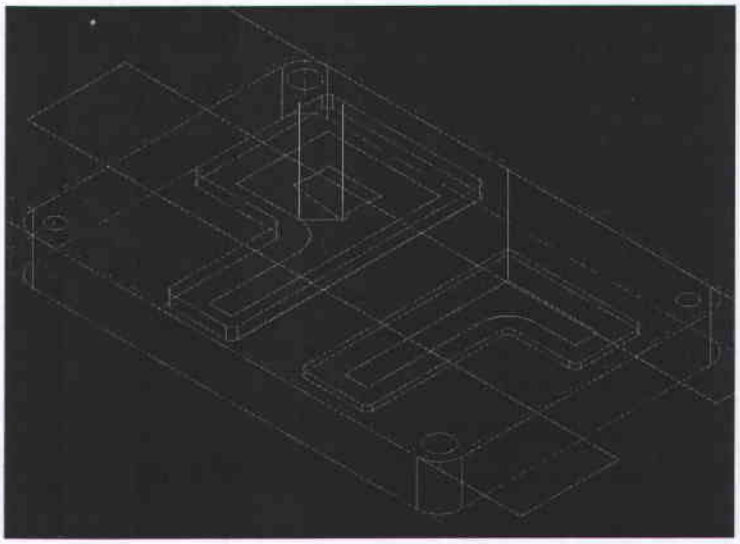
Depth value sets the final machining depth for the pocket operation. The value is set to **0** and **incremental** and is measured from the two geometry chains that we selected. This insures that both of them are going to be machined to the appropriate depth.
Choosing **Incremental** tells the system to calculate the value relative to either the current top of stock (as with **Clearance** parameter), relative to the selected geometry (as with **Top of stock** and **Depth** parameters), or relative to the depth of each cut (as with **Feed plane** and **Retract**).

➢ Select the **Roughing/Finishing parameters** tab.

Stepover percentage sets the distance between roughing passes in the XY axis as a percentage of the tool diameter and will automatically update the stepover distance.
Spiral inside to outside enabled allows you to spiral from the center to the pocket wall.
Finish area enabled allows the tool to make another cut around the pocket walls to "contour" them.



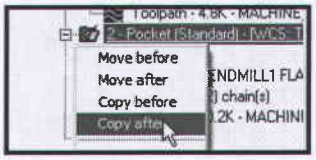
➢ Select the **Constant Overlap Spiral** as the **Cutting method**.
➢ Disable the **Finish** area.
➢ Select the **OK** button to exit **Pocket parameters**.



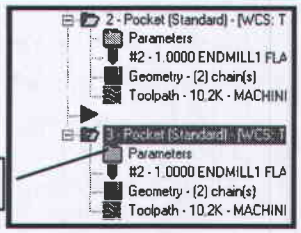
STEP 10:
REMACHINING THE TWO POCKETS.

Note that the 1.0" Flat End Mill could not clean the 0.125" radius fillets. Using a 1/8" Flat End Mill to remove all the material inside the pocket will not be efficient. We will remove the remaining material using an 1/8" Flat End Mill tool with the remachining pocket style.

➢ Select **Toolpath Manager**.
➢ **Right-mouse click** and hold it down on the folder icon in front of the **Pocket toolpath**.
➢ Drag the mouse down and release it.
➢ Select **Copy after.**



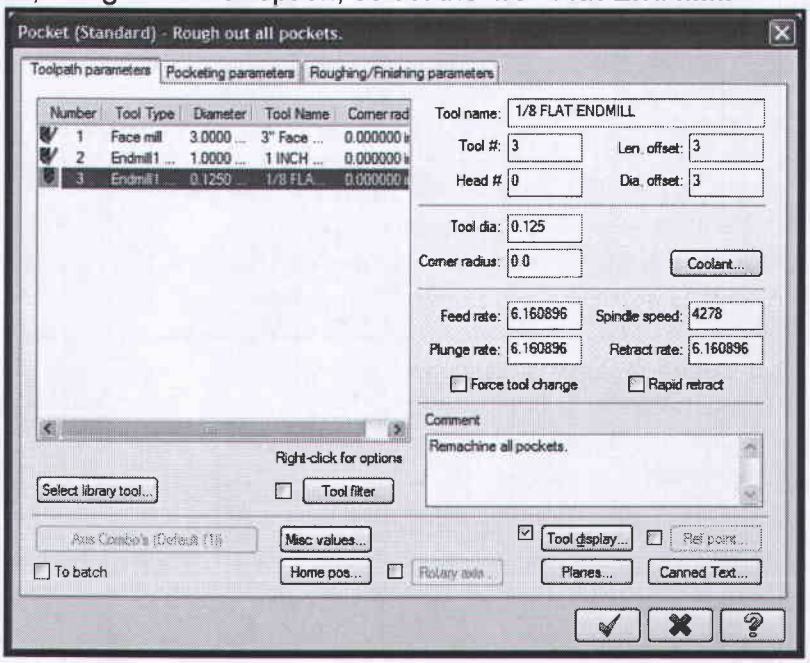
- You should now have two pocket toolpaths.
- Left-click on the second pocket **Parameters**.



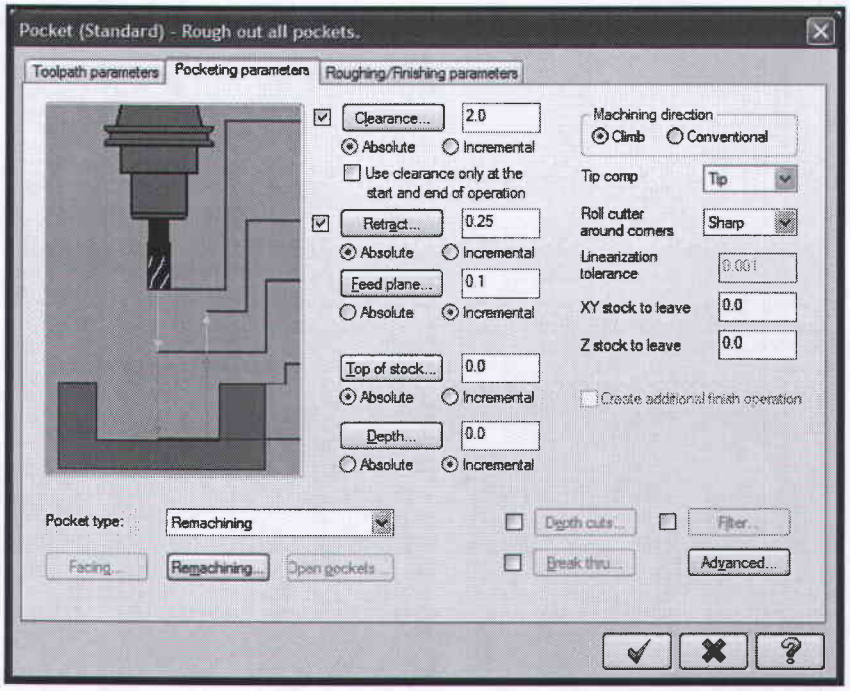
- Select the **Toolpath parameters** page.
- Click on **Select library tool**.

Select Parameters

- Following the steps outlined earlier, using the **Filter** option, select the **1/8" Flat End Mill.**

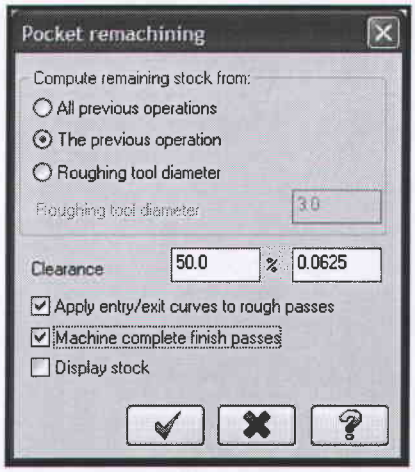


- Select and change the parameters in the **Pocket Parameter** dialog boxes, as shown below.

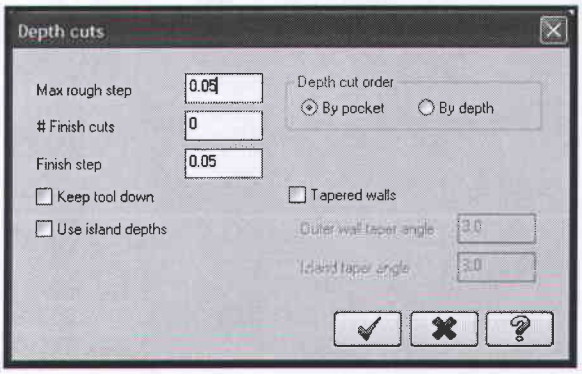


➢ Select the drop-down arrow in the **Pocket type** field and select **Remachining**.
➢ Select the **Remachining** button and match the parameters with the following screenshot.

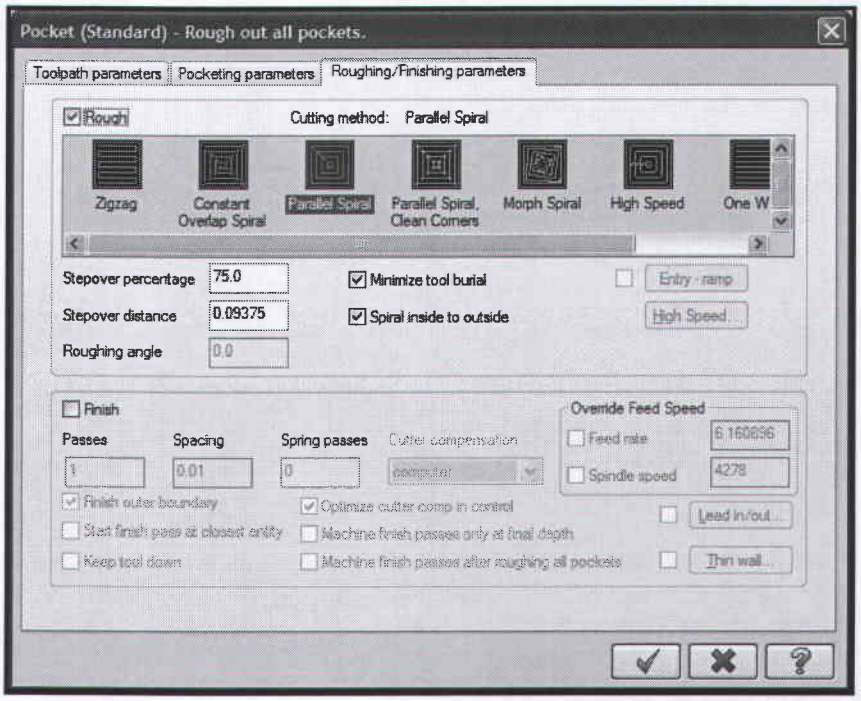
Compute remaining stock from The previous operation enables the system to calculate the remaining stock for remachining by determining the stock left after the previous operation.
Clearance, set as a percentage of the tool diameter, allows you to expand the remachining area at the beginning and at the end to prevent a cusp of material remaining.
Apply entry/exit curves to rough passes allows you to use the Lead in/out parameters.
Machine complete finish passes allows you to finish the entire part.



➢ Select the **OK** button to exit.
➢ Select the **Depth cuts** button and change the parameters as shown.
➢ Select the **OK** button to exit.

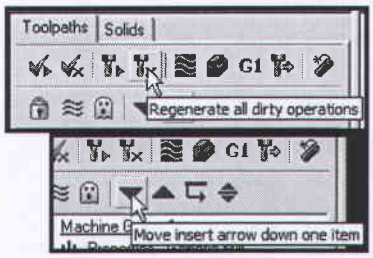


➢ Check the parameters in the **Roughing/finishing parameters** page to match the following screenshot.

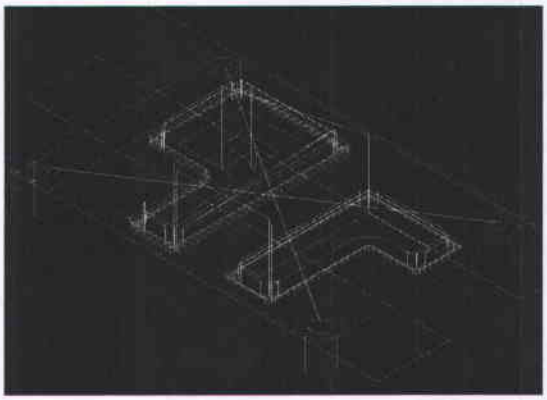


➢ Select the **OK** button to exit.

➢ Select the **Regenerate all dirty operations** icon.

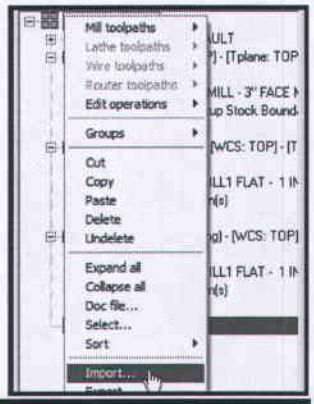


➢ Select the **Move insert arrow down one item** icon.

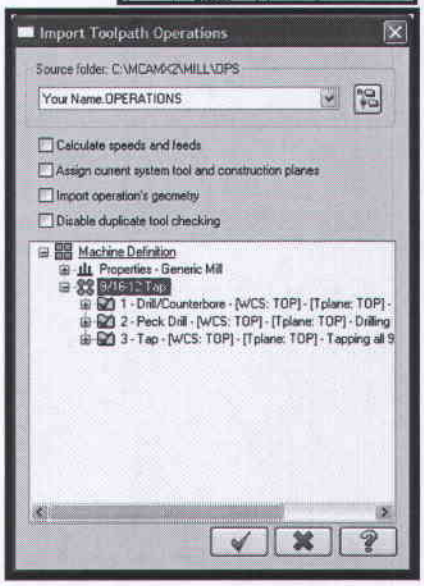


STEP 11:
IMPORT FROM LIBRARY 9/16 TAP HOLES.

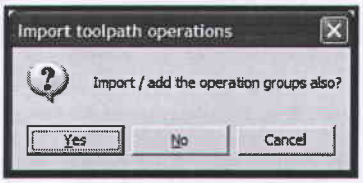
➢ **Right-mouse click** in **Toolpaths Manager** and select **Import**.



➢ Select the drop-down arrow in the **Source folder** field and select **Your name. Operations**.
➢ The three drilling operations used for the 9/16-12 tap will be listed.
➢ Make sure that all operations are selected (click on toolpath group).
➢ Select the **OK** button.



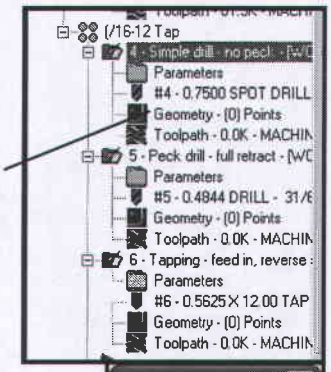
Select the **Yes** button to import the operation group too.



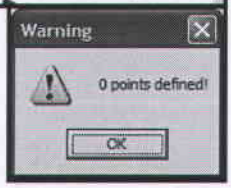
➢ Select the **OK** button.
➢ Select the upper **red X** button to exit **Import Toolpath Operations**.

Note that all operations have no geometry. You are going to add the center points and regenerate the toolpaths as shown in the following steps.

➢ Select **Geometry** in the "Simple drill - no peck" operation.

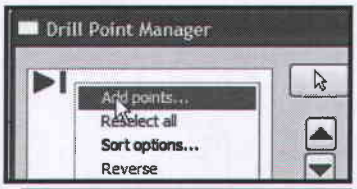


Select Geometry

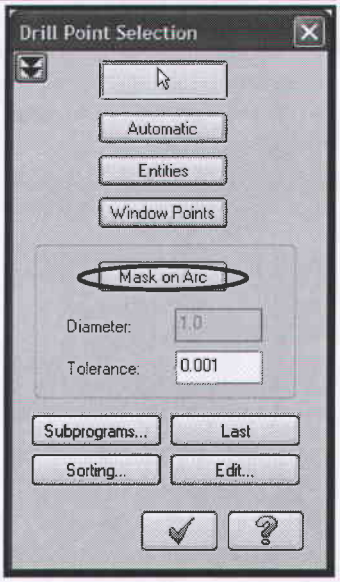


➢ Select the **OK** button to accept the warning.

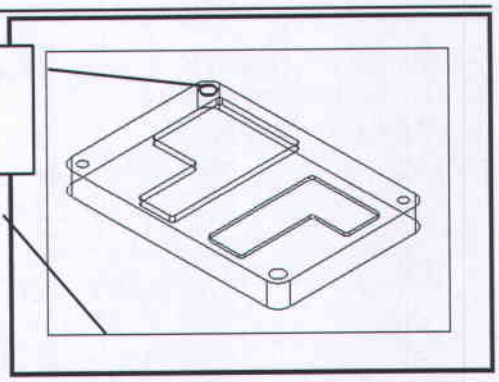
➢ **Right-mouse click** in the **Drill Point Manager** and select **Add points**.



➢ Select the **Mask on Arc** button.



➢ [Select arc to match]: Select the 9/16-12 arc.



➢ Make a window around the part as shown in the picture to the right.

➢ Hit **Enter** when finished.

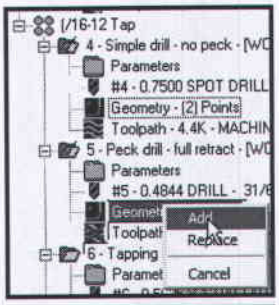
➢ Select the **OK** button to exit **Drill Point Selection**.

The **Drill Point Manager** should look as shown to the right.

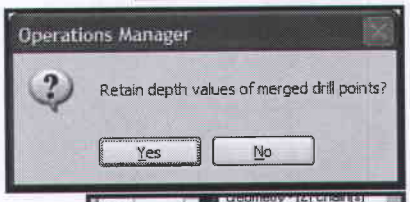
➢ Select the **OK** button to exit **Drill Point Manager**.

➢ Select the **Regenerate all selected operations** icon.

➢ **Right-mouse click** on the **Geometry** of the spot drilling operation, and hold down the button.
➢ Drag the **Geometry** of the spot drilling on the top of the drilling operation **Geometry**.
➢ Release the right-mouse button, and select **Add**.



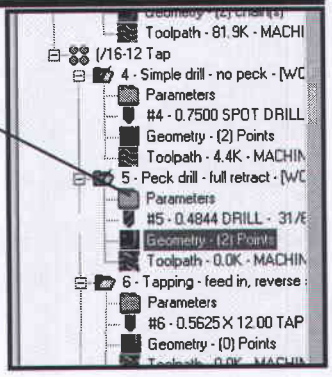
➢ Select **No.** We will need to change the drilling depth.



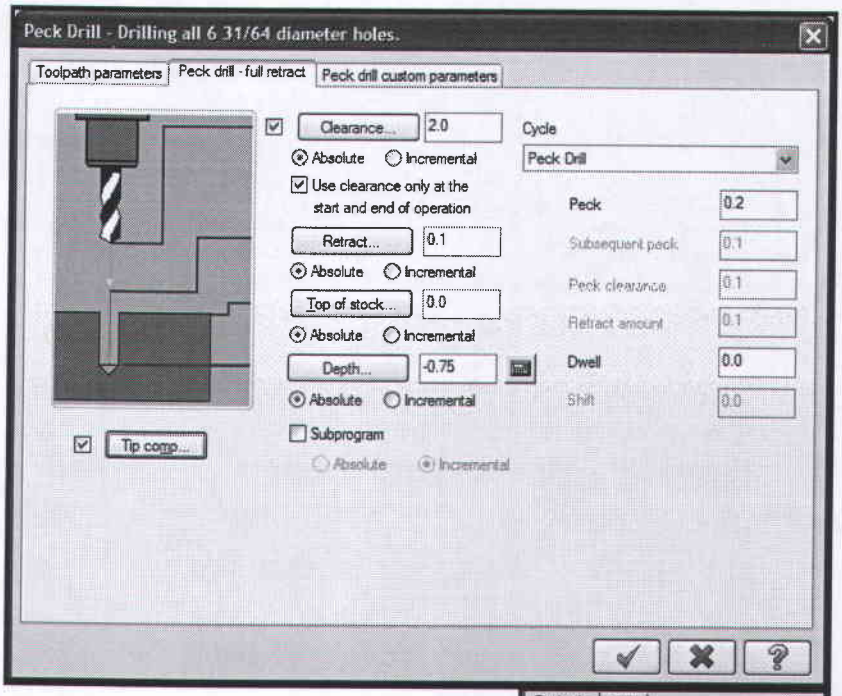
➢ Select **Drilling** operation **Parameters**.

➢

Select Parameters

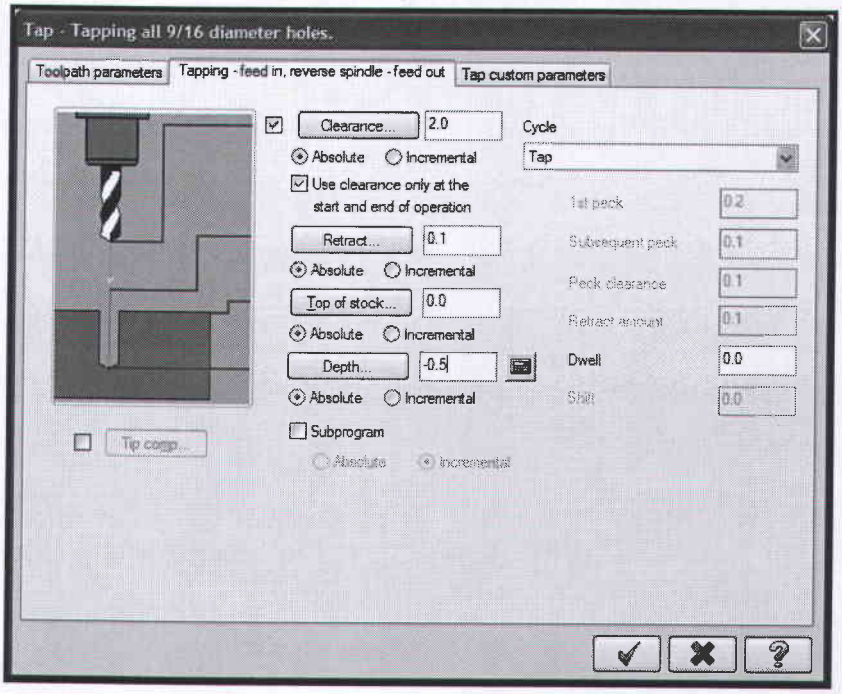


➢ Change the **Depth** to -0.75 and the **Drilling cycle** to **Drill/Counterbore**.



➢ Select the **OK** button to exit.

➢ Select the **Regenerate all selected operations** icon.
➢ Follow the steps outlined earlier and add the **Geometry** for the tapping operation.
➢ Select the tapping operation **Parameters** and change the depth to -0.5.



➢ Select the **OK** button to exit.

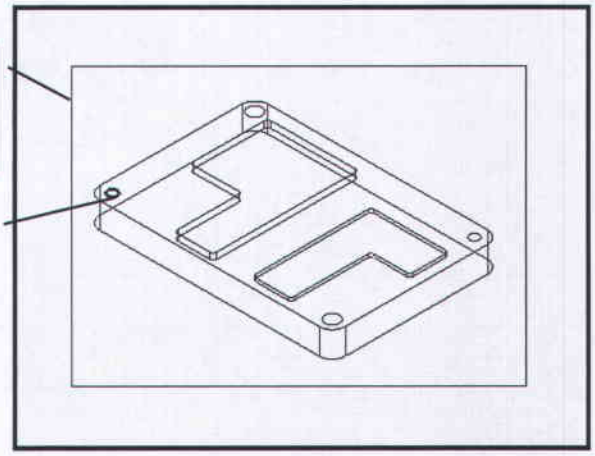
➢ Select the **Regenerate all dirty operations** icon.

STEP 12:
DRILLING 3/8 HOLES.

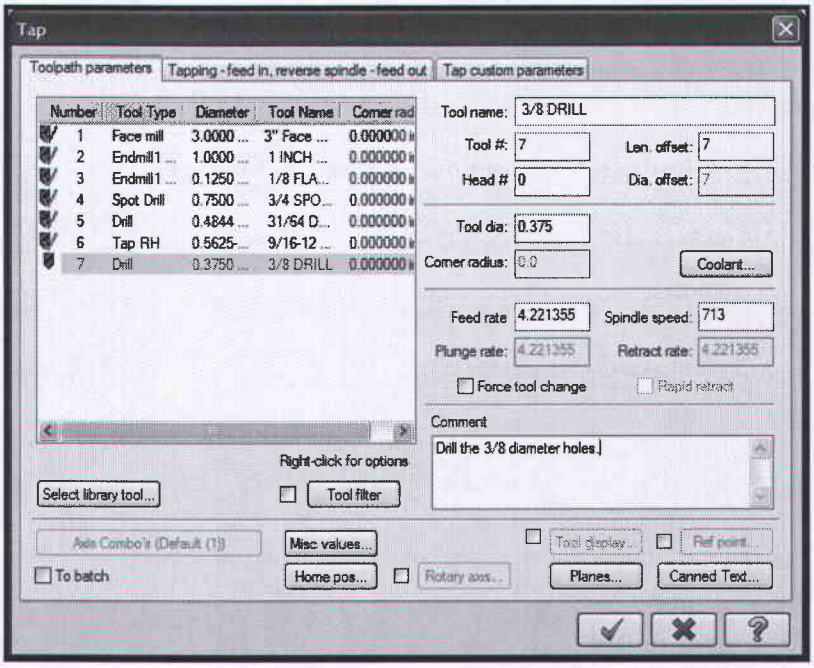
Toolpaths

- **Drill Toolpath**
- Select the **Mask on Arc** button.

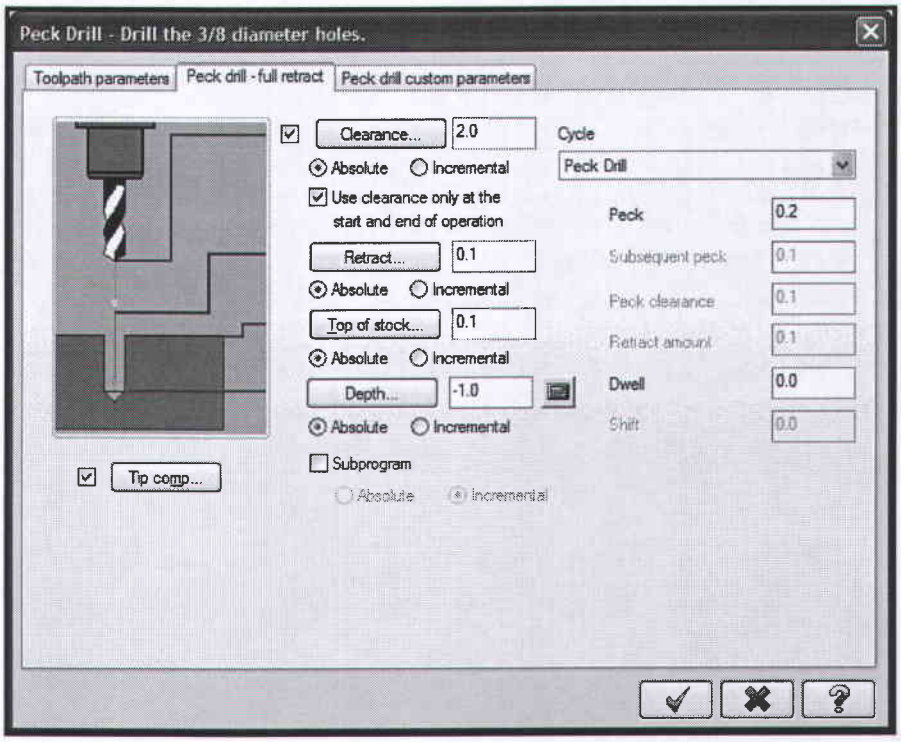
- [Select arc to match]: Select the 3/8 diameter arc.



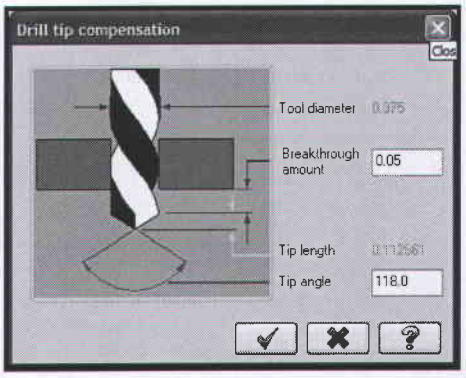
- Make a window around the part as shown in the picture to the right.
- Hit **Enter** button.
- Select the **OK** button to exit **Drill Point Selection**.
- Click on **Select library tool**.
- Using **Filter** select the **3/8 Drill** as shown in the previous steps.
- Change the parameters in the **Toolpath parameters** as shown below.



➢ Select the second page tab and change the parameters as shown.



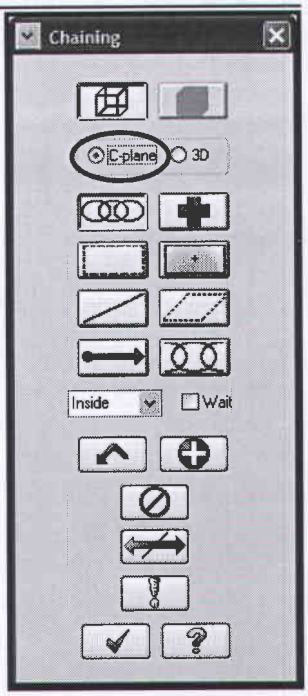
➢ Select the **Tip comp...** button and make the changes as shown.



Breakthrough amount value allows you to give an extra amount for the tool to go deeper than the final depth to prevent any remaining material for the cut-outs.
Tip length value is automatically calculated by the system based on the diameter and tip angle of the tool. The value is added to the final depth.

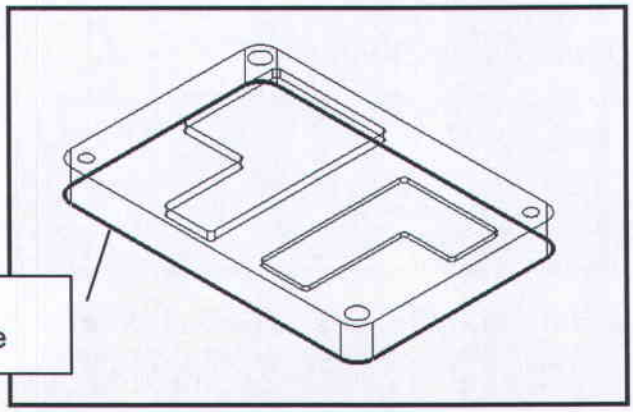
➢ Select the **OK** button twice to exit drilling parameters pages.

STEP 13:
CONTOUR THE OUTSIDE PROFILE.

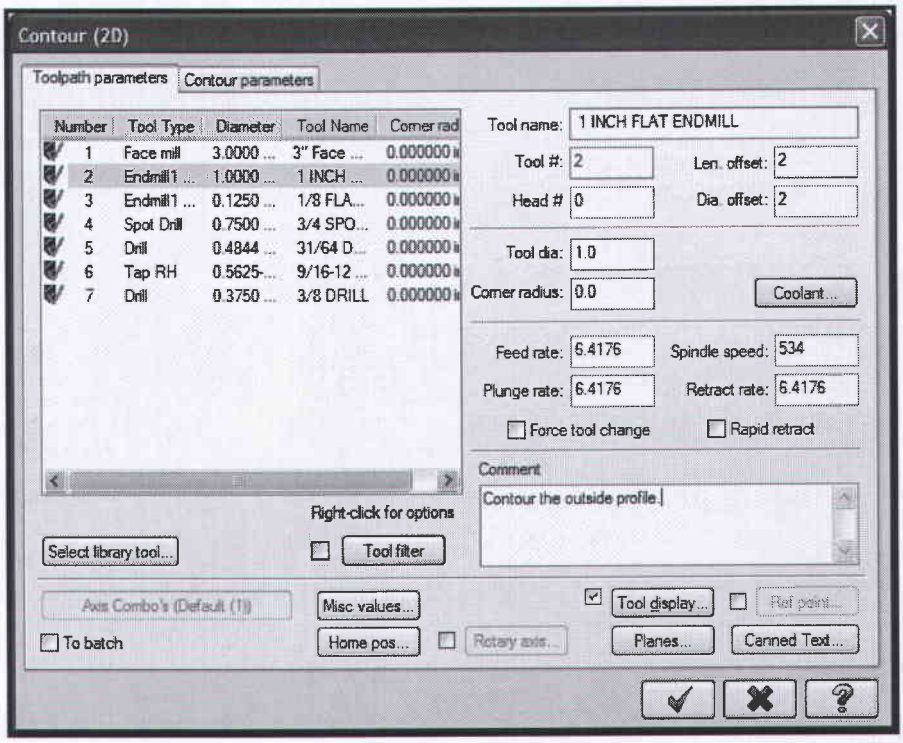


Toolpaths

- **Contour Toolpath**
- Make sure that **C-plane** is selected in the **Chaining** dialog box.
- Select the contour at the bottom.
- Select the first entity in the contour, as shown.
- Be sure to chain the contour in a CCW direction. Otherwise select the **Reverse** button.

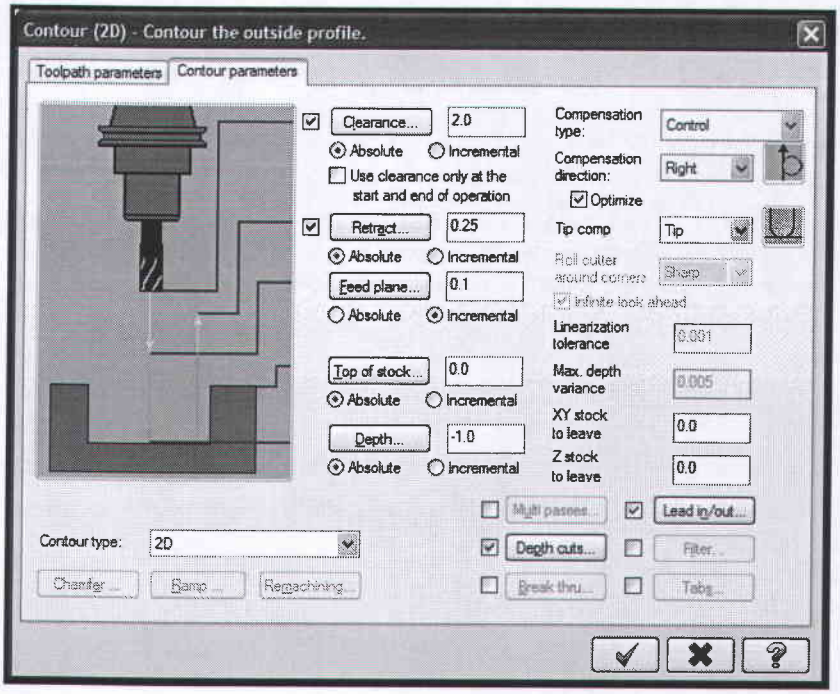


- Select the **OK** button to exit **Chaining**.
- Select the existing **1" Flat End Mill** and make all the necessary changes in the **Toolpath parameters** page.



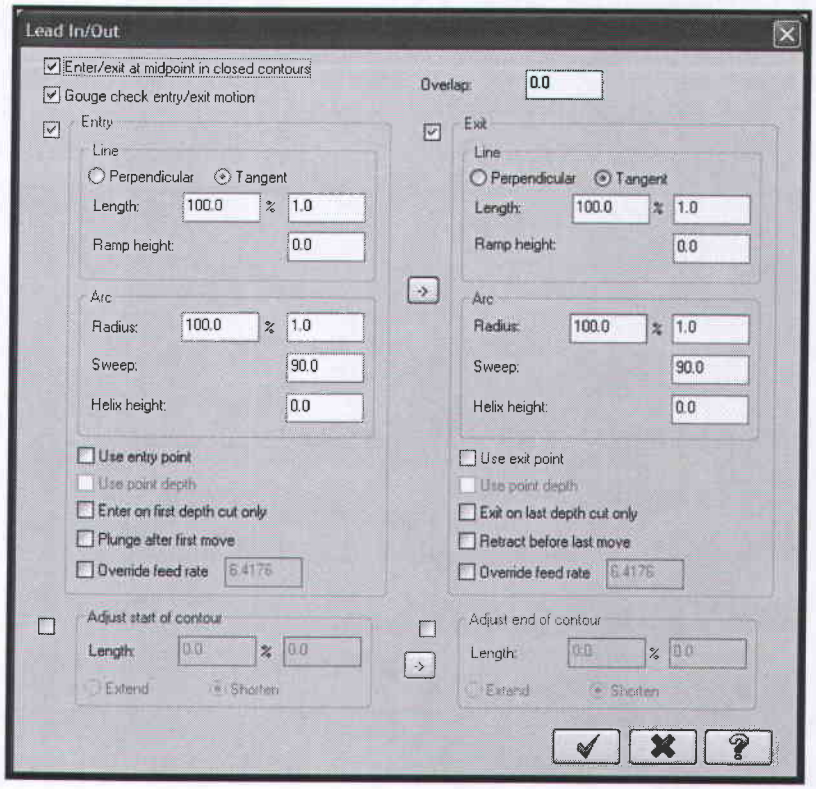
- The **Feed rate, Plunge rate, Retract rate** and **Spindle speed** are based on the tool definition. Change them as desired.

➢ Select the **Contour parameters** and make the changes as shown.



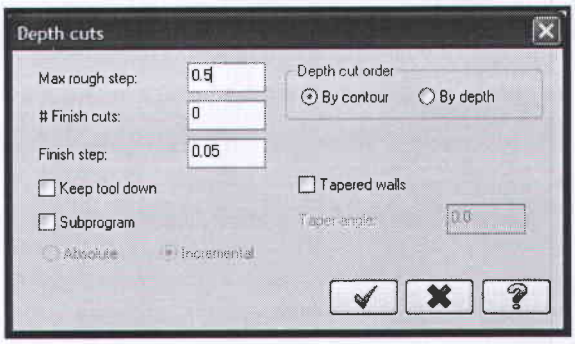
- Cutter compensation in control requires the Lead in/out parameter to be turned on; most of the CNC machines need a linear move at the beginning of the program to compensate the cutter diameter.
- If you use the arc option too, make sure that the radius is larger than the radius of the tool.

➢ Enable the box in front of the **Lead in/out** button and select a combination of one **Line** and an **Arc** at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting the part.

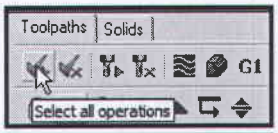


➢ Select the **OK** button twice to exit contour parameters.

➢ Select the **Depth cuts** button and change the max rough step to 0.5



STEP 14:
BACKPLOT THE TOOLPATH.

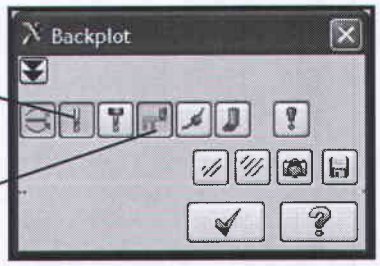


➢ Select the **Select all operations** icon to select all operations.

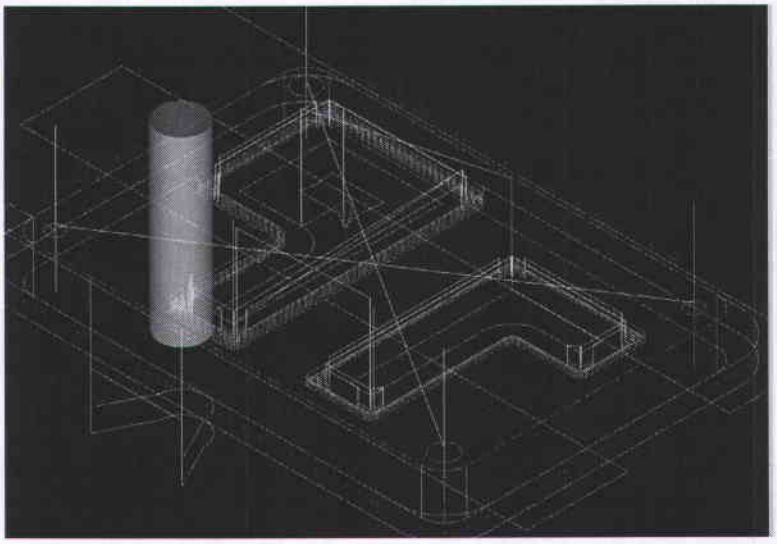
➢ Select the **Backplot** selected operations button.

➢ Make sure that you have the following buttons turned on (they will appear pushed down).

➢ **Display tool**
➢ **Display rapid moves**



➢ Select the **Play** button.

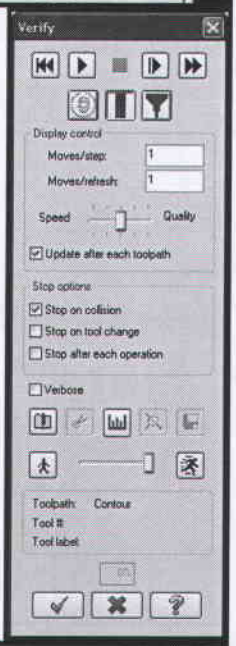


➢ Select the **OK** button to exit **Backplot**. 

VERIFY- TOOLPATH VERIFICATION

STEP 15: VERIFY.

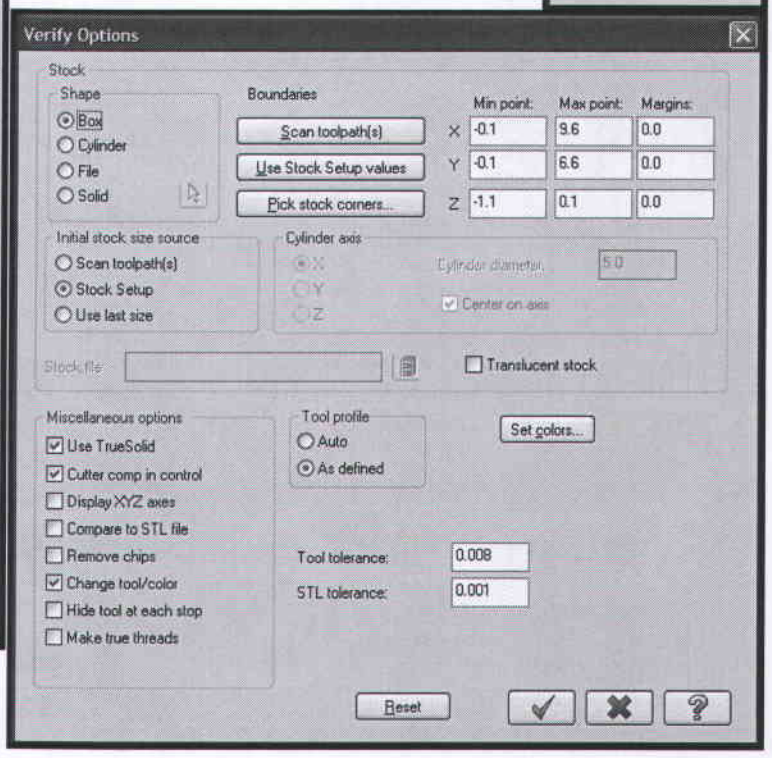
➢ Select the **Verify selected operations** button.



➢ Select the **Configure** button.

➢ Make the changes as shown.

Initial stock size source should be set to **Job Setup** to use the stock information from Stock Setup.
Use True Solid allows you, after verifying the part, to rotate and magnify it to more closely check features, surface finish, or scallops.
Cutter comp in control allows Verify to use the information regarding the tool diameter and to simulate the cutter compensation.
Change tool/color to change the color of the cut stock to indicated tool changes in the toolpath.

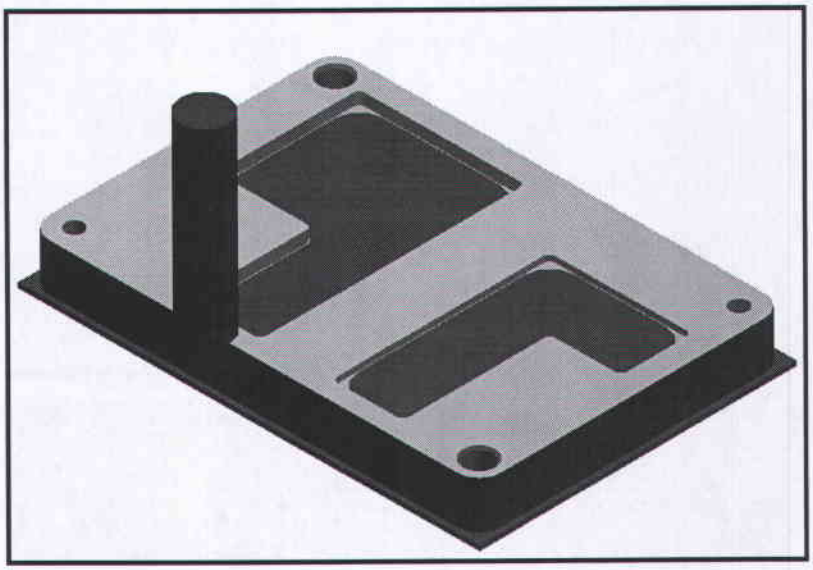


➢ Select the **OK** button to exit **Verify Options**.

➢ Set the **Verify speed** by moving the slider bar in the speed control bar. 

➢ Select the **Machine** button to start simulation. 

➢ The finish part should appear as shown in the picture to the left.



➢ Select the **OK** button to exit **Verify.** 

STEP 16: POST PROCESS THE FILE.

➢ Make sure that all operations are selected.

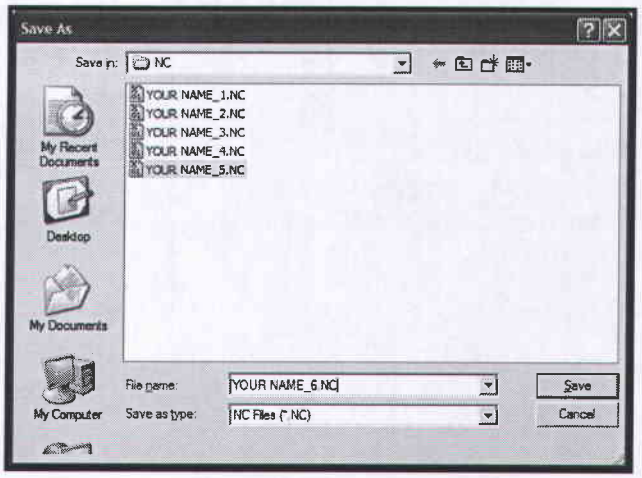
➢ Select the **Post selected operations** button from **Toolpath Manager**.

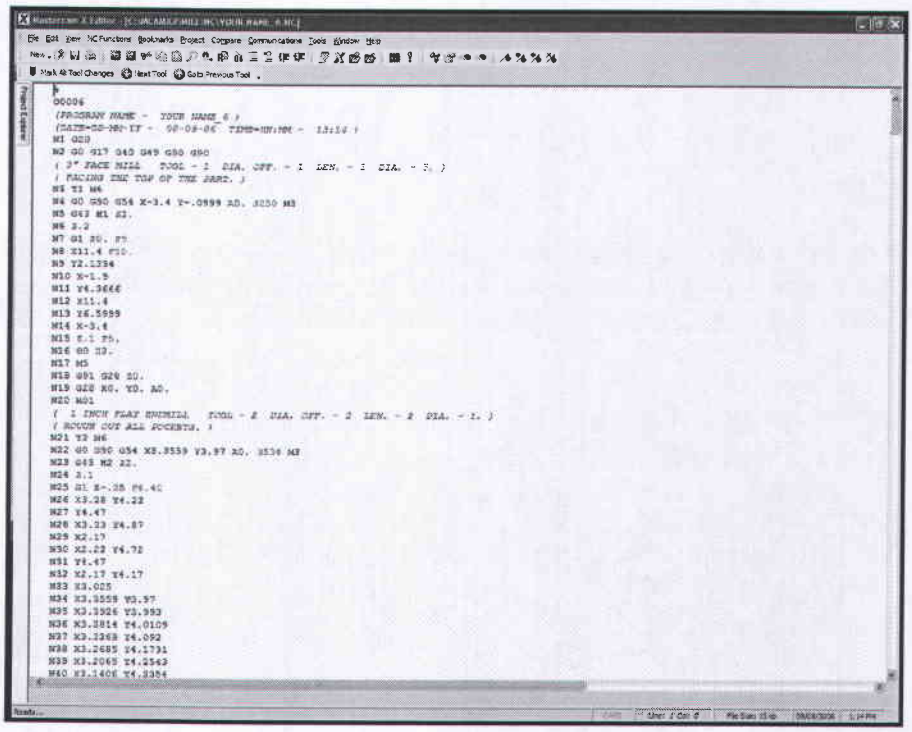
➢ In the **Post processing** window, make all the necessary changes as shown to the right.

NC file enabled allows you to keep the NC file and to assign the same name as the MCX file.
Edit enabled allows you to automatically launch the default editor.

➢ Select the **OK** button to continue. 

- Enter the same name as the geometry name in the **NC File** name field.
- Select the **Save** button.

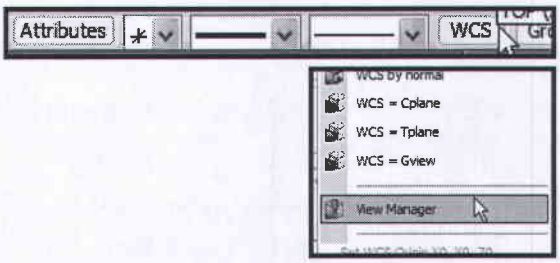




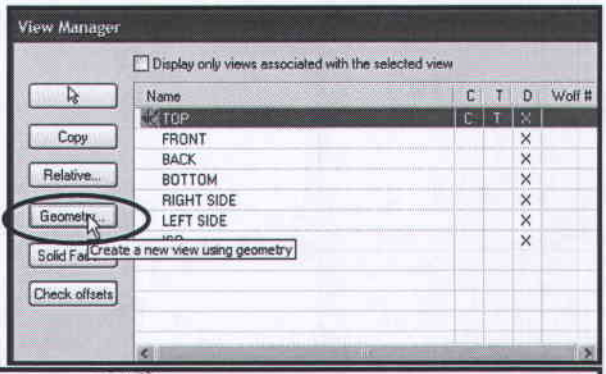
- Select the red **X** box at the upper right corner to exit the **Editor**.

MACHINE THE BOTTOM OF THE PART.
STEP 17:
SET THE NEW TOOL PLANE, CONSTRUCTION PLANE AND ORIGIN.

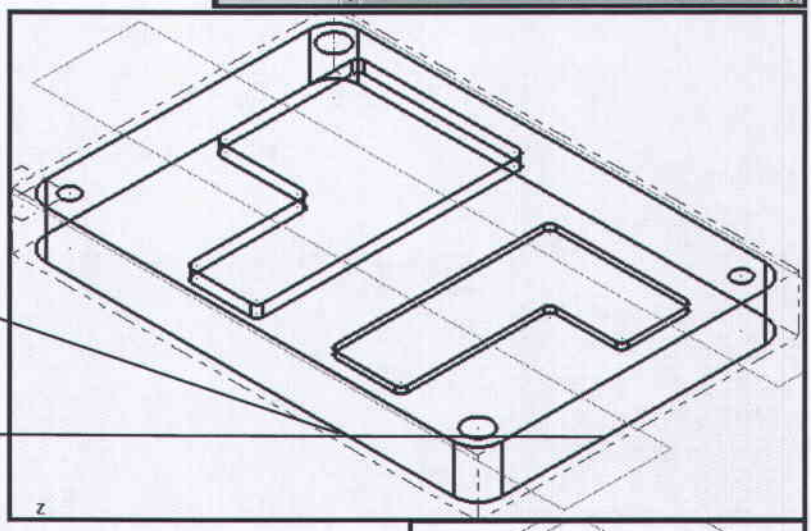
- Select **WCS** in the **Status Bar.**
- Select **WCS Manager.**



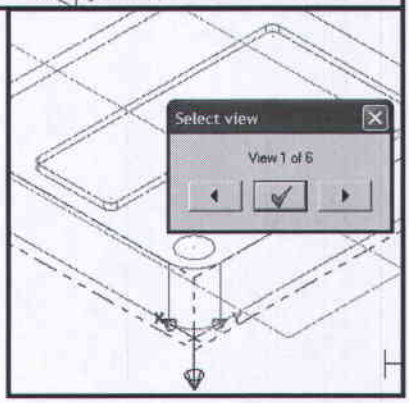
- Select **Geometry**

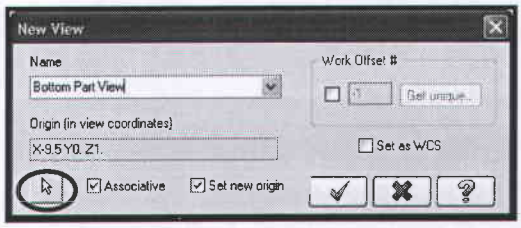


- Select the lines as shown below.

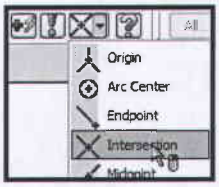


- Select the **OK** button to accept the view.

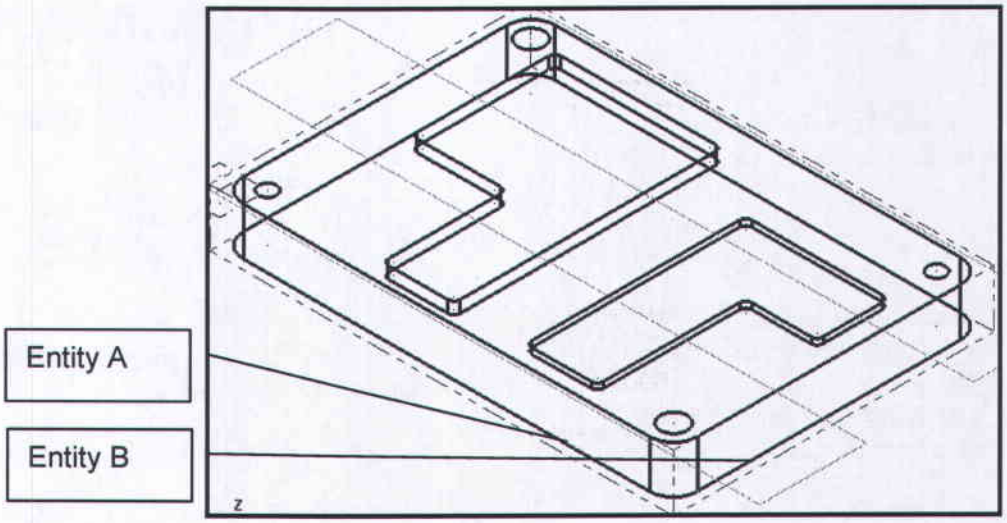




- Enter the New View name
- Make sure that Associative and Set new origin are enabled.
- Click on **Select** button to select the new **Origin** from the graphic area.

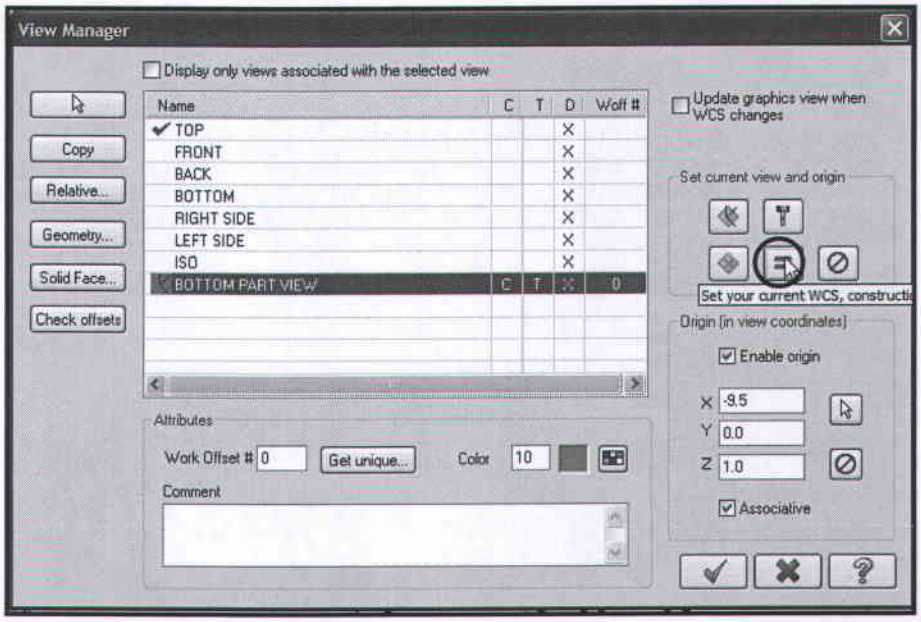


- Select the drop down arrow from the **AutoCursor Ribbon Bar** and select **Intersection**
- [Select line, arc or spline]: Select Entity A
- [Select line, arc or spline]: Select Entity B



- Select the **OK** button.

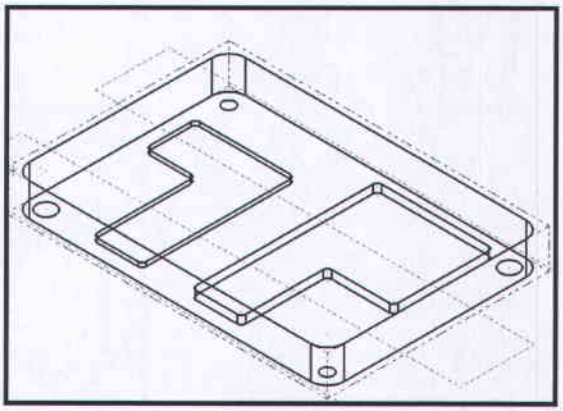
- Select **Bottom Part View** and click on **Set your current WCS, construction plane and tool plane with their origins to the selected view** button.



- Select the **OK**

button.

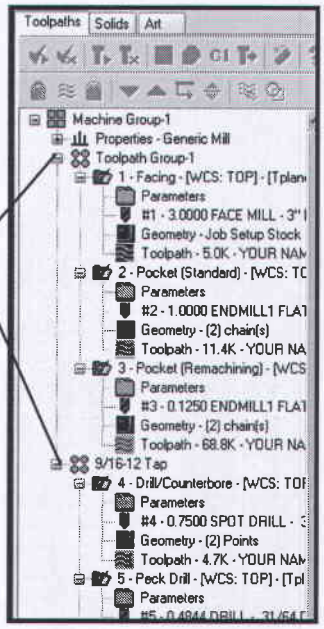
➢ Select the **Isometric view**.
➢ The part should look as shown to the right.



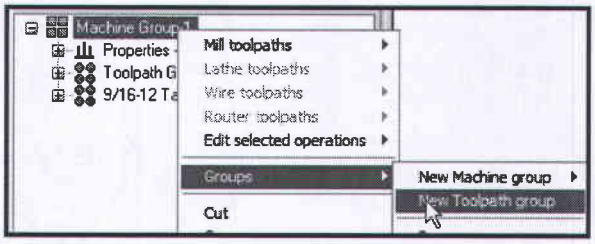
STEP 18:
FACE THE BOTTOM OF THE PART

➢ Click on the minus in front of Toolpath Group-1 and 9/16 -12 Tap to collapse all the operations.

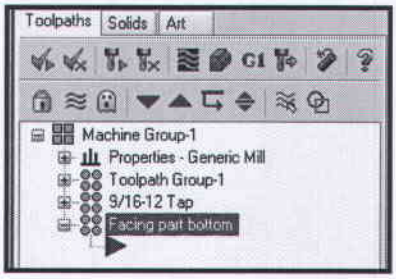
Select the minus sign in front of the toolpath groups



➢ Right-mouse click on the **Machine Group-1** and select **Groups** and **New Toolpath group**



➢ Enter the name of the group : "Facing part bottom"

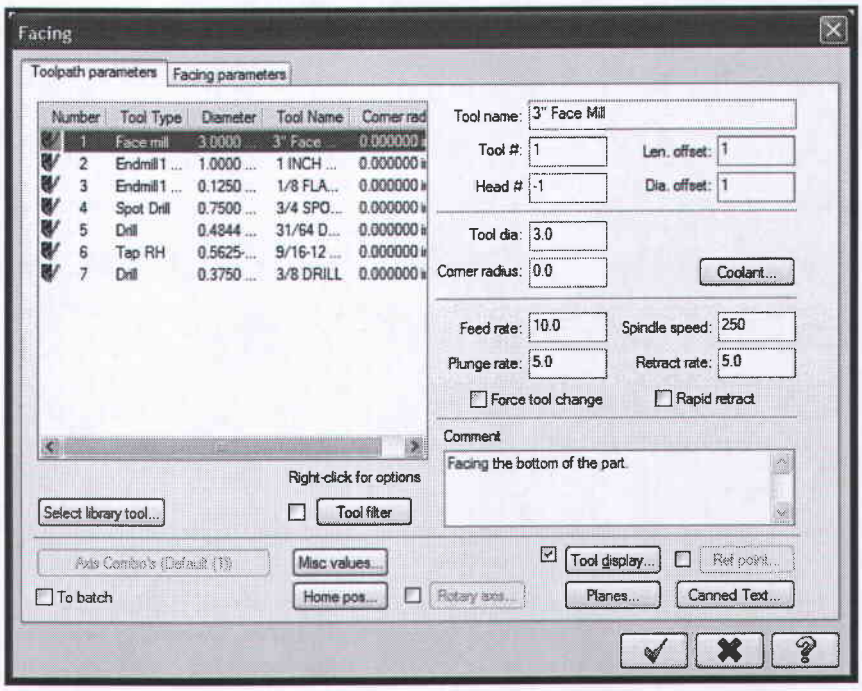


Toolpaths

➢ **Face Toolpath**

➢ [Select OK to use defined stock or select chain1]: Select the **OK** button to use defined stock.

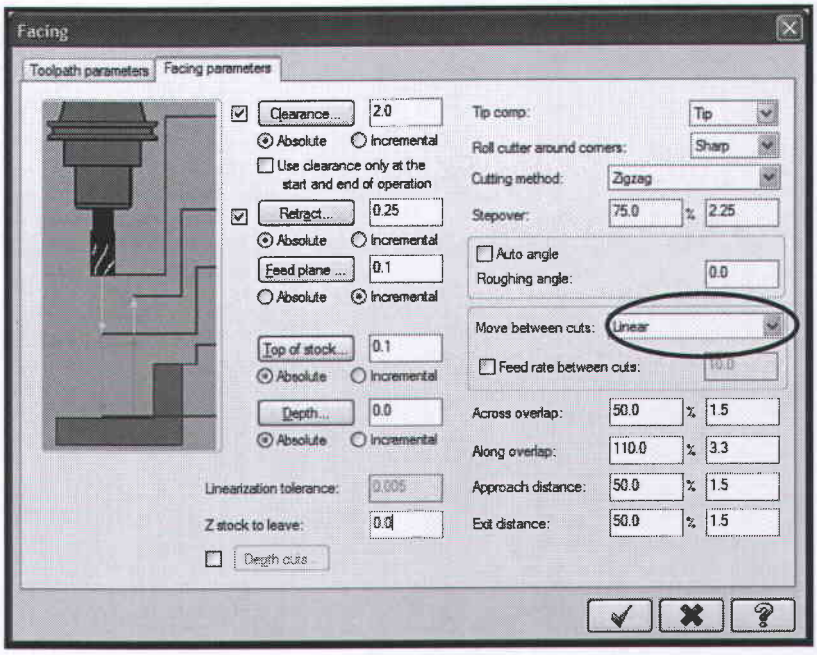
➢ Select the **3" Face Mill** and change the parameters in the **Toolpath Parameters** page as shown.



➢ Select the **Facing parameters** page and change the parameters as shown.

➢ Select the drop-down arrow and change the **Move between cuts** to **Linear**.

➢ Select the **OK** button to exit.



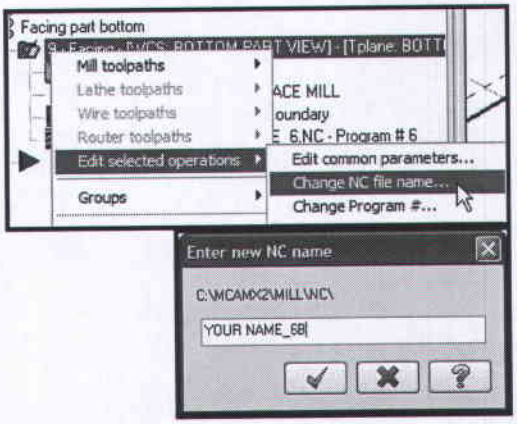
STEP 19:
BACKPLOT AND VERIFY THE OPERATION.

- Follow the instructions at pages 6-30 to 6-32

STEP 20:
POST THE LAST OPERATION ONLY.

- To be able to create a new program we need to rename the NC file.

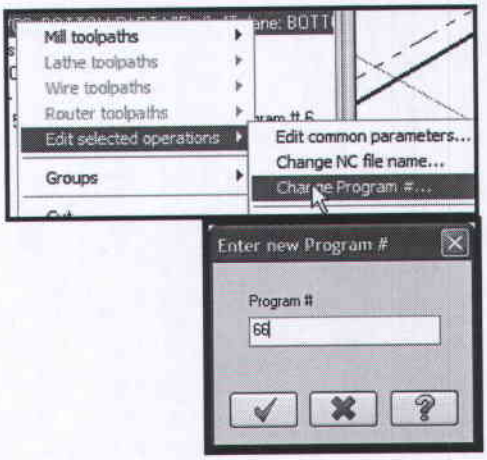
➢ Right –mouse click on the **Facing** operation and select **Edit selected operations** and **Change NC file name...**



➢ Enter the new name as shown. Your Name_6B

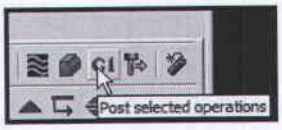
➢ Select the **OK** button.

➢ Right –mouse click on the **Facing** operation and select **Edit selected operations** and **Change Program #...**



➢ Enter a new number.

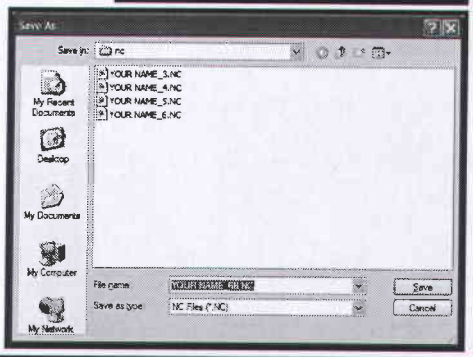
➢ Select the **Post selected operations** button from **Toolpath Manager**.

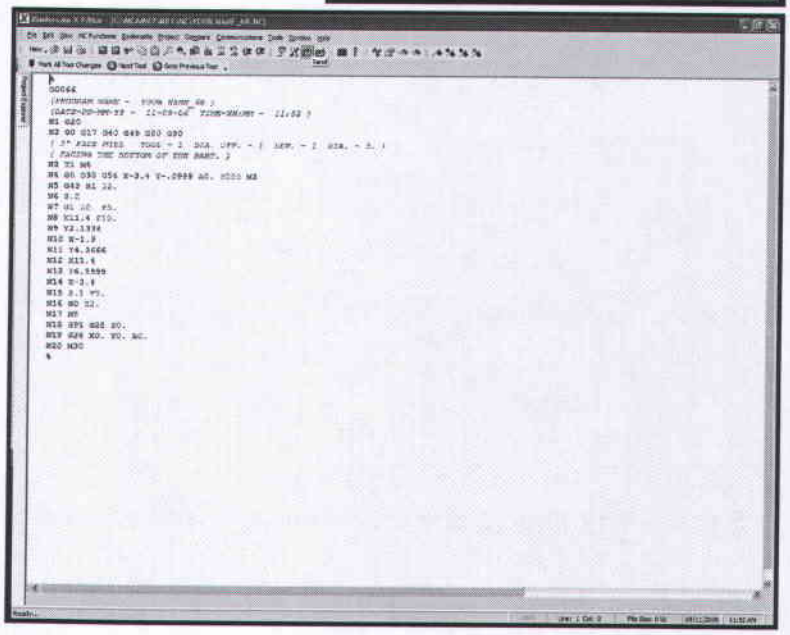


➢ In the **Post processing** window, make all the necessary changes as shown to the right.

➢ Select the **OK** button to continue.

➢ Select the **Save** button.

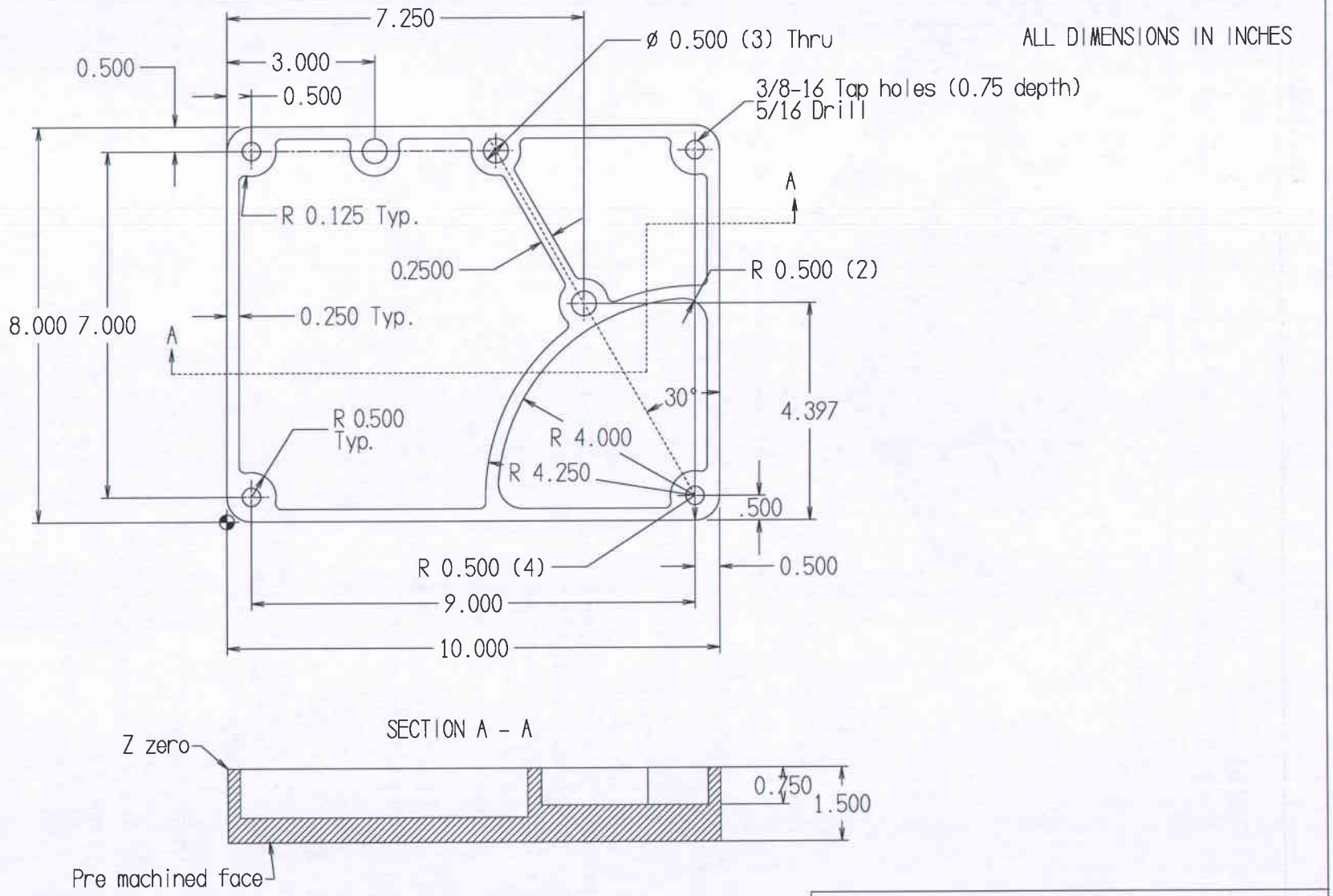




➢ Select the red **X** box at the upper right corner to exit the **Editor**.

STEP 21:
SAVE THE UPDATED MCX FILE.

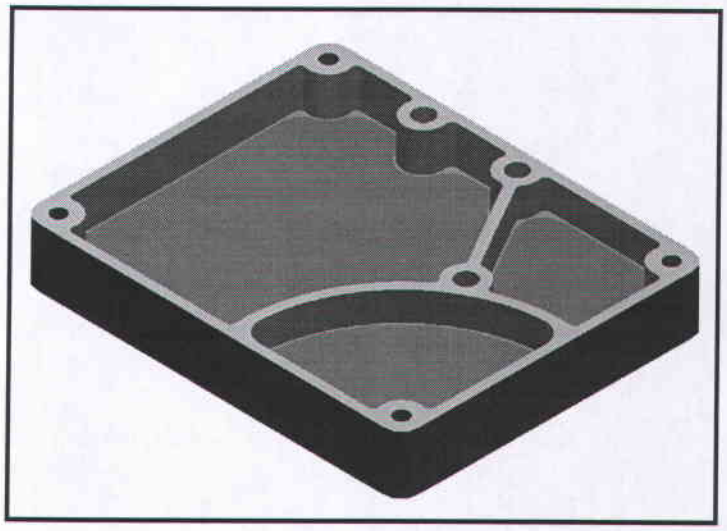
➢ Select the **Save** icon.



TITLE TUTORIAL 6 - EXERCISE 1	
MATERIAL ALUMINUM T6061	
DATE: NOVEMBER 12, 2005	eMastercam.com

REVIEW EXERCISES.

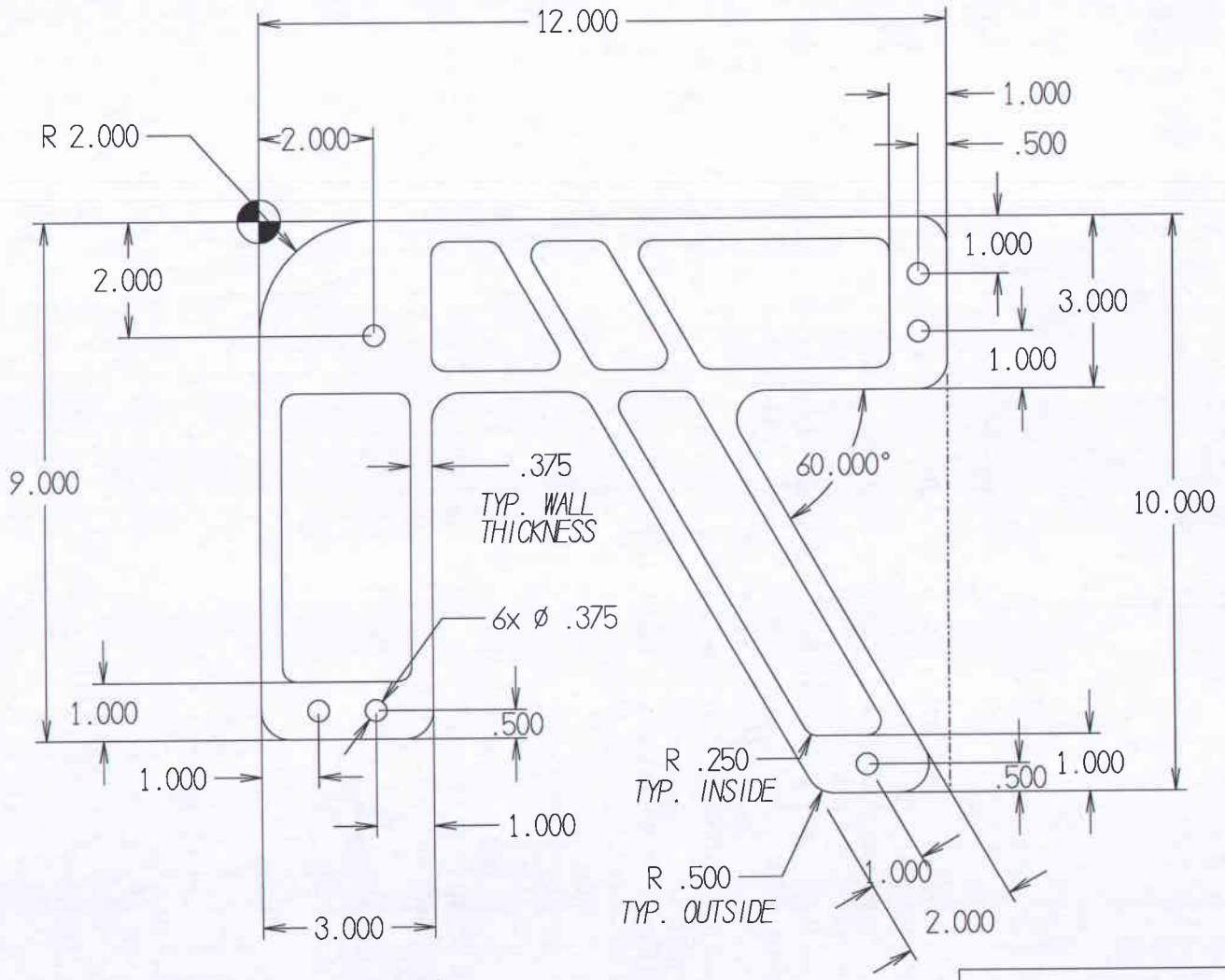
Student practise. Create the Toolpath for Exercise 1-Tutorial 6 as per the instructions below;



☞ **Tips:**

Create the 3D geometry using Xform/Translate.
Create each pocket with a different depth.
Stock size use **Bounding box** to establish X, Y & z sizes and
give extand X, Y, and Z 0.2
Face the top of the part using 3" Face Mill (create a new tool or get it from the library)
Spot Drill the 3/8" diameter holes using 1/2" Spot Drill
Drill the 3/8" diameter holes using 5/16" Drill
Tap the 3/8" diameter holes using 3/8 -16" Tap
Drill the 1/2" diameter holes using 1/2" Drill
Add the ½ holes center points in the **Spot Drill** and edit the depth of the holes.(See Tutorial 3)
Rough Pocket the part using 1" Flat End Mill
Select each pocket at the bottom
Depth = 0 (incr)
Max rough step (depth cuts) = 0.25
Stock to leave XY = 0.05
Use Parallel spiral clean corners cutting method
Remachining Pocket the part using 1/8" Flat End Mill
Contour the part using 1.5" Flat End Mill
2 depth cuts
1 rough pass; spacing = 0.1
1 finish pass; spacing = 0.05
Backplot and Verify the toolpaths.
Post process the file.

ALL DIMENSIONS IN INCHES



PART THICKNESS = 1.000"
MACHINE ALL POCKETS 0.750 DEEP

TITLE TUTORIAL 6 - EXERCISE 2	
MATERIAL ALUMINUM T6061	
DATE: JUNE 12, 2000	eMastercam.com

Student practise. Create the Toolpath for Exercise 2-Tutorial 6 as per the instructions below;

☞ **Tips:**

Create the 3D geometry using Xform/Translate.
Create each pocket with a different depth.
Stock size use **Bounding box** to establish X &Y sizes and
give Z = 1.2" and the stock origin Z =0.2
Face the top of the part using 3" Face Mill (create a new tool or get it from Big.tl8 library)
Center Drill the 3/8" diameter holes using 1/4" Center Drill
Drill the 3/8" diameter holes using 3/8" Drill
Contour the part using 1.5" Flat End Mill
2 depth cuts
Rough Pocket the part using 1" Flat End Mill
Select each pocket at the bottom
Depth = 0 (incr)
Max rough step (depth cuts) = 0.25
Stock to leave XY = 0.05
Use Parallel spiral cutting method
Remachining Pocket the part using 3/8" Flat End Mill
Finish Pocket the part using 3/8" Flat End Mill
Backplot and Verify the toolpaths.
Post process the file.

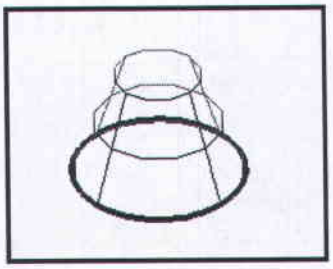
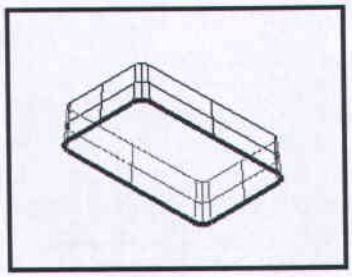
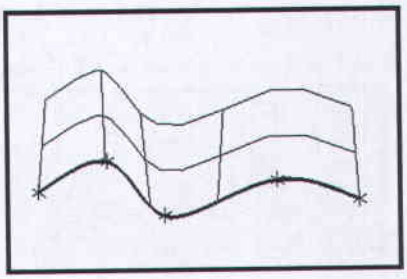
SURFACE MODELING

GEOMETRICAL SURFACES are surfaces that have a constant shape, such as blocks, cylinders, cones, spheres, or draft surfaces and revolved surfaces.

Draft Surface: a surface generated by extruding one or more contours, along a line defined by an angle and a length.
Applications: used to create an angled surface, a tapered wall, or cones and cylinders.

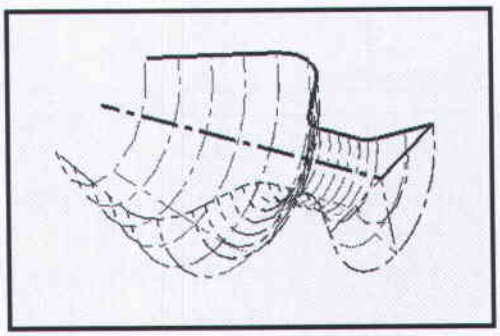
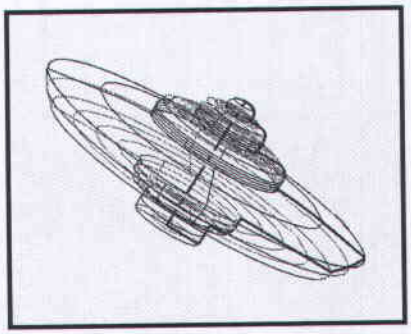
- A contour is defined as a chain of entities. Mastercam entities include points, lines, arcs, and splines. In order to form a chain, the distance between the entities should be less than 0.0001mm.

See **Tutorial 12** and **Tutorial 14**.
Examples

Revolved Surface: a surface generated by rotating a contour around an axis or a line.
Applications: on parts that require arc or circular cross sections.

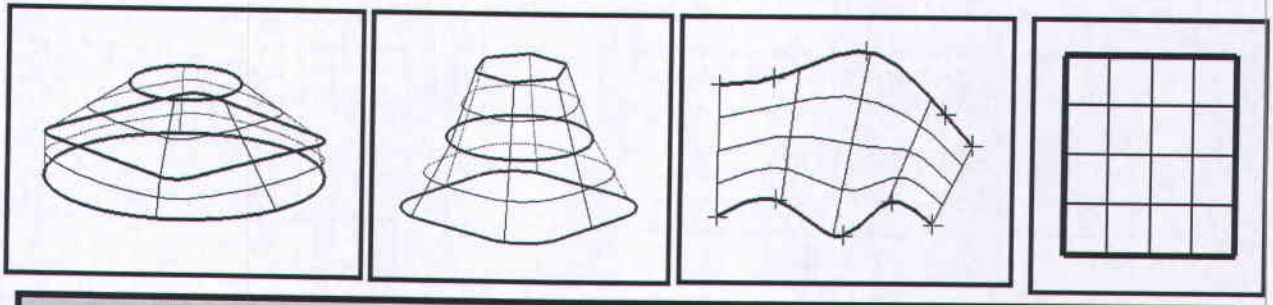
See **Tutorials 8, 14 and 15**.
Examples

FREE FORM SURFACES are surfaces generated from arbitrary curve segments and follow no set geometrical pattern.

Ruled Surface: a surface generated by connecting straight lines between two or more open or closed contours. As a result the surface has sharp edges at the intermediate contours.
Applications: any time a surface must be fit between two or more open or closed contours.

See **Tutorials 9, 12, 13 and 15.**
Examples

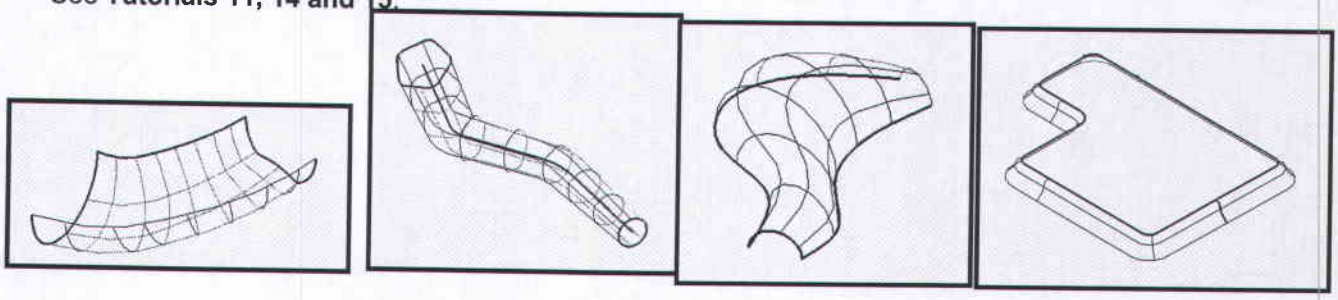


Loft Surface: a surface generated by blending together more than 3 open or closed contours. As a result the surface has rounded edges at the intermediate contours.
Applications: any time a surface must be parabolically blended through more than two curves.
- The system will automatically create a ruled surface through two chains of entities.

Examples

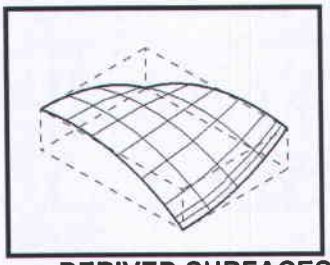
Swept Surface: a surface generated by translating or rotating one or more contours (across curves) along one or two other contours (along curves).
Applications: used when the across section of the surface at any point is constant (when the surface is generated from one across contour and one along contour).
Also used when the across section at any section is not constant (when the surface is generated from two or more across contours and one or two along contours).

See **Tutorials 11, 14 and 15.**

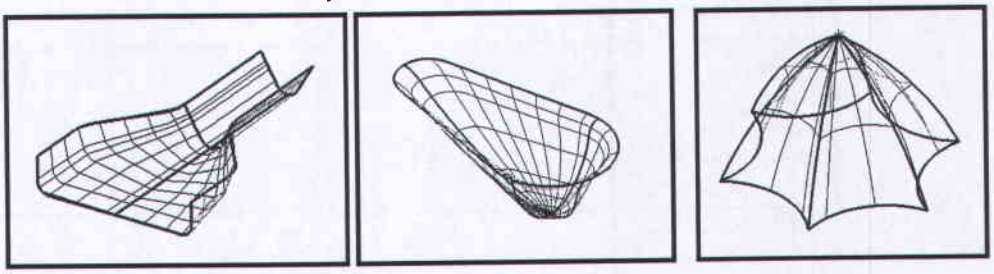


Net Surface: a surface created from a network of intersecting curves.
Application: used to create a surface from a grid of curves.

See **Tutorial 10**.
Examples
Single patch



Multiple Patches

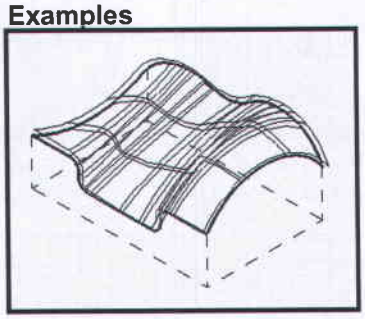


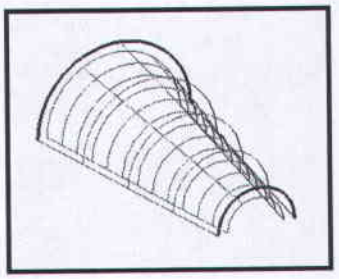
DERIVED SURFACES

Offset Surface: a surface created by offsetting an existing surface with a given distance. Each point from the offset surface is at a fixed, normal distance from the original surface.
Applications: used to create a surface offset at a given distance from the original surface.

are surfaces generated from existing surfaces.

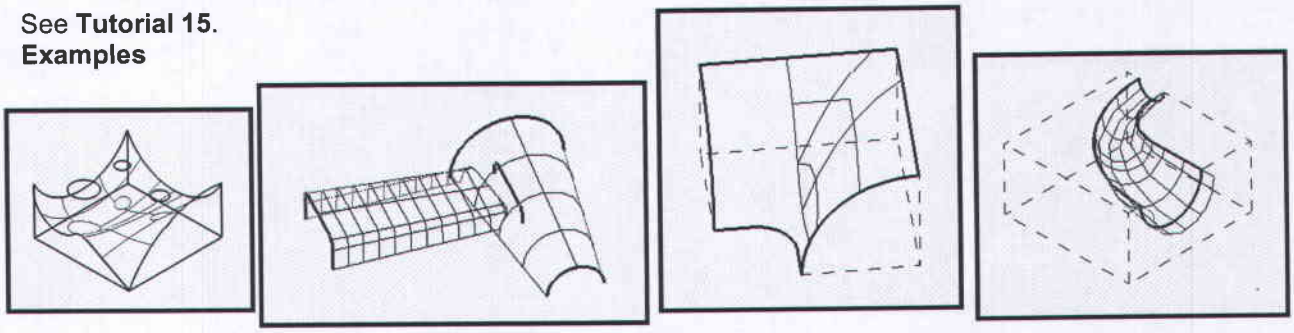
See **Tutorial 13**.
Examples





Trim Surface: a surface generated by trimming an existing surface to a specific boundary, to another surface, or to a given plane.
Extend Surface: a surface generated by extending an existing surface to a plane or by a given length.
Applications: used to redefine the surface edges.

See **Tutorial 15**.
Examples



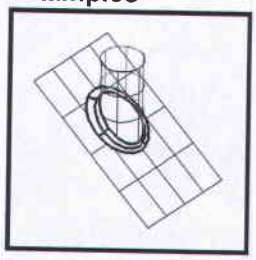
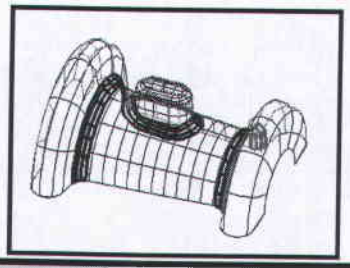
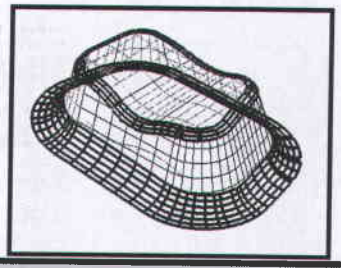
Flat Boundary Surface: a trimmed surface, generated by trimming a flat surface to a specific boundary.
Applications: use to create a flat surface on a part inside a closed boundary.

See **Tutorial 12**.

Fillet Surface: a surface generated by creating fillets (radius) that are tangent to the original surfaces. You can also create a fillet surface between a surface and a plane, and between a surface and a curve.
Application: used to smooth sharp edges.

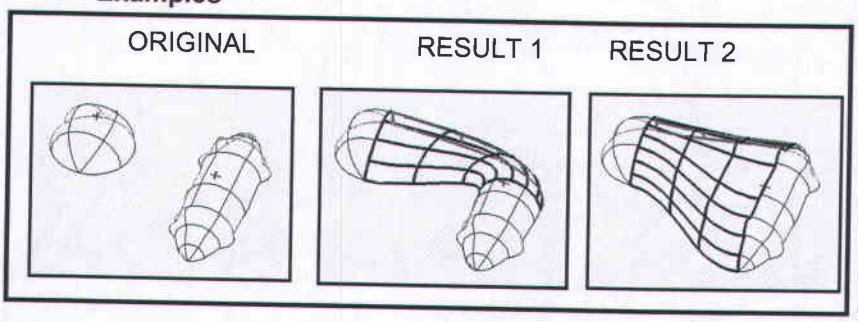
See **Tutorial 12**.

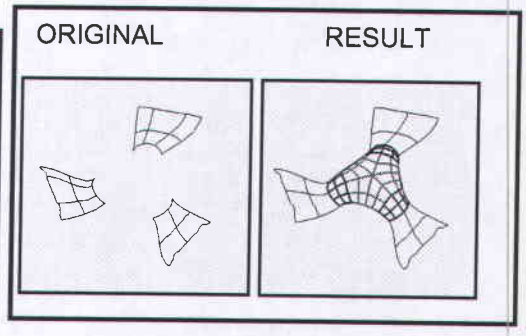
Examples

Blend Surface: a surface that blends two or three existing surfaces together, and is tangent to each parent surface.
Application: used to blend surfaces together.

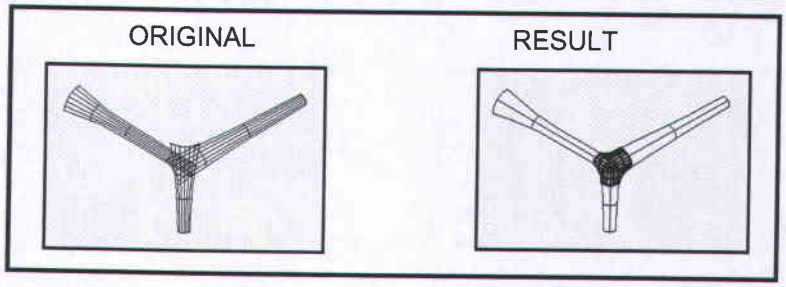
Examples





Fillet Blend Surface: a surface that blends three intersecting fillet surfaces together.
Application: used to blend intersecting fillet surfaces together.

Examples



SURFACE TYPES

Mastercam allows you to create three different types of surfaces: Parametric, NURBS (**N**on-**U**niform **R**ational **B**-**S**pline), and Curve-generated. Each type uses a different method to calculate

Parametric: the system generates the surface by translating each curve segment (parametric spline) in a different direction forming a patch. A patch is the surface area bounded by four contours (chains of entities).

and store mathematical data about the surface.

Characteristics:
- Compatible with IGES and VDA conversion.
- Not associated with their generating curves.
- Require a large amount of data storage.

NURBS: the system generates the surface by translating the string of control points in a second direction, resulting in a grid.

Characteristics:
- Compatible with IGES conversion. Can be output (from Mastercam) to a VDA file format.
- Not associated with their generating curves.
- Require less data storage than parametric surfaces but slightly more computing time.

Curve-generated: the system stores exact data about their generating curves.

Characteristics:
- Associated with their generating curves.
- Require less data storage than parametric or NURBS surfaces.
- Entity association refers to the dependent relationship between the entity and the original entity or group of entities from which the entity was generated. If you delete the original entity or group of entities the system deletes the surface too.