Mastercam®

Router Training Tutorial

X9

Mastercam X3

Router Training Tutorials

To order more books:
Call 1-800-529-5517 or
Visit www.inhousesolutions.com or
Contact your Mastercam Dealer

TABLE OF CONTENTS

TUTORIAL SERIES FOR

Mastercam X3

HOW TO USE THIS BOOK

This book provides a comprehensive step by step approach to learning Mastercam. It contains pages of projects, helpful hints. The book covers nine tutorials.

The material covered includes 2D Geometry, Drilling, Contouring, Pocketing, Nesting and Slot Milling. Explanations are given for the use of Nesting, chaining parameters to Metric, as well as proper Stock Setup and Tool Settings.

The Router Training Tutorials also include a General Notes chapter with useful tools and shortcuts that make the software easier to use. An alphabetic description of the 2D toolpath parameters, chaining and toolpath manager are also covered in the General Notes.

Each tutorial walks you through all the procedures from Geometry Creation to Toolpath instructions, Verification and G-Code Generation.

LEGEND:

Step to follow to complete the tutorial

Additional explanation for the current step

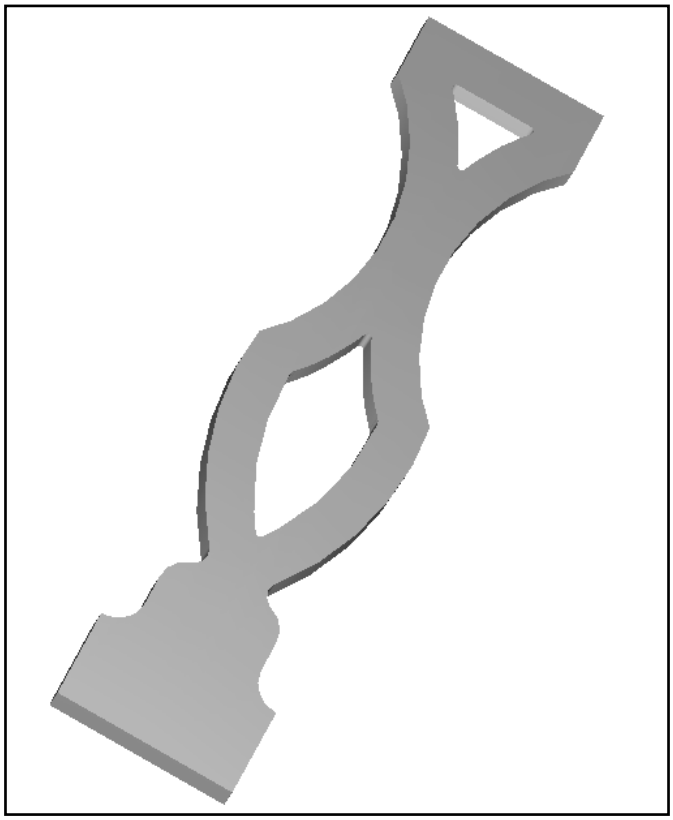
Callouts that give direction on how to complete the task

Bold text (usually) represents Mastercam terminology

TUTORIAL SERIES FOR

Mastercam X3

TUTORIAL # 2
2D GEOMETRY, POCKET & CONTOUR



Objectives:

The Student will design a 2-dimensional wireframe drawing by:

- Creating a rectangle using the 1 point method.
- Creating parallel lines, by defining the offset direction and distance.
- Creating arcs using polar command.
- Creating a line knowing the endpoints of the line.
- Trimming 1 entity to another existing entity.
- Creating arcs knowing the radius and the center points of the arcs.
- Trimming two entities up to the intersection point between the entities.
- Deleting extra construction lines.
- Offsetting entities with a given distance.
- Mirroring existing geometry to complete a part.
- Modifying an entity by breaking it in two pieces.
- Creating fillet radii.

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

- A 2 dimensional contour.
- Machine the two pockets in one operation.
- Remachining the two pockets using a smaller tool that will remove the material only in the area where the previous tool could not fit.

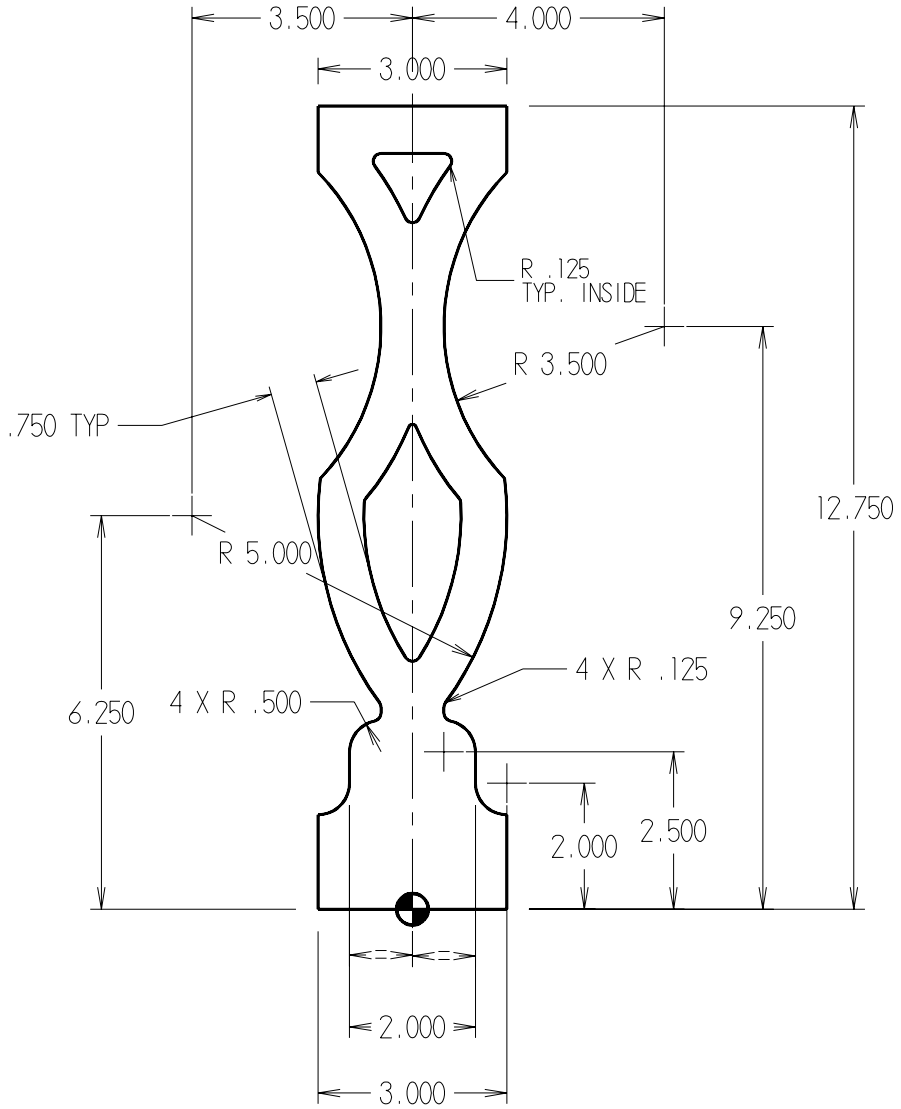
The Student will check the toolpath using Mastercam's Verify verification module by:

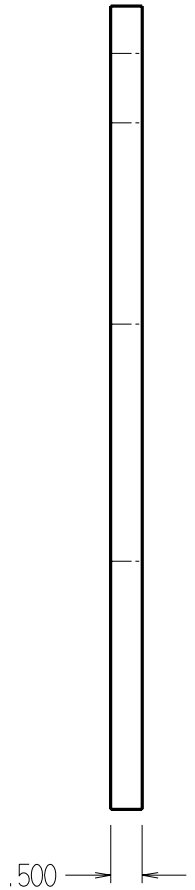
- Running the Backplot function.
- Defining a 3 dimensional rectangular block the size of the work piece.
- Running the Verify function to simulate machining the part on the screen.

The Student will generate the NC file by:

- Running the Post processor.

ALL DIMENSIONS IN INCHES





TITLE TUTORIAL #2	
MATERIAL WOOD	
DATE: JUNE 19, 2001	eMastercam.com

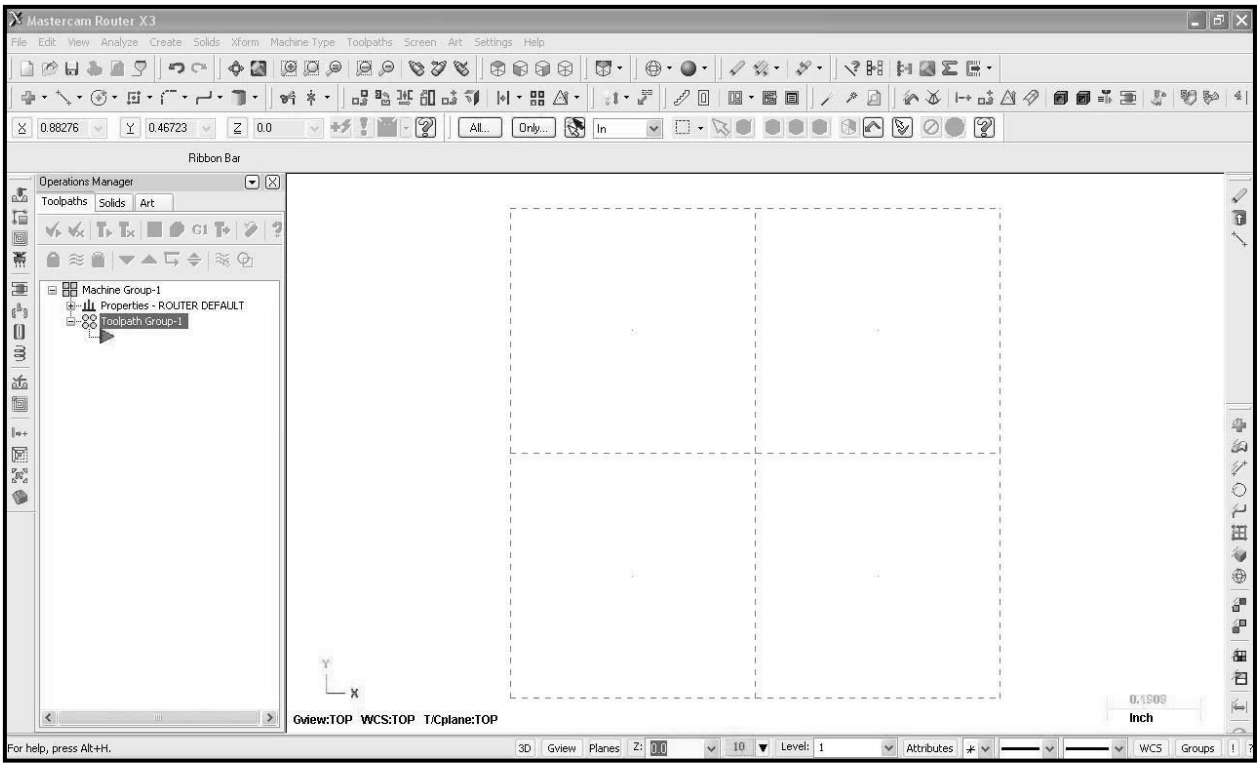
GEOMETRY CREATION

Setting the toolbar states

- Before starting the geometry creation we should customize the toolbars to see **Setting the Toolbar States** to create the geometry and machine a 2D part. on page A-4 in the **User Notes**.

- Make sure that the **Grid** is enabled as it will show you where the part origin is. See **Setting the Grid** on page A-5

- **Operations Manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to hide it.

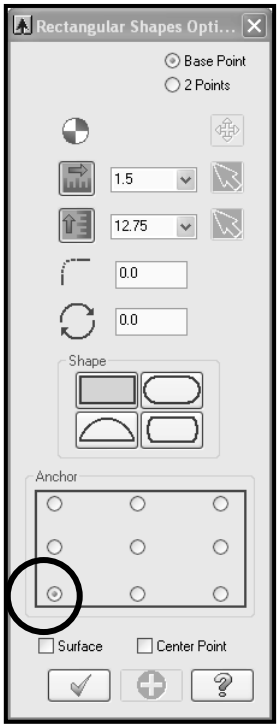


- Due to the fact that this drawing is symmetrical in the **Y**-axis, you will only draw 1/2 of the total part and use the mirror function to complete the part

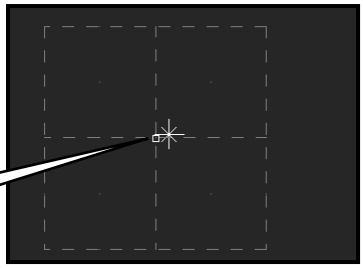
STEP 1: CREATE A 1.5 X 12.75 RECTANGLE KNOWING THE WIDTH AND HEIGHT.

Create

- **Rectangular Shapes**
- Enter the **Width** and the **Height** as shown in the following screenshot.
- Select the arrow to expand the **Rectangular Shapes Options** as shown



- 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** to exit **Rectangular Shapes Options** dialog box
- Use **Fit** icon to fit the drawing to the screen.



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

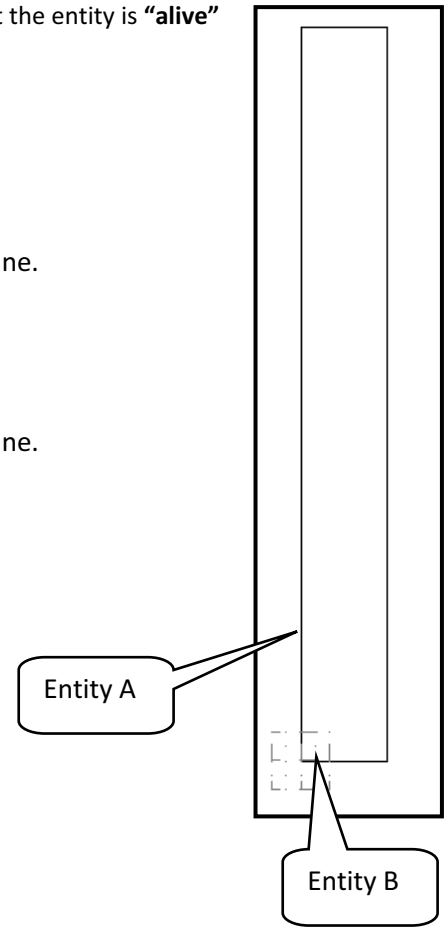
STEP 2: CREATE PARALLEL LINES (KNOWING THE SIDE AND THE DISTANCE BETWEEN THE LINES) TO ESTABLISH THE CENTER POINTS OF THE 0.500 RADIUS ARCS.

Create

- **Line**
- **Parallel** 
- [Select a line] Select **Entity A**.
- Indicate the offset direction] Pick a point to the right of the selected line.

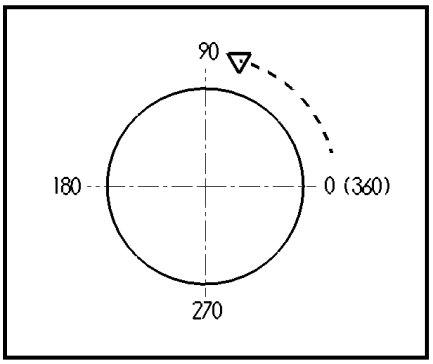
  - Note that the color of the geometry is **cyan** which means that the entity is **"alive"** and you can still change the line parameters.

- Enter the **Distance**  **0.5** (Enter)
- Select apply button to continue with the same command 
- [Select a line]: Select **Entity B**
- [Indicate the offset direction]: Pick a point above the selected line.
- Type the **Distance**  **2.0** (Enter)
- Select apply button to continue with the same command 
- [Select a line]: Select **Entity B**
- [Indicate the offset direction]: Pick a point above the selected line.
- Type the **Distance**  **2.5** (Enter)
- Select the **OK** button. 



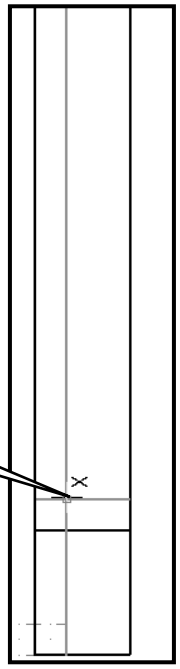
STEP 3: CREATE THE 0.5 RADIUS ARCS USING POLAR COMMAND (KNOWING THE CENTER POINT, RADIUS AND INITIAL ANGLE AND FINAL ANGLE).

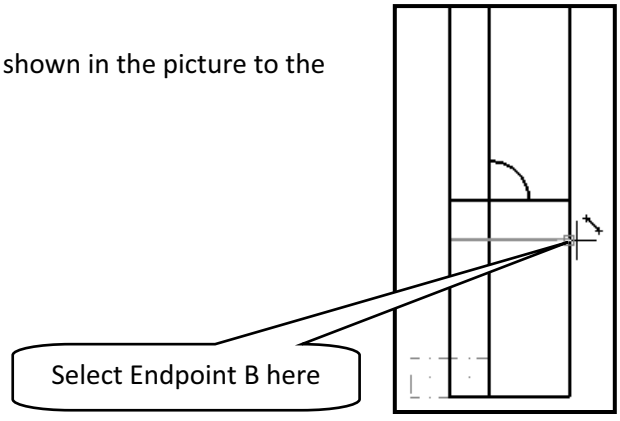
- To create a polar arc you have to know the radius of the arc, the initial angle and the final angle. Positive angles in Mastercam are measured in **CCW** direction as shown in the following diagram.



Create

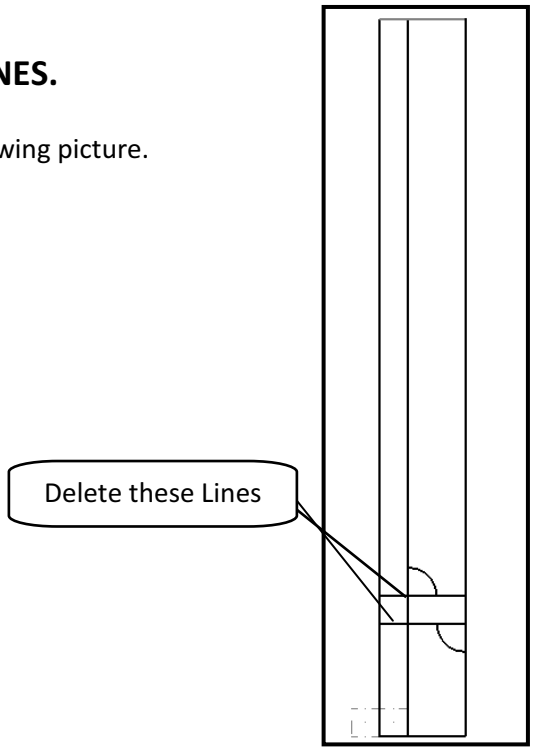
- **Arc**
- **Arc Polar**
- Enter the **Radius** **0.5** (Tab)
  - To set the other parameters of the arc use Tab key. Note that the diameter value is automatically changed by the system based on the radius.
- Enter the **Start Angle** **0** (Tab)
- Enter the **End Angle** **90** (Enter)
- [Enter the center point]: Select **Point A** at the intersection as shown.
- Select the **Apply** button to continue using the same command.
- Enter the **Radius** **0.5** (Tab)
- Enter the **Start Angle** **180** (Tab)
- Enter the **End Angle** **270** (Enter)
- [Enter the center point]: Select the **Endpoint B** as shown in the picture to the right.
- Select the **OK** button.





STEP 4: DELETE THE EXTRA CONSTRUCTION LINES.

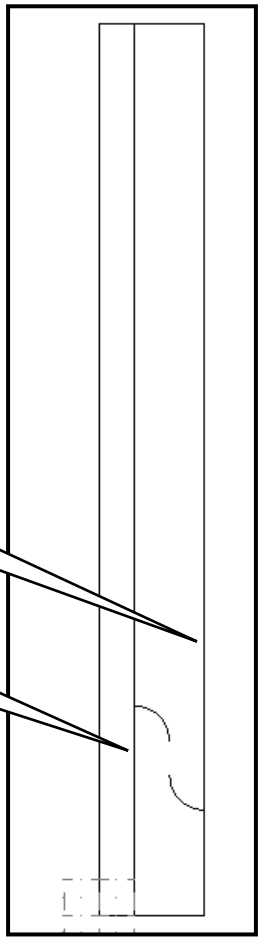
- Pre-select the two horizontal lines as shown in the following picture.
- Select **Delete** entity icon.



STEP 5: TRIM 1 ENTITY USING DIVIDE.

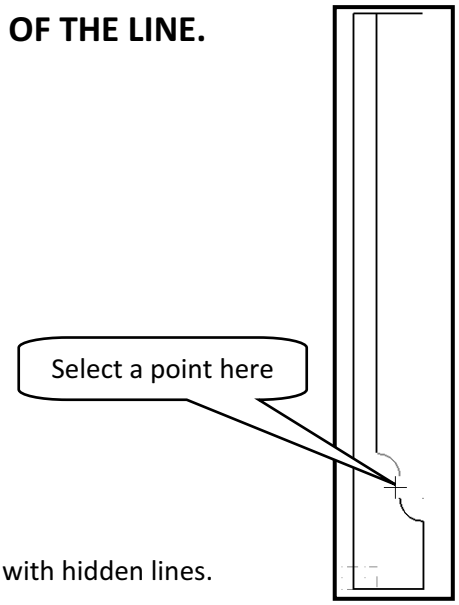
Edit

- **Trim/Break**
- **Trim/Break/Extend**
- Select the **Divide** icon from the trim ribbon bar.
- [Select the curve to divide]: Select **Entity A**
  - When using divide, select the entity on the side that you want to remove.
- [Select the curve to divide]: Select **Entity B**
- Select the **OK** button.

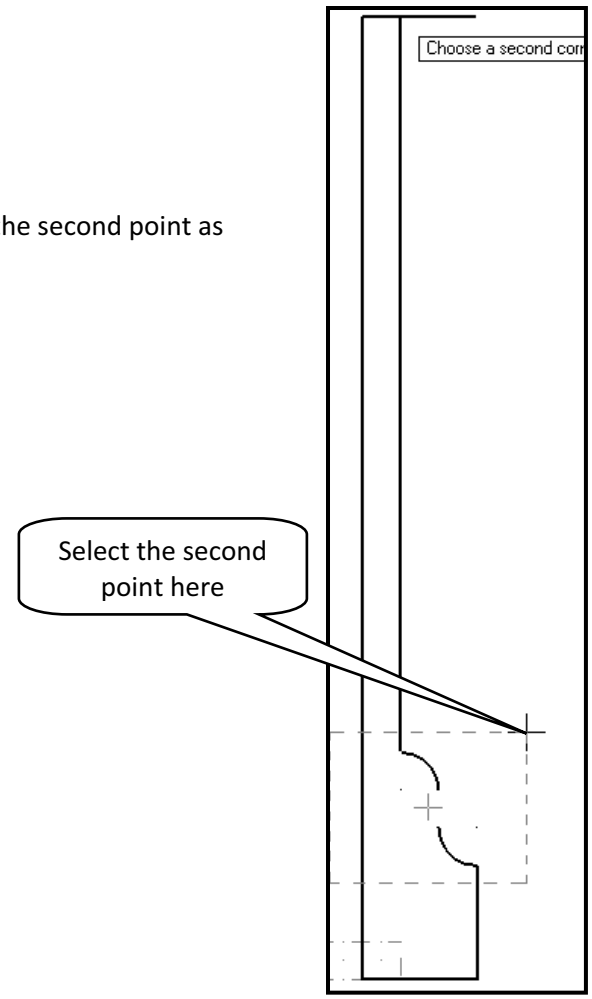


STEP 6: CREATE A LINE KNOWING THE ENDPOINTS OF THE LINE.

- Select **Zoom Target** icon.
- [Pick point to zoom from]: Select a point as shown



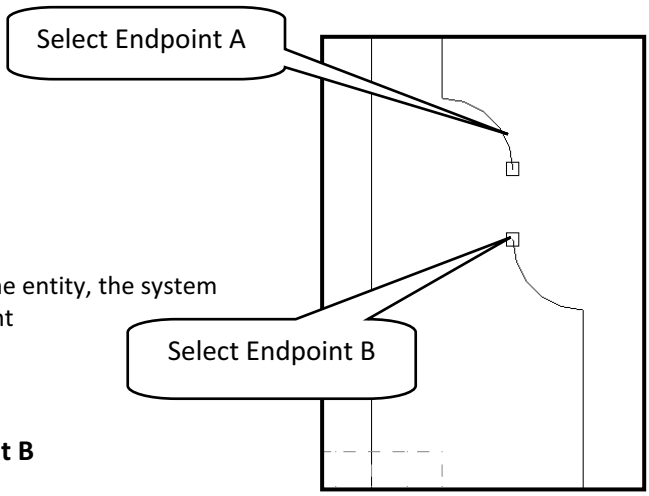
- Drag the cursor to make a window around the part as shown with hidden lines.



- [Choose a second corner for your zooming] Select the second point as shown

Create

- **Line**
- **Endpoints**
- [Specify the first endpoint]: Select **Endpoint A**
  - By moving the cursor to the endpoint of the entity, the system will display a small square and the endpoint symbol .
- [Specify the second endpoint]: Select **Endpoint B**
- Select the **OK** button.
- Use **Fit** icon to fit the drawing to the screen.



STEP 7: CREATE THE 5“ RADIUS AND 3.5“ RADIUS ARCS, KNOWING THE RADIUS AND THE CENTER POINT OF THE ARC.

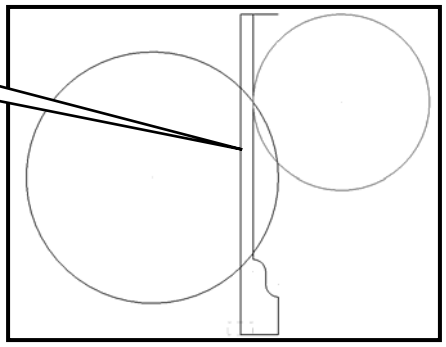
Create

- **Arc**
- **Circle Center Point**
- Enter the **Radius** value **5.0** (Enter)
- [Enter the center point]: Select the **Fast point** icon and enter the coordinates: **-3.5, 6.25** (Enter)
  - The first coordinate value is the X-coordinate of the point. The second coordinate value is the Y-coordinate of the point. The values are measured from the geometry origin, and they are separated by a comma.
- Select the **Apply** button to continue using the same command.
- Enter the **Radius** value **3.5**(Enter)
- [Enter the center point]: Select the **Fast point** icon and enter the coordinates: **4.0, 9.25**
- Select the **OK** button to exit the command.

STEP 8: CREATE A PARALLEL LINE KNOWING THE SIDE AND THE DISTANCE TO COMPLETE THE TOP GEOMETRY.

Create

- **Line**
- **Create Line Parallel**
- [Select a line]: Select **Entity A**.
- [Indicate the offset direction]: Pick a point to the right of the selected line.



  - Note that the color of the geometry is cyan which means that the entity is "alive" and you can still change the line parameters.

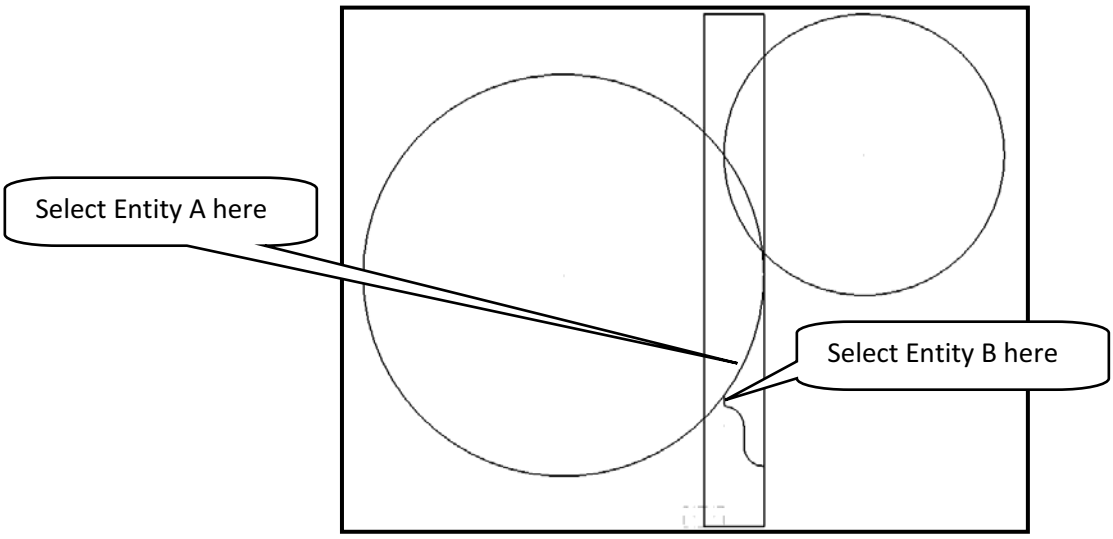
- Enter the **Distance** **1.5** (Enter)
- Select the **OK** button.

STEP 9: TRIM 2 ENTITIES UP TO THEIR INTERSECTION POINT SIMULTANEOUSLY.

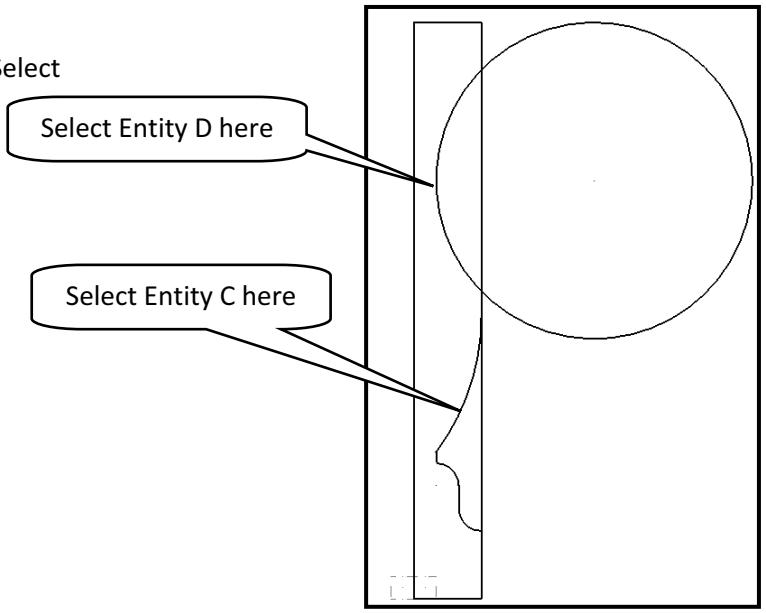
- Trimming can result in either trimming or extending up to the intersection point. Always select the entities to trim on the side that you want to keep after trimming.

Edit

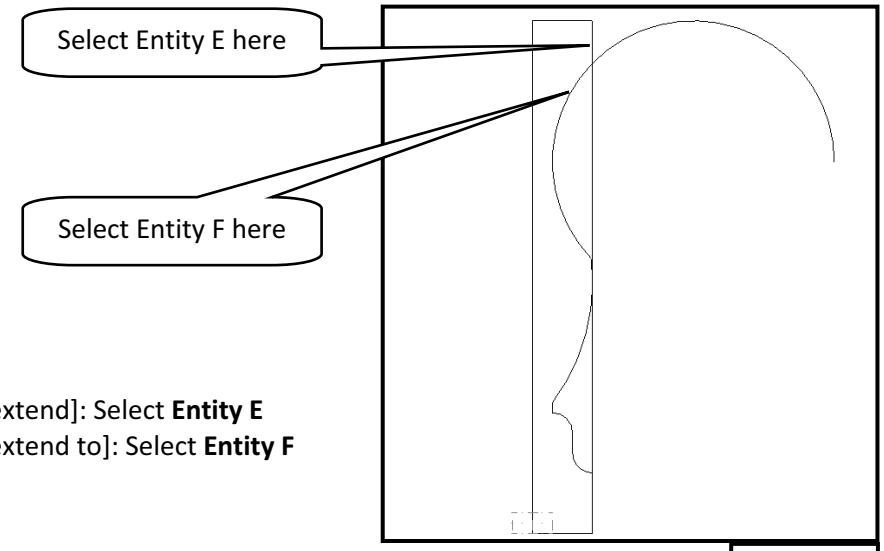
- **Trim/Break**
- **Trim/Break/Extend**
- Select the **Trim 2 Entities**
- [Select entity to trim/extend]: Select **Entity A**
- [Select the entity to trim/extend to]: Select **Entity B**



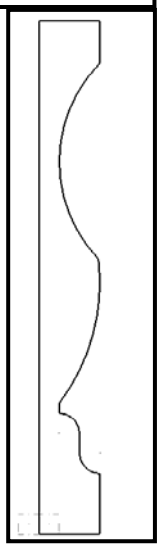
- [Select the entity to trim/extend]: Select **Entity C** (make sure that the arc is highlighted)



- [Select the entity to trim/extend to]: Select **Entity D**



- [Select the entity to trim/extend]: Select **Entity E**
- [Select the entity to trim/extend to]: Select **Entity F**



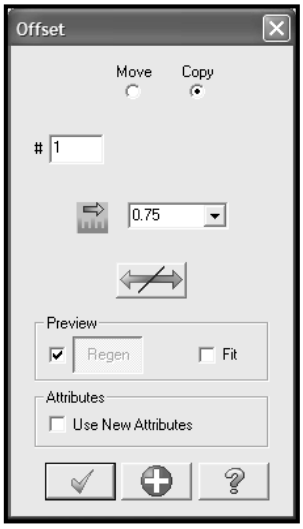
- Select the **OK** button.
- Use **Fit** icon to fit the drawing to the screen.

The part should look as shown to the right.

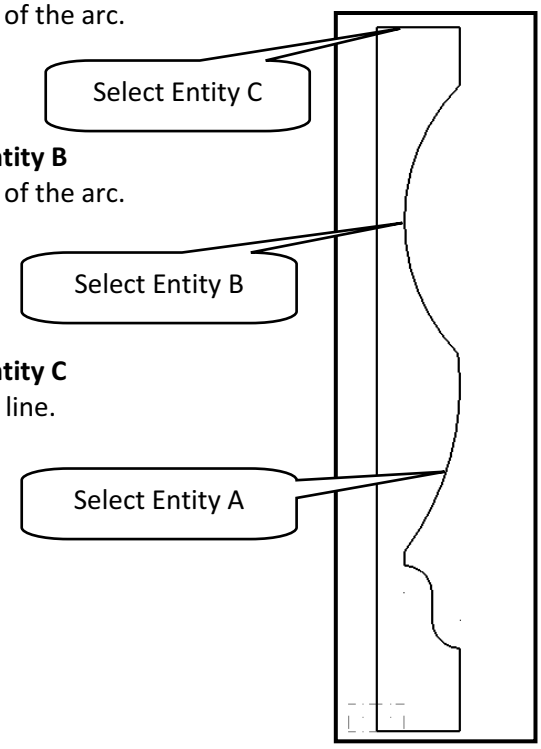
STEP 10: OFFSET THE ENTITIES TO CREATE THE INSIDE SHAPES.

Xform

- **Offset**
- Make the changes as shown in the following screen shot.(Enable **Copy** and change the offset distance to **0.75**)



- [Select the line, arc, spline or curve to offset]: Select **Entity A**
- [Indicate the offset direction]: Select a point to the left of the arc.



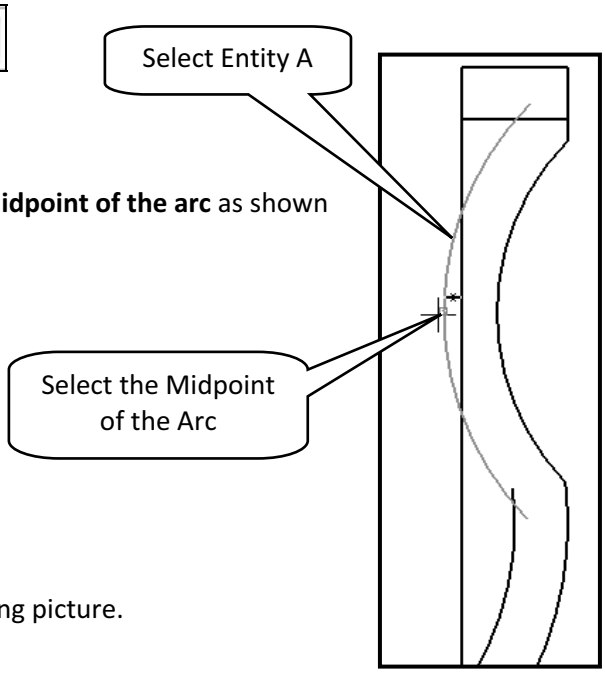
- [Select the line, arc, spline or curve to offset]: Select **Entity B**
- [Indicate the offset direction]: Select a point to the left of the arc.
- [Select the line, arc, spline or curve to offset]: Select **Entity C**
- [Indicate the offset direction]: Select a point below the line.
- Select the **OK** to exit **Offset** dialog box

STEP 11:BREAK THE OFFSET ARC OF THE 3.5 RADIUS ARC IN TWO PIECES.

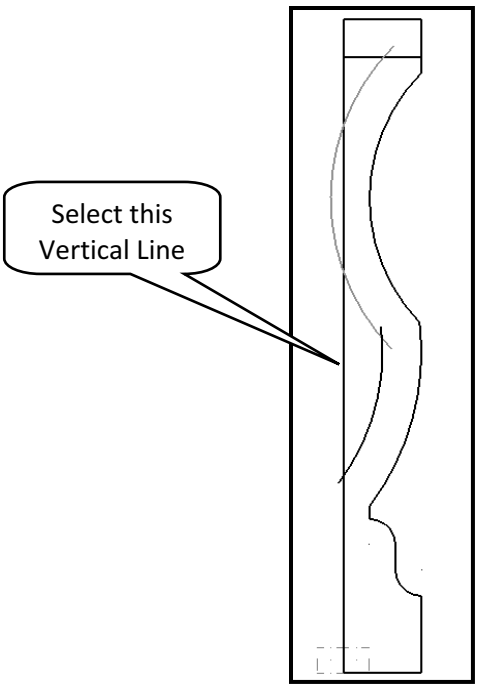
- The offset arc is used to create two separate pockets. So we have to break it in two pieces.

Edit

- **Trim/Break**
- **Trim/Break/Extend** 
- Select the **Trim to Point** 
- Enable **Break** button. 
- [Select entity to break/extend]: Select **Entity A**
- [Indicate the break/extend location]: Select the **Midpoint of the arc** as shown in the picture.



- Select the **OK** button. 
- Pre-select the vertical line as shown in the following picture.
- Select **Delete** entity icon. 

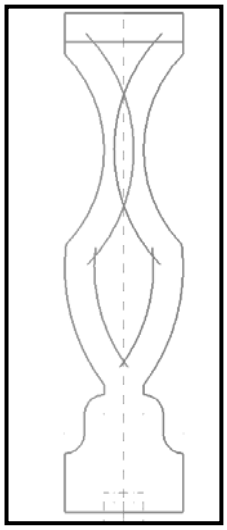


STEP 12: TRANSFORM THE GEOMETRY TO REPRESENT THE WHOLE PART.

Xform

- **Mirror**
- [Mirror: select entities to mirror]: Select **All** button
- Select the **OK** button to exit **Select All** dialog box.
- Select **End Selection** button.
- Make the changes in the Mirror dialog box as shown in the following screenshot.

The part should look as shown below.



- Select the **OK** button to exit **Mirror** dialog box.

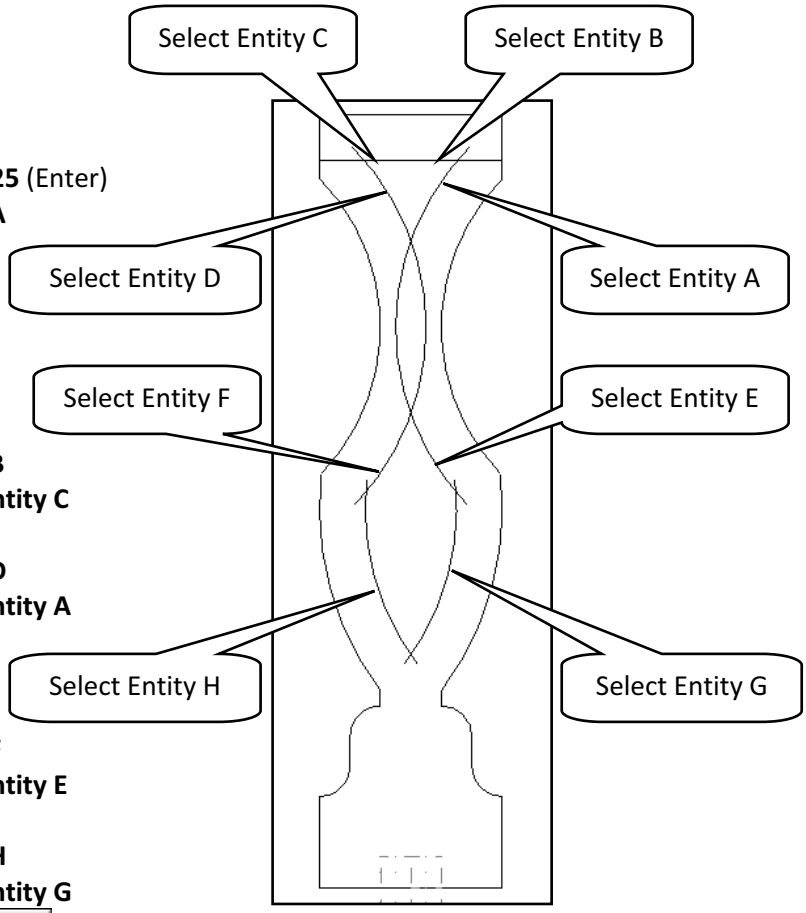
Screen

- **Clear Colours**

STEP 13: FILLET THE POCKETS WITH A .0625 RADIUS.

Create

- **Fillet**
- **Entities**
- Enter the fillet **Radius** **0.0625** (Enter)
- [Select an entity] Select **Entity A**
- [Select another entity] Select **Entity B**



- [Select an entity] Select **Entity B**
- [Select another entity] Select **Entity C**

- [Select an entity] Select **Entity D**
- [Select another entity] Select **Entity A**

- [Select an entity] Select **Entity F**
- [Select another entity] Select **Entity E**

- [Select an entity] Select **Entity H**
- [Select another entity] Select **Entity G**
- Select the **OK** button to exit.

STEP 14: TRIM SIMULTANEOUS 2 ENTITIES UP TO THEIR INTERSECTION POINT.

Edit

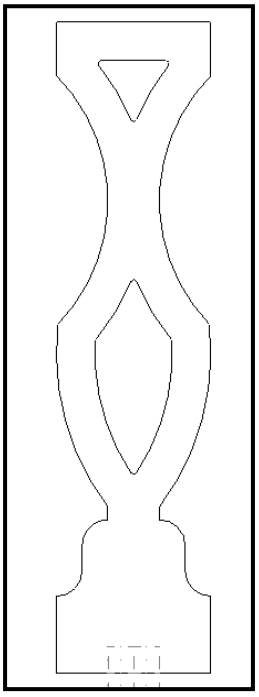
- **Trim/Break**
- **Trim/Break/Extend**

- Select the **Trim 2 Entities**
- [Select entity to trim/extend]: Select **Entity A**
- [Select the entity to trim/extend to]: Select **Entity B**

- Select entity to trim/extend]: Select **Entity C**
- [Select the entity to trim/extend to]: Select **Entity D**
- Select the **OK** button to exit.

Trim

STEP 15: SAVE THE FILE.



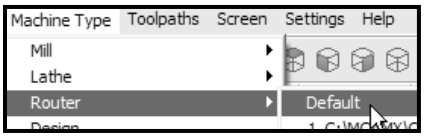
File

- **Save as**
- **File name**: "Your Name_2"
- Select the **Save** button.

TOOLPATH CREATION

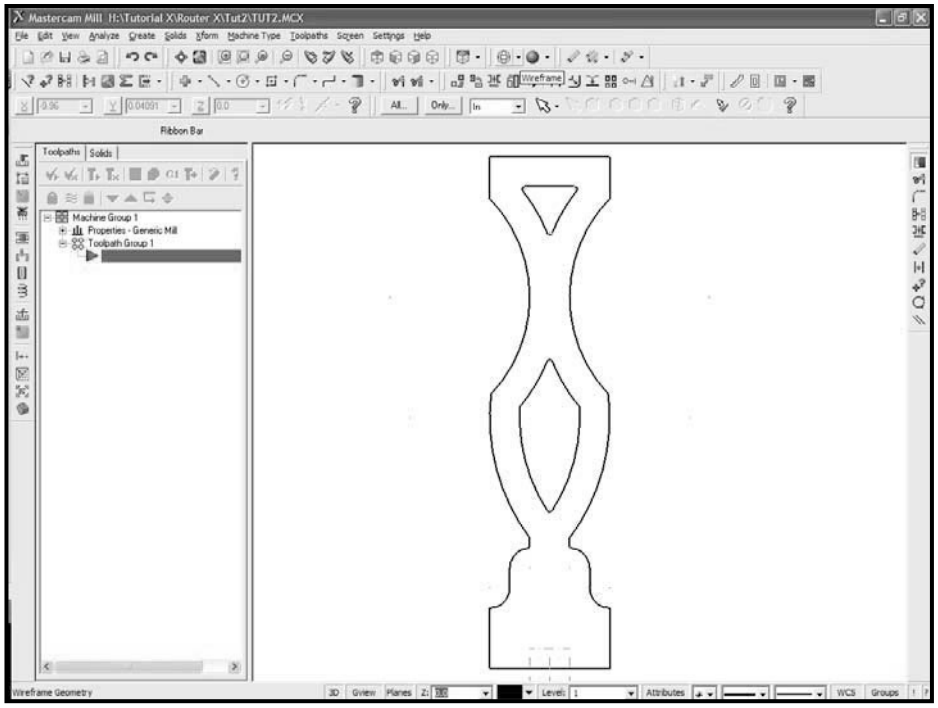
STEP 16: SET UP THE STOCK TO BE MACHINED.

- To display the **Toolpaths Operation Manager** press Alt + O.



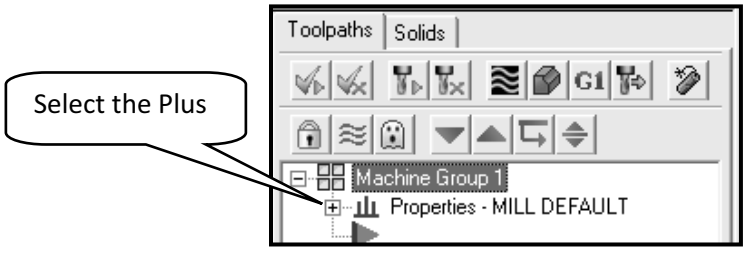
Machine type

- **Router**
- **Default**

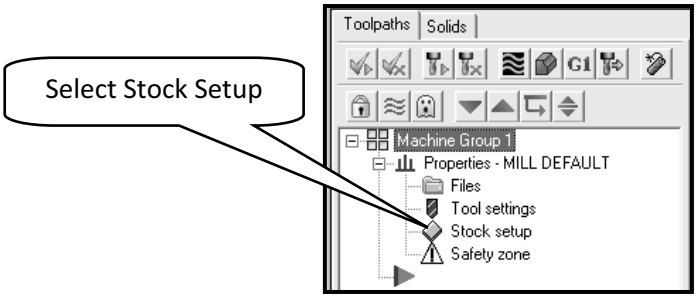


- Use **Fit** icon to fit the drawing to the screen.

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

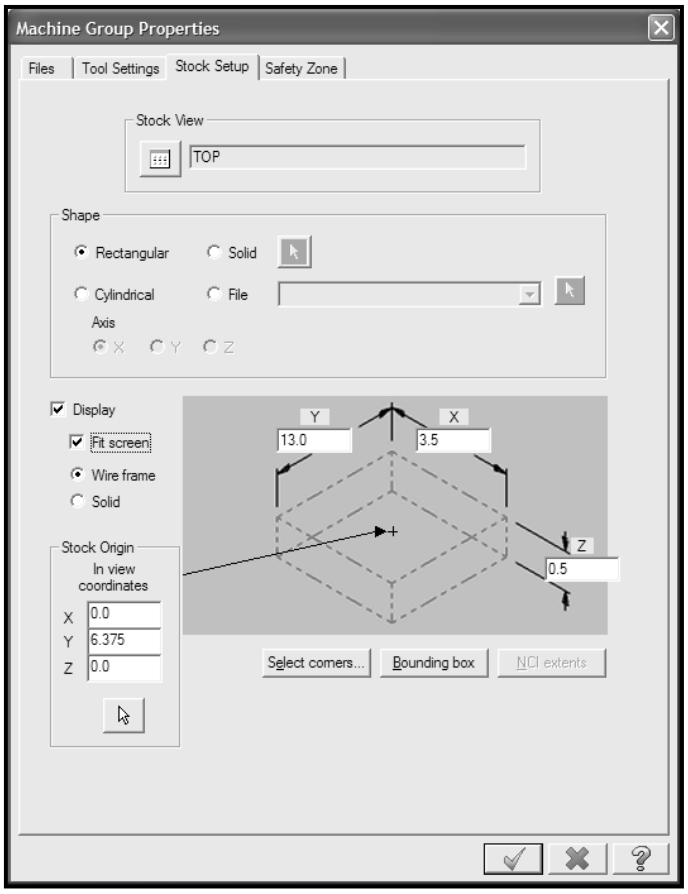


- Select **Stock setup**.

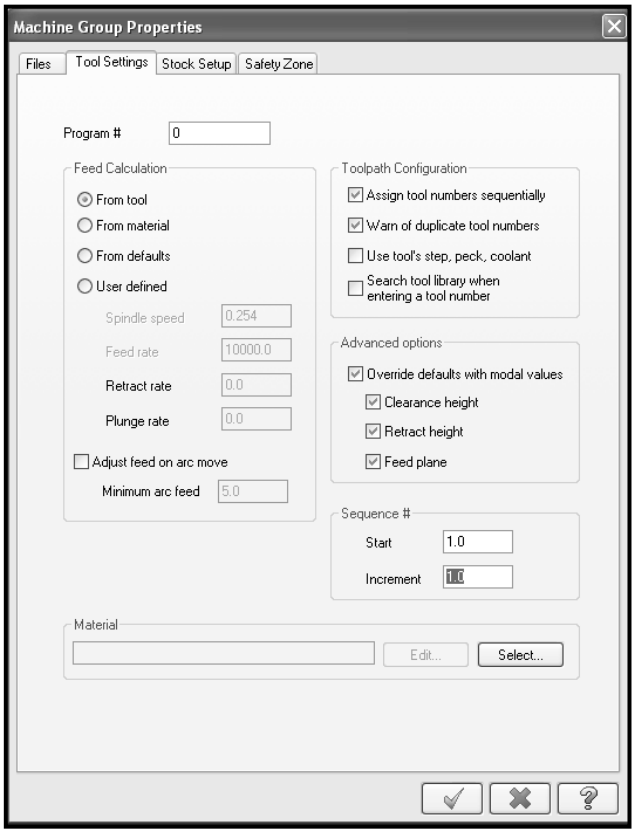


- Change the parameters to match the screenshot to the right.

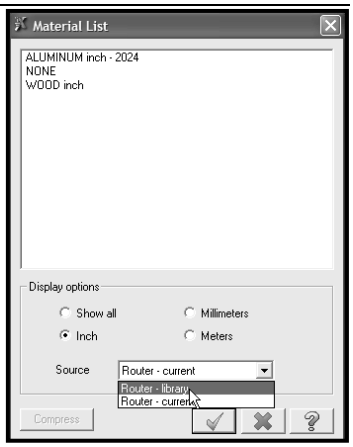
The **Stock Origin** values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finish part.
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen.(**Fit Screen**)



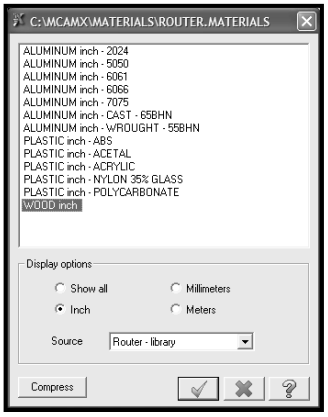
- Select the **Tool Settings** tab to set the Toolpath parameters and the part material.
- Change the parameters to match the following screenshot.
- Click on **Select** button.



- Change the **Source** to **Router library.**



- Select the **Wood inch.**
- Select the **OK** button to exit the library.



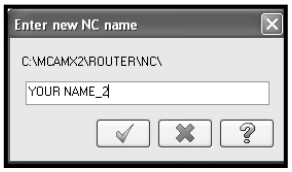
- Select the **OK** button to exit **Toolpath Group Properties**. 
- Select the **Isometric View** from the view toolbar to see the stock.

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

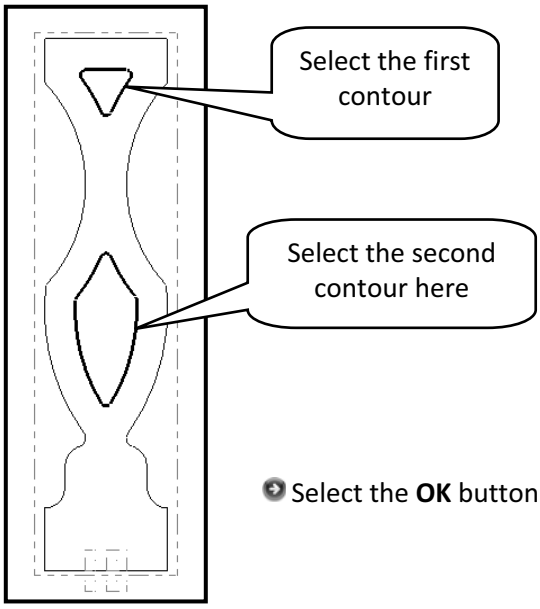
STEP 17: POCKET THE INSIDE PART USING ½“ STRAIGHT BIT

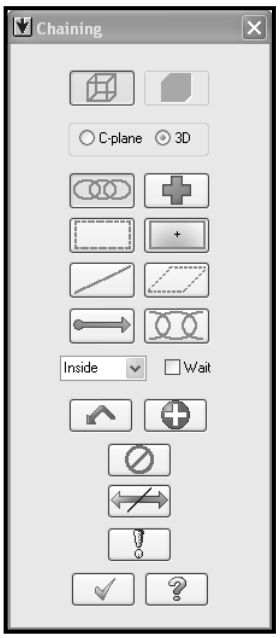
Toolpaths

- **Pocket** 



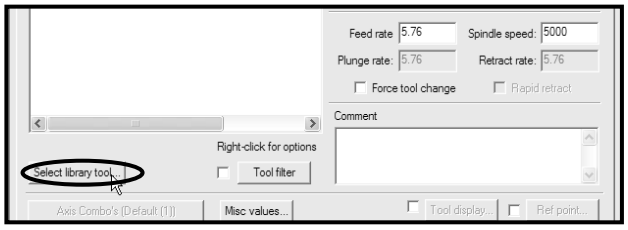
- Select the **OK** button to accept the NC name. 
- Select the pocket chains as shown.



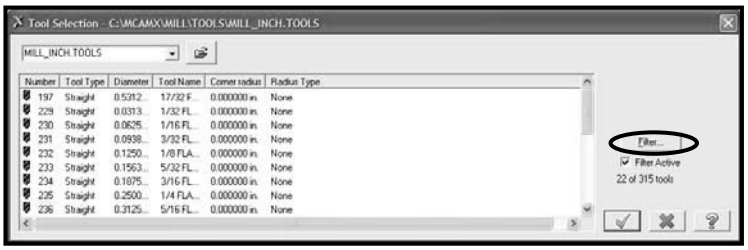


- Select the **OK** button to exit **Chaining**. 

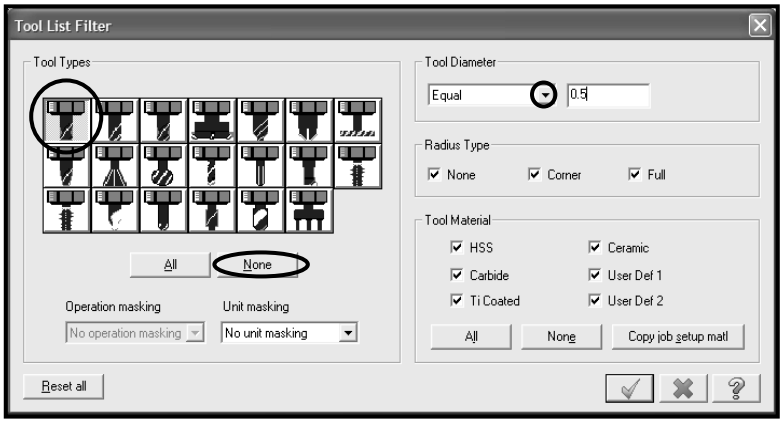
- Click on **Select library tool** button.



- Select **Filter** button.

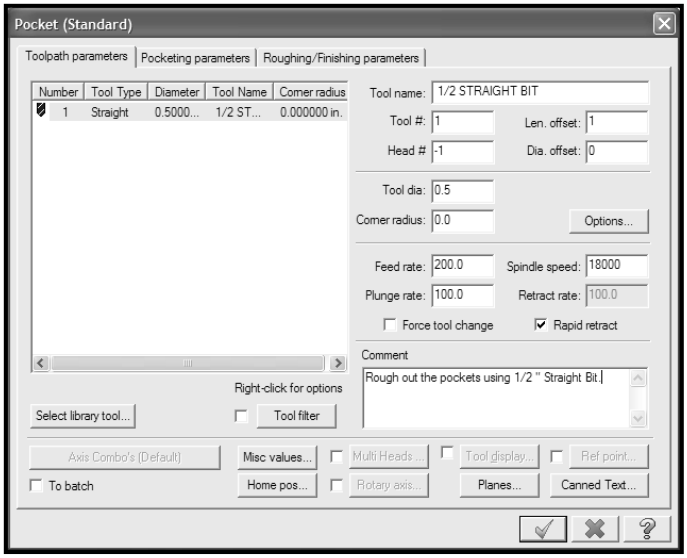


- Select the **None** button to disable any previous tool selection as shown.
- Select the **Straight** in the **Tool Types** list.
- Select the **drop down arrow** in the **Tool Diameter** field and select **Equal**.
- Enter 0.5 in the **Tool Diameter** value box.
- Select **OK** button to exit **Tool List Filter.**



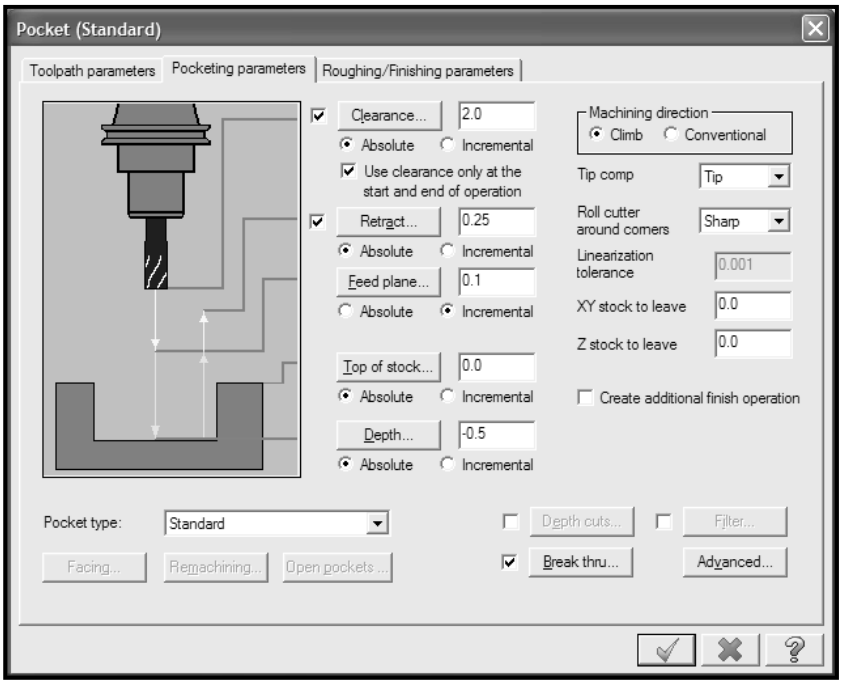
- Make sure that the tool is selected (highlighted) in the **Tool Selection** window.
- Select **OK** button to exit **Tool Selection.**

- Make the changes to match the following screenshot.



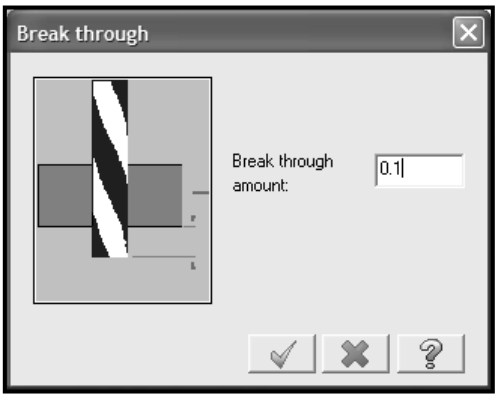
- The Toolpath parameters dialog box allows you to select the tool used in this operation. It also allows you to change the Spindle speed, the Feed rate, Plunge rate and Retract rate. You can insert a comment that will be output in the NC file after running the post processor.

Select the second page and fill out the parameter screen as shown in the following screen shot.



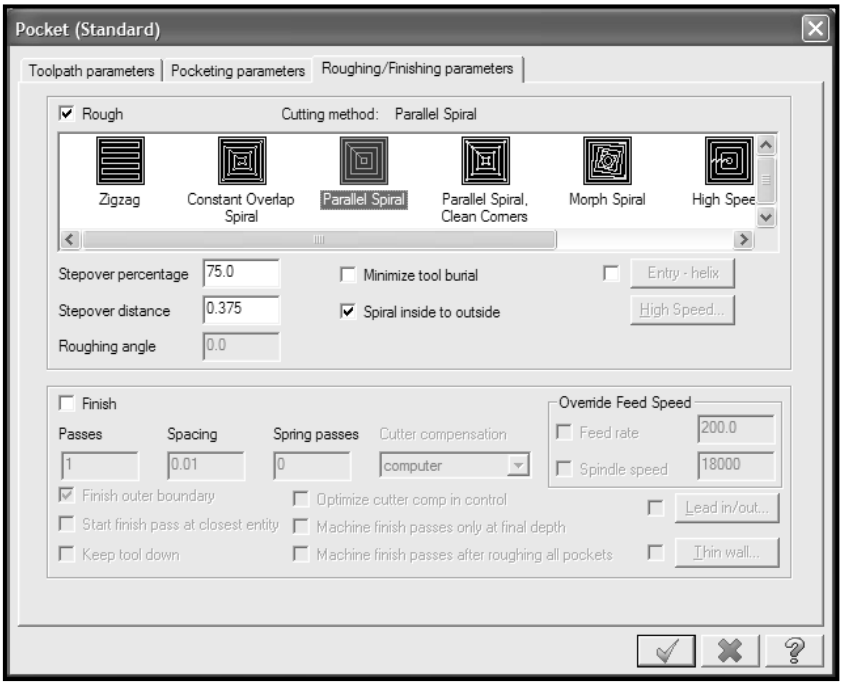
- The **Pocket parameters** dialog box allows you to establish the heights for rapid movement **Clearance** and **Retract**; the height from where the tool moves with feedrate **Feed plane**; the **Top of stock** and the final depth (**Depth**). You can use **Break through** amount to cut deeper than the final depth. You can also establish the **Machining direction** and the **Pocket type**. You can leave stock on the walls (**XY stock to leave**) or on the floor (**Z stock to leave**) that will require another operation to be removed.

Enable and select **Break through** button and set the break through amount to 0.1". This makes the cut deeper than the final depth to prevent any tabs remaining attached to the part.



Select the **OK** button to exit **Break through** window.

- Select the third page and fill out the parameter screen as shown in the following screen shot.



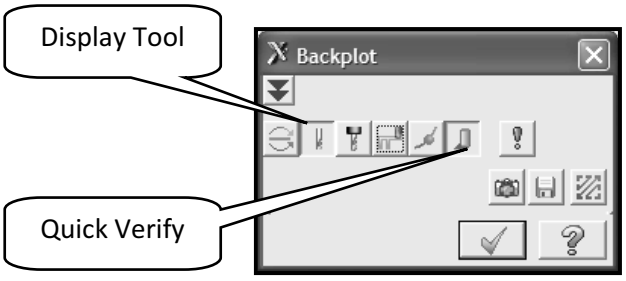
  - The **Roughing/Finishing parameters** dialog box allows you to establish the **Cutting method**, the **Stepover distance**, cutting method option **Spiral inside to outside** and if you are using contour to finish the pocket walls or not (**Finish** area).

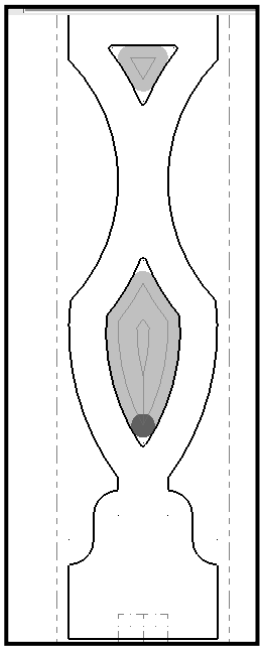
- Change the settings as shown above and select the **OK** button to exit pocket parameters screens.

STEP 18: CHECK THE TOOLPATH USING BACKPLOT.

- Click on the **Toolpaths Manager** tab.
- Select **Backplot selected operations** button.

- Make sure that you have the following buttons selected. (They look pushed-down)
- Enable **Display tool**
- Enable **Quick verify**
- Select **Play** button.

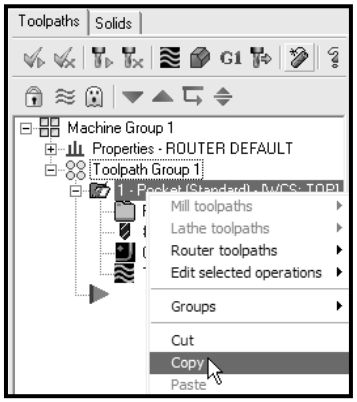




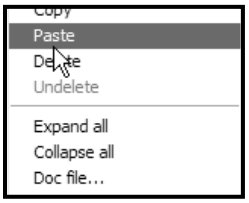
- Select the **OK** button to exit **Backplot.**

STEP 19: REMACHINE THE POCKETS USING 1/8“ STRAIGHT BIT

- The Remachining pocket operation allows you to cut areas where the previous tool diameter did not fit. We are going to copy the existing pocket operation in the **Toolpaths Manager** and modify the second pocket **Parameters**.



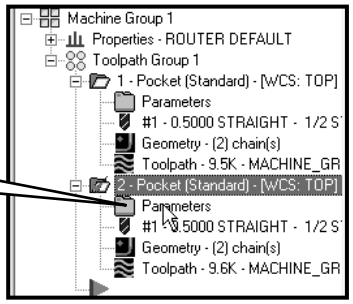
- Select the **Toolpaths** tab
- Right-mouse click on the pocket and select **Copy**



- Right-mouse click below the pocket, and select **Paste**.

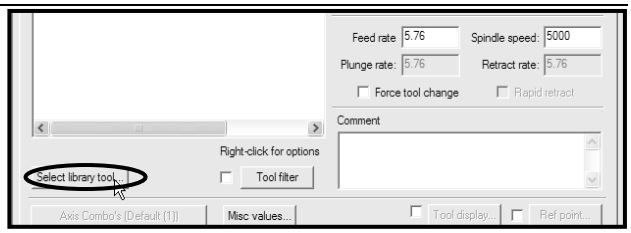
- We are going to use a 1/8“ **Straight Bit** tool to remove the material only in the area where the previous tool did not fit.

- To change the tool and the parameters for remachining, select **Parameters** in the second operation.

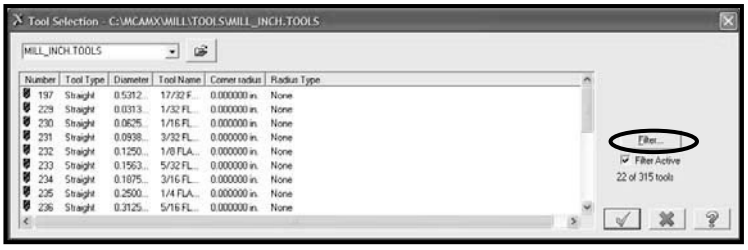


Select Parameters

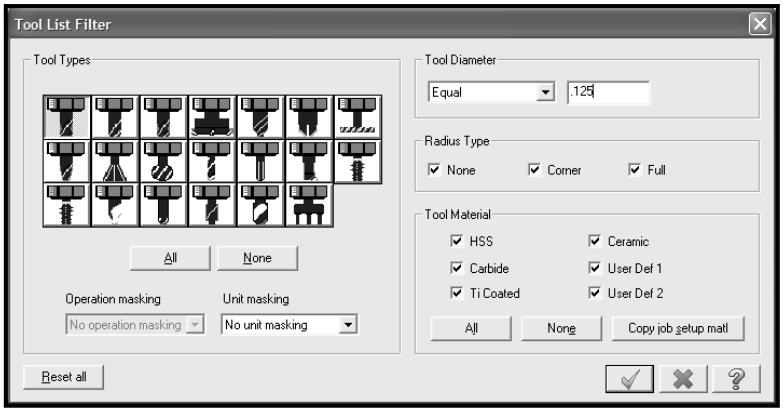
- Click on **Select library tool** button.



- Select **Filter** button.

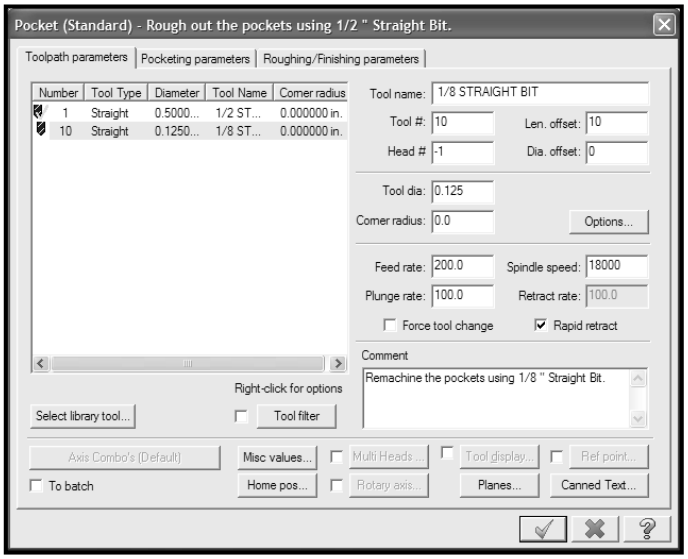


- Select the **Straight** in the **Tool Types** list.
- Enter 0.125 in the **Tool Diameter** value box.
- Select **OK** button to exit **Tool List Filter.**

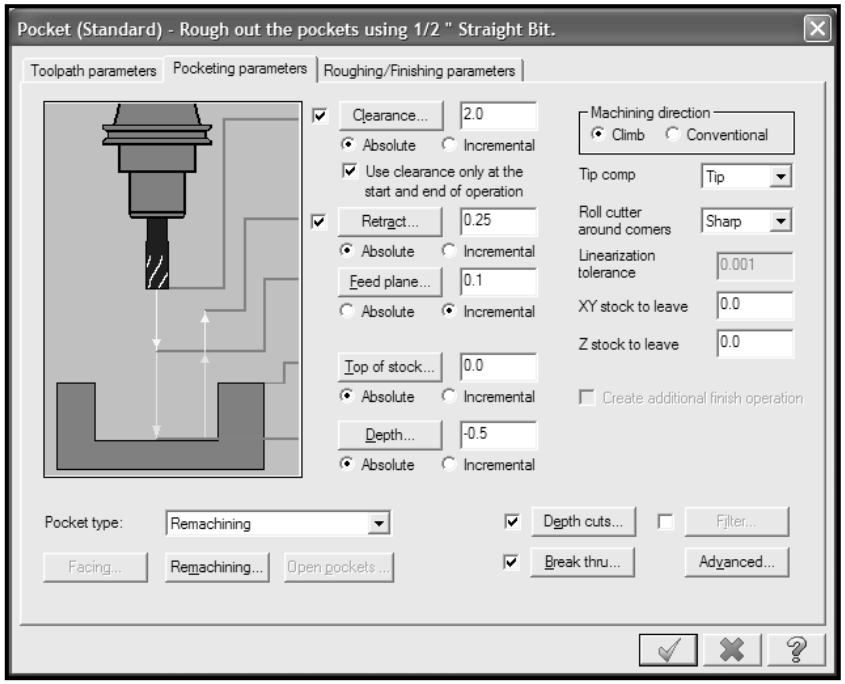


- Make sure that the tool is selected (highlighted) in the **Tool Selection** window.
- Select **OK** button to exit **Tool Selection.**

- Make the changes to match the following screenshot.



- Select the second page and change the pocket type to remachining.

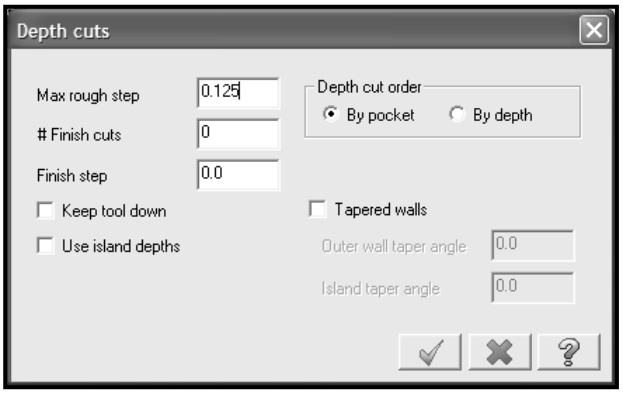


- Select the **Remachining** button.

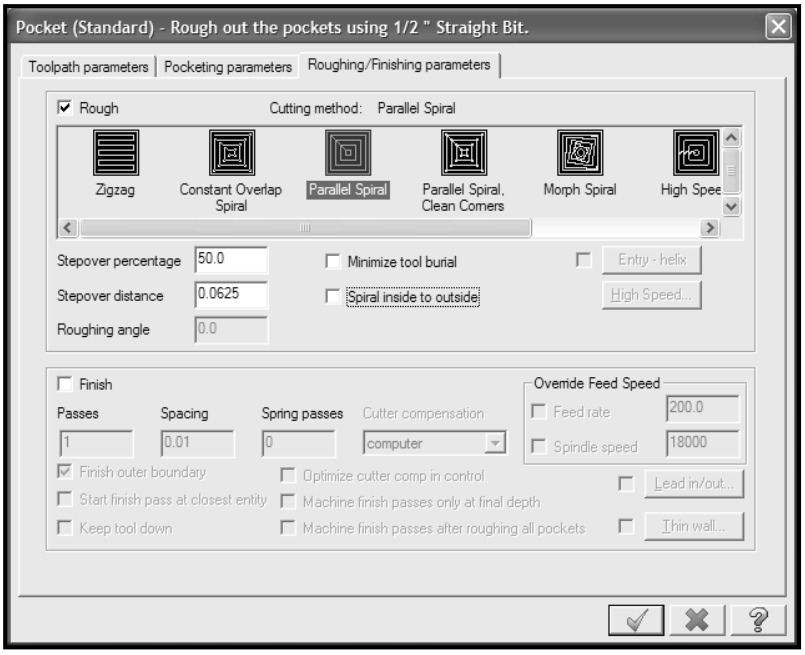
  - Compute remaining stock from roughing tool diameter enables the system to calculate the area for remachining based on the size of the **Roughing tool diameter**.

- **Clearance** extends the entry and exit area for remachining to prevent any remaining cusps.
- **Applying entry/exit curves** will ensure that the tool will have a smooth entry/exit from an already machined area.

- Select **Depth** cuts button and change the parameters as shown.



- Select the **Roughing/Finishing parameters** page.



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

- Select the **Regen all dirty operations** button to recalculate the toolpath taking in account the changes that you made.

- Repeat **Step 18 page 2-20** if you want to verify the remachining toolpath using **Backplot**.

STEP 20: CONTOUR THE OUSIDE PART USING ½“ STRAIGHT BIT

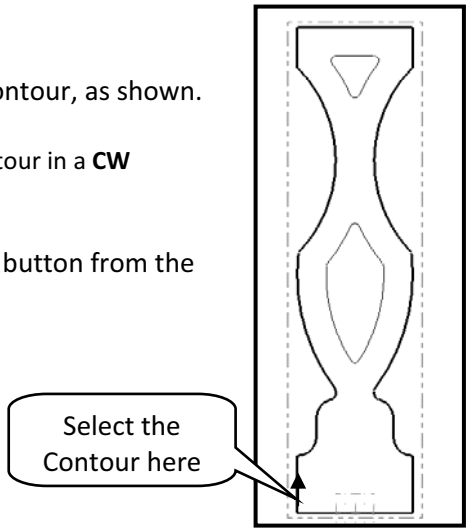
Toolpaths

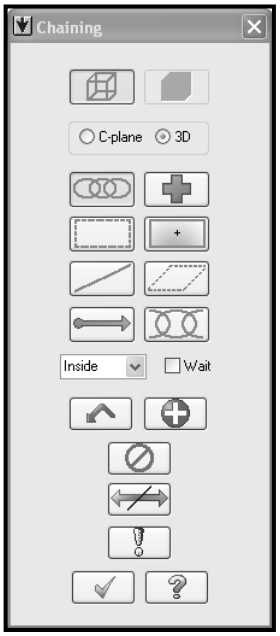
- **Contour**
- Select the first entity in the contour, as shown.

  - Be sure to chain the contour in a **CW** direction.

- Otherwise select the **Reverse** button from the **Chaining Dialog** box.

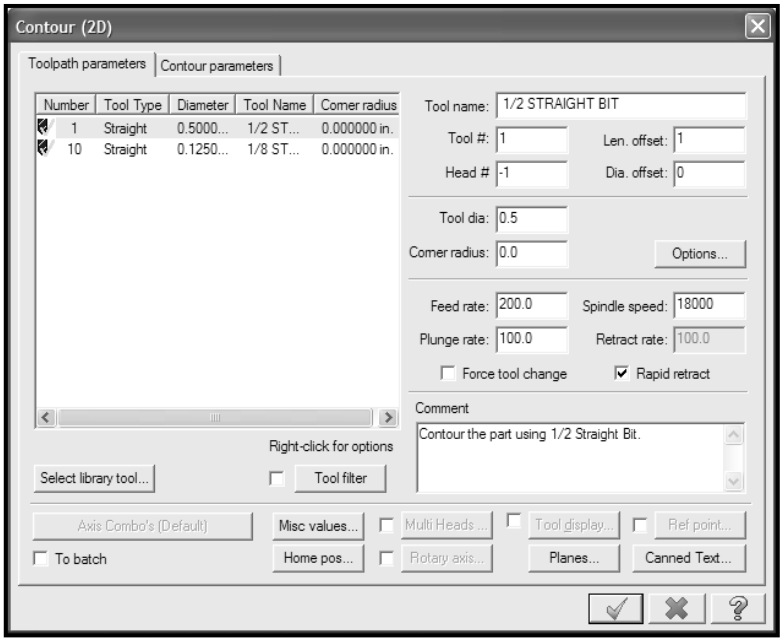
- Select the **OK** button to exit **Chaining**.





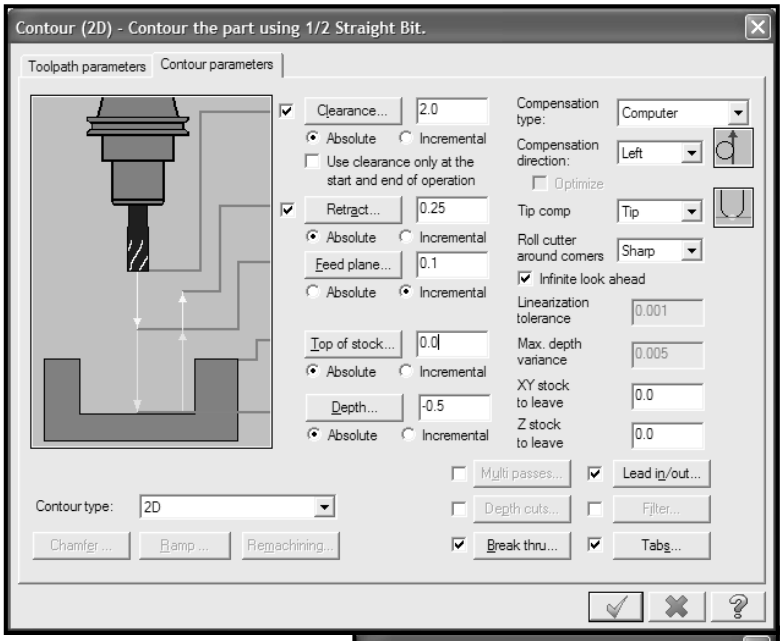
- Click on **Select library tool** button.
- Select the existing ½ " Straight Bit.

  - The **Toolpath parameters** dialog box allows you to select the tool used in this operation. It also allows you to change the **Spindle speed**, the **Feed rate**, and **Retract rate**. You can insert a comment that will be output in the NC file after running the post processor.

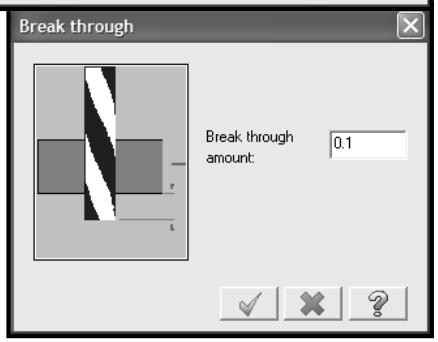


- Select the second page and fill out the parameter screen as shown in the diagram.

  - The **Contour parameters** dialog box allows you to establish the heights for rapid movement **Clearance**, from where the tool moves with feedrate **Feed plane**, the **Top of stock** and the final **depth**. You can use **Break through** parameters to cut further down. You can also establish the cutter **Compensation** direction, **Lead in/out** parameters and the **Tabs** to keep the part fixed on the machine table.

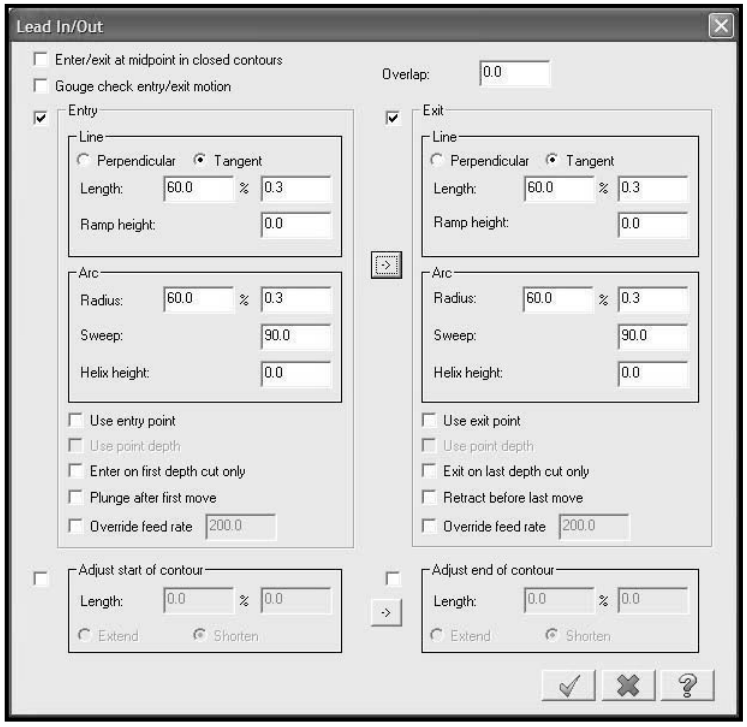


- Select the **Break through** button and set the **Break through amount** to 0.1 as shown in the following screen shot.



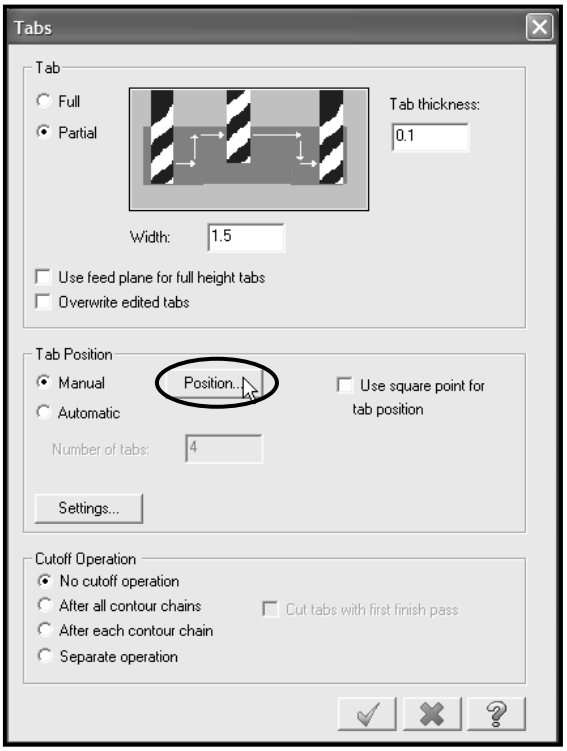
- Select the **OK** button. 

- Select **Lead in/out** button and set the parameters as shown in the screenshot to the right, to establish a smooth entry/exit into material.



- Select the **OK** button to exit **Lead in/out** dialog box. 

- Select the **Tabs** button from the parameter screen and follow next instructions to establish the tabs size and locations on the part.

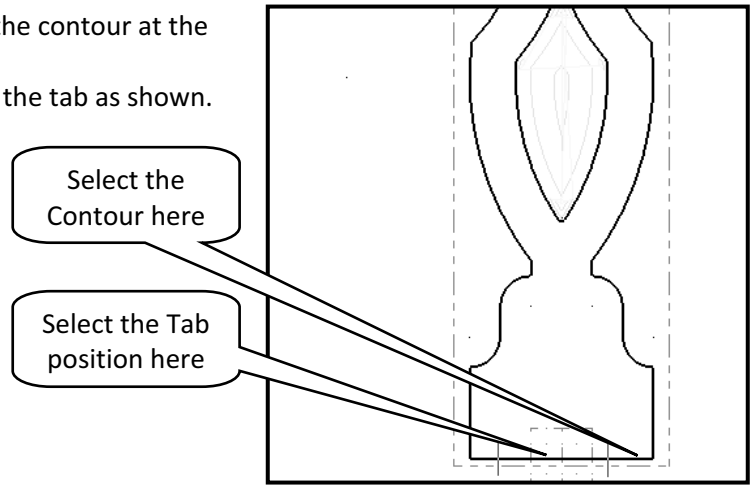


  - The Tabs are used to hold down the part on a small part of the main body of the material because the vacuum clamping might not sufficiently hold it down. The tabs could be either cut off later or broken by hand.

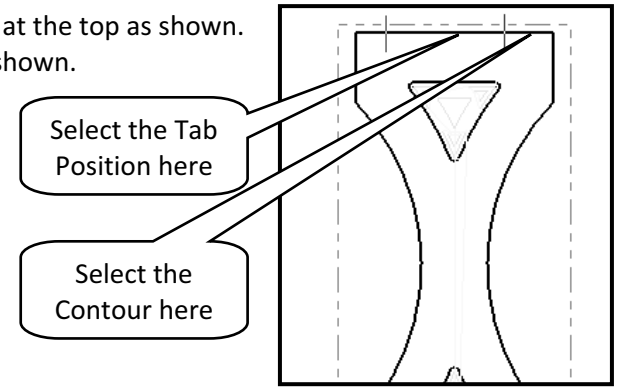
- Choose **Partial tab**, the **Width** and the **Tab thickness**.
- For the **Tab Position** choose **Manual**.
- Select the **Position** button.

  - The graphic mode will be enabled, and your are prompted to choose the contour and the location of the tab.

- [Select chains to add tabs to]: Select the contour at the bottom as shown.
- [Place tab at desired position]: Select the tab as shown.



- [Select chains to add tabs to]: Select the contour at the top as shown.
- [Place tab at desired position]: Select the tab as shown.

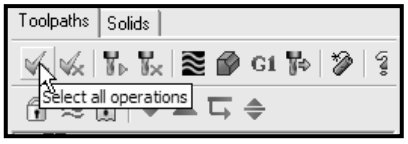


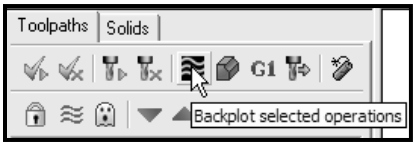
- Press **Enter** to exit.
- Press the **OK** button to exit **Tabs** settings.

- Select the **OK** button from the parameter screen to exit contour settings. 

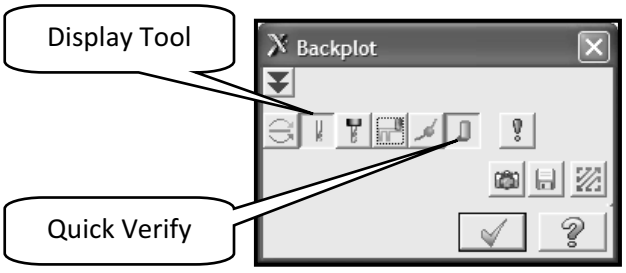
STEP 21: BACKPLOT THE TOOLPATH.

- Click on the **Select all operations** button in the **Toolpaths Manager** tab and**.**

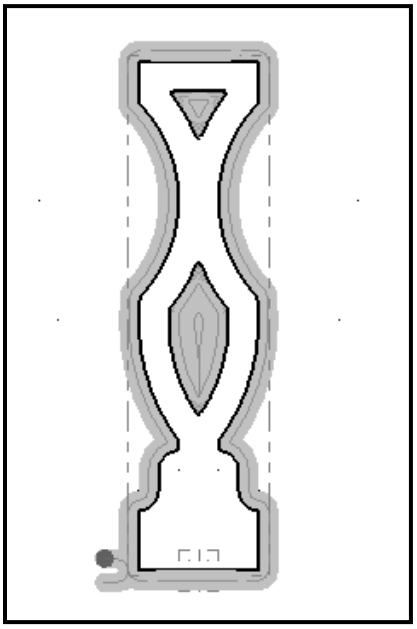




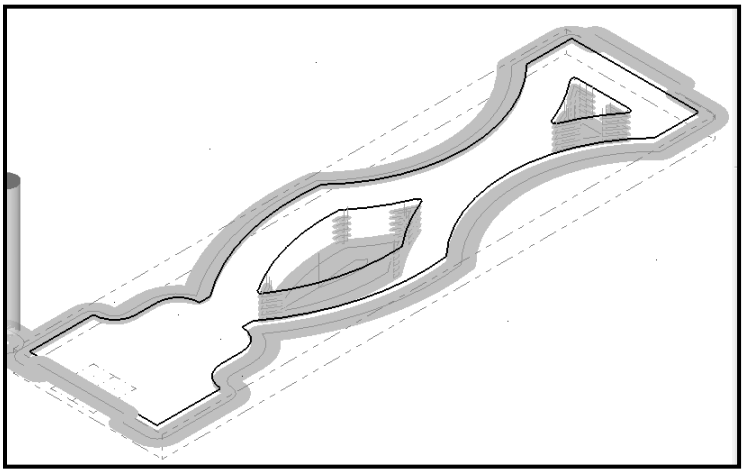
- Select **Backplot selected operations** button.
- Make sure that you have the following buttons turned on. (They look as push-down buttons)
- Enable **Display tool**
- Enable **Quick verify**



- Select **Play** button.



- Change the graphic view to **Isometric**. 



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

STEP 22:VERIFY

- Select **Verify selected operations** button.

Update after each toolpath updates the stock after each operation.

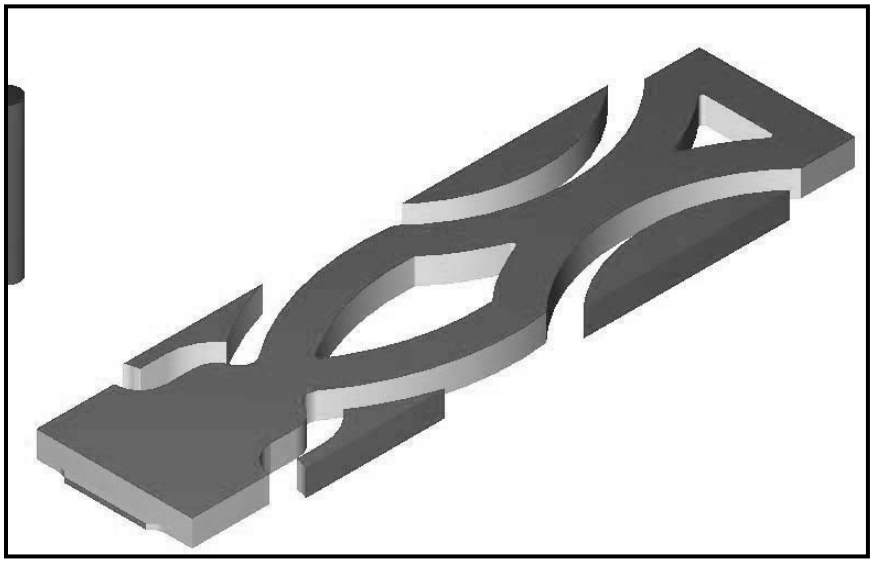
Stop on collision pauses the verification when the tool touches the part with a rapid move.

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

- Select **Play** button to start simulation.

- The computer will now simulate the process of the part being machined.

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



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

STEP 23: SAVE THE UPDATED MCX FILE.

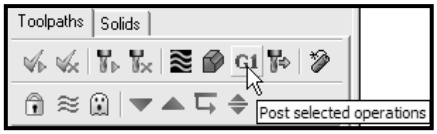
- Select the **Save** icon.

STEP 24: POST THE FILE

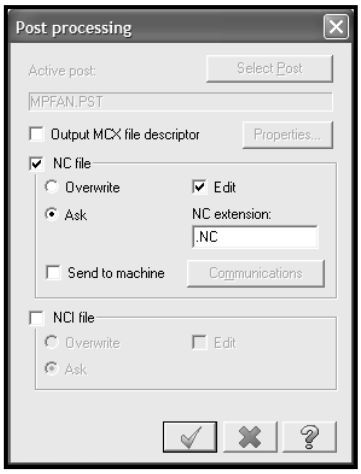
- The final step in Mastercam is to generate the G-codes that the CNC machine controller uses to cut the part. The Post processor is a translator that will translate the NCI file that contains the toolpath information, into the machine controller language.

In this example we are using a generic Fanuc controller.

- Make sure that all operations are selected, otherwise:
- **Select all operations**
- Select **Post selected operations** button from **Toolpath Manager**.



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

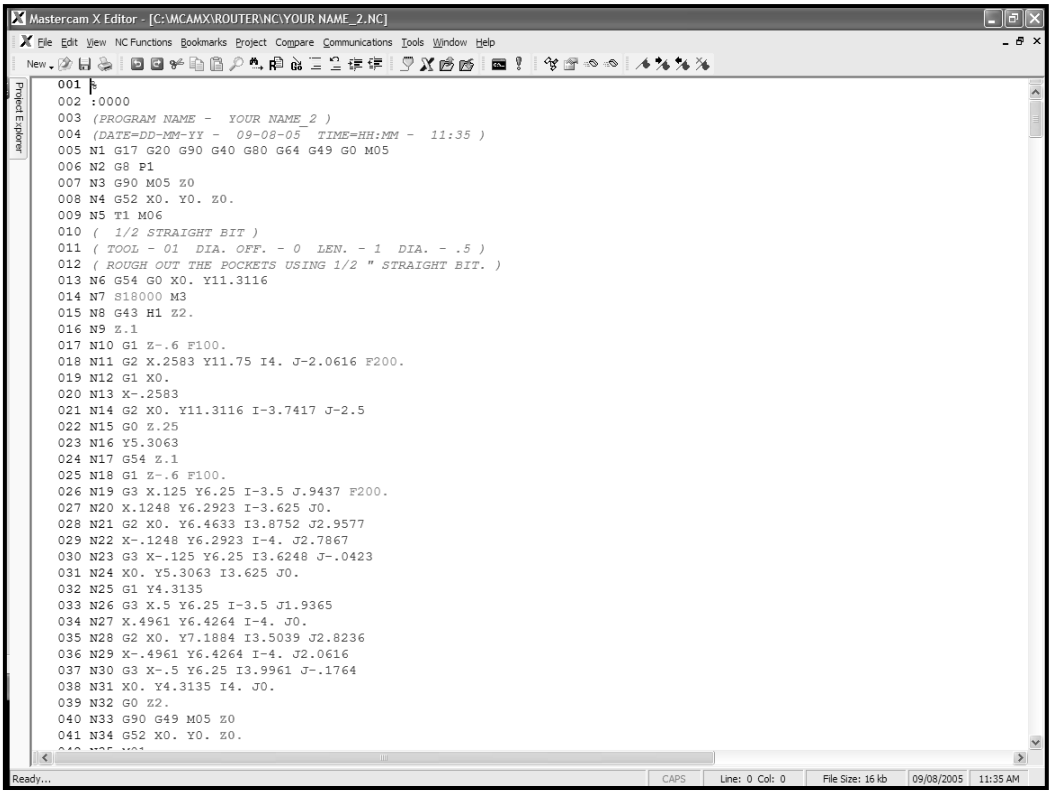


NC file enabled allows you to keep the NC file and to assign the same name as the MCX file
Edit enabled allows you to automatically launch the default editor.

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

- Enter the same name as the geometry name " Your Name _2" in the **NC File name** field.

- Select **OK** button to save the NC file.



- The G code that you have created will appear on the screen; if the code looks okay you can shut the window down without saving it. If you need to change the code, make sure you save it before you exit.

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

STEP 25: SAVE THE UPDATED MCX FILE

Select **Save** icon.

NOTES:

TUTORIAL 2 QUIZ

- What does a pocket operation allow you to do?

- What does a post processor do?