Mastercam®

Mill Level 3 Training Tutorial

X9

Mastercam X3

Mill Level 3 Training Tutorials

TABLE OF CONTENTS

Getting Started **A-1**
Graphic User Interface A-1
Navigate Through the System A-2
Setting the Toolbar States A-4
Setting the Grid A-6

Surface Modeling **B-1**

TUTORIALS **1-1**
Tutorial #1, Revolved & Flat Boundary Surfaces, Pocket, Parallel & Contour Toolpaths 1-1
Tutorial #2, Ruled and Flat Boundary Surfaces, Radial and Scallop Toolpaths 2-1
Tutorial #3, Net Surface, Pocket and Restmill & Finish Blend Toolpaths 3-1
Tutorial #4, Swept & Flat Boundary Surfaces, Rough Flowline & Plunge, Finish Blend & Pencil Toolpaths 4-1
Tutorial #5, Revolved & Fillet Surfaces, HS Core Roughing & Finish Scallop, Pocket facing, Pencil & Project Toolpaths 5-1
Tutorial #6, Revolved surface, HS Area Roughing, Finish Contour, Leftover and Project for Machining Raised Letters 6-1
Tutorial #7, Draft, Swept, Revolve and Fillet Surfaces and High Speed Toolpaths: Core Roughing, Horizontal Area, Waterline and Scallop Rest Passes 7-1
Tutorial #8, Ruled, Swept, Revolve and Fillet Surfaces and High Speed Toolpaths: Core Roughing, Horizontal Area, Scallop, Scallop Rest Passes and Pencil 8-1

General Notes **C-1**
Default Key Assignments C-2
Customizing C-3
Key Mapping C-7
Data Entry Shortcuts C-9
Surface Toolpath Parameters C-10
Create Geometry in 3D C-39
Solids Menu C-44
Solids Manager C-45
Chaining C-47
Window Selection C-50
Chaining and Window Options C-51
Toolpaths Manager C-53
Properties C-58
Milling G-Codes C-61

TUTORIAL SERIES FOR

Mastercam X3

HOW TO USE THIS BOOK

This book provides a comprehensive step-by-step approach to learning Mastercam Mill 3D. The book includes eight projects, an additional 8 exercises and quizzes for each project to test your knowledge.

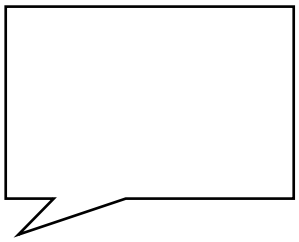
The material covered includes 3D Wireframe and Surface geometry creation, Surface Finish and Surface Roughing Toolpaths. It also contains the advanced Surface High Speed Toolpaths. Explanations are given for the proper use of Stock Setup and Tool Settings. It teaches you how to create an STL Stock and how to use it in the solid model verification. C-hooks and tips on how to select and organize the part are also covered to complete the learning experience.
The Mill Level 3 Training Tutorials also include General Notes with useful tools and shortcuts that make the software easier to use. A description of the 3D toolpath parameters and the Operations Manager are also covered in this section.

Each tutorial walks you through all the procedures from 3D Geometry Creation to Surface 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



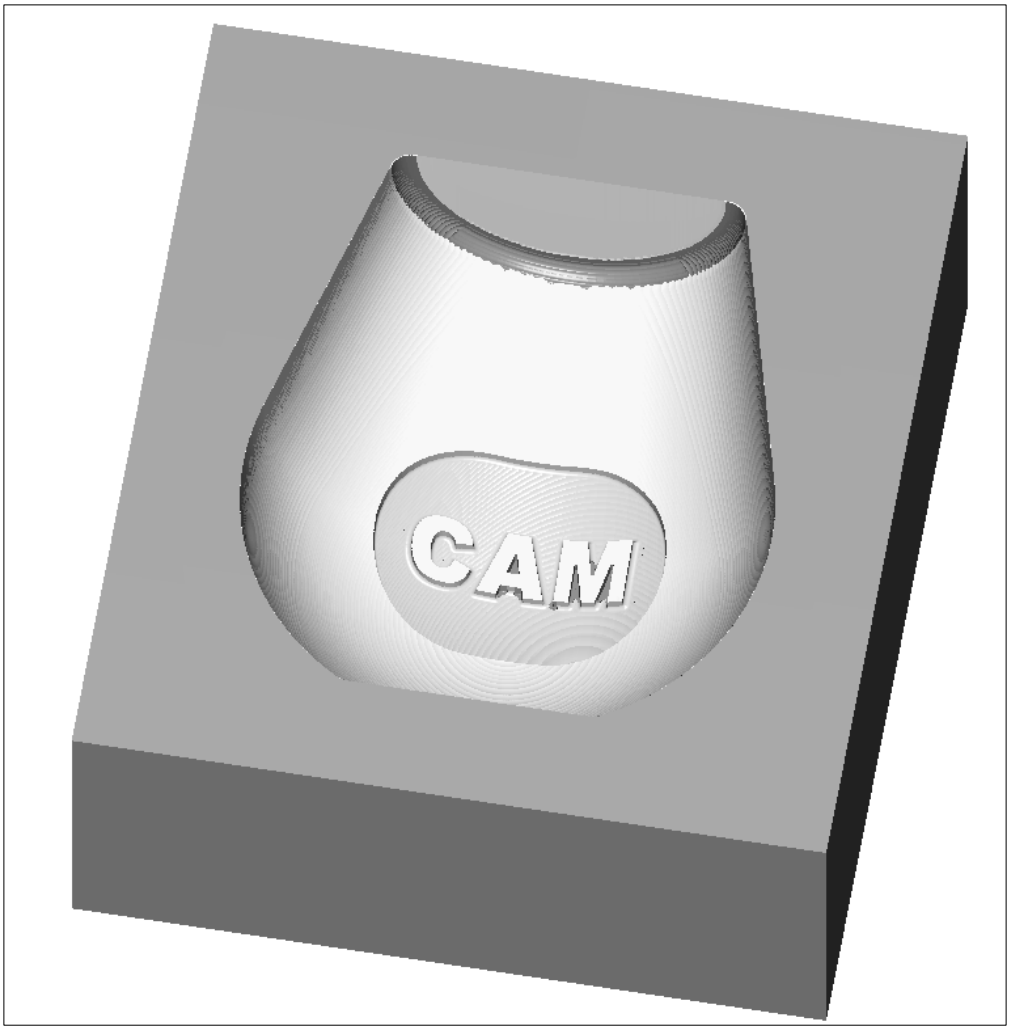
Callouts that describe the parameters used in the current step

Bold text (usually) represents Mastercam terminology

TUTORIAL SERIES FOR

Mastercam X3

TUTORIAL 6
REVOLVED SURFACE HIGH SPEED AREA ROUGHING, FINISH CONTOUR, FINISH LEFTOVER, AND FINISH PROJECT TOOLPATHS TO MACHINE RAISED LETTERS



Objectives:

The Student will design a 2-dimensional drawing by:

- Creating a rectangle.
- Creating parallel lines.
- Creating line endpoints.
- Creating arc tangent through a point.
- Creating fillets.
- Trimming the geometry.
- Moving the geometry.
- Creating the letters.
- Creating an obround shape.

The Student will design a 3-dimensional drawing by:

- Creating the revolved surface.
- Creating a flat surface using rectangle command.
- Trim surface to curve.
- Creating the bounding box.
- Creating an offset surface.

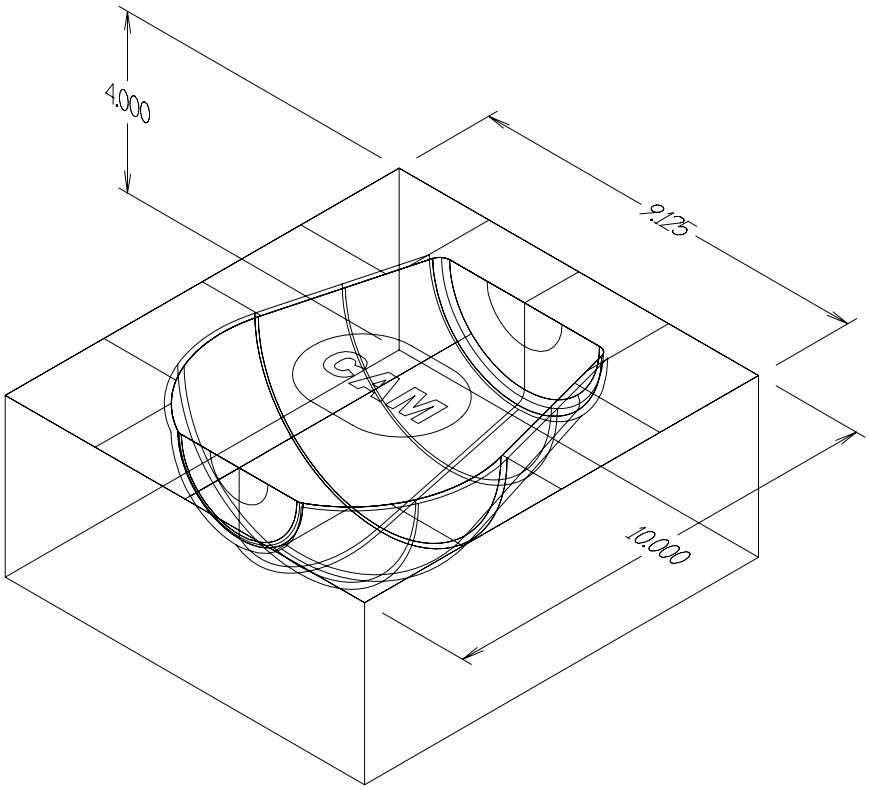
The Student will create a 3-dimensional milling toolpath consisting of:

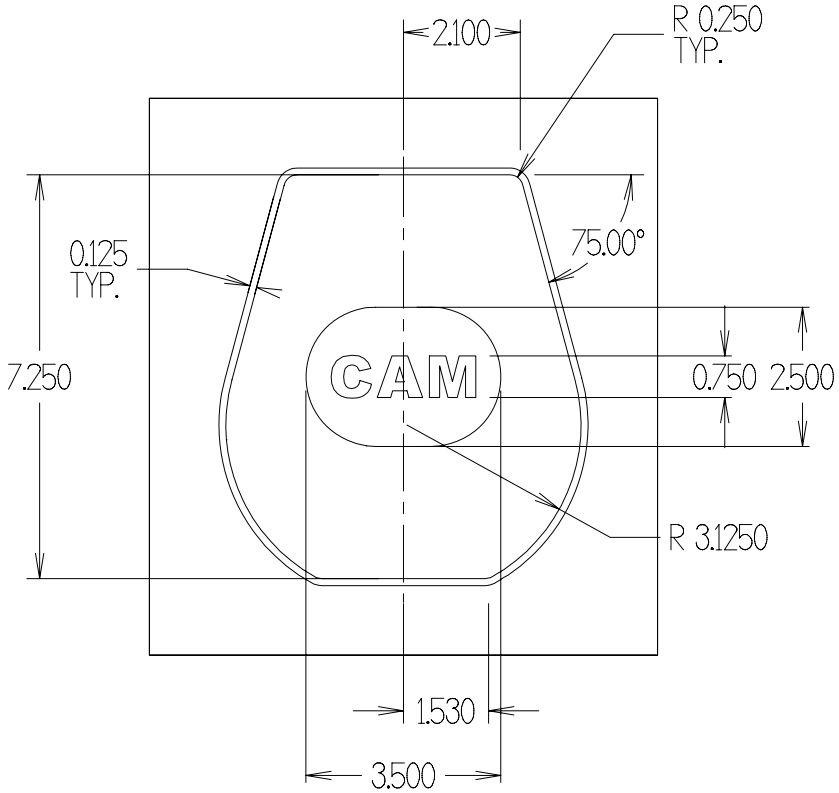
- Roughing the cavity using surface high speed area mill toolpath.
- Finishing the cavity using surface finish contour toolpath.
- Finishing the fillets using surface finish leftover.
- Creating two 2D pockets toolpaths to be used in surface finish project.
- Using finish project toolpaths to machine the raised letters.

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

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

ALL DIMENSIONS IN INCHES





TRUE TYPE FONT ARIAL BLACK
LETTER HEIGHT = 0.750 "
DISTANCE BETWEEN LETTERS = 0.1
STARTING POSITION OF LETTERS: X-1.30, Y-0.336

TITLE	TUTORIAL 6
MATERIAL	ALUMINUM T6061
DATE: JUNE 12, 2008	eMastercam.com

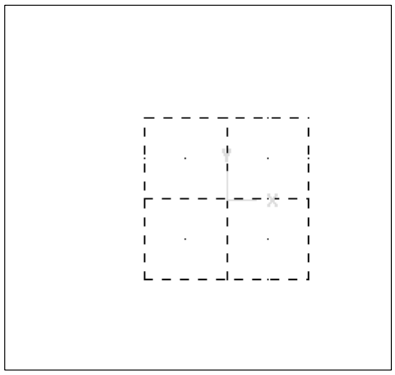
GEOMETRY CREATION

To start a new file from Mastercam:

File

New

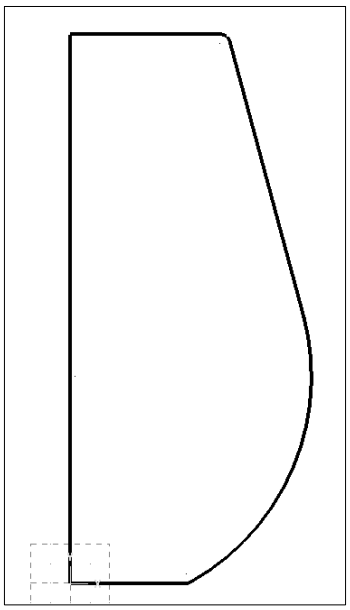
- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 3D part. See **Getting started**
- **Operation manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.
- Before starting the geometry make sure that the **Grid** is enabled. It will show you where the part origin is. See **Getting started** page A-6 for details.



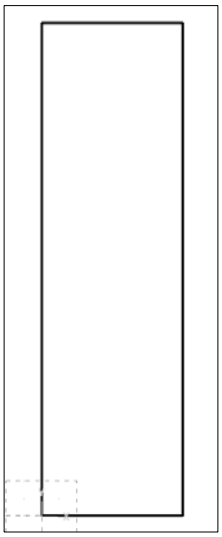
STEP 1: CREATE THE 2D PROFILE TO BE REVOLVED.

- Note that for the revolved surface we will only need half of the 2D wireframe.

Step Preview:

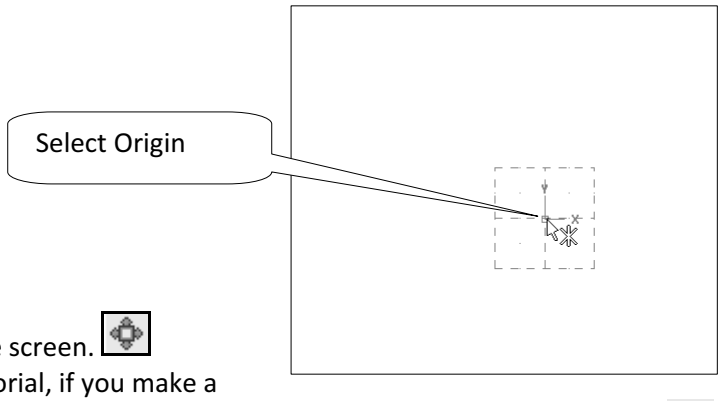


1.1 Create the 2.0 " by 7.0 " rectangle.
Sub Step Preview:



Create

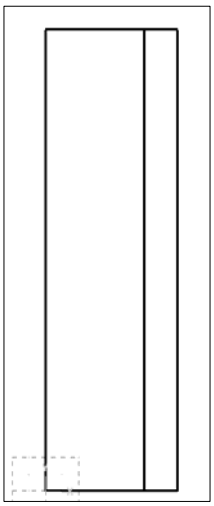
- **Rectangle**
  - Enter the **Width** 2.1 (Tab).
  - Enter the **Height** 7.25 (Enter).
    - Make sure that Anchor to center is not enabled.
  - [Select position of first corner]: Select the **Origin** (center of the grid) as shown.



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

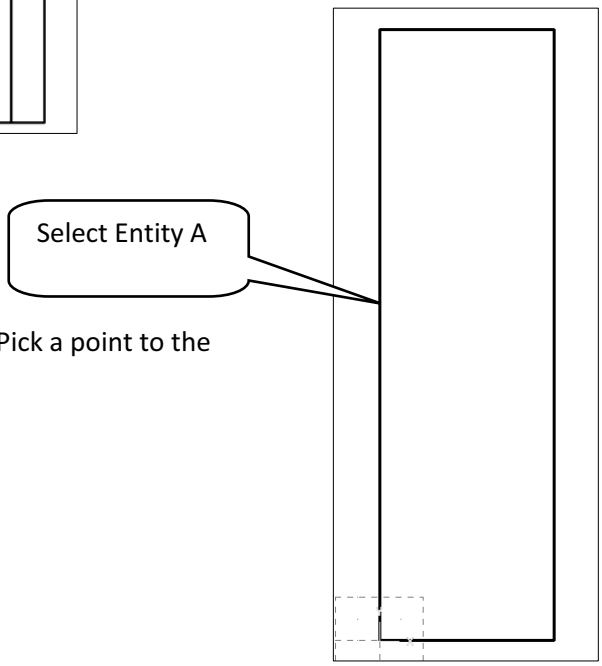
- During the geometry creation of this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.
  - Select the **OK** button.

1.2 Create a line parallel
Sub Step Preview:

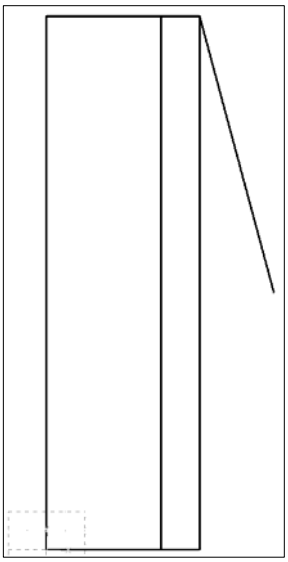


Create

- **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** 1.53 (Press Enter).
  - Select the **OK** button.

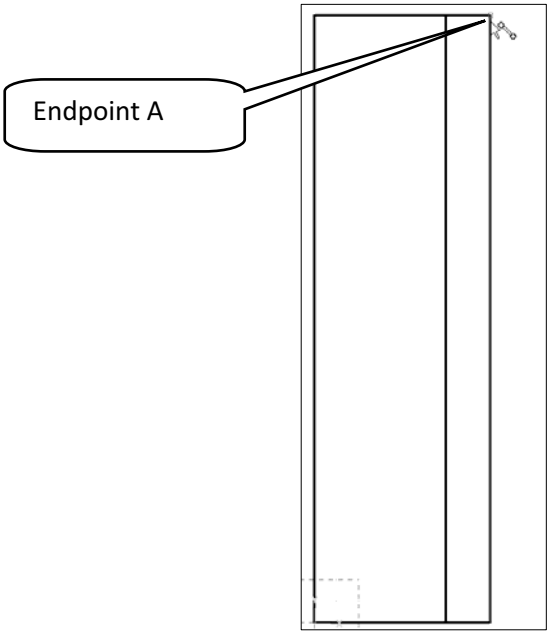


1.3 Create a line knowing one endpoint, the length and the angle
Sub Step Preview:

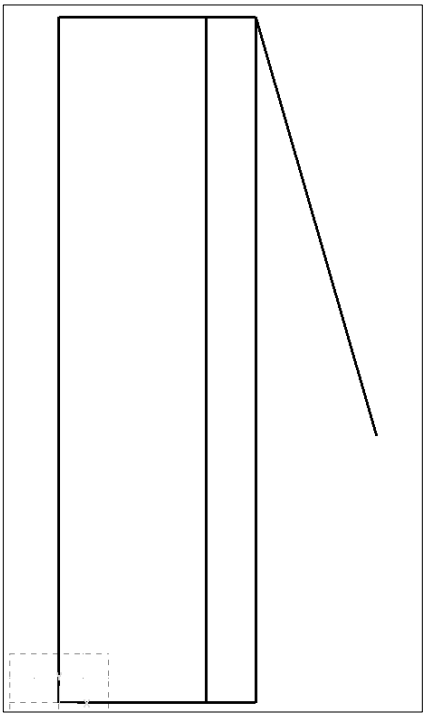


Create

- **Line**
- **Endpoint**
  - [Specify the first point]: Select Endpoint A as shown:

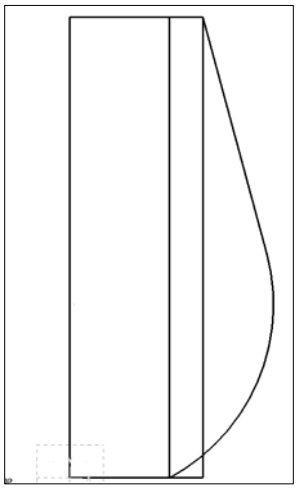


- Sketch the line as shown.
- In the **Ribbon Bar**, change the **Angle** to 285 and press Enter



- Select the **OK** button.

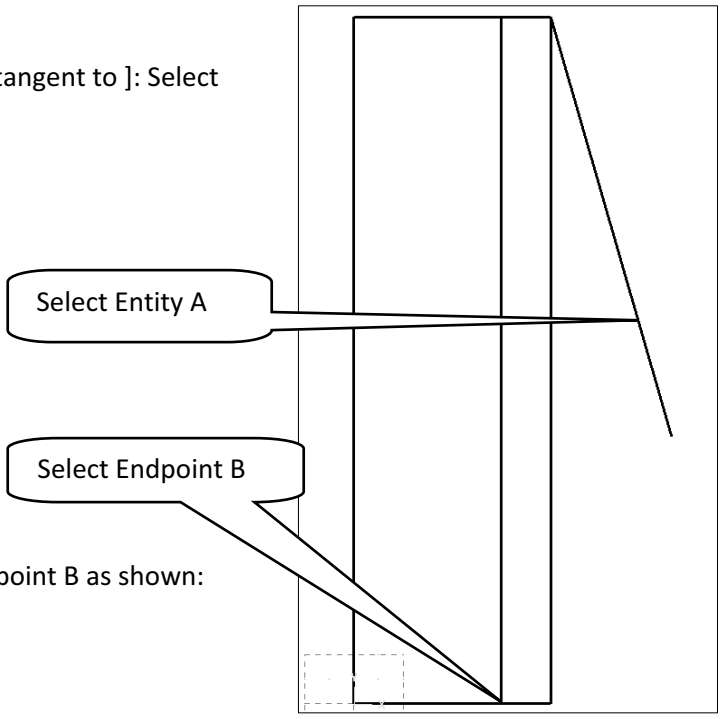
1.4 Create an arc tangent through a point
Sub Step Preview:



Create

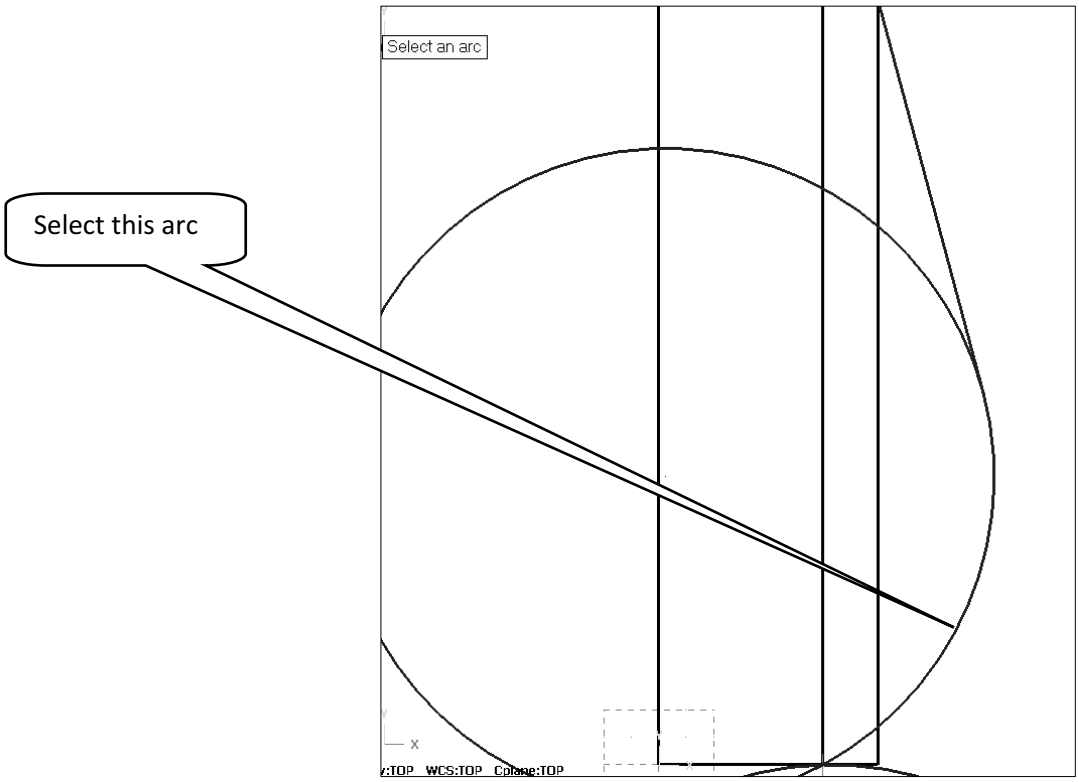
- **Arc**
- **Arc Tangent**

- Enable **Tangent point.**
- Enter the **Radius** 3.125
- [Specify the entity that the arc is to be tangent to]: Select Entity A as shown.



- [Specify the tangent point]: Select Endpoint B as shown:

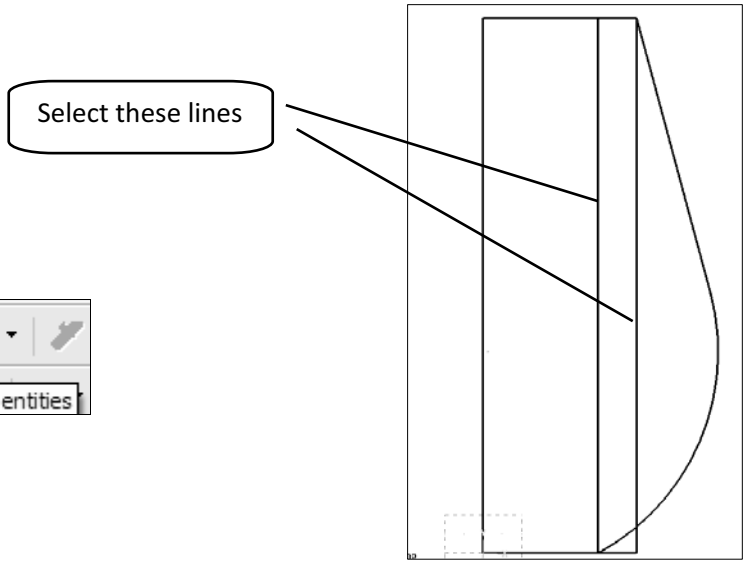
- [Select an arc]: Select the arc as shown:



- Select the **OK** button.

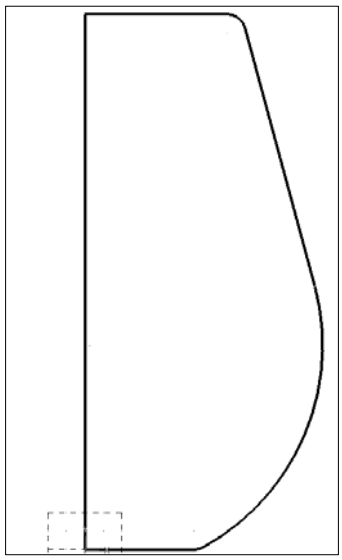
1.5 Delete the extra construction lines

- Select the two lines as shown:



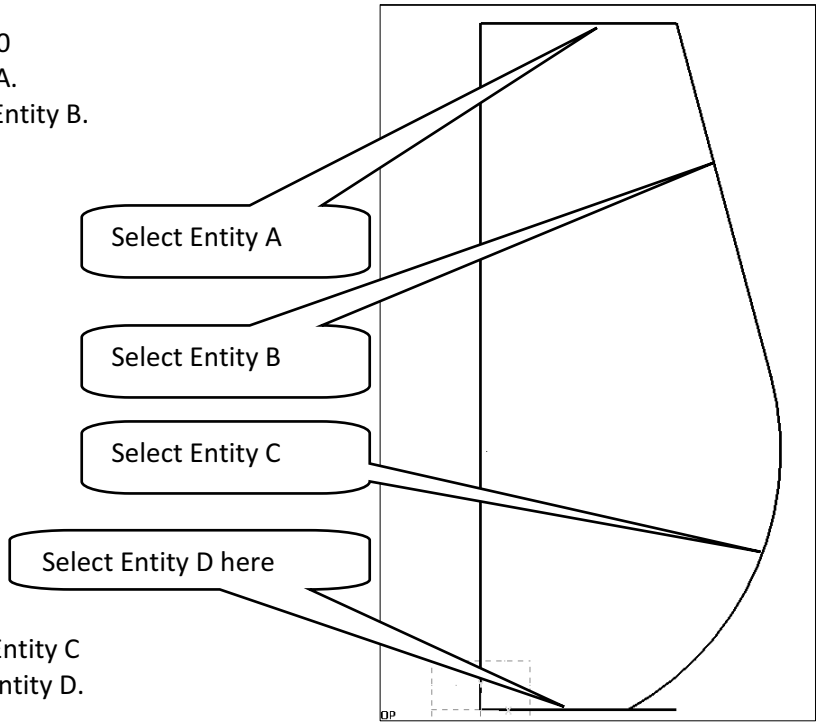
- Select **Delete entities** icon. 

1.6 Fillet the corners with the 1/8 " radius.
Step Preview:



Create

- **Fillet**
- **Entities**
  - Enter the fillet **Radius** 0.250
  - [Select an entity]: Select Entity A.
  - [Select another entity]: Select Entity B.

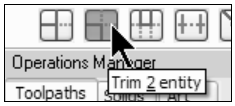


  - [Select an entity]: Select again Entity C
  - [Select another entity]: Select Entity D.
  - Select the **OK** button.

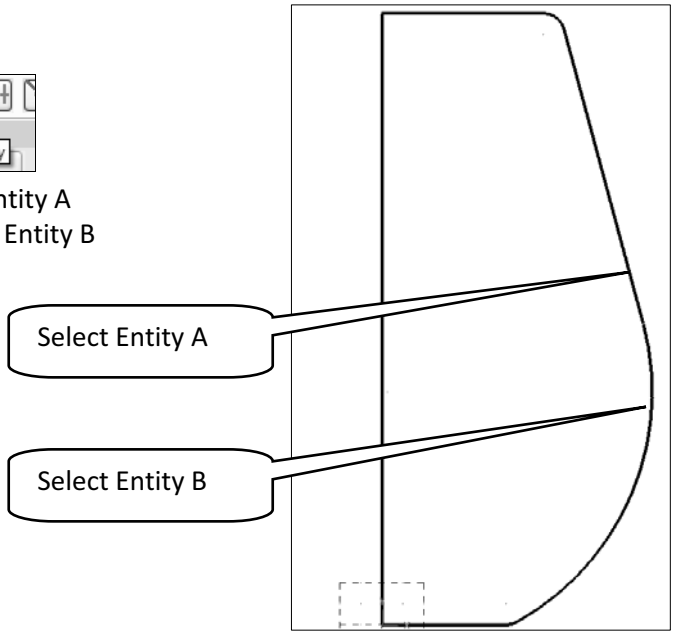
1.7 Trim two entities

Edit

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



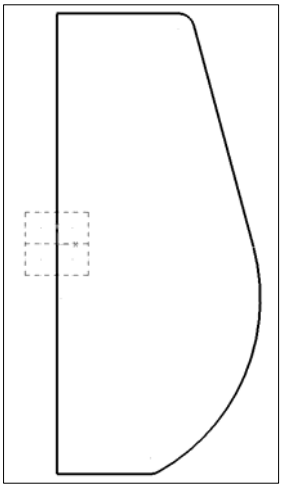
- Enable Trim 2 entities icon.
- [Select the entity to trim/extend]: Select Entity A
- [Select the entity to trim/extend to]: Select Entity B
-



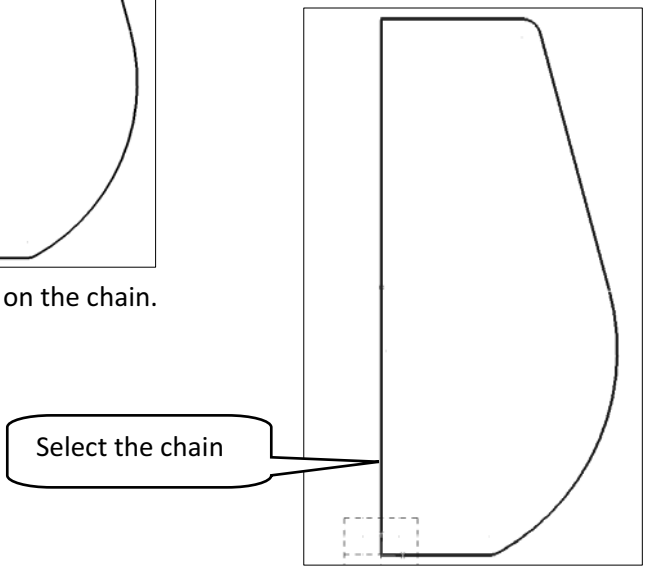
- Select the **OK** button.

STEP 2: MOVE THE GEOMETRY USING MOVE TO ORIGIN

Step Preview:

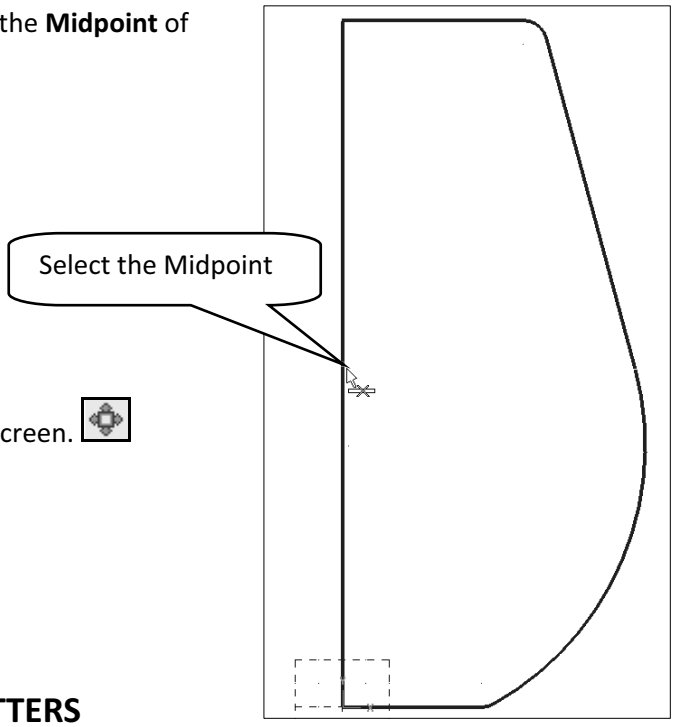


- Hold-down the **Shift** key and click somewhere on the chain.



Xform

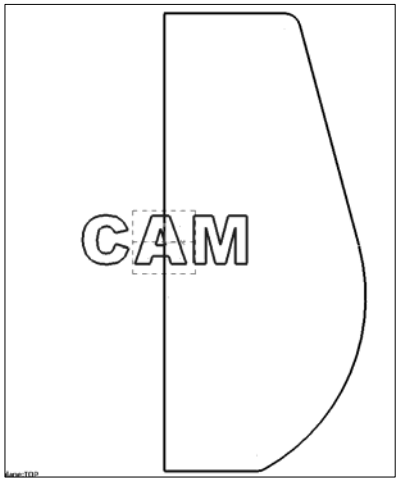
- **Move to origin**
  - [Select the point to translate from]: Select the **Midpoint** of the vertical line as shown:



  - Select the **Fit** icon to fit the drawing to the screen.
  - Select **Clear Colors** icon.

STEP 3: CREATE AND CENTER THE LETTERS

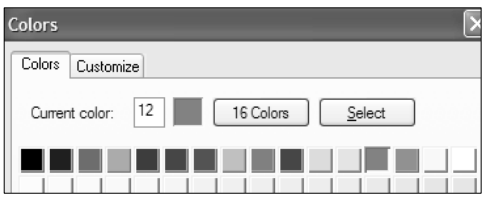
Step Preview:



3.1 Change the main color to red (No. 12)

- Select the **Color** in the **Status Bar**

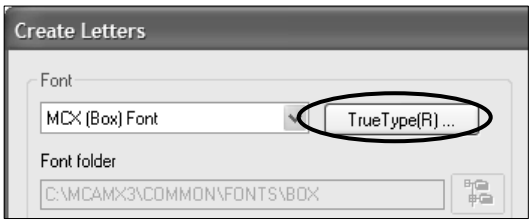
- Select color red as shown:



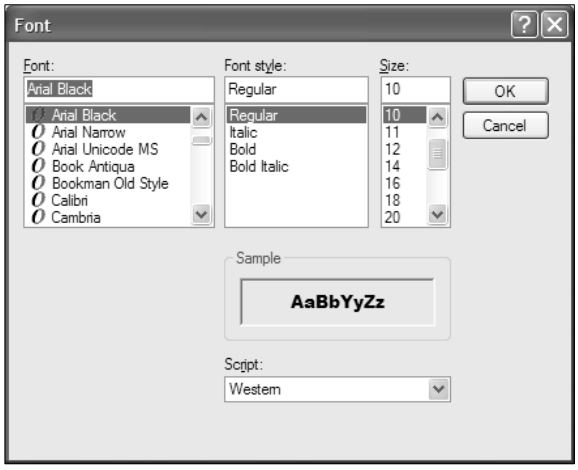
3.2 Create the letters

Create

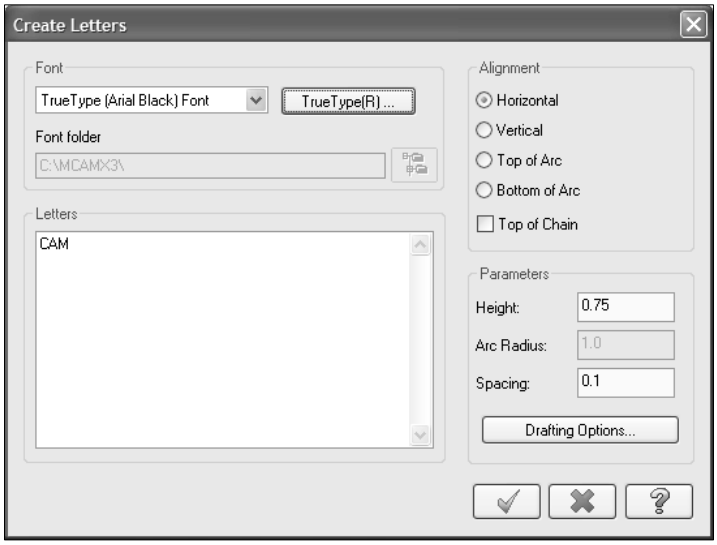
- **Letters**
  - Select **True Type** button.



  - Select the **Arial Black** font as shown.
  - Select the **OK** button.



  - Enter the **CAM** and change the **Height** and **Spacing** as shown.
  - Select the **OK** button. 

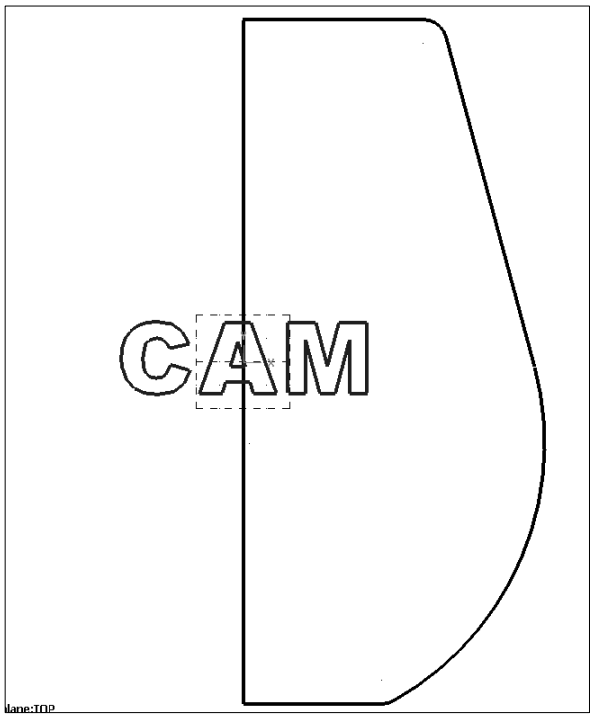


  - [Enter starting location of letters]: Select the **Fast Point** icon.
  - Enter the **X**-axis, **Y**-axis values as shown: 

  - Press **Enter** to finish.
  - Press **Esc** to exit the command.

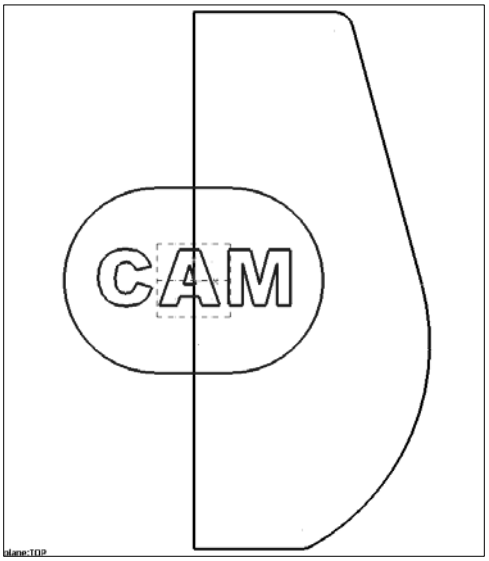
Select the **Fit** icon to fit the drawing to the screen.

The geometry should look as shown:



STEP 4: CREATE THE OBROUND SHAPE

Step Preview:

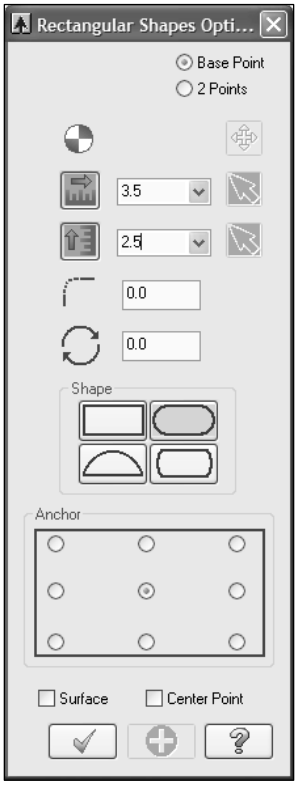


Create

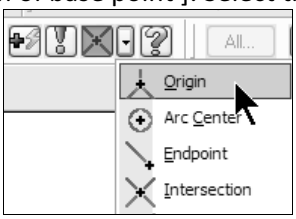
Rectangular Shapes

- Expand the dialog box if needed by selecting the down arrow.

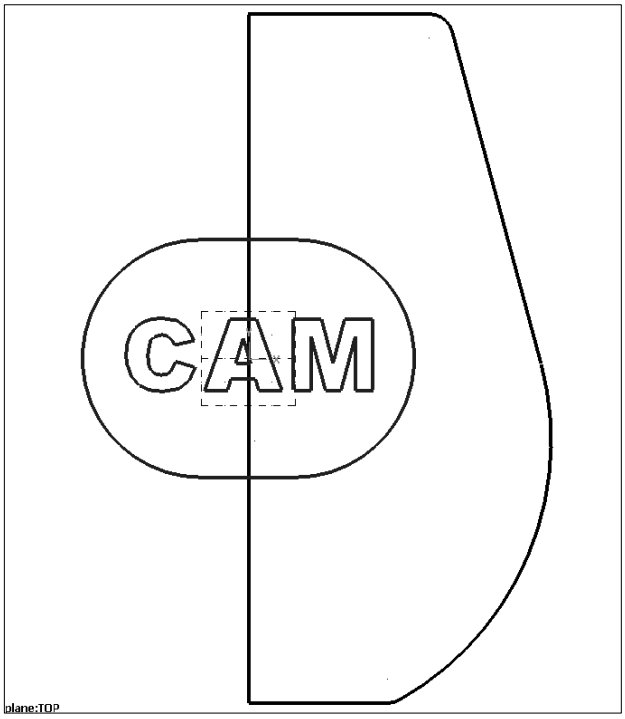
- Change the parameters as shown.



- [Select position of base point]: Select the **Origin** from the **Autocursor** drop-down list.

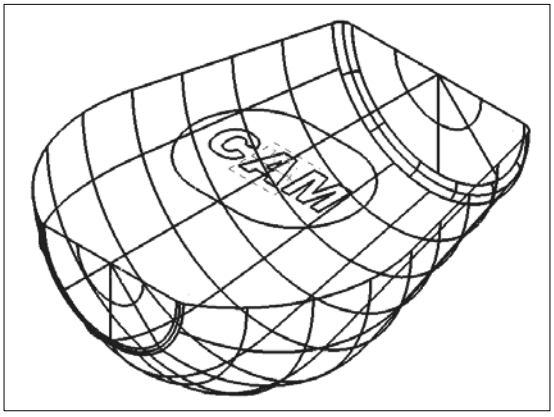


- Select the **OK** button to exit **Rectangular Shapes Options** dialog box 



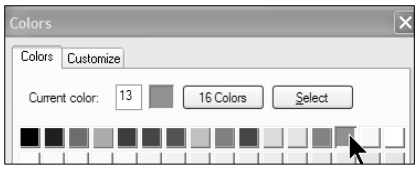
STEP 6: CREATE THE REVOLVED SURFACE(S).

Step Preview:



6.1 Change the main color to magenta (No. 13)

- Select the **Color** in the **Status Bar** as shown in the previous step
- Select color magenta

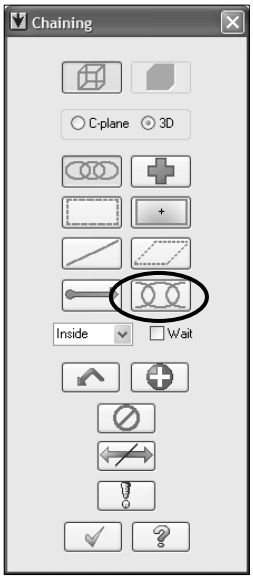


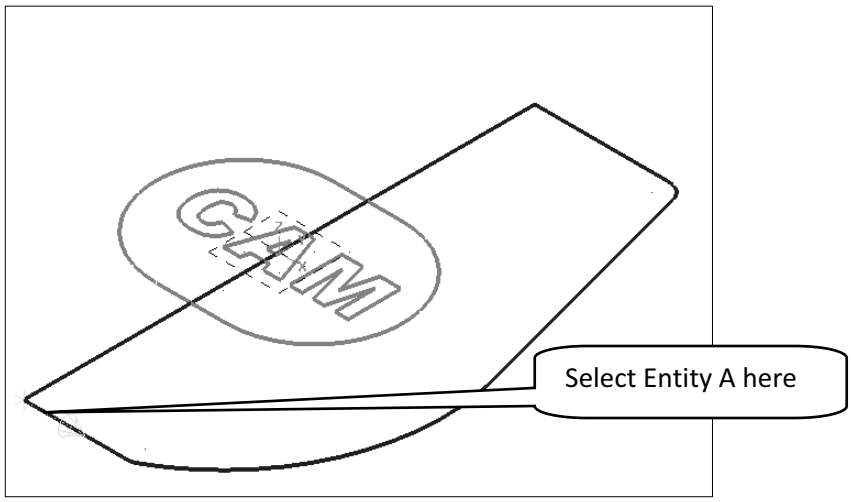
- Change the graphic view to **Isometric**. 

6.2 Create the Revolved surface

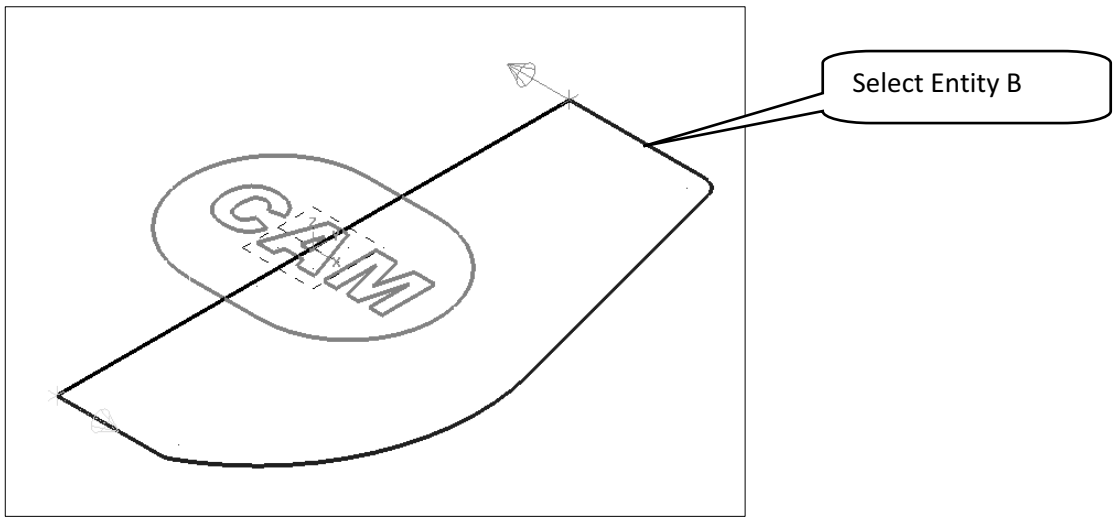
Create

- **Surface**
- **Revolved**
  - Select **Partial** in the **Chaining** dialog box
  - [Select the first entity]: Select Entity A as shown:
    - Make sure that the chaining direction is CCW; otherwise select **Reverse** button. 

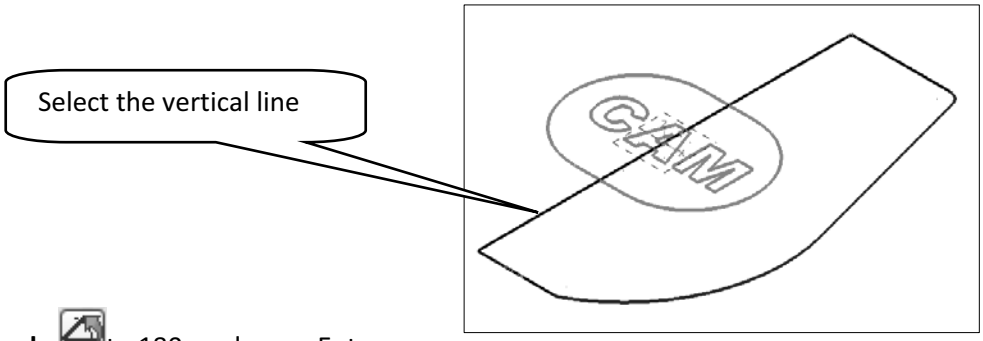




- [Select the last entity]: Select Entity B as shown:

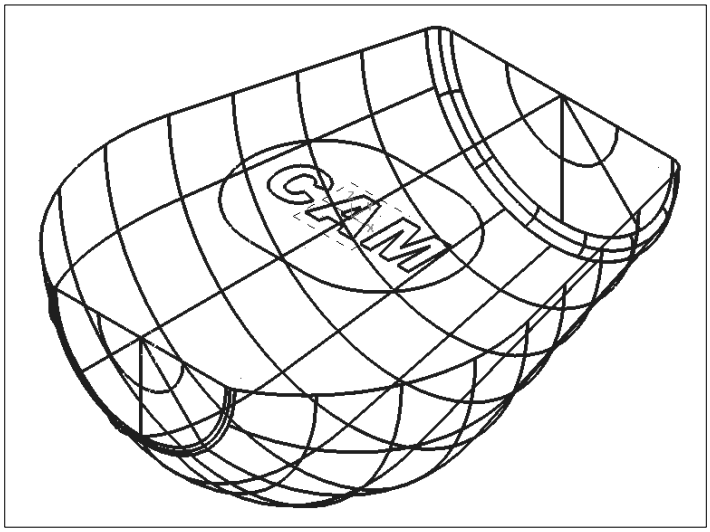


- Select the **OK** button to exit **Chaining** dialog box.
- [Select the axis of rotation]: Select the vertical line



- Change the **End Angle** to 180, and press Enter

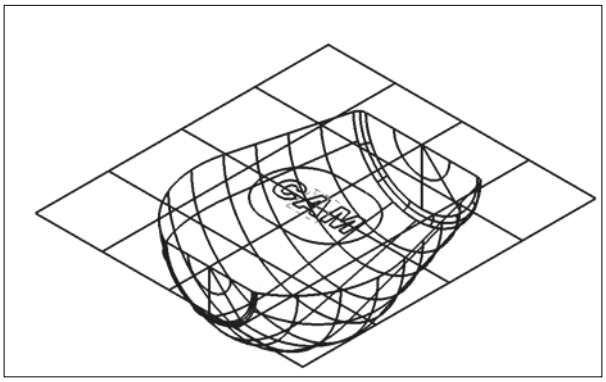
*The surface should look as shown, otherwise, select the **Reverse** button.*



- Select the **OK** button.

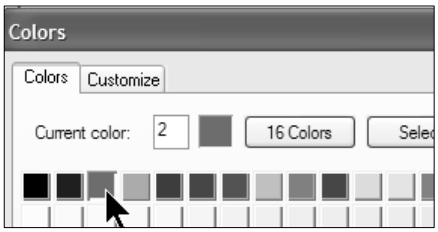
STEP 7: CREATE THE FLAT SURFACE INSIDE OF A RECTANGLE

Step Preview:



7.1 Change the main color to dark green (No. 2)

- Select the **Color** in the **Status Bar** as shown in the previous step
- Select the dark green color



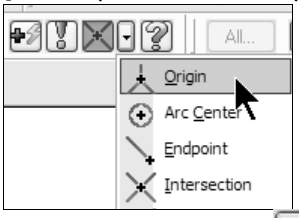
- Geometry color must be different then live entity color.

7.2 Create a rectangle with a surface inside of it

Create

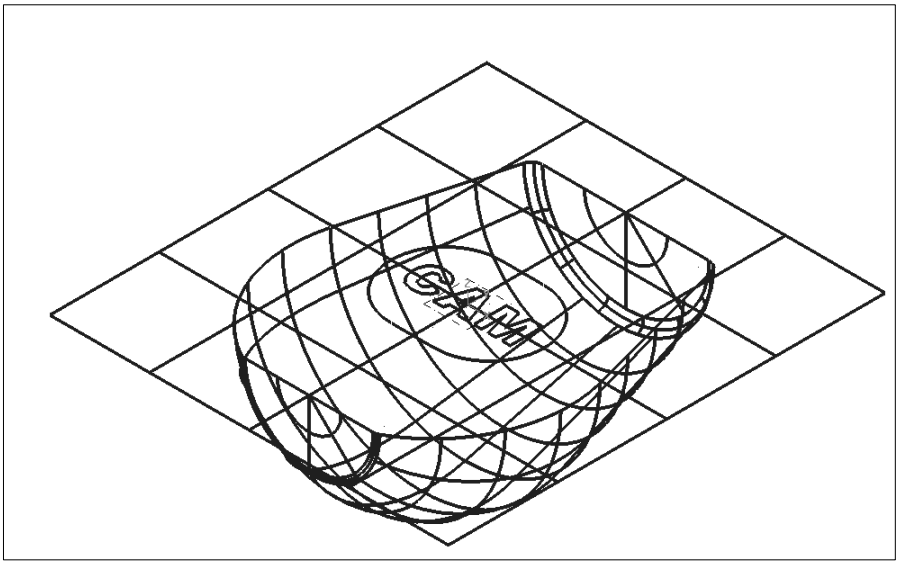
- **Rectangle**
  - Enter the **Width** 9.125 (Tab).
  - Enter the **Height** 10.0 (Enter).
  - Enable Anchor to center and Create Surface icons in the Ribbon Bar.

  - [Select position of base point]: Select the **Origin** from the **Autocursor** drop-down list.



  - Select the **OK** button.

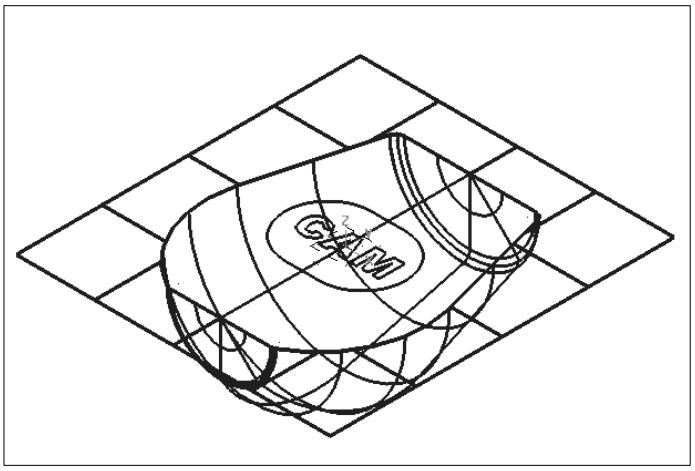
Select the **Fit** icon to fit the drawing to the screen.



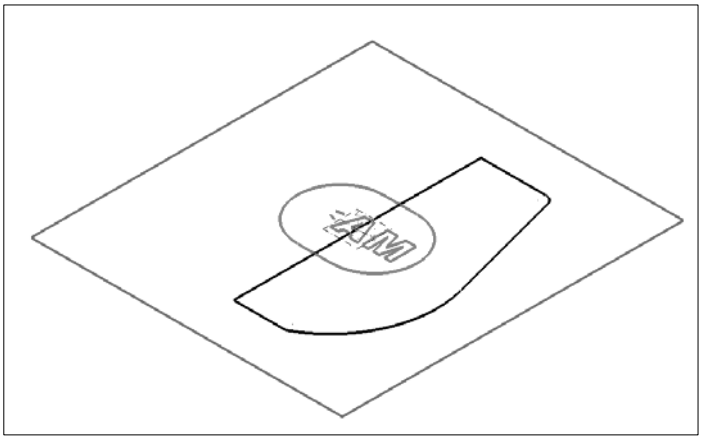
STEP 8: TRIM THE FLAT SURFACE TO CURVE

To trim a surface or a set of surfaces to a chain of curves we need a closed boundary (Mastercam understands curves to be lines, arcs or splines). You can trim a surface or set of surfaces with an open chain only if the chain completely divides the original surfaces in two.

Step Preview:



8.1 Use the Hide entity command to keep on the screen only 2D wireframe
Sub Step Preview:

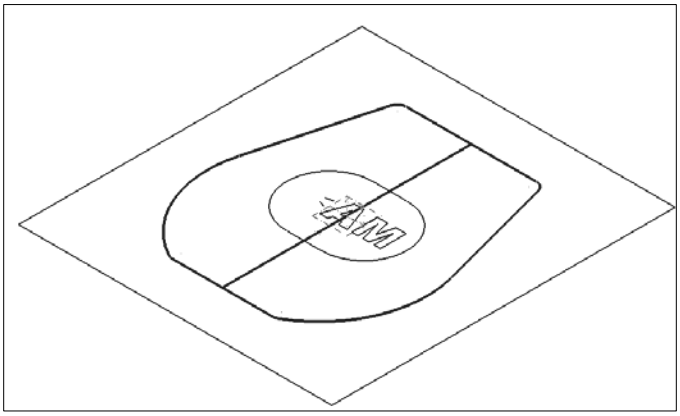


- Click on the **QM Lines** and on the **QM Arcs** icons from the **QM toolbar** at the left of the graphic window.
  - Note that all lines and all arcs from the graphic window are selected.
- Select the **Hide entity** icon. 

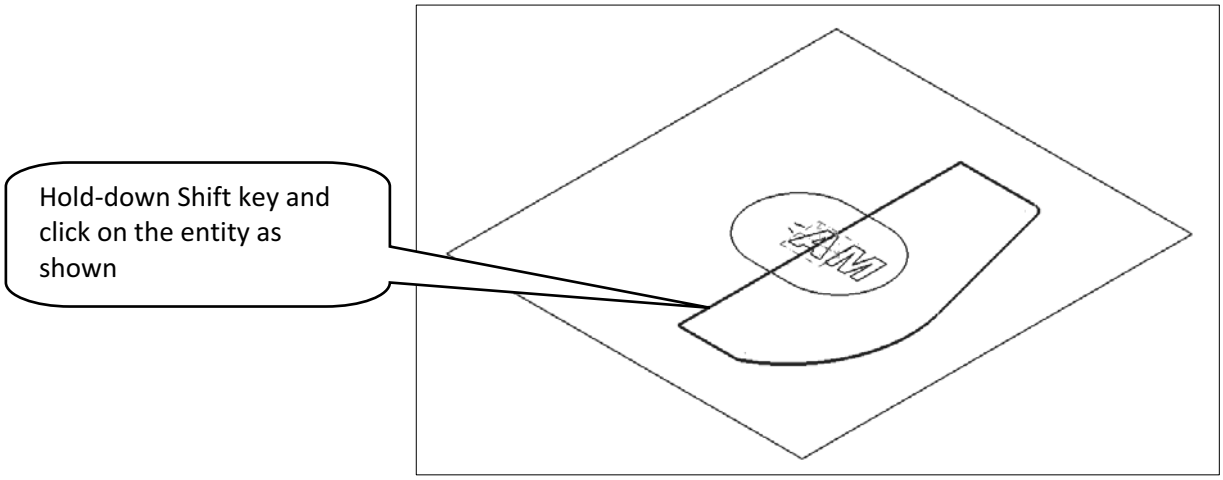
8.2 Mirror the geometry

- We are going to mirror the 2d profile used to create the revolved surfaces.

Sub Step Preview:



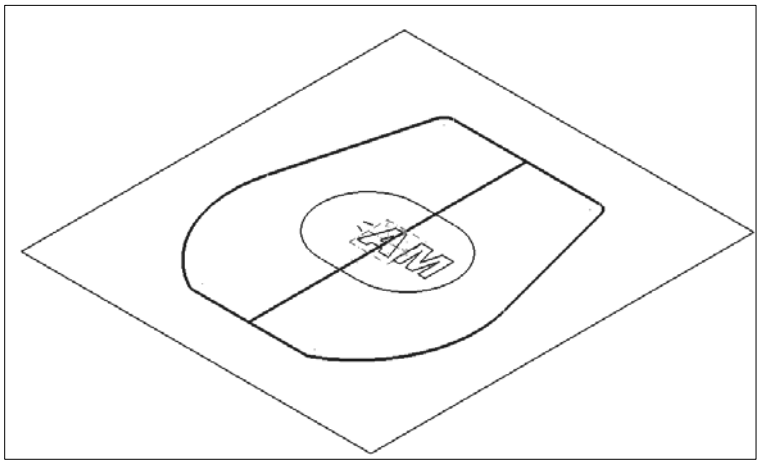
- Select the profile by holding down the **Shift** key and selecting one entity of the chain as shown.

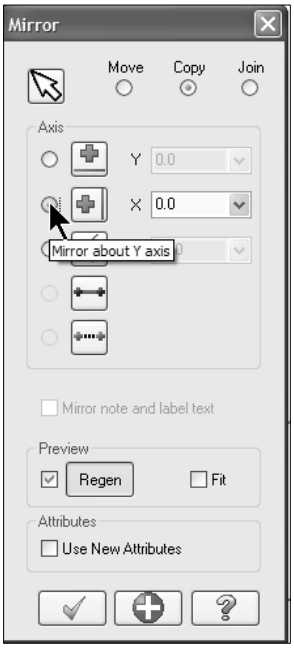


Xform

- **Mirror**
  - Enable **Mirror about Y axis** as shown.

The preview should look as shown:

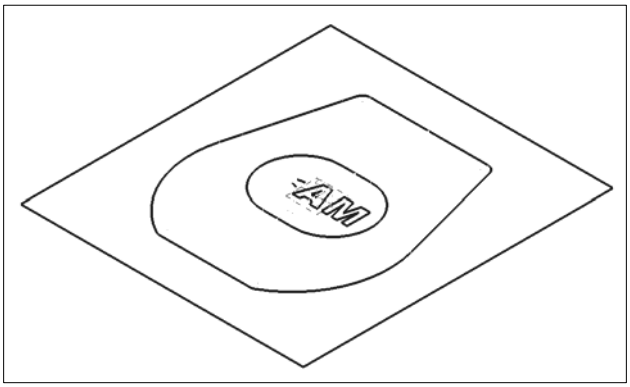




- Select the **OK** button to exit **Mirror**. 
- Select **Clear Color** icon. 

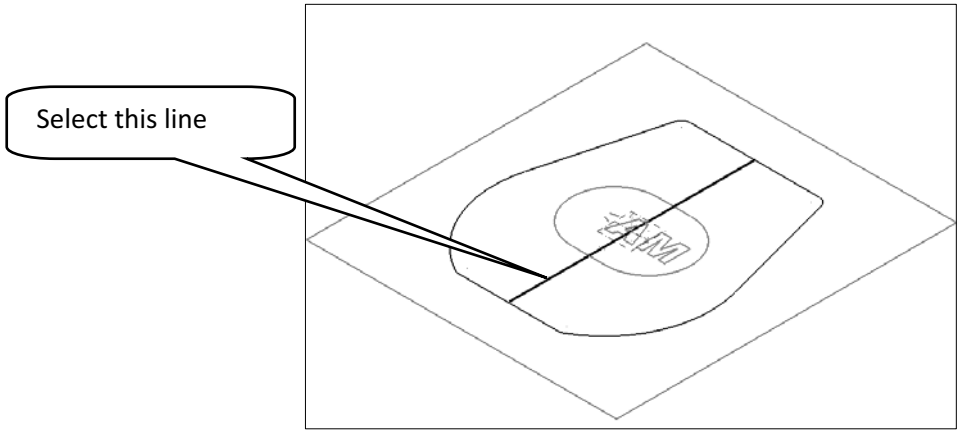
8.3 Delete the center lines.

Sub Step Preview:



- Note that because we selected the center line while selecting the chain to be mirrored, another center line was created on the top of the existing one.

- Select the line as shown:

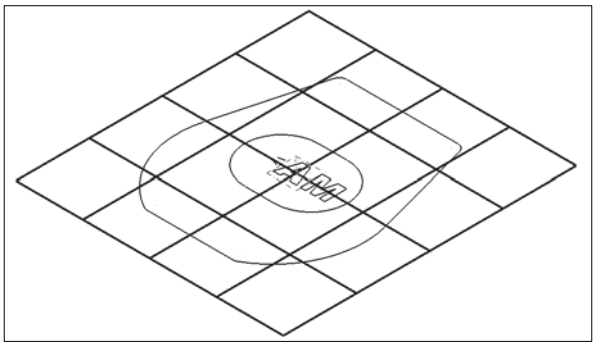


- Select **Delete entities** icon. 

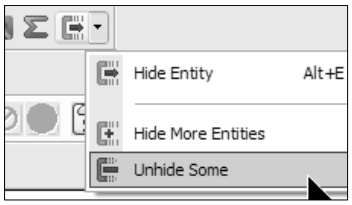
Repeate the step to delete the other line.

8.4 Unhide the flat surface

Sub Step Preview:

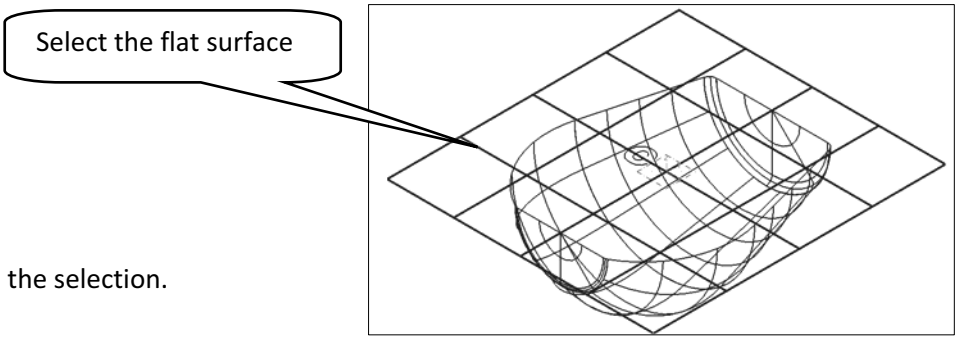


- Select **Unhide Entities** from the drop-down list.



  - Note that the rest of the entities appear on the screen.

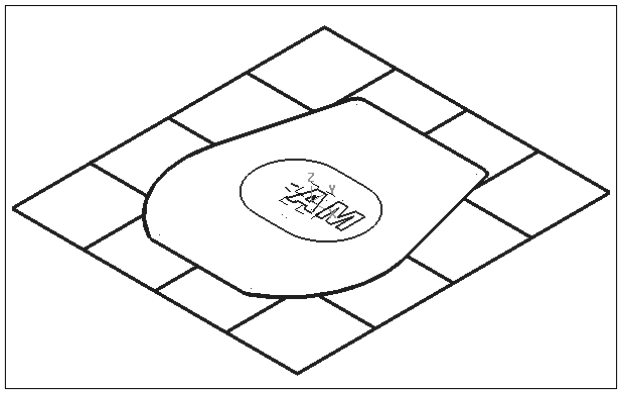
- [Select entities to keep on the screen]: Select the top flat surface.



- Press **Enter** to finish the selection.

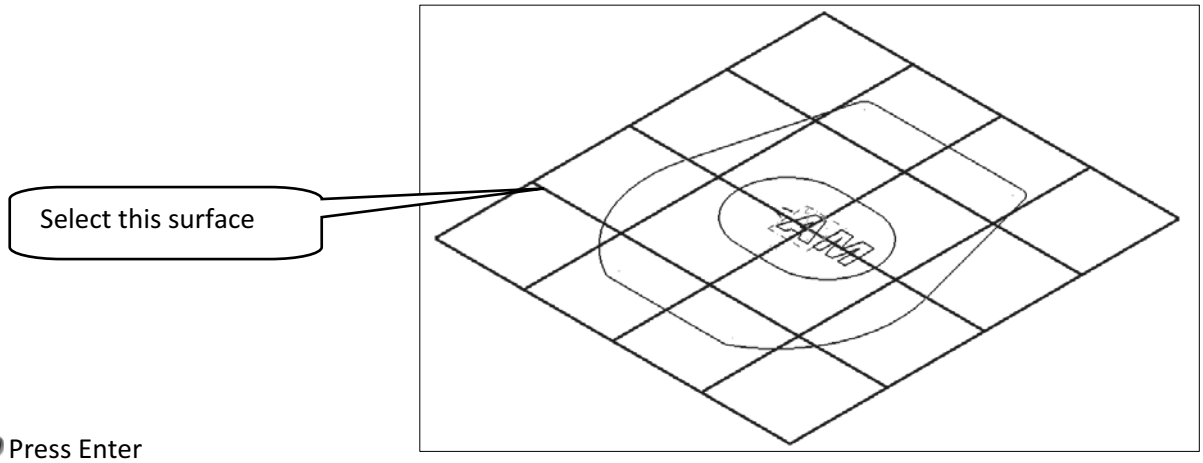
8.5 Trim the surface to a chain (closed)

Sub Step Preview:

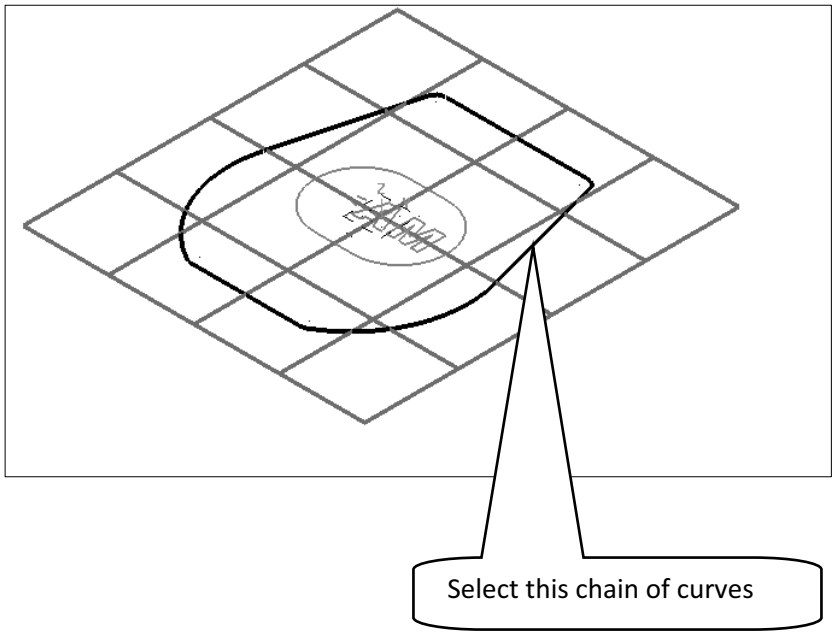


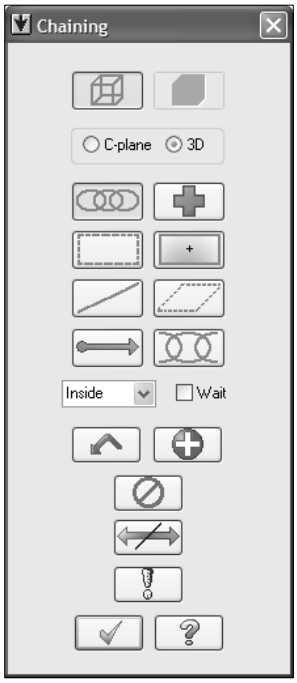
Create

- **Surface**
- **Trim**
- **To curve**
  - [Select surfaces and press Enter to continue]: Select the flat surface as shown.



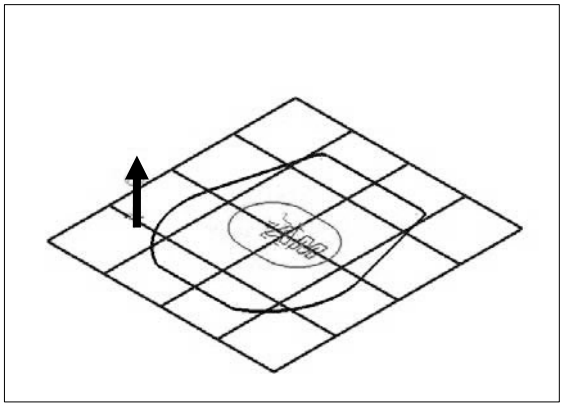
  - Press Enter
  - [Select Curves 1]: Select the chain of curves as shown below.

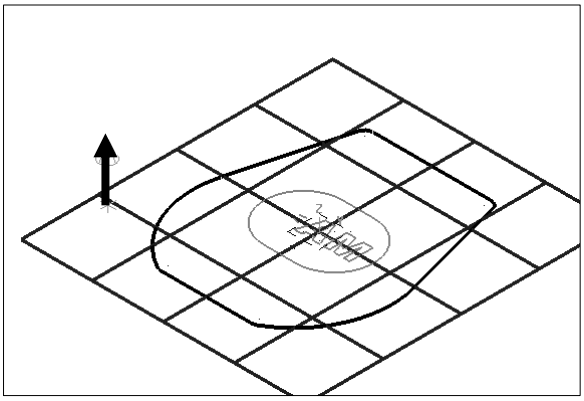




  - Select the **OK** button to exit **Chaining** dialog box. 

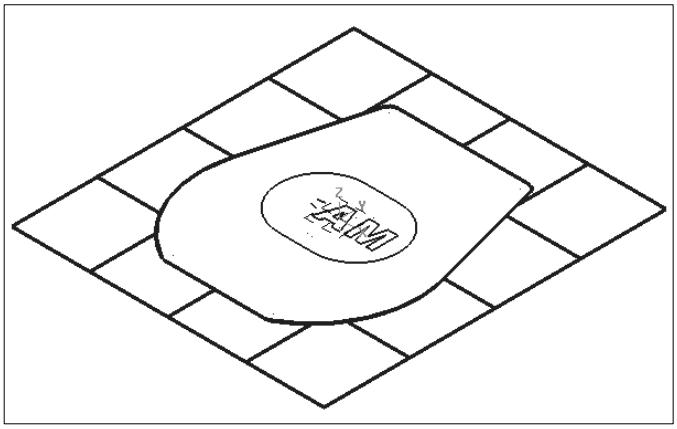
- [Indicate area to keep – select a surface to be trimmed]: Select two points on the surface outside of the chain as shown.



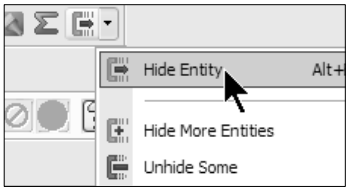


- Select the **OK** button 

The surface should look as shown:

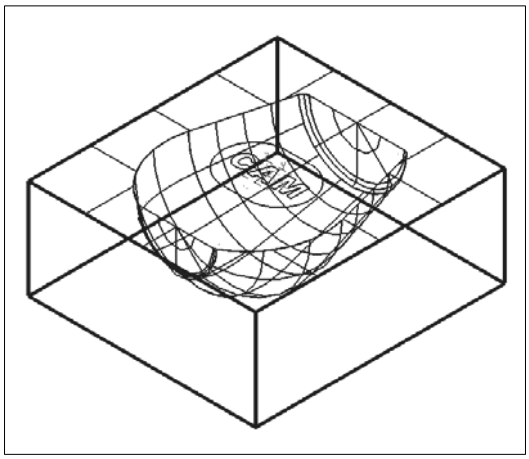


- Click on **Hide Entity** from the drop-down list to bring back the rest of the geometry.

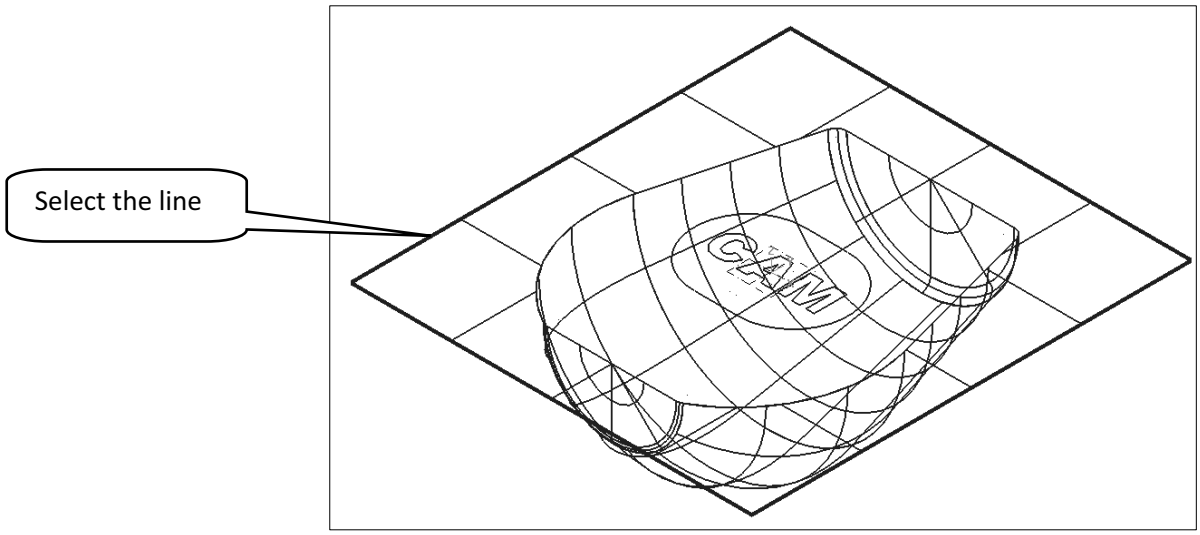


STEP 9: CREATE THE A BOX THAT REPRESENTS THE STOCK

Step Preview:



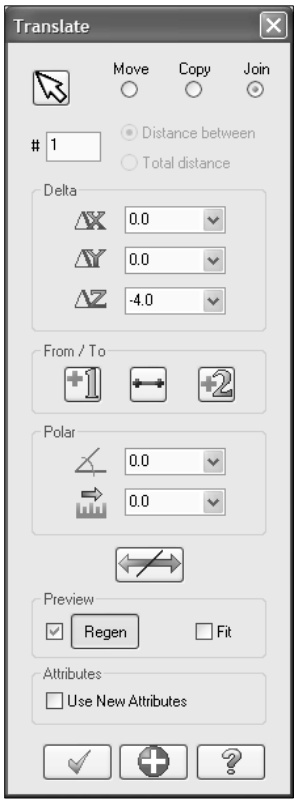
- Pre-select the chain by holding-down the Shift key and selecting one line as shown.



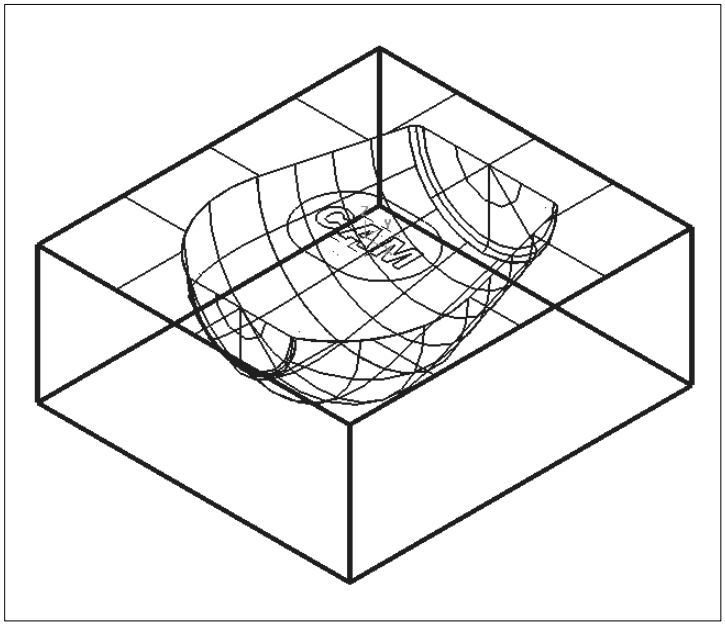
Xform

Translate

Change the parameters in the **Translate** dialog box as shown.



The geometry should look as shown:



STEP 10: SAVE THE FILE.

File

Save as

File name: "Your Name_6"

Select the **OK** button.

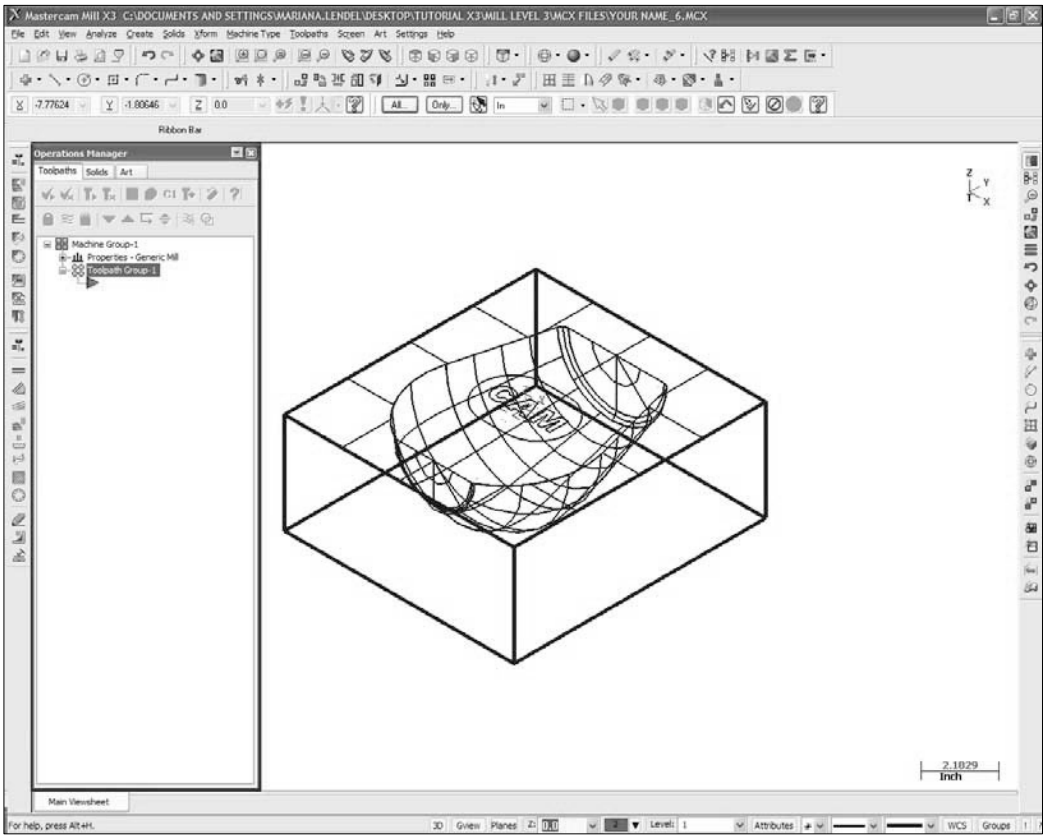
TOOLPATH CREATION
STEP 11: SET UP THE STOCK TO BE MACHINED.

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

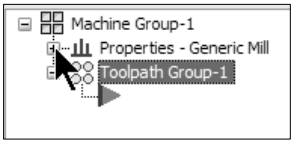
Make sure that no machine is already selected.

Machine Type

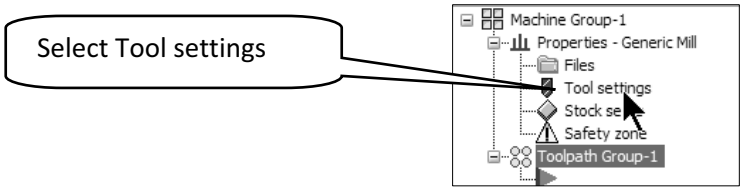
- **Mill**
  - Select **Default.**



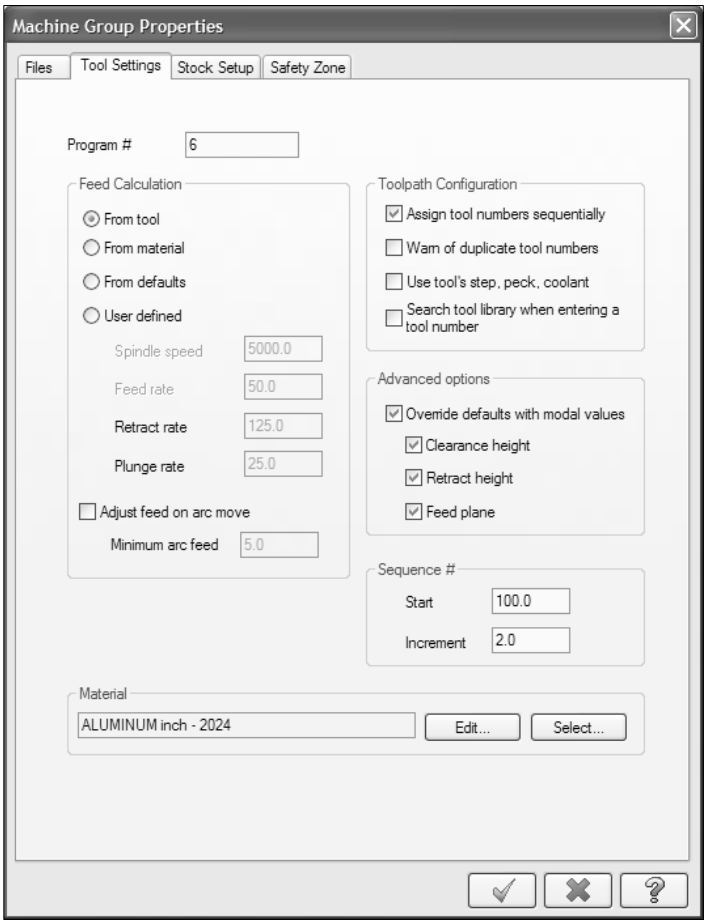
- Use the **Fit** icon to fit the drawing to the screen.
- Select the plus sign in front of **Properties** to expand the **Toolpaths Group Properties**.



- Select **Tool settings**.

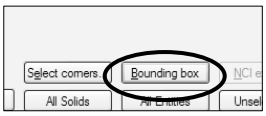


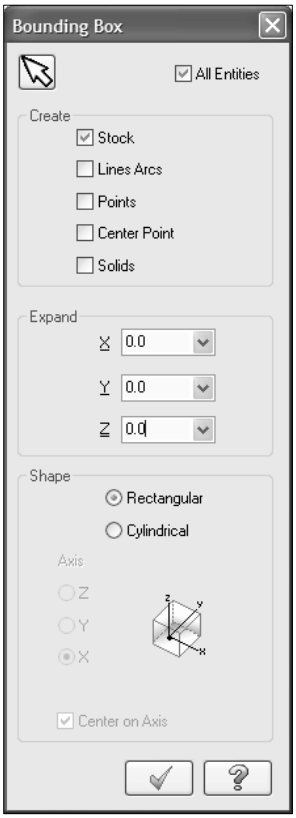
- Match the parameter with the ones in the screenshot below.



- Select **Stock Setup** tab.

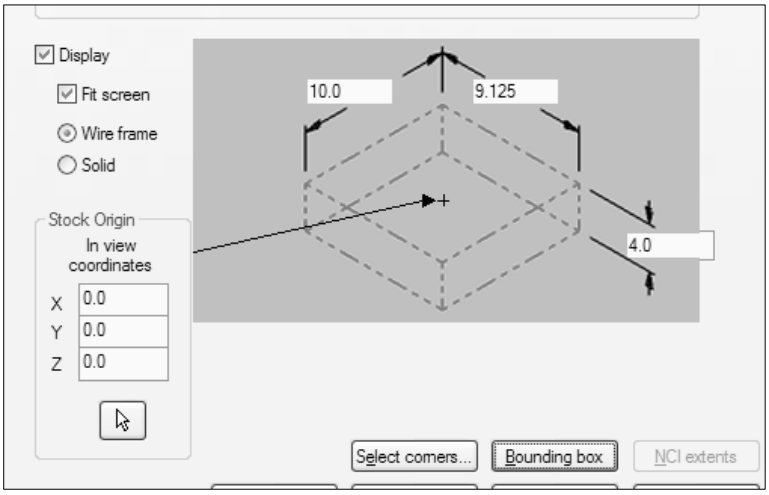
- Change the parameters to match the following screenshot.
- Select **Bounding box** button to automatically find the part extents.





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

- The stock values should look as shown:

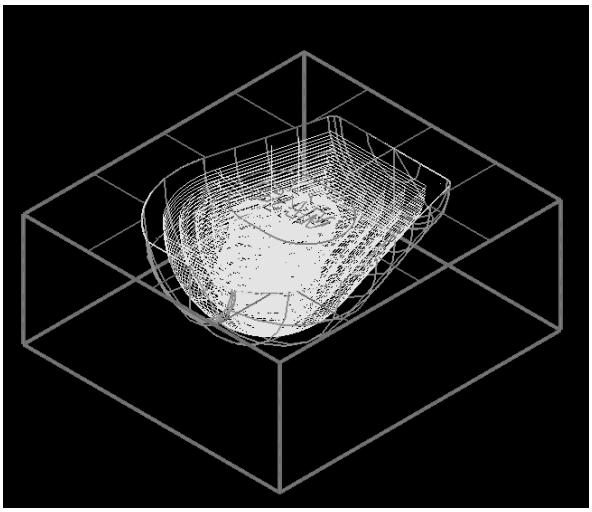


- Select the **OK** button to exit **Toolpath Group Properties.**

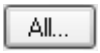
STEP 12: ROUGH OUT THE SURFACE USING SURFACE HIGH SPEED ROUGHING (AREA MILLING).

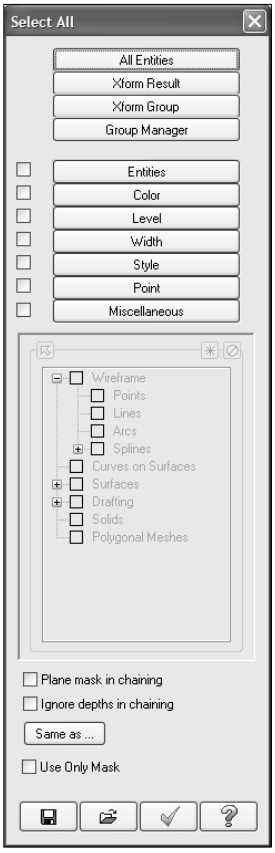
- **Area roughing toolpaths** are designed to rough out cavities, pockets, or other areas that can be most efficiently machined with an inside to outside toolpath. They are generated from a set of surface profiles that describe the shape of your surfaces at different Z heights, plus a set of offset profiles that rough out stock as the tool moves away from the center.

Toolpath Preview:

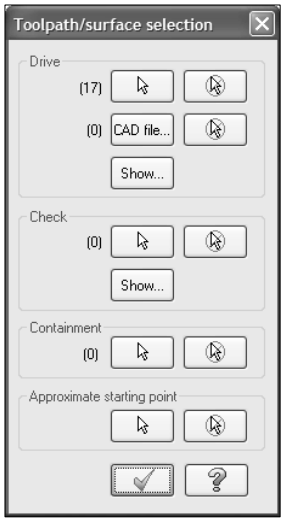


Toolpaths

- **Surface High Speed** 
  - Select the **OK** button to accept the NC name 
  - [Select Drive Surface]: Select the **All** button. 
  - Select the **OK** button to exit. 

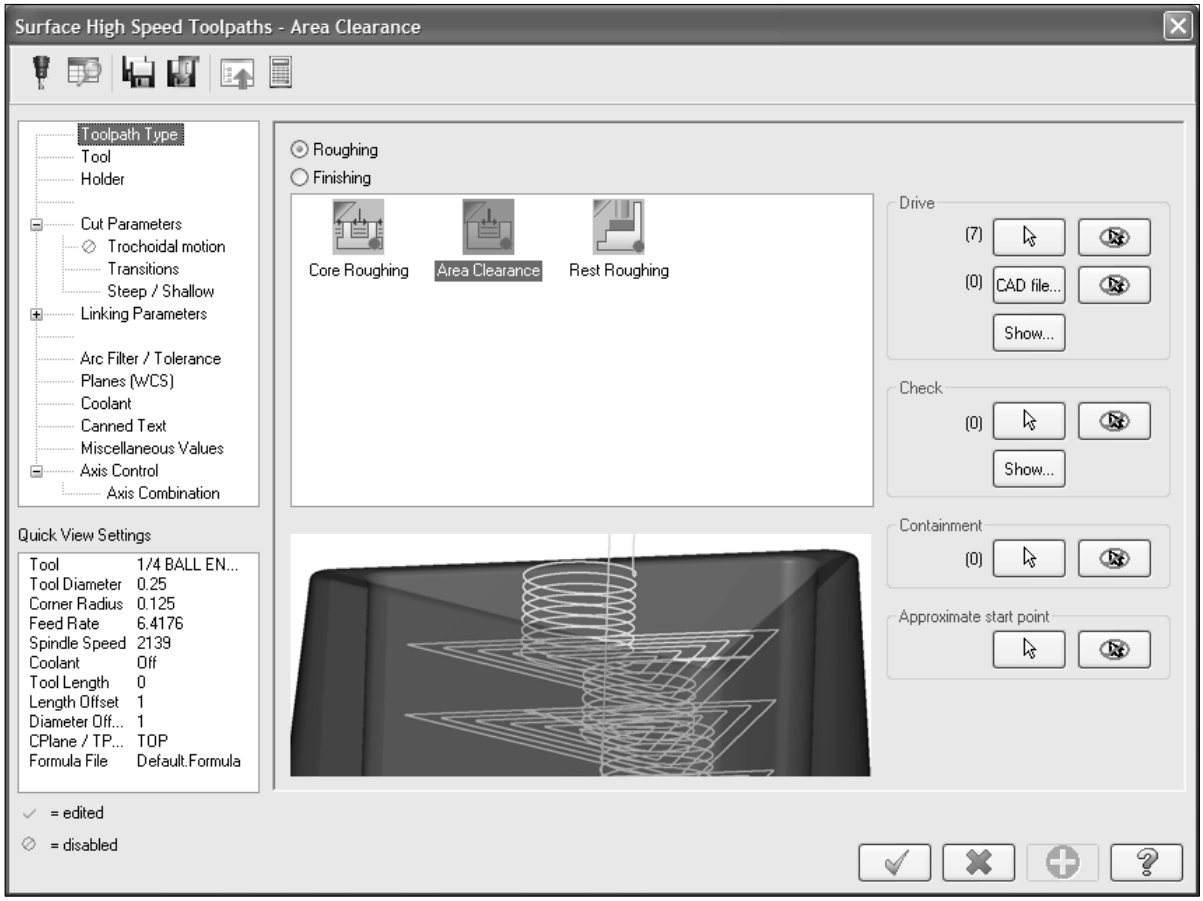


- Press **Enter** key.



- Select the **OK** button to exit **Toolpath/surface selection.**

- Select **Toolpath Type** and enable **Roughing**
- Select **Area Roughing**

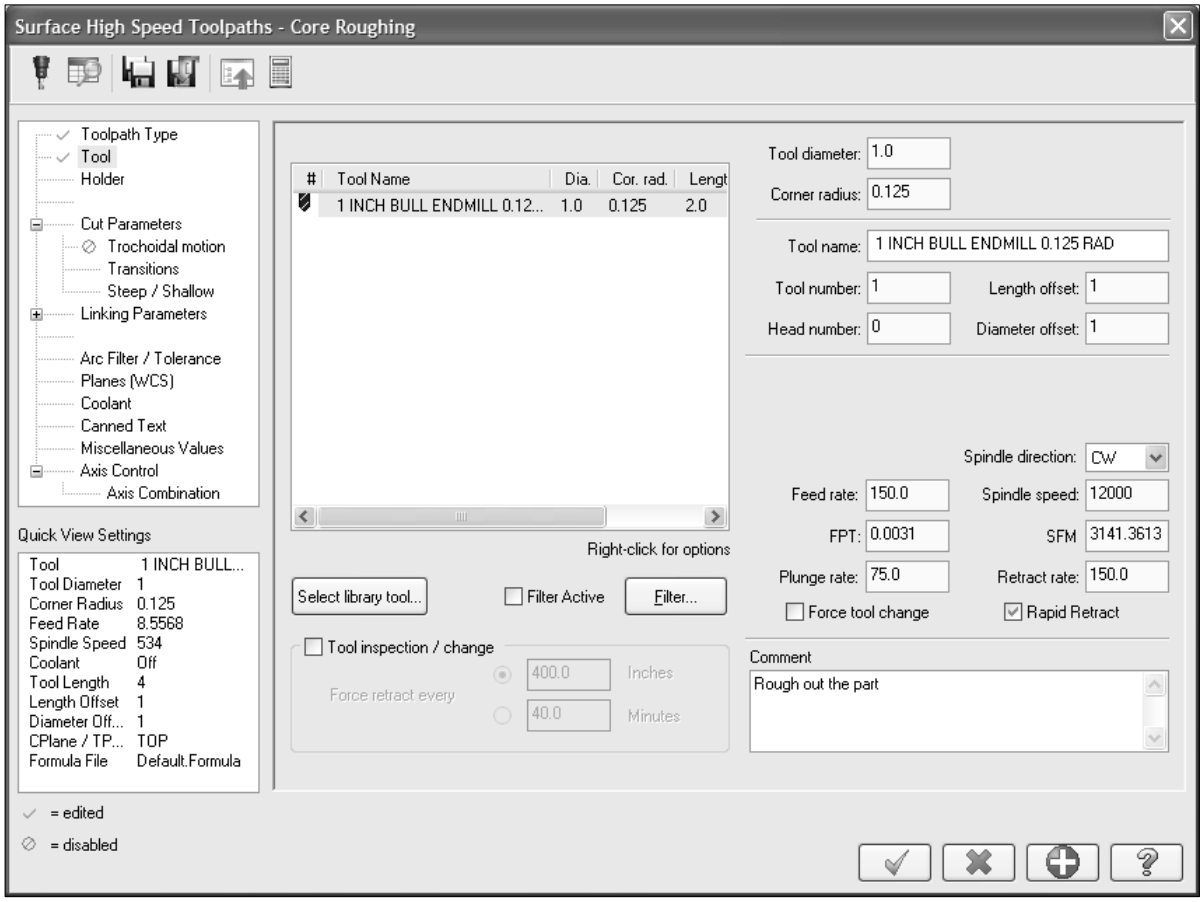


- Select **Tool**.

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

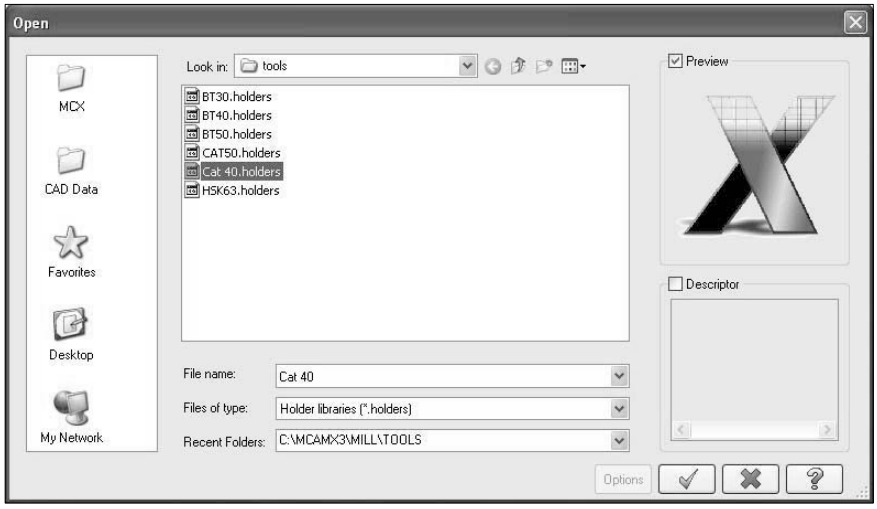
- Use the **Filter** button in to select the 1.0 " Bull Nose Endmill with corner radius 1/8 "

- Make the necessary changes in the **Toolpath parameters** to match the following screenshot.



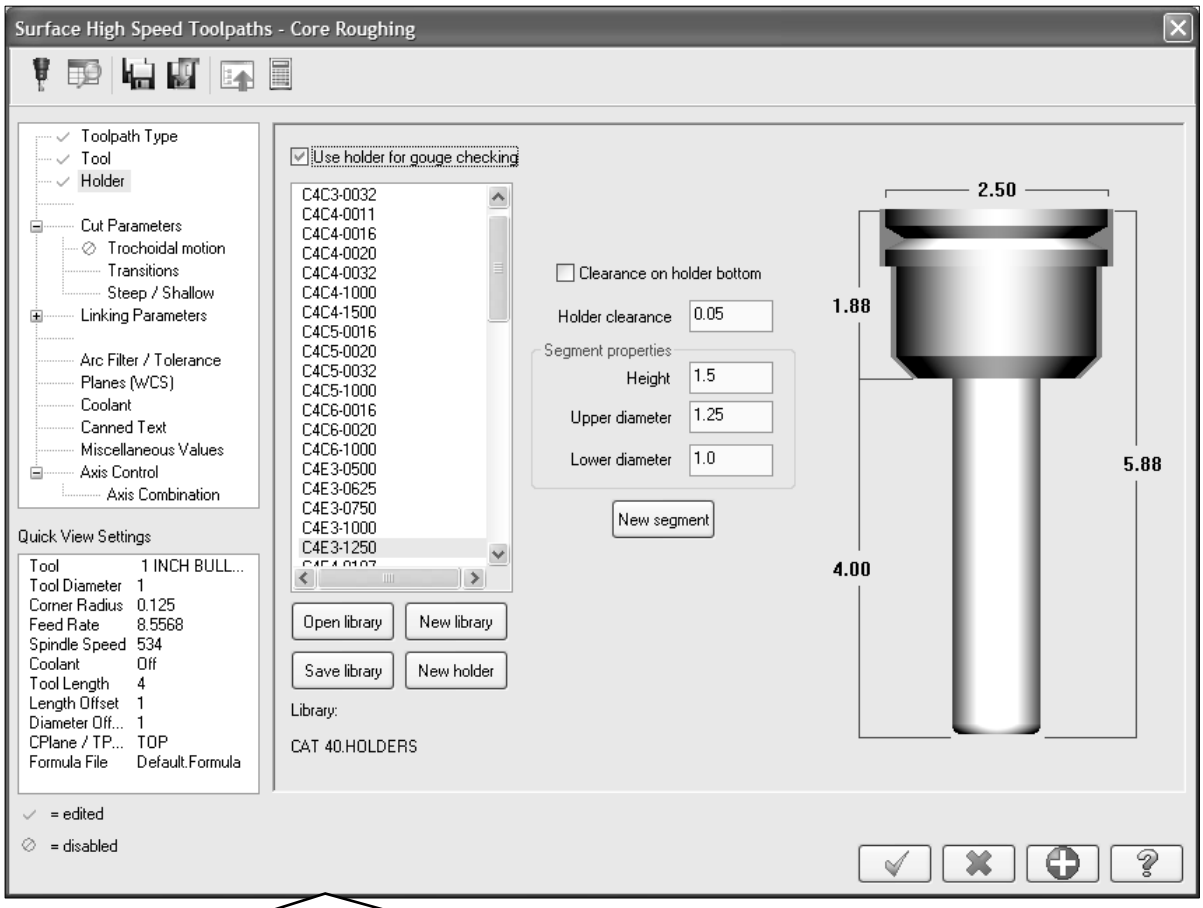
- Select the **Holder**.
- Select **Open library button**. 

- Select the **Cat 40.holders**



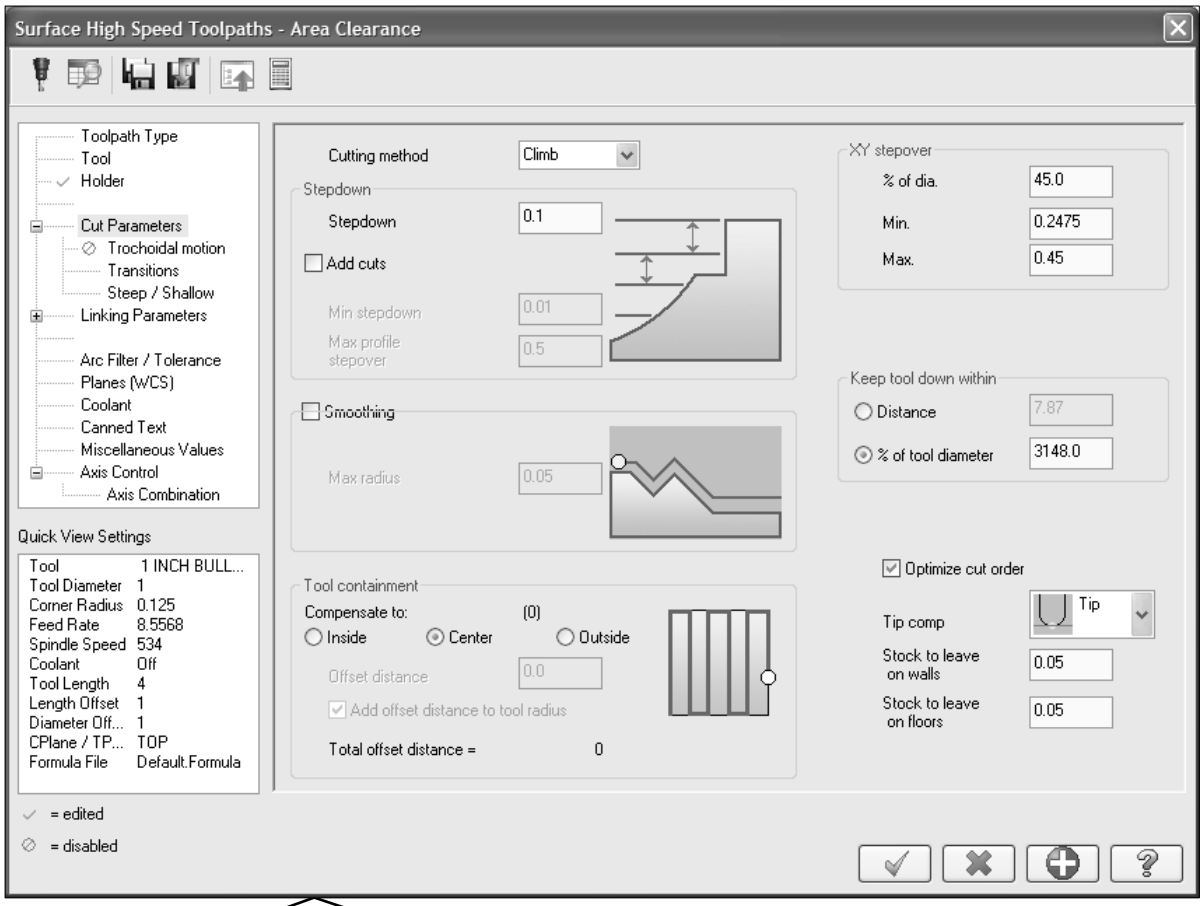
- Select the **OK** button 

Select the **C4E3-1250** holder



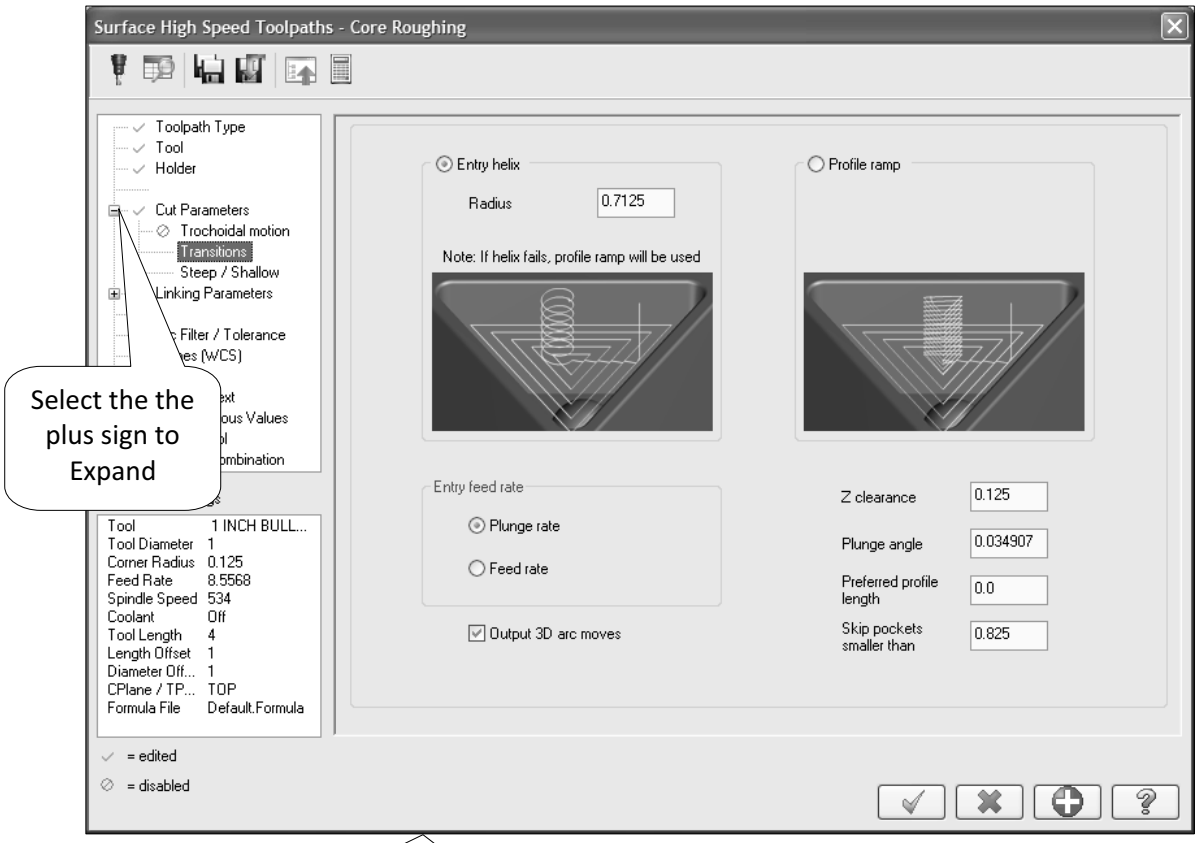
Holder page allows you to create a holder definition, load a holder from an existing library or edit the holder after it has been selected
Use holder for gouge checking when enabled,activates the gouge checking feature. Mastercam will check to make sure that the holder does not come into contact with any part geometry.
Holder clearance field establishes the minimum separation between the holder and your surface model. Set the clearance bigger than the stock to leave on the walls.

Select the **Cut Parameters** page and make the changes to match the following screenshot.



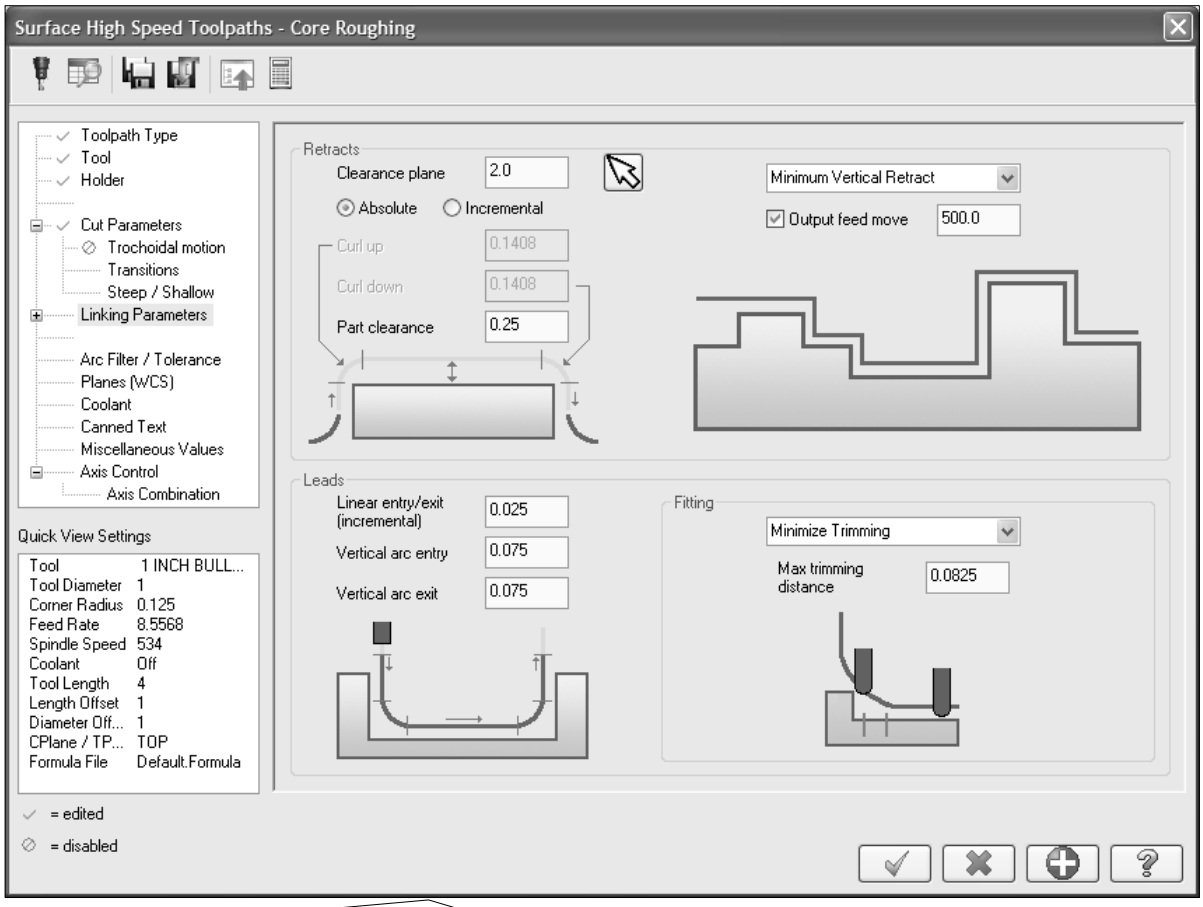
Step-down options allow you to configure how Mastercam spaces the cuts in Z.
Step-down value sets a constant Z spacing between cutting passes.
Add cuts feature allows you to insert additional cutting passes in areas of your part where the profile is close to flat. Mastercam will add new cuts to maintain the maximum profile stepover, while spacing them each by at least as much as the minimum step-down.
XY stepover settings allow you to configure the spacing between the passes at the same Z. Mastercam will use the largest value possible (up to the maximum XY stepover) that does not leave unwanted material between the passes. However, it will not separate the passes by less than the minimum stepover.
Stock to leave on your drive surfaces lets you enter separate values for the wall and floor surfaces. Note that the stock to leave on walls must be greater than or equal to the stock left on the floor. For surfaces that are not exactly horizontal or vertical, Mastercam will interpolate between the wall and floor values.
Keep tool down prevents the tool from retracting if the distance between the end of one pass and the start of the next pass is less than the specified settings.

Expand **Cut Parameters** if needed, and select **Transitions** to set the **Entry helix**.



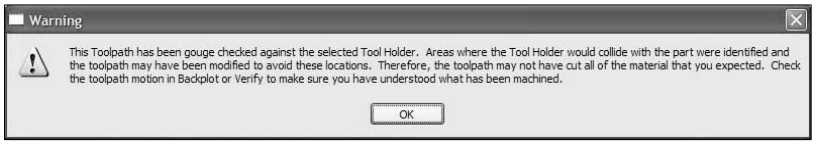
Transitions allow you to configure the entry move that the tool will make as it transitions to new Z levels. You can choose to create either a ramp entry, or helical entry move. If the profile is too small to create a helix of this size, Mastercam will create a ramp move instead.

Select the **Linking Parameters** page and change the parameters to match the following screenshot.

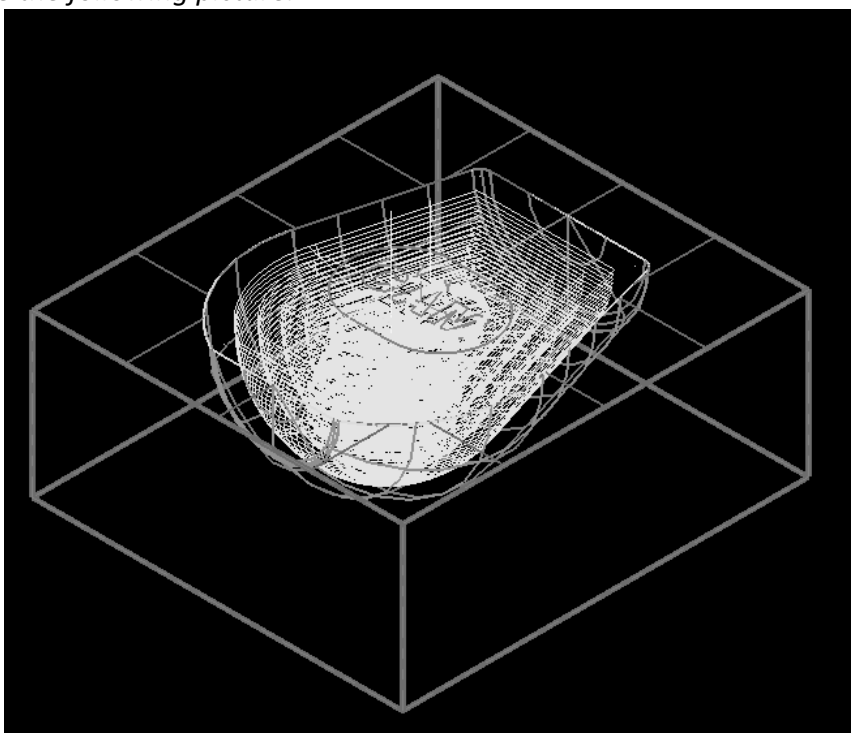


Linking options allow you to configure how Mastercam links air moves when the tool is not in contact with the part
Minimum vertical retract is a vertical retract and constant-Z move at the Part clearance height.
Leads fields set the tool moves onto and off of the part at the start and end of each cutting pass. These moves are applied to each pass no matter which cutting pass is selected.
Fitting settings allow you to choose how the entry and exit arcs fit to the ends of the cutting passes.
Minimize trimming sets the path to retract to be as close to the surface as possible, maintaining a minimum distance from the surface to fit the arc.
Max trimming distance parameter limits the amount of trimming applied to non-horizontal passes.

Select the **OK** button to exit parameter pages. 
Select the **OK** button to continue.

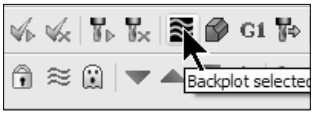


The toolpath should appear like the following picture.

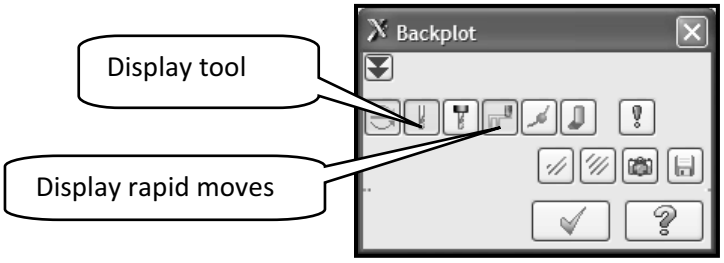


STEP 13: BACKPLOT THE TOOLPATH

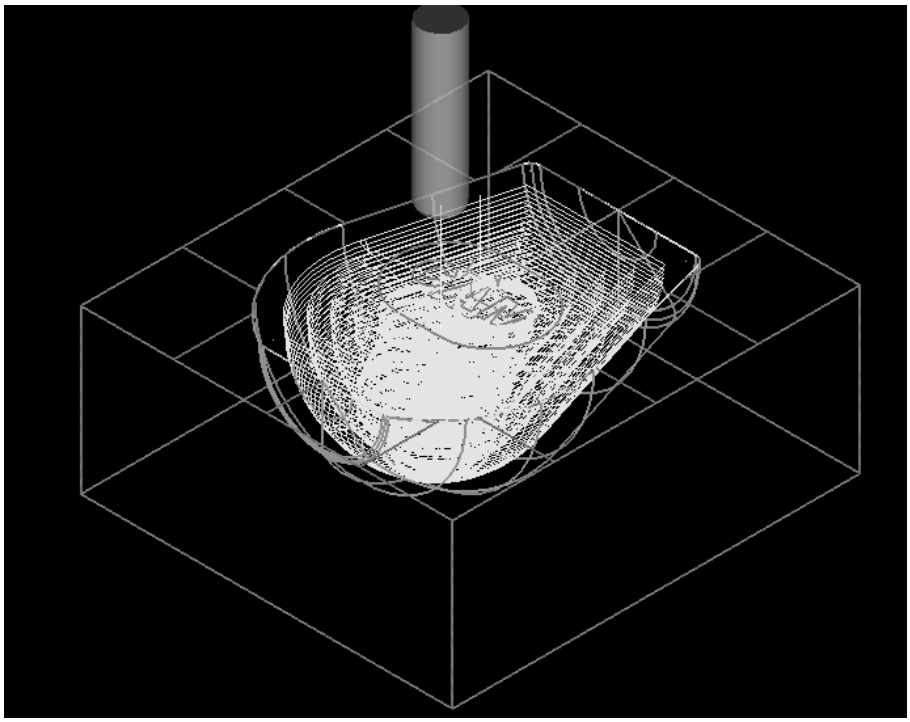
- Select the **Backplot** selected operations button.
- Make sure that you have the following buttons turned on (they will appear pushed down).



- **Display tool**
- **Display rapid moves**



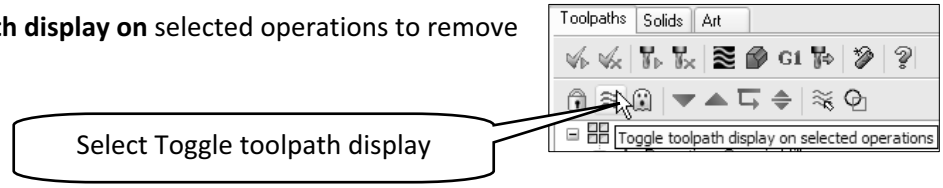
- Select the **Play** button.



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

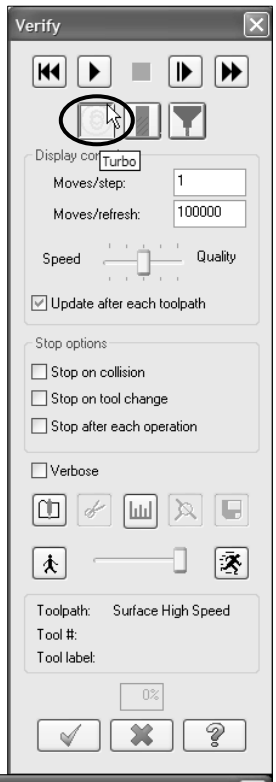
STEP 14: VERIFY THE ROUGH TOOLPATH AND SAVE IT AS A STL FILE TO BE USED AS AN INTERMEDIATE STOCK.

- Select **Toggle toolpath display on** selected operations to remove the toolpath display.

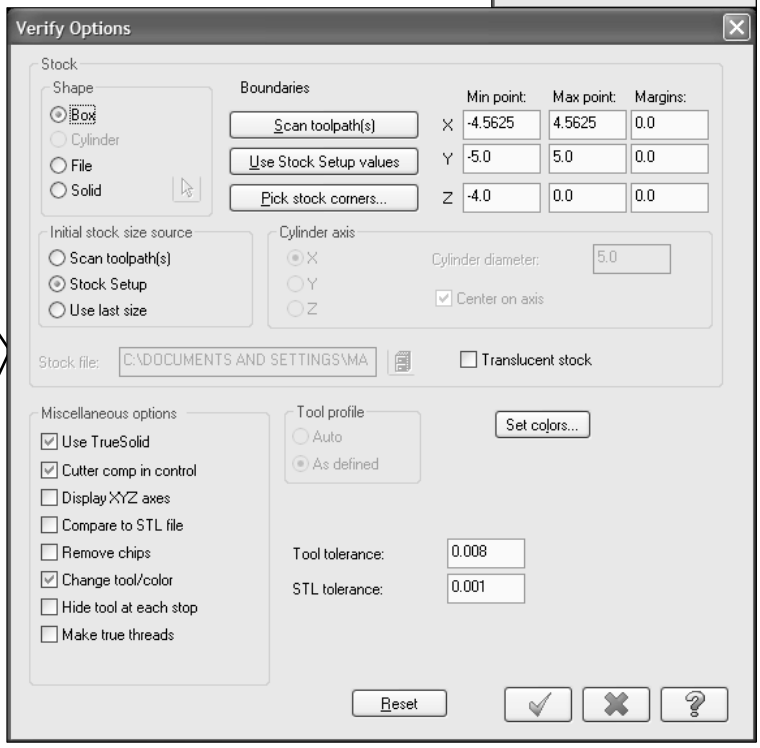


- Select the **Verify selected operations** button.

- Enable **Turbo** mode.



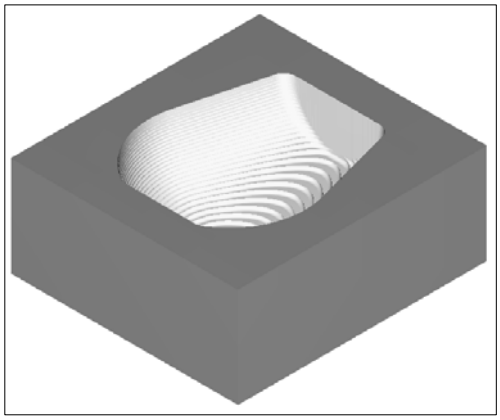
- Select the **Configure** button. 



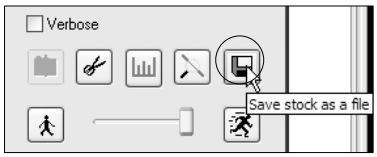
Initial stock size source should be set to **Job Setup** to use the stock information from Stock Setup.
Use True Solid allows you, after verifying the part, to rotate and magnify it to more closely check features, surface finish, or scallops.
Change tool/color to change the color of the cut stock to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

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

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



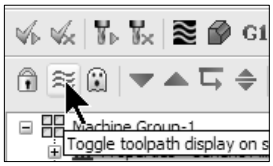
The part should appear as shown to the right.



- Select the **Save stock as a file** button.
- Save the **STL file** with the same name at a known location.
- Select the **OK** button.

  - To speed up the Verification, we will use this stl file when we verify the rest of the operation.

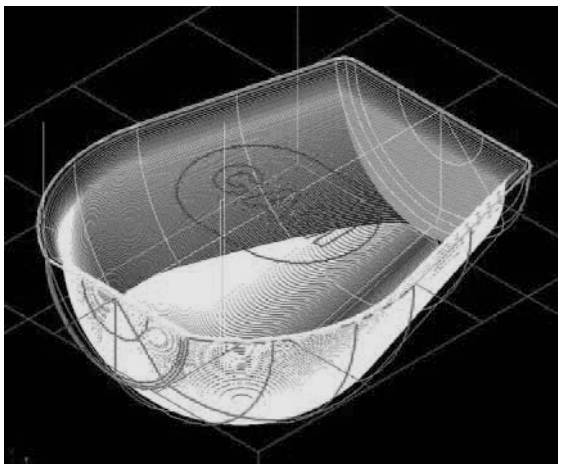
- Select the **OK** button to exit **Verify**.
- Select **Toggle toolpath display on selected operations** icon to hide the toolpath from the screen.

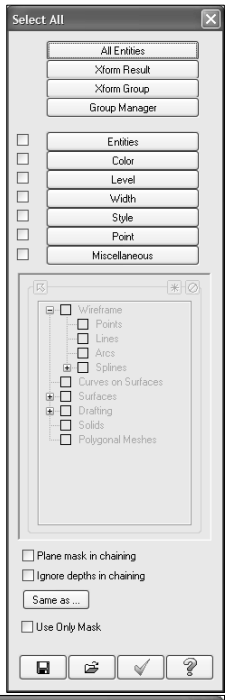


STEP 15: FINISH THE SURFACES USING SURFACE FINISH CONTOUR.

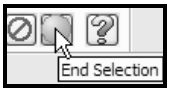
Contour Rough and Finish Toolpaths perform multiple cuts at constant Z levels. Both toolpaths are recommended for parts with steep walls. You should avoid using this toolpath for parts with flat surfaces. To machine flat areas use shallow or parallel surface toolpaths.

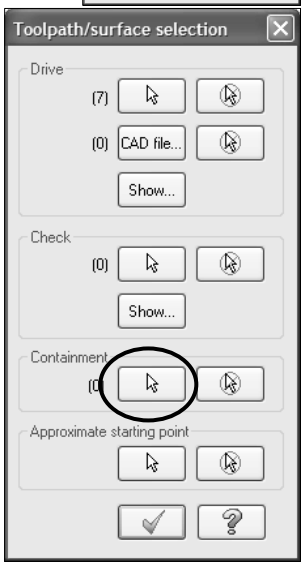
Toolpath Preview:



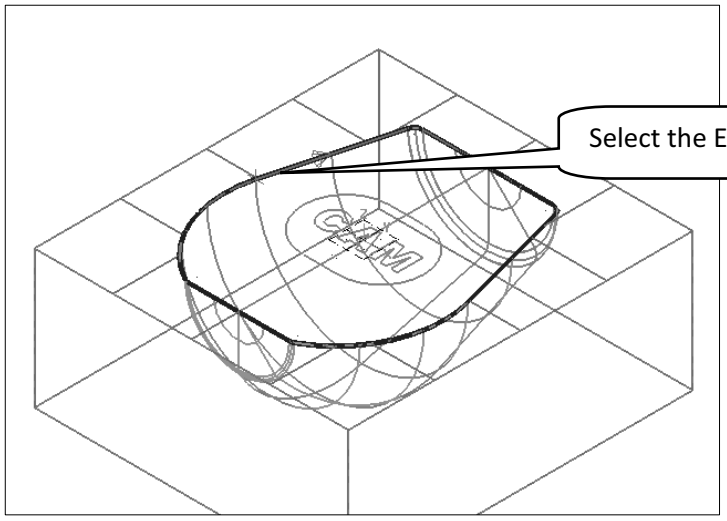


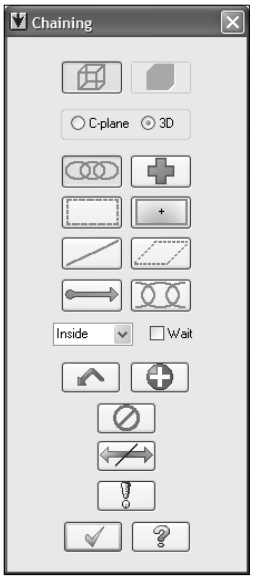
Toolpaths

- **Surface Finish**
- **Contour**
  - [Select Drive Surface]: Select the **All** button. 
  - Select the **OK** button to exit. 
  - Select the **End Selection** button. 
  - Select the **Containment** selection button.

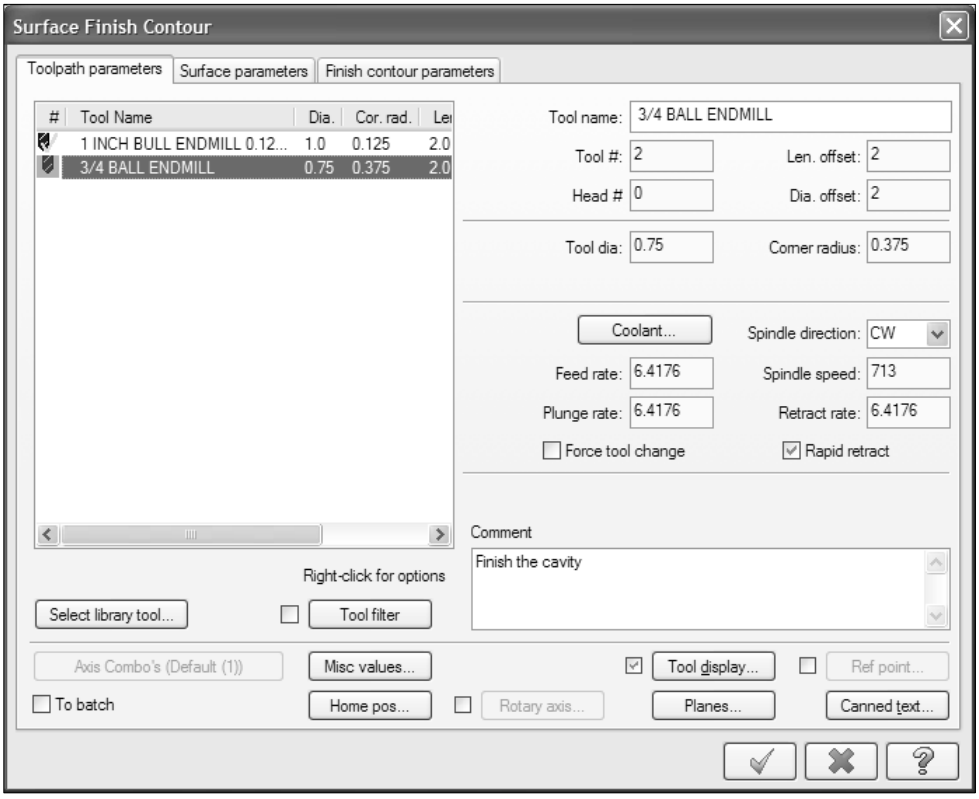


- [Chain 2D tool containment boundary #1]: Select Entity A.

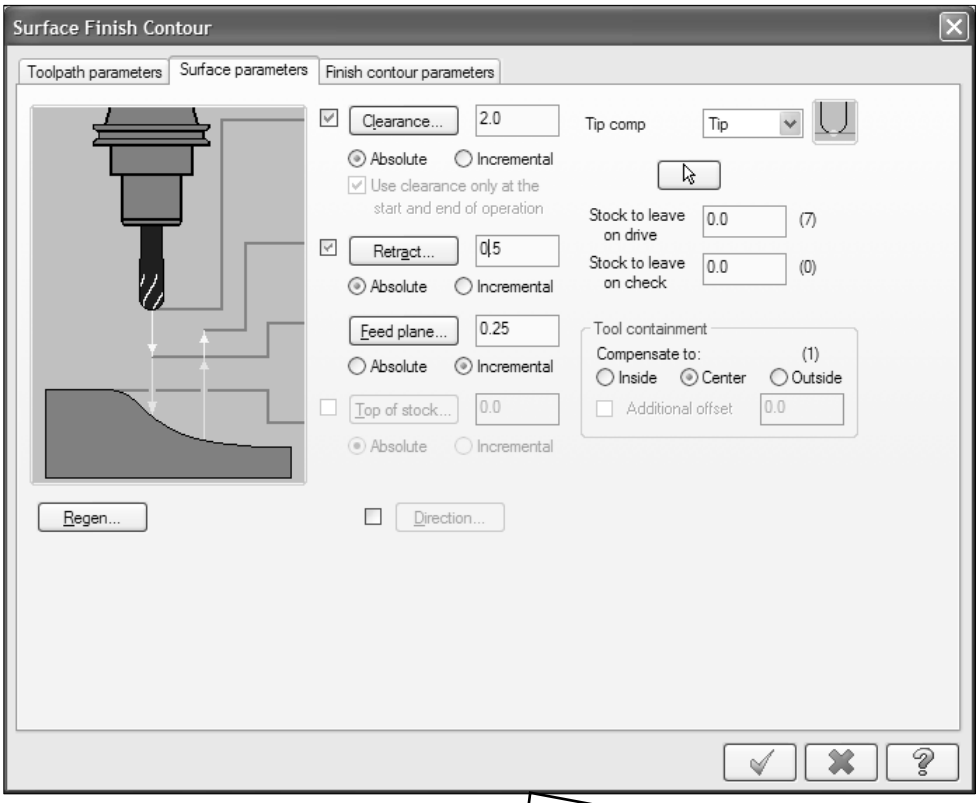




- Select the **OK** button twice to exit the **Toolpath/surface selection** dialog box. 
- Click on the **Select library tool** button and use **Filter** to select ¾ Ball Endmill.
- Make any necessary changes as shown in the following screenshots.

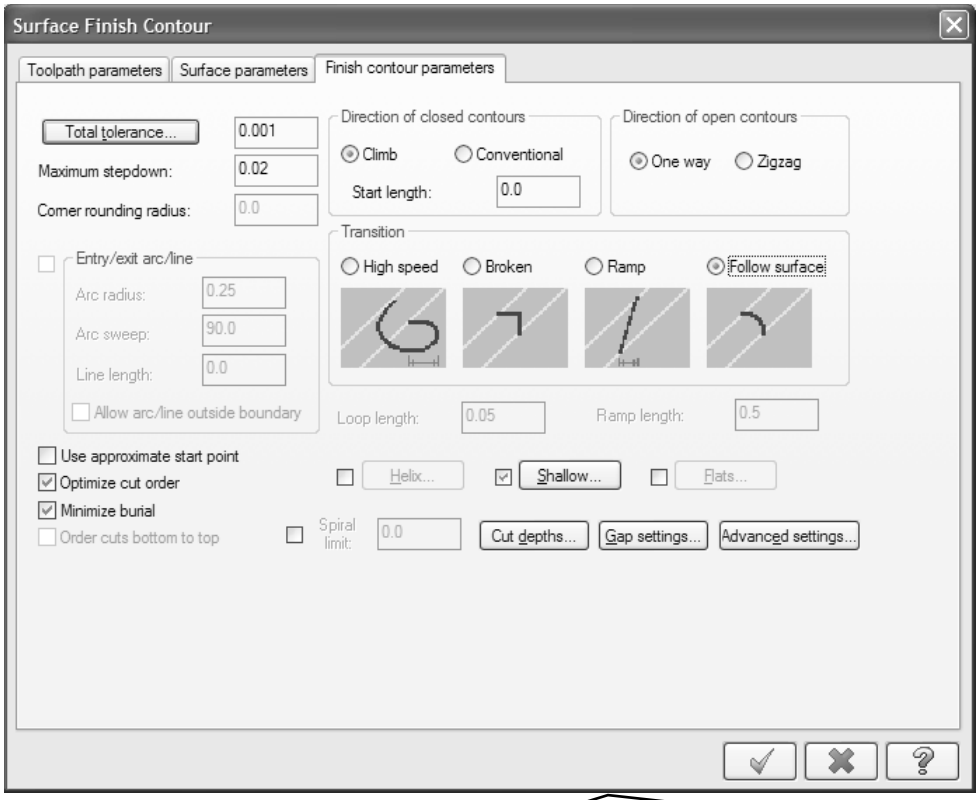


Selec the **Surface parameters** and change the parameters as shown.



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 tool pass.
Feed plane value sets the height the tool rapids to before changing to the plunge rate.
Stock to leave (on **Drive surface**) sets the amount to leave for the finish operation as a constant value all the way around the drive surfaces.

Select the **Finish contour parameters** and change the parameters as shown.

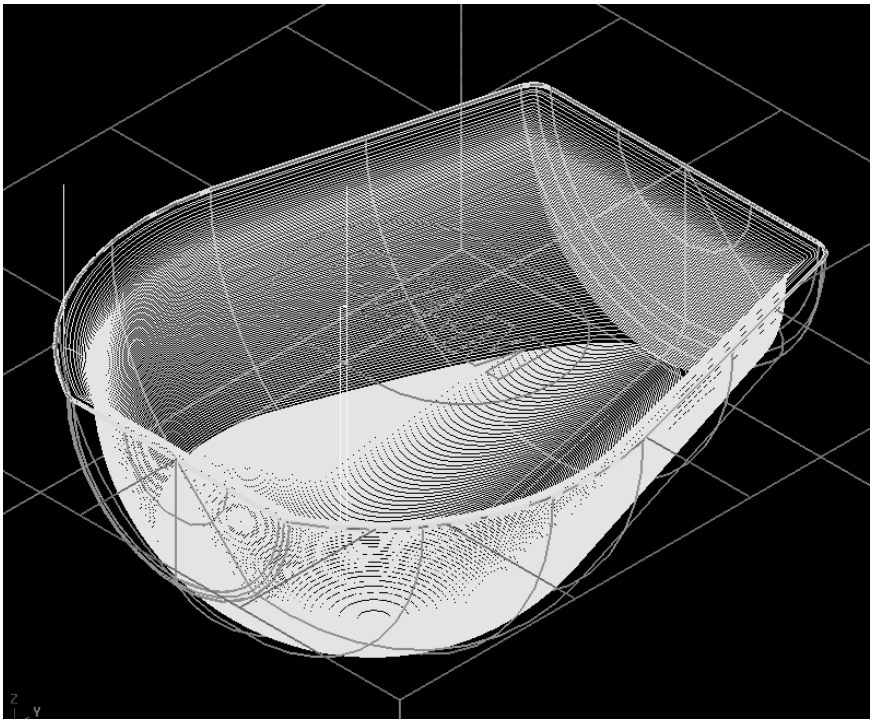


Optimize cut order cleans one area completely before moving to another area.
One way allows the tool to move around the shape in the same direction.
Minimize tool burial generates additional retract and plunge moves when the tool could be engaging material on both sides, as when machining between two bosses.
Shallow removes tool moves or adds tool moves in the shallow areas.

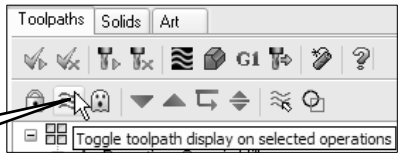
Select **Shallow** button and change the parameters as shown.

Add cuts to shallow area allows the system to generate additional cuts between adjacent cuts when the adjacent cuts are further apart than the limiting stepover value.
Limiting angle sets the angle that defines the shallow areas on the part. Mastercam adds or removes cuts in an area that ranges between 0 to 45 degrees.
Limiting stepover is used as the minimum stepover when adding cuts to the shallow areas and as maximum stepover when removing cuts from shallow areas.

- Select the **OK** button to exit **Shallow** dialog box.
- Select the **OK** button to exit **Finish contour parameters**.



- Select **Toggle toolpath display on** selected operations to remove the toolpath display.

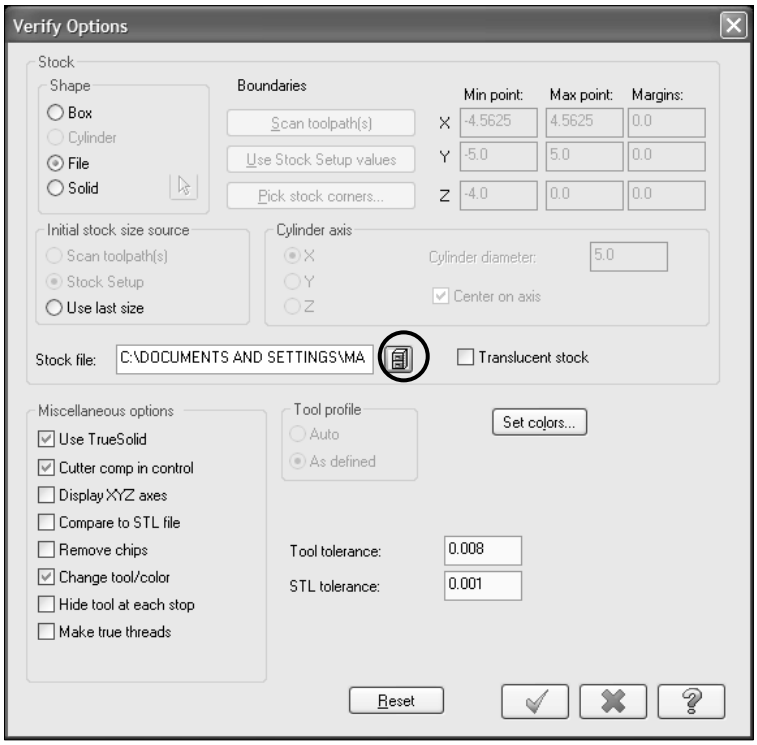


STEP 16: VERIFY THE FINISH TOOLPATH

- Select only the **Surface Finish Contour** operation.
- Select the **Verify selected operations** button.

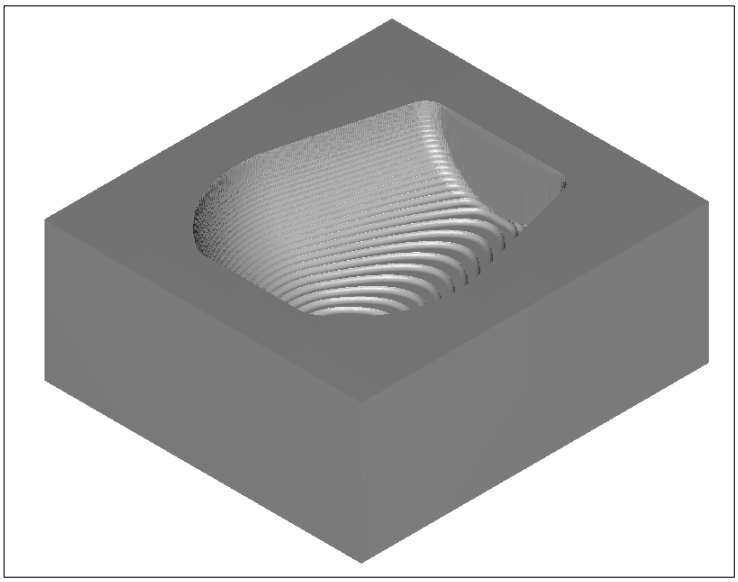
- Enable **Turbo** mode.

- Select the **Configure** button. 
- Enable **File** in the **Stock Shape**, and select the **Stock file** browse button.



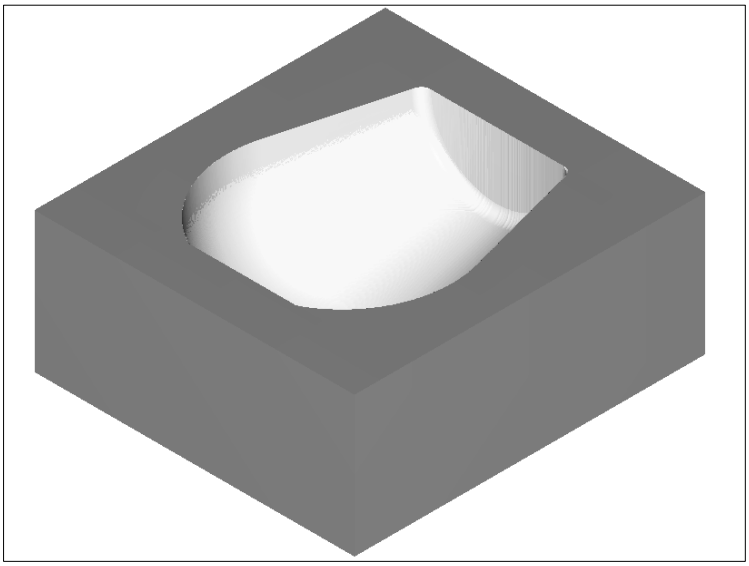
- Find the Stl file that you saved in a previous step (Your Name_6.STL).
- Select the **OK** button twice to exit **Verify Options**.

The stock should look as shown:



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

The part should appear as shown to the right:



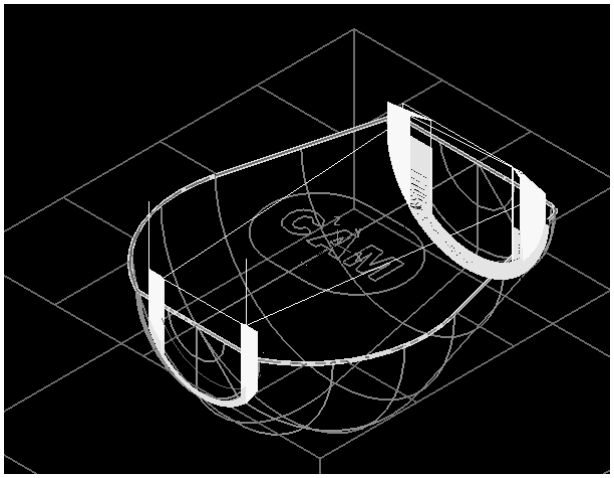
- Select the **OK** button to exit **Verify**.

STEP 17: FINISH THE FILLET SURFACES USING SURFACE FINISH LEFTOVER TOOLPATH

- Note that because the fillet surface radius is 0.25 " the fillet surfaces were not cleaned.

- **Surface finish leftover toolpath** removes remaining stock that Mastercam calculates based on the dimensions of a tool used in a previous operation. Finish leftover uses a smaller tool than the roughing tool. Mastercam looks at the part, calculates where the roughing tool could not fit, and creates tool motion to remove stock from these areas.

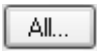
Toolpath Preview:

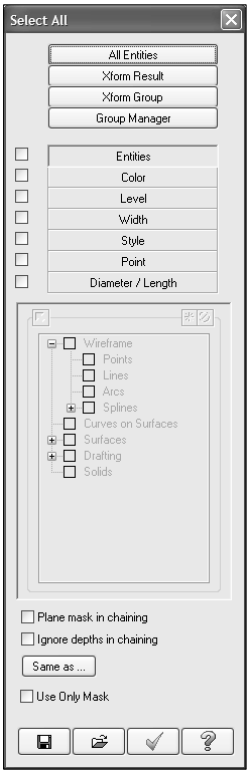


Toolpaths

Surface Finish

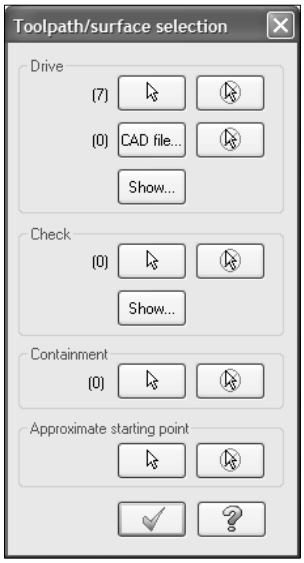
Leftover 

[Select Drive Surfaces]: Select the **All** button. 

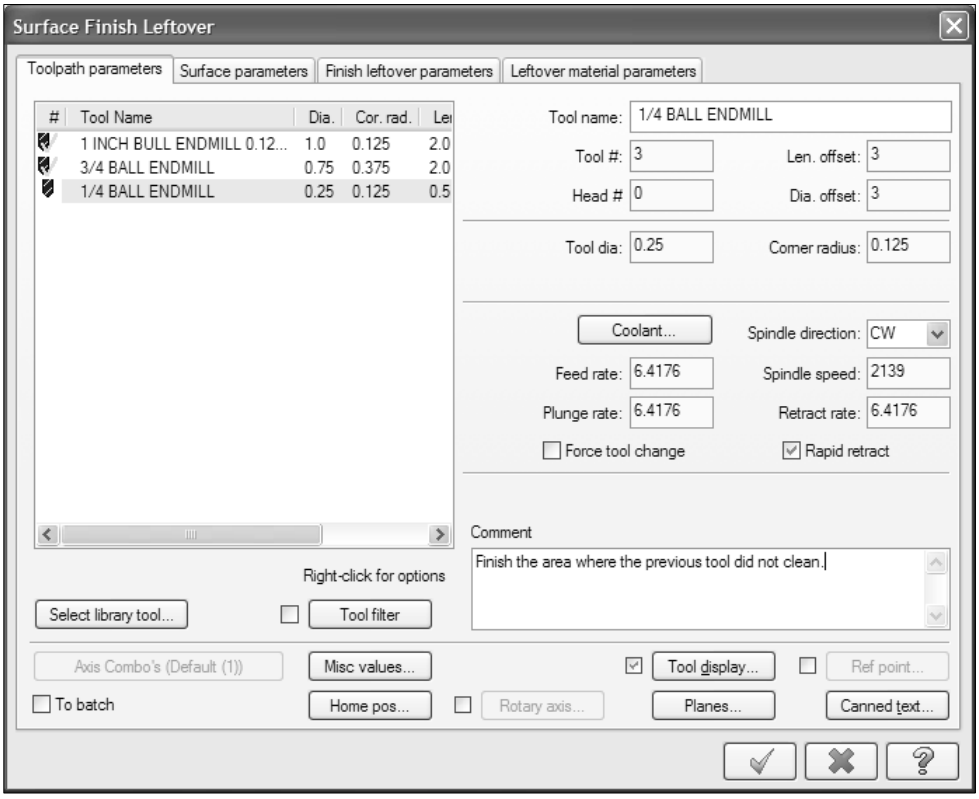


Select the **OK** button to exit. 

Select the **End Selection** button. 

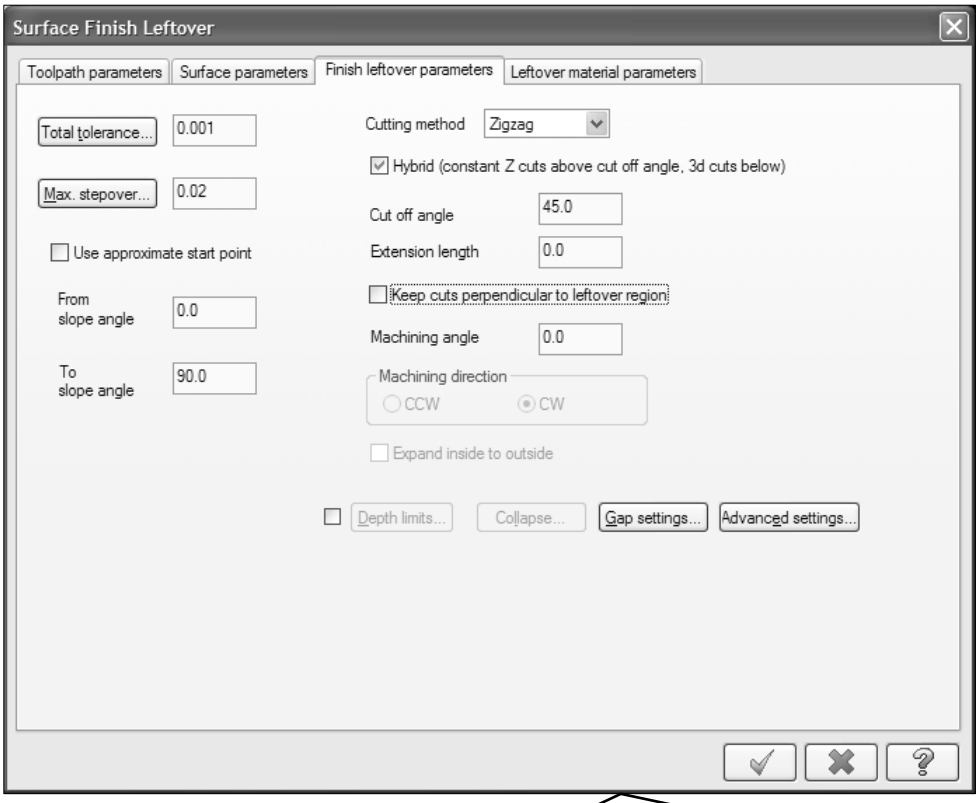


- Select the **OK** button to exit **Toolpath/surface selection**.
- Click on **Select library tool** in the **Toolpaths parameters** dialog box.
- Use the **Filter** to select the ¼ " Flat Endmill.
- Make the necessary changes in the **Toolpath parameters** to match the following screenshot.



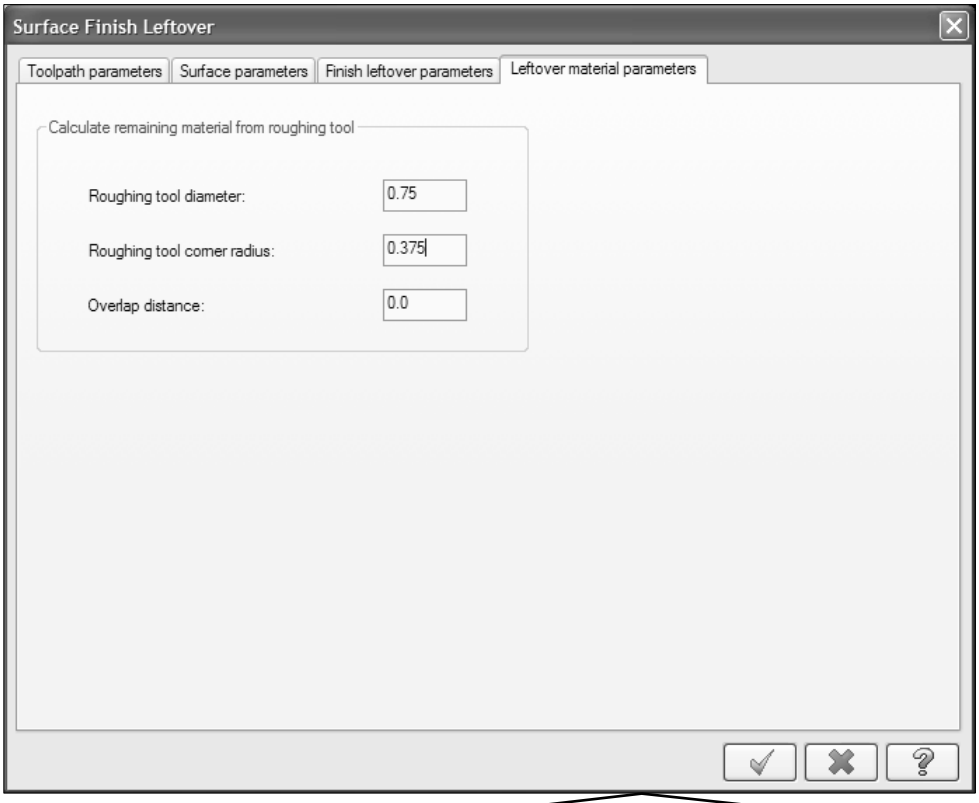
- Note that because we enabled the Advanced options in the Tool Settings parameters (Properties); the Clearance, Retract and Feed plane, will be the same as in the previous operation.

Select **Finish leftover parameters** tab and change the parameters as shown.



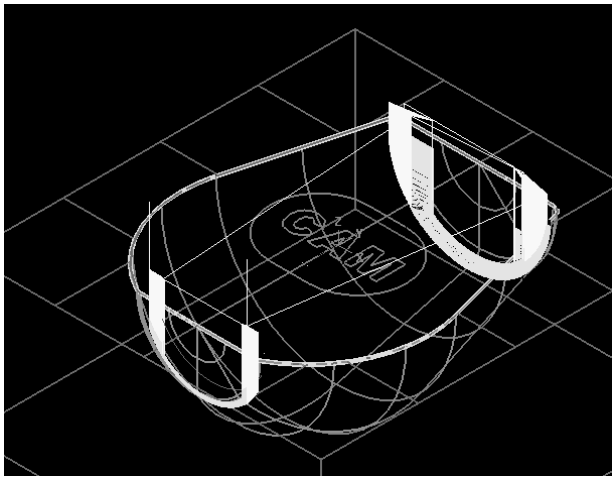
Total tolerance is the sum of the filter tolerance and cut tolerance. You can adjust the ratio of the filter tolerance to the cut tolerance, change the tolerance amounts, and select arc options.
Maximum stepover sets the size of the step between XY cuts in a surface toolpath.
From slope angle/To slop angle set the area to be machined.
Hybrid leftover cutting combines 2D and 3D cuts where cuts above the cutoff angle (usually the steepest area) are constant Z and cuts below are 3D.
Keep cuts perpendicular to leftover region can improve the finish by adding more moves, but it will increase the machining time.

- Select the **Lefover material parameters** page and change the parameters as shown in the following screenshot.



Roughing tool diameter is the diameter of the previous tool that was used to machine the part (in our case the 0.75 " Ball endmill used in the finish contour toolpath).
Roughing tool corner radius is the corner radius of the previous tool that was used to machine the part.
Overlap distance is an additional offset applied to the previous tool shape. This value causes the leftover material to be calculated as if it had been cut by a larger tool.

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



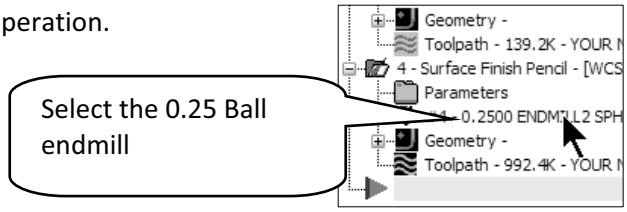
- Select **Toggle toolpath display on** selected operations to remove the toolpath display.

STEP 18: USE CHECK HOLDER TO FIND THE MINIMUM TOOL LENGTH REQUIRED FOR THE ¼ " BALL ENDMILL

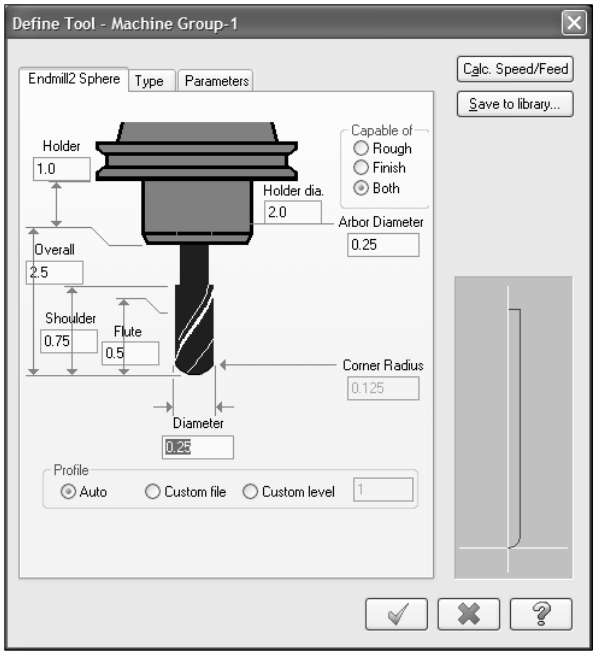
- **CheckHolder** C-Hook is used to check an operation's tool holder for interference with the part. It calculates areas where there is interference, and tells you the minimum tool length required to avoid it.

18.1 Check the current tool lenght

- Click on the 0.25 "Endmill Sphere in the pencil operation.



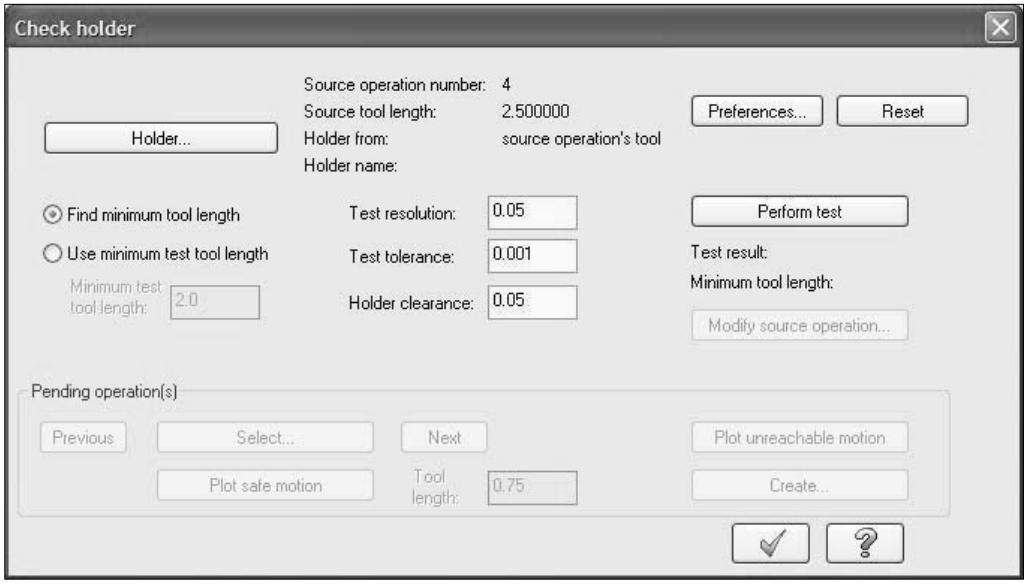
- Note the Overall value of 2.5 "



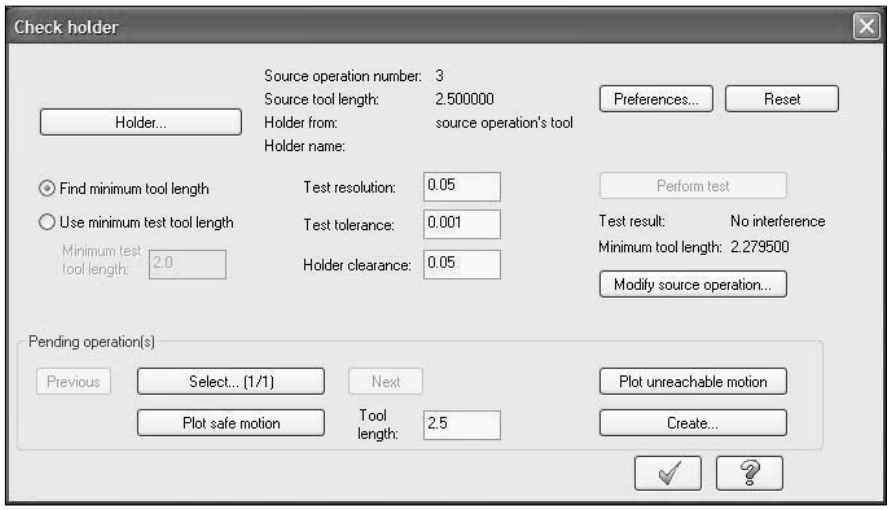
- Select the **OK** button to exit.

18.2 Use the Check Holder C-hook to to check the tool holder for interference with the part.

- Make sure that only the **Surface Leftover** is selected in the **Toolpaths manager**.
- Enter **Alt** + **C** to open the C-hook directory.
- Select **CheckHolder.dll** from the list.
- Enter the **Holder clearance** of 0.05 and keep the other settings as shown:



- Select the **Perform test** button.
  - Note that the test result is **No interference.**



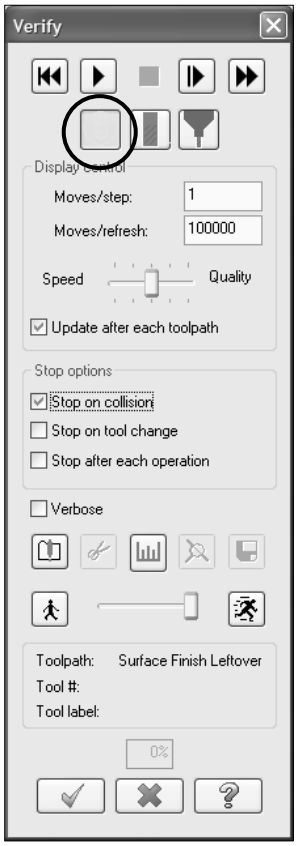
- Select the **OK** button to exit **Check holder** parameters. 

STEP 19: VERIFY THE SURFACE FINISH CONTOUR AND THE SURFACE FINISH LEFTOVER TOOLPATHS

- Select only the **Surface Finish Contour** and the **Surface Finish Leftover** operations.
  - Click on the **Surface Finish Contour** first and then hold down the **Ctrl** key and select the **Surface Finish Leftover**.

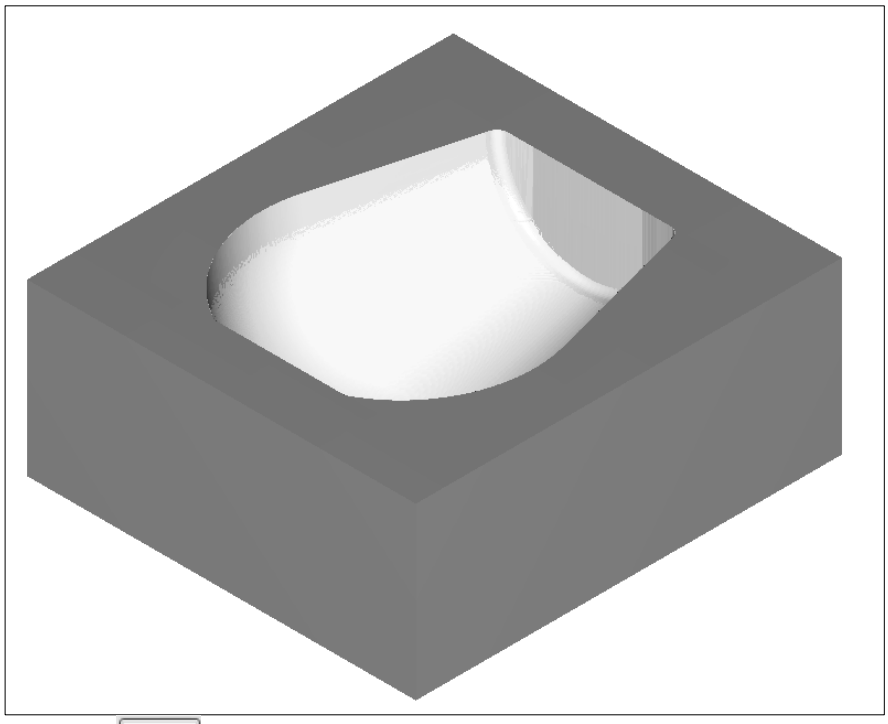
- Select the **Verify selected operations** button.

- Enable **Turbo** mode and Stop on collision.



Select **Machine** button to start simulation.

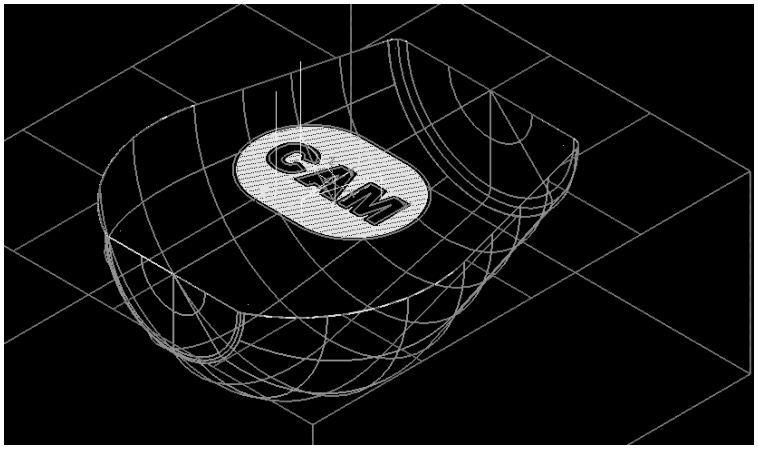
The part should appear as shown below:



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

20. CREATE THE 2DPOCKET TOOLPATH THAT WILL BE PROJECTED ONTO THE SURFACES.

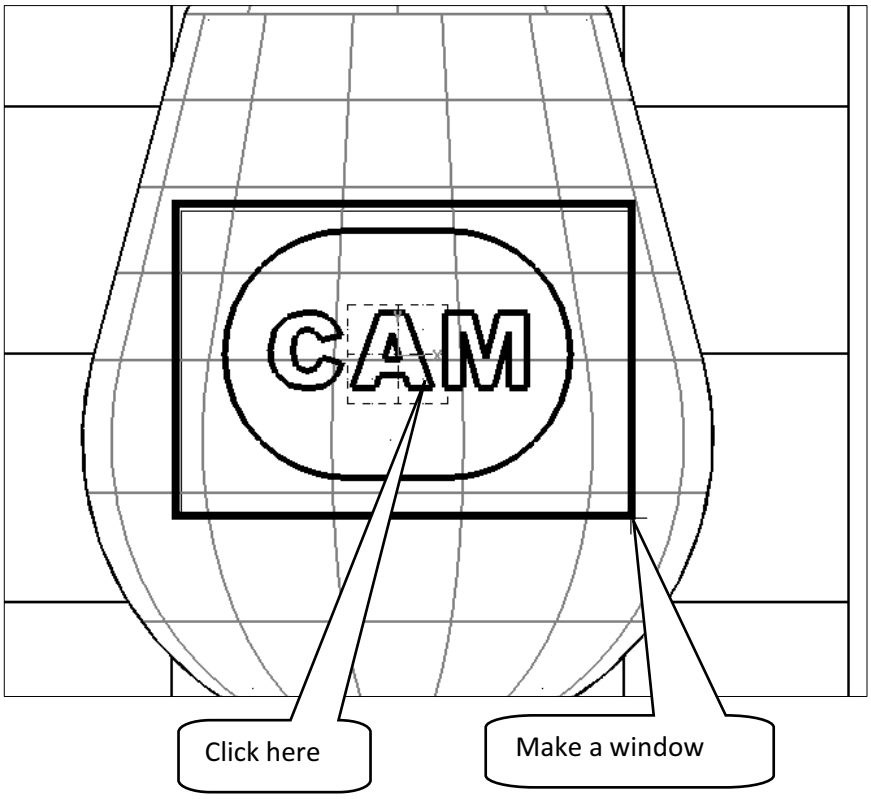
Sub Step Preview:

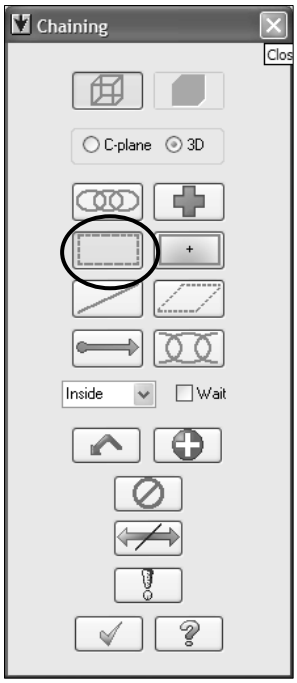


- Change the graphic view to **Top**. 

Toolpaths

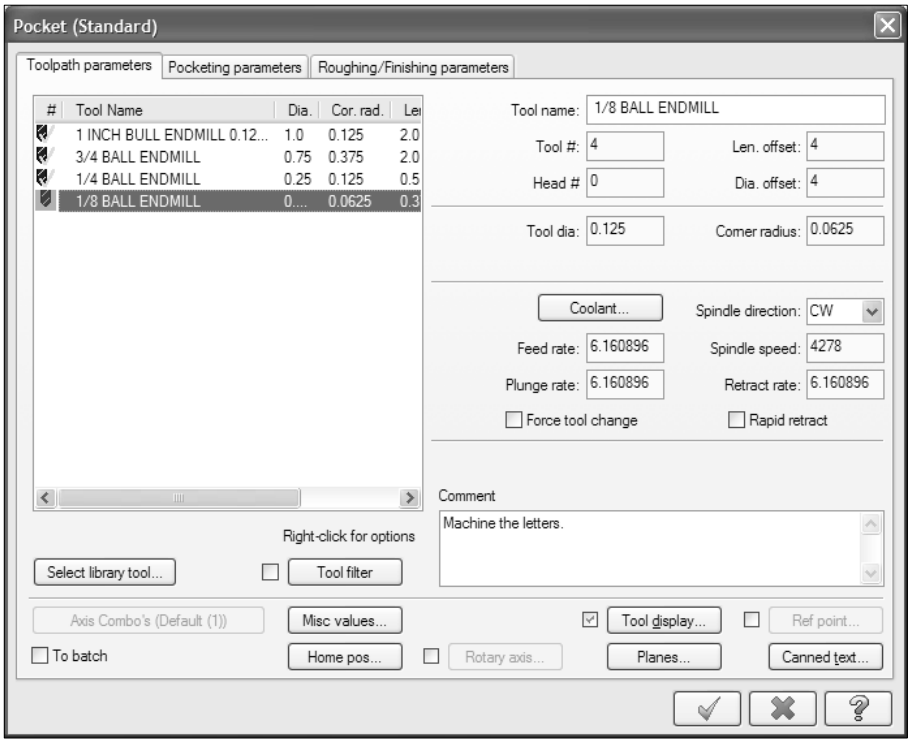
- **Pocket**
  - [Select pocket chain 1]: Make a window around the obround as shown:
  - [Sketch approximate start point]: Click inside the obround as shown:



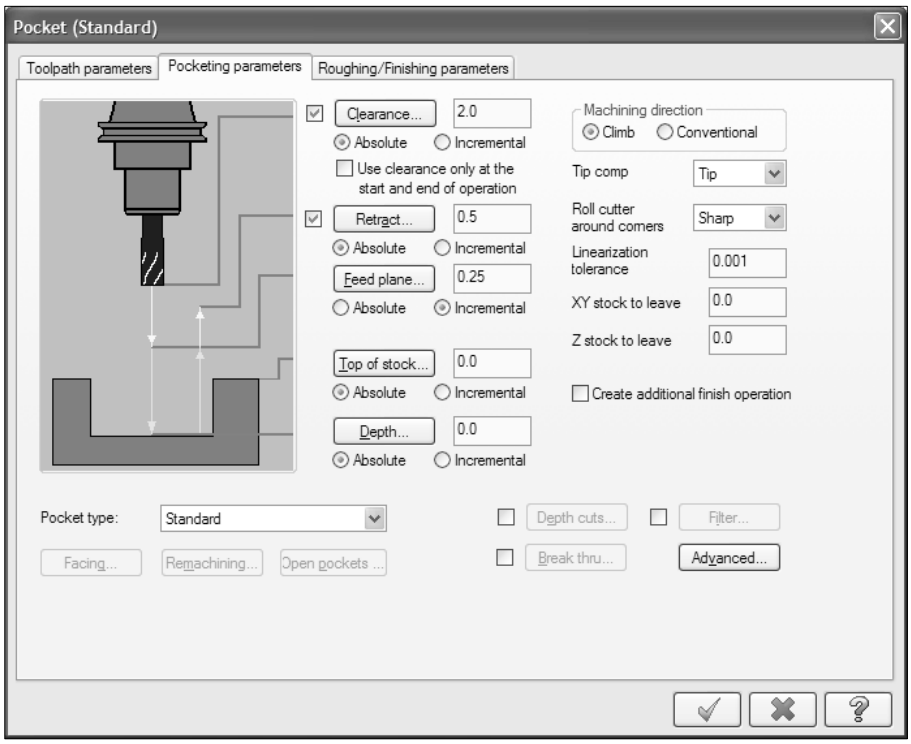


- Select the **OK** button to exit **Chaining** dialog box. 

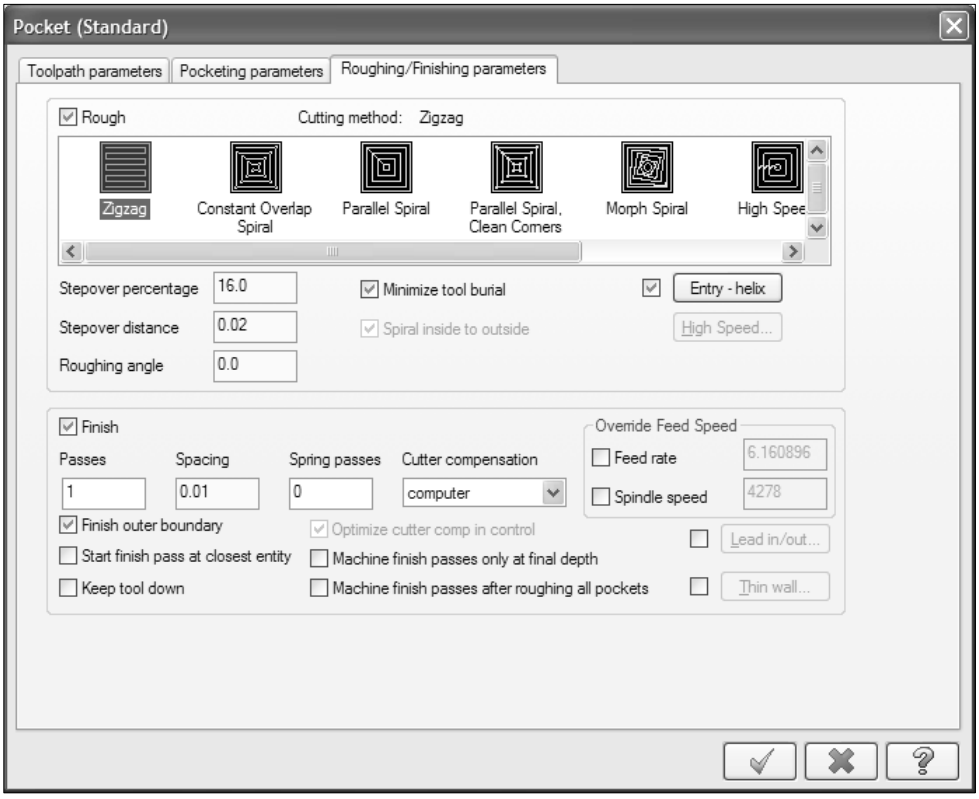
Select the ¼ " Ball endmill and match the parameters in the **Toolpath parameters** as shown.



Select the **Pocket parameters** page and check the parameters to match the following screenshot.



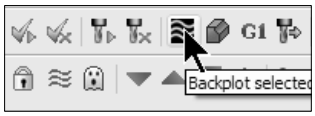
Select **Roughing/Finishing parameters** tab and made the changes as shown.



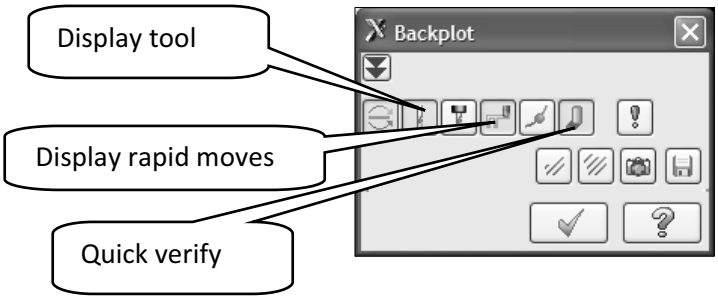
Select the **OK** button to exit **Roughing/Finishing parameters**.

STEP 21: BACKPLOT THE TOOLPATH

- 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**
- **Quick verify**



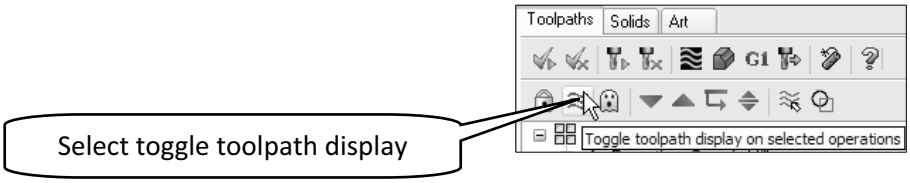
- Select the **Play** button.

- Click in the middle of the pocket and using the mouse wheel, Zoom in.

- To **Zoom in** you can also use **Zoom target** icon 

; click in the middle of the pocket and then, move the cursor out and select the corner of the zoom window when the pocket is included.
- Note the areas (inside the A letter and around M letter) that the 1/8 " Ball endmill could not machine. We will remachine the pocket using an 1/16 Ball endmill to clean up these areas.
- Select the **OK** button to exit **Backplot.** 

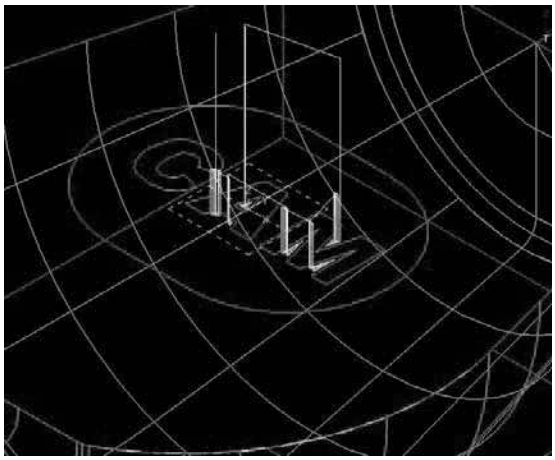
- Select **Toggle toolpath display on** selected operations to remove the toolpath display.



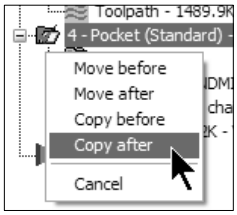
STEP 22: REMACHINE THE POCKET.

- **Remachining** calculates areas where the roughing tool could not machine the stock and creates a second toolpath to clear the remaining material.

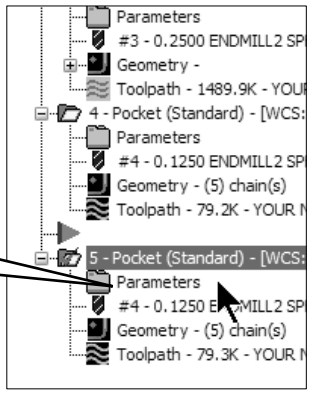
Toolpath Preview:



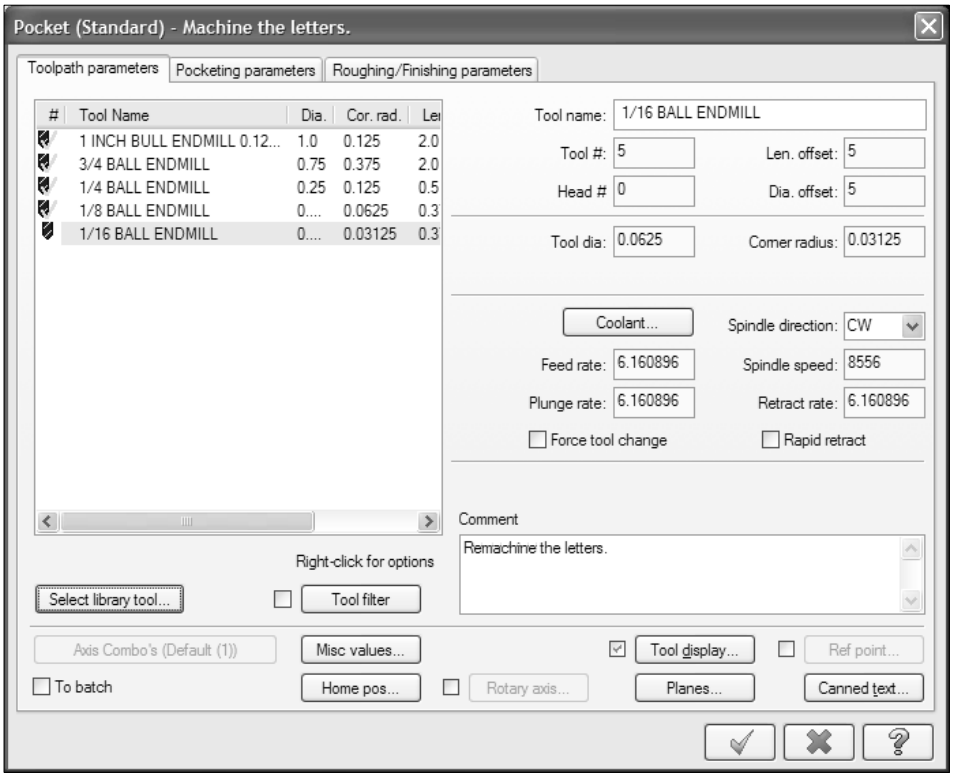
- Click on the Pocket (Standard) operation in the Toolpath Manager and make sure that it is the only operation selected.
- Right-mouse click on the Pocket, hold it down and drag
- Select **Copy after** from the drop-down list.



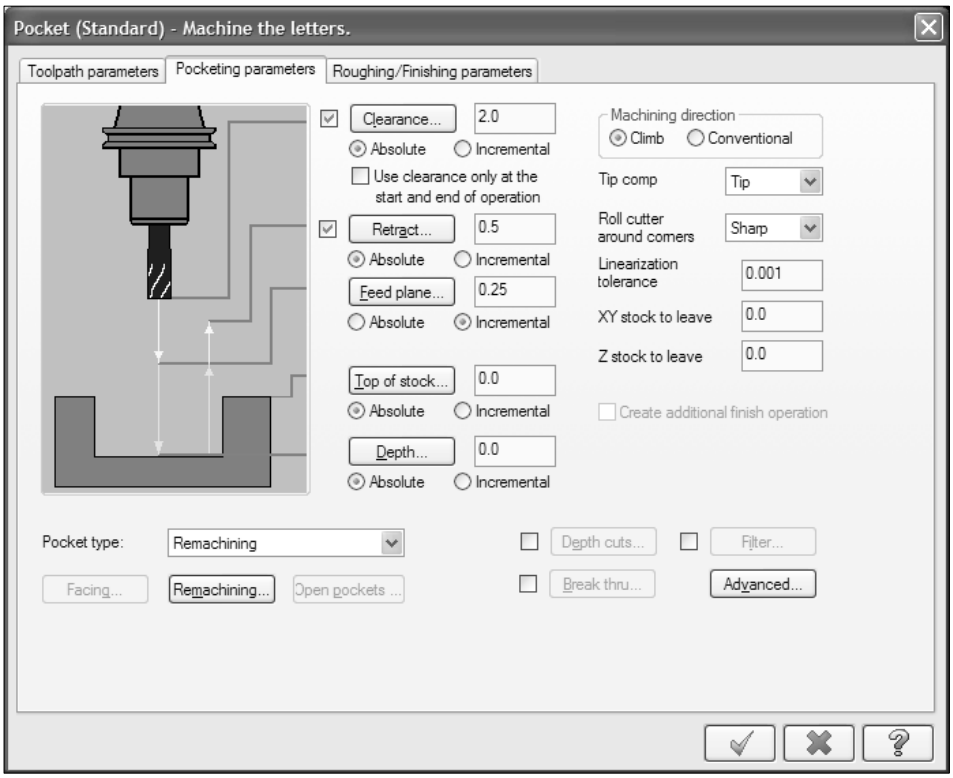
- Select **Parameters** in the second Pocket operation as shown.



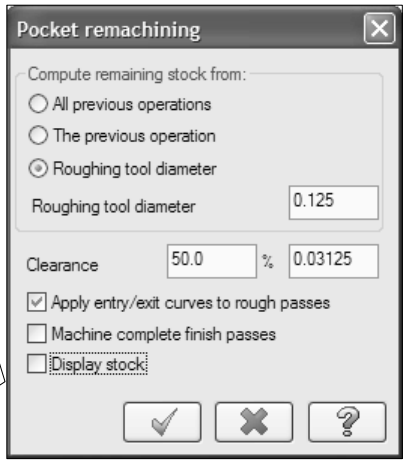
Select the **Toolpath parameters** tab and select from the library the 1/16 " Ball endmill.



Select the **Pocketing parameters** tab and select **Remachining** as the **Pocket type.**

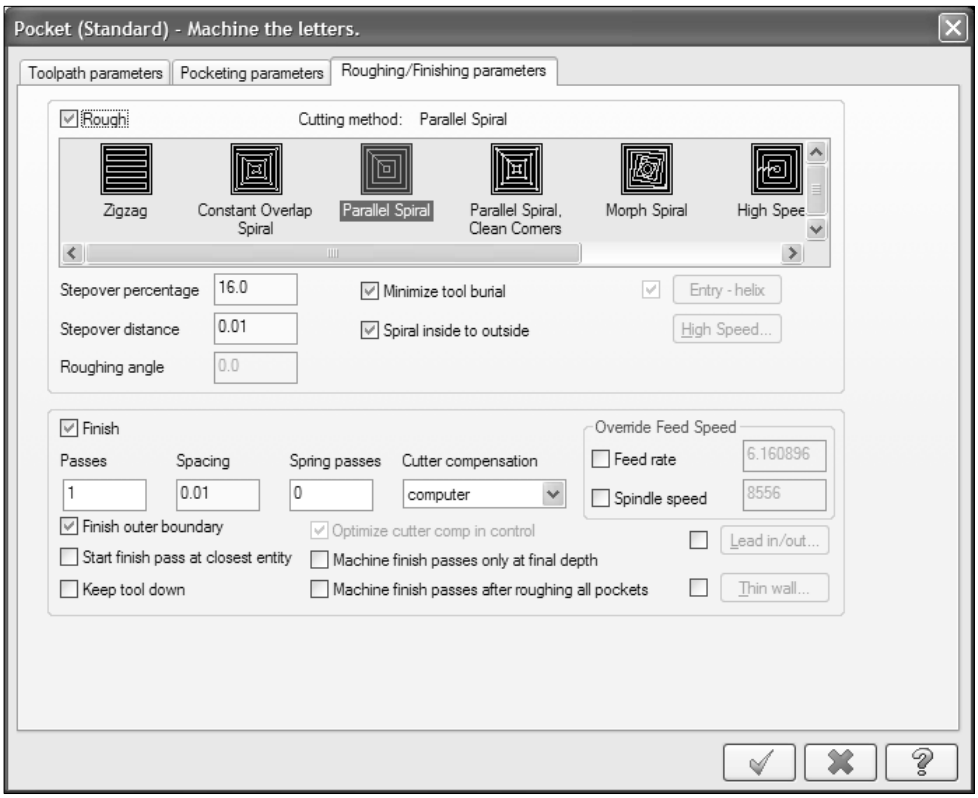


- Select the **Remachining** button and change the parameters as shown.



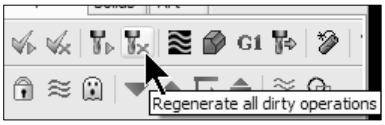
Remachining calculates the remaining stock from: the stock left by all previous operations,the stock left by the most recent previous operation or the size of the roughing (previous toolpath) tool diameter.
Clearance extend the remachining toolpath at the beginning and end to prevent cusps of material from being not machined.
Apply entry/exit curves to rough passes takes the values from **Lead in/out** dialog box to make a smooth entry/exit with each pass.

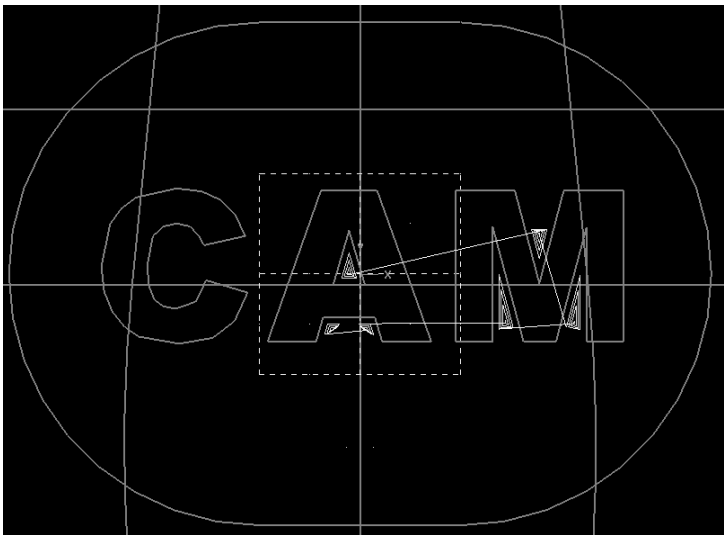
- Select the **OK** button to exit the **Pocket remachining**.
- Select the **Roughing/Finishing parameters** to make sure that the parameters are matching the following screenshot.



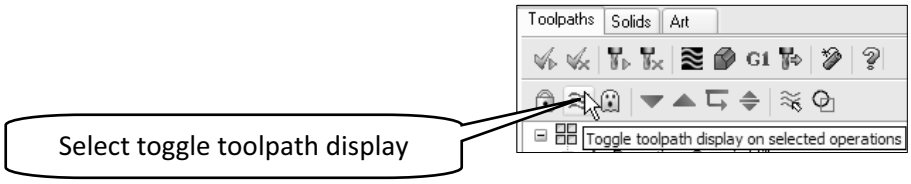
- Select the **OK** button to exit the **Pocket parameters.**

- Select **Regenerate all dirty operations** icon to regenerate the toolpath.





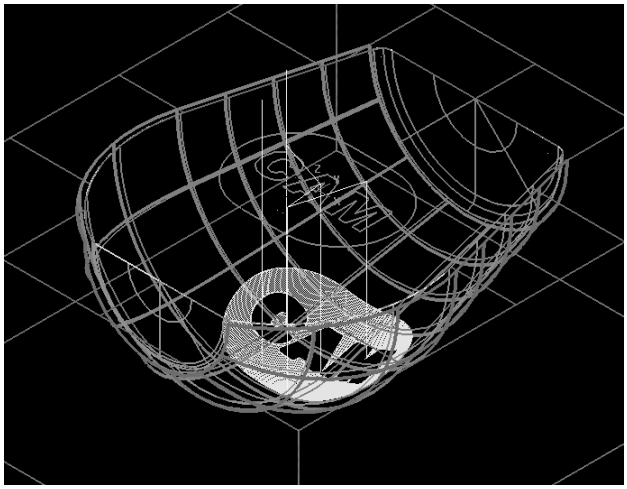
- Select **Toggle toolpath display on** selected operations to remove the toolpath display.



STEP 23: USE SURFACE FINISH PROJECT TO MACHINE THE RAISED LETTERS

- **Project Rough and Finish Toolpaths** allow you to project curves, points, or another NCI file onto selected surfaces. These toolpaths can closely match the cut motion to the shape of the part and can be used for engraving.

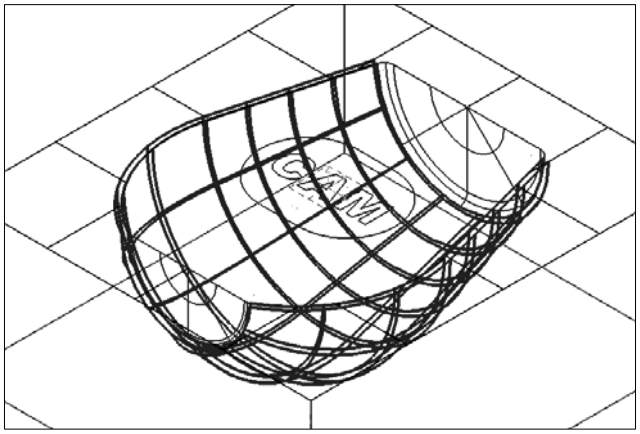
Toolpath Preview:



- To be able to machine the raised letters we need to offset the existing surfaces to project the toolpaths on them.

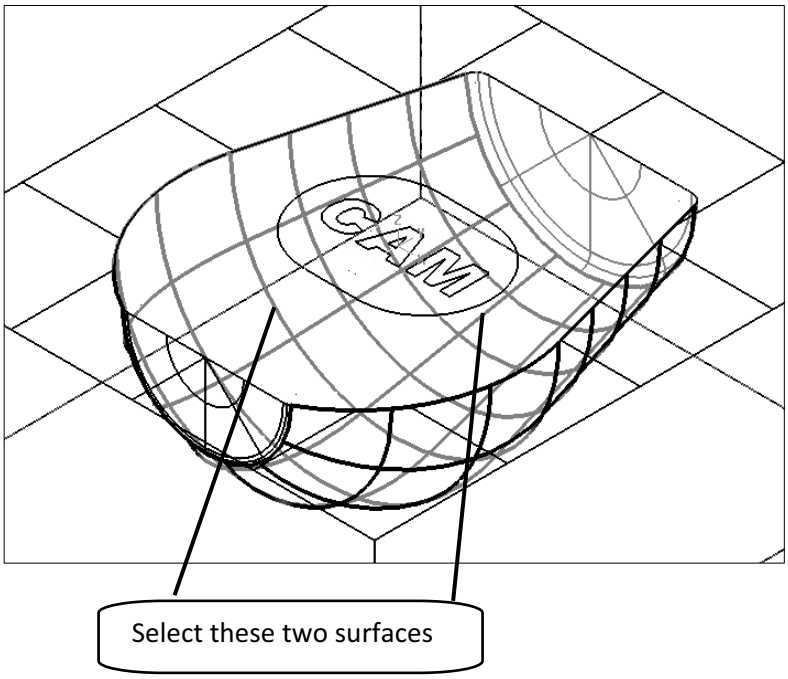
23.1 Create the offset surfaces

Sub Step Preview:



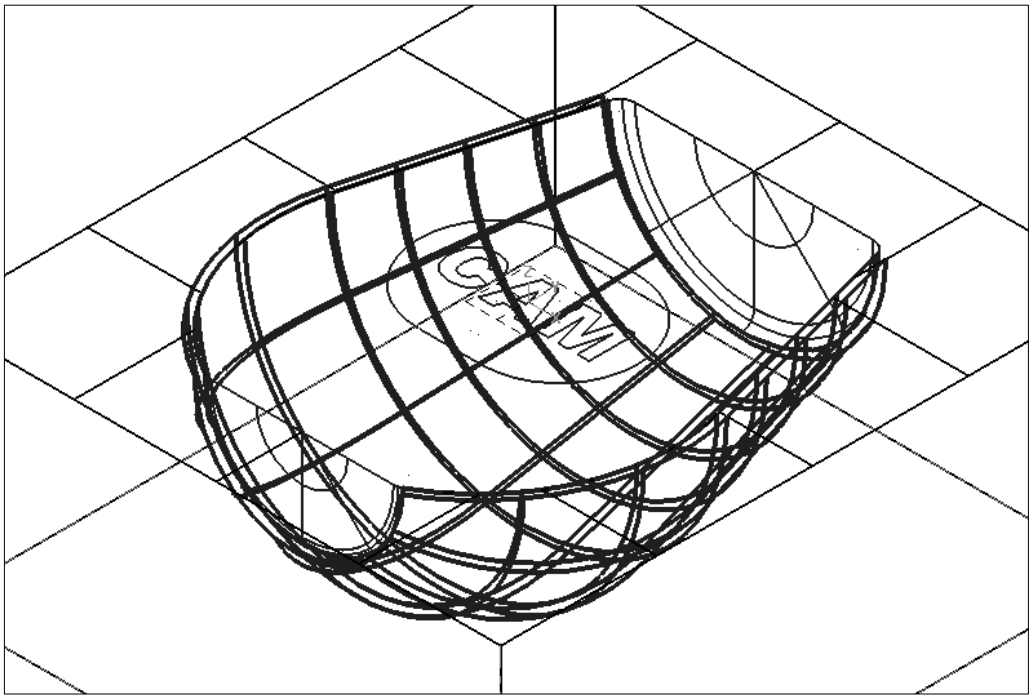
Create

- **Surface**
- **Offset**
  - [Select surfaces to offset]: Select the two surfaces as shown



  - Press Enter to finish the selection.
  - Change the **Offset distance** to 0.125“ end press **Enter**.

- Make sure that the offset surfaces are created below the original ones as shown.



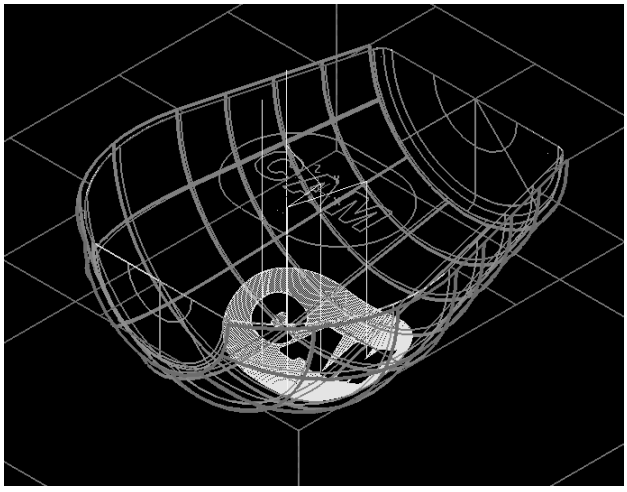
- Otherwise; click on the **Cycle/Next** button  first, and then the **Flip**  button to change the surface normal orientation. Repeate the procedure for the second surface if necessary.

Select the **OK** button to exit **Offset surface** command. 

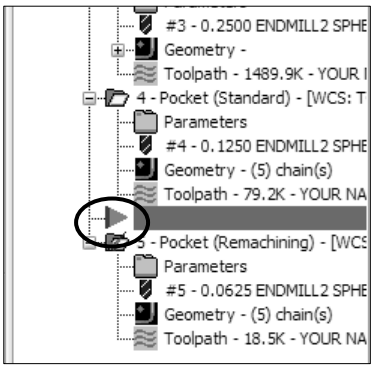
Select **Clear Color** icon. 

23.2 Surface Finish Project

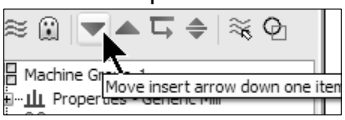
Sub Step Preview:



- Note the red insert arrow location in the **Toolpaths Manager**. The next operation is going to be created between the two pockets if the arrow is not moved.



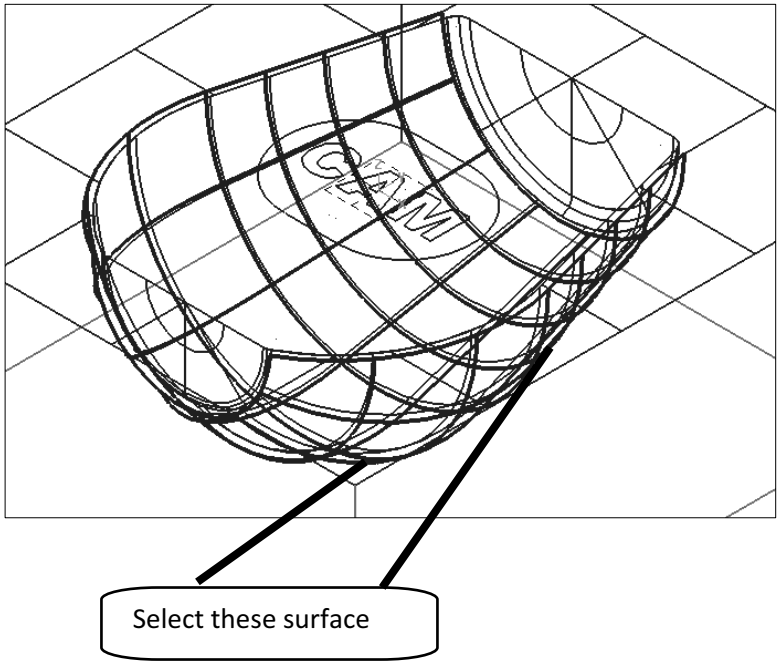
- Click on the **Move the insert arrow downe one item**. This ensures that the next toolpath will be created at the end and not in the middle of the program.



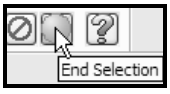
- Change the **Graphic View** to **Isometric**.

Toolpaths

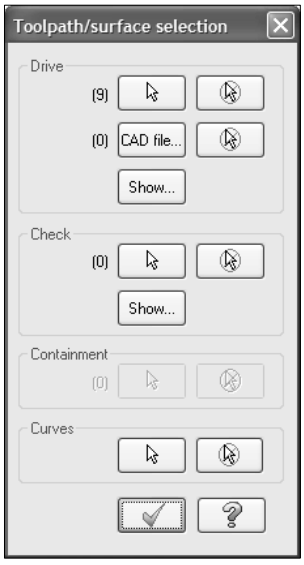
- **Surface Finish**
- **Project**
  - [Select Drive Surface]: Select the two offset surfaces that you just created.



- Select the **End Selection** button.

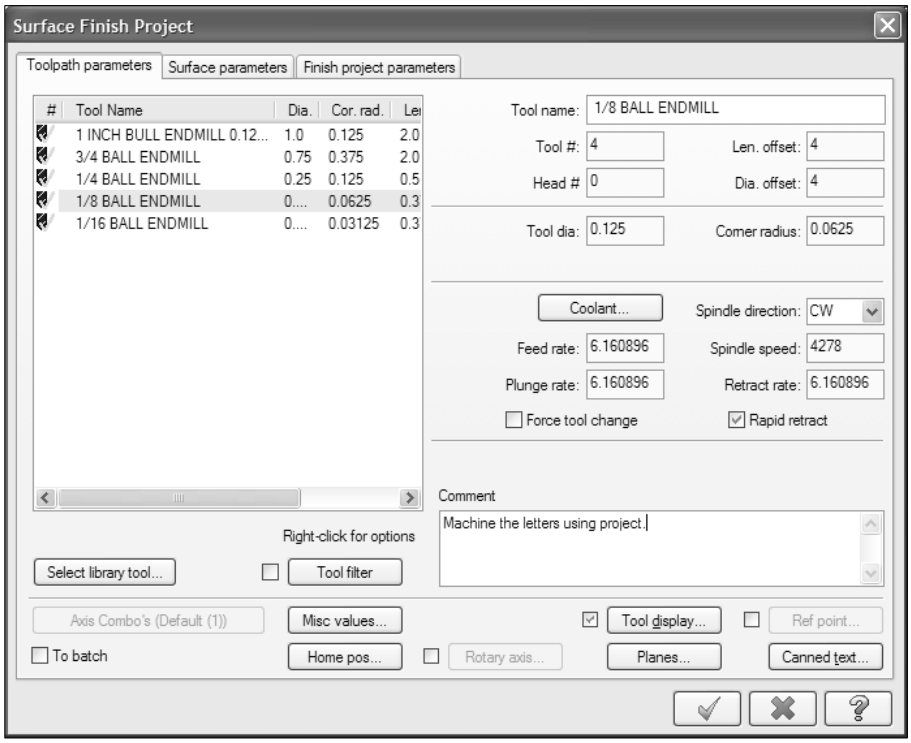


- Select the **OK** button to exit.

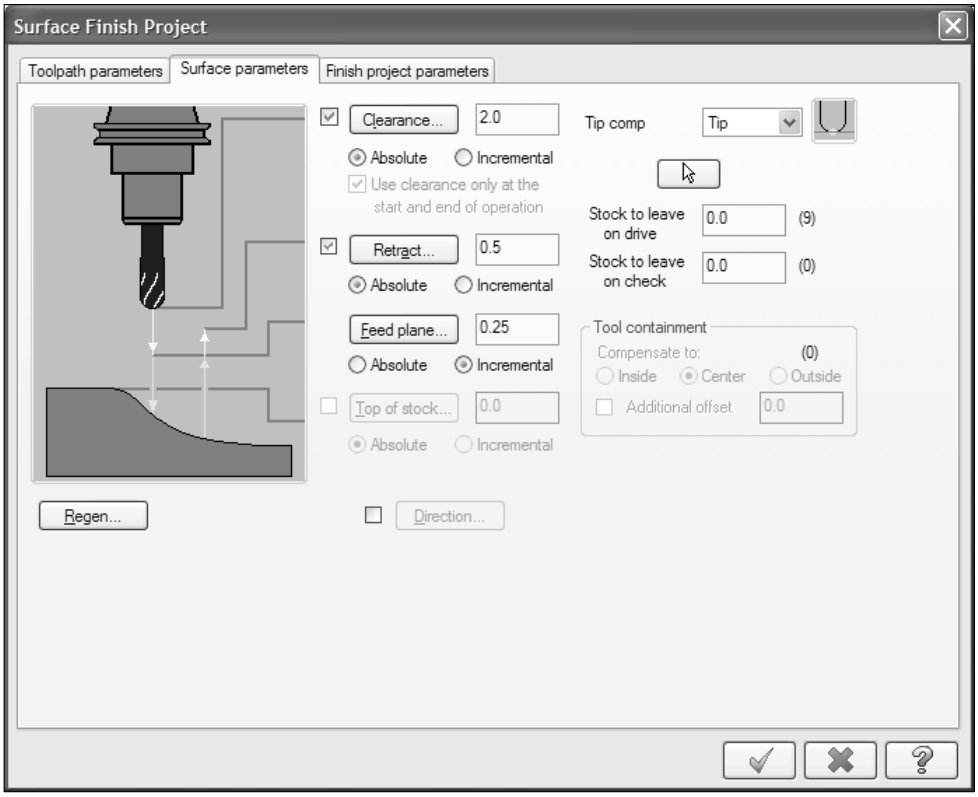


- Select the **OK** button to exit **Toolpath/Surface** selection.

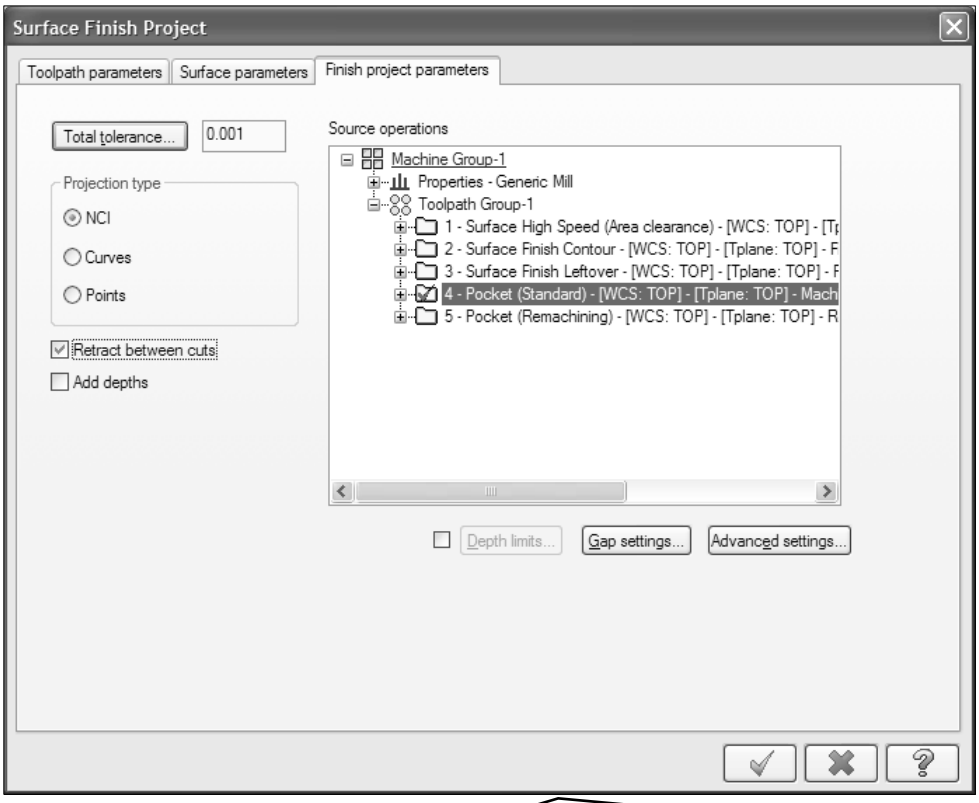
- Select the 1/8 " Ball endmill and make the changes as shown in the following screenshots.



Select **Surface parameters** and change the parameters as shown.



- Select the Finish project parameters and make sure that Projection type is set to NCI.
- Enable operation 4- Pocket (Standard) in the Source operations and enable Retract between cuts.

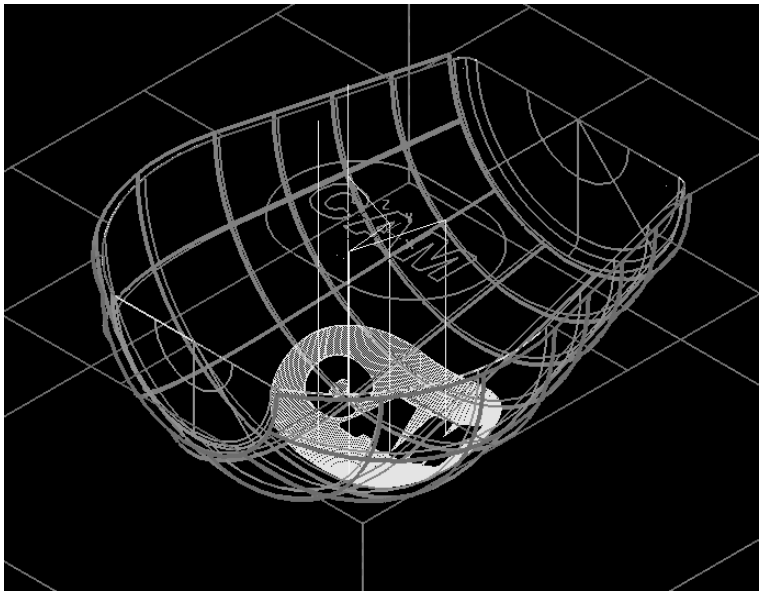


Projection type

- **NCI** allows you to project an existing 2D toolpath (contour or pocket) onto selected surfaces.
- **Curve** allows you to project 2D chains of entities (lines, arcs, splines) onto selected surfaces. Mastercam prompts you to select the chains after you exit project parameters.
- **Points** allows you to project points onto selected surfaces. Mastercam prompts you to select the chains after you exit project parameters.

Retract between cuts forces a retract move between cuts and when engraving letters (curve projection type) it allows the tool to retract between letters.
Add depths allows you to add cut depths for the rough project. It is enabled only with projection type NCI.

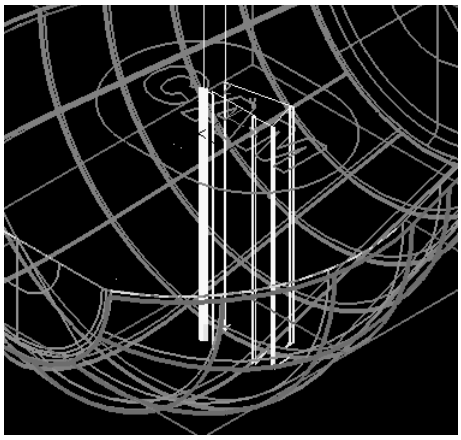
- Select the **OK** button to exit. 



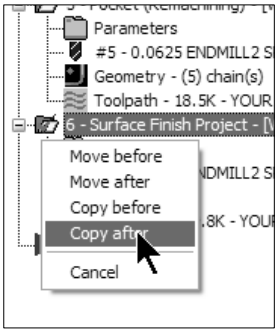
- Select **Toggle toolpath display on** selected operations to remove the toolpath display.

STEP 24: USE SURFACE FINISH PROJECT TO CLEAN UP THE LETTERS.

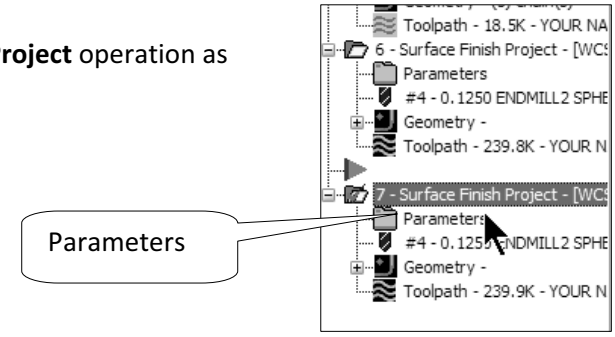
Toolpath Preview:



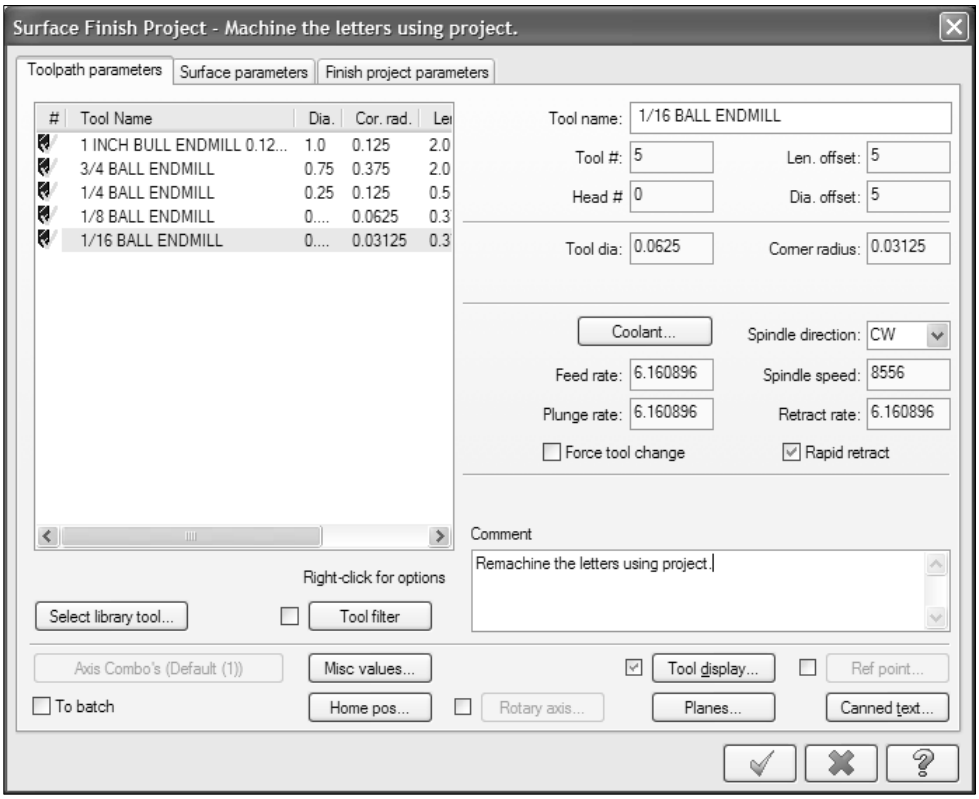
- Click on the **Surface Finish Project** operation in the **Toolpath Manager** and make sure that it is the only operation selected.
- Right-mouse click on the **Surface Finish Project**, hold it down and drag
- Select **Copy after** from the drop-down list.



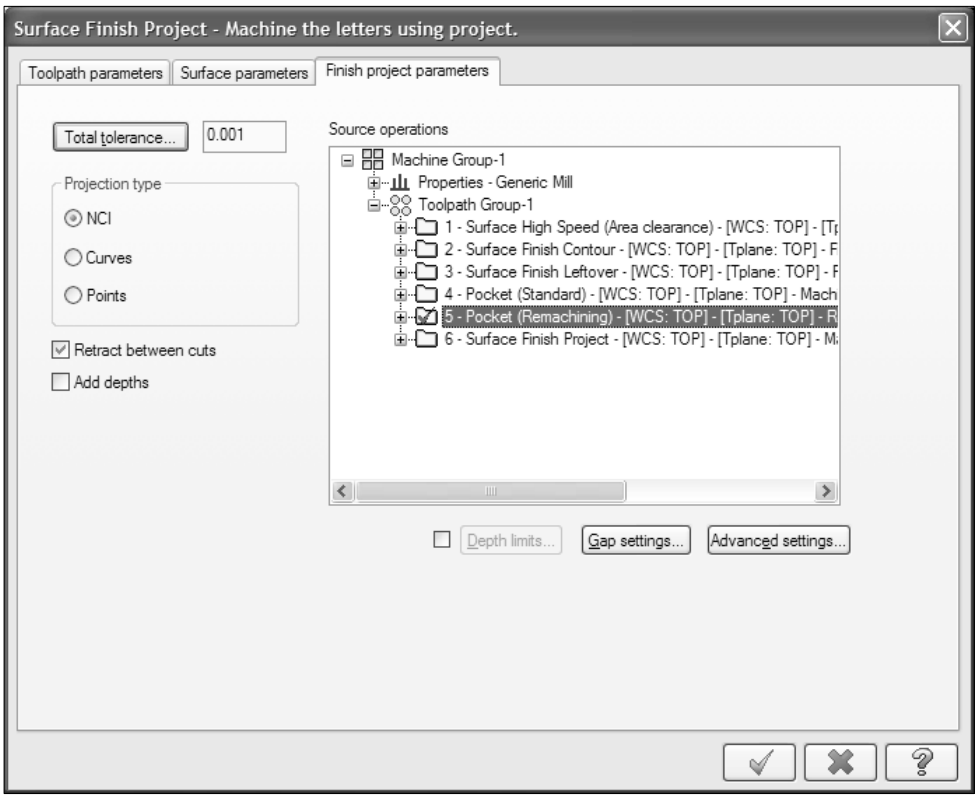
- Select **Parameters** in the second **Surface Finish Project** operation as shown.



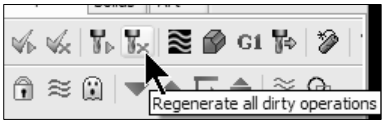
- Select **Toolpath parameters** and select the 1/16 Ball endmilll



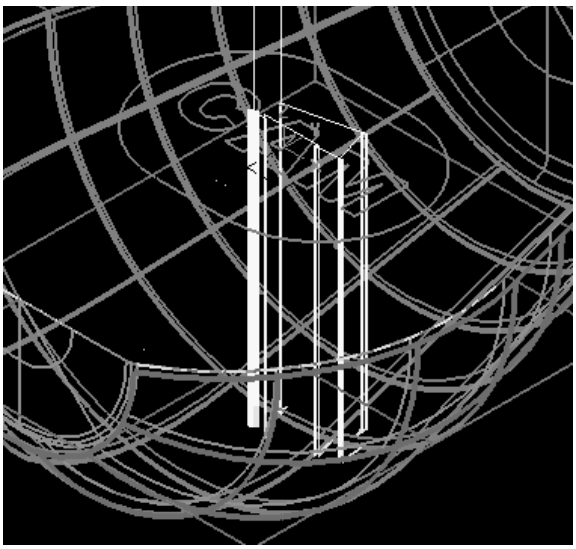
- Select the Finish project parameters and select operation 5- Pocket (Remachining) from the list.



- Select the **OK** button to exit the toolpath.
- Select **Regenerate all dirty operations** icon to regenerate the toolpath.

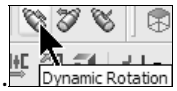


The toolpath should look as shown below:



STEP 25: VERIFY

- Based on Step 19, verify the Surface Finish Contour, the Surface Finish Leftover and the two Surface Finish Project operations only, and make sure you don't select any of the 2D Pockets.

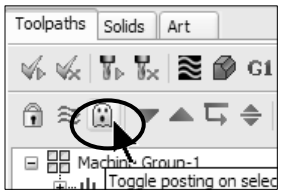
- To rotate the part, select the **Dynamic Rotation** icon. 

- [Pick a point to begin dynamics]: Select a point around the middle of the part.
- Slightly move the cursor and click on the part is in the proper position.

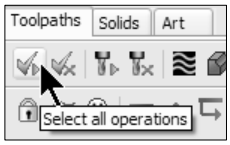
The part should look as shown:

STEP 25: POST PROCESS THE FILE.

- Click on the **Pocket (Standard)** operation to select only this operation and then, holding down the Ctrl key, select the **Pocket (Remachining)** operation.
- Click on **Toggle posting on selected operation** to turn the post off for these toolpaths.

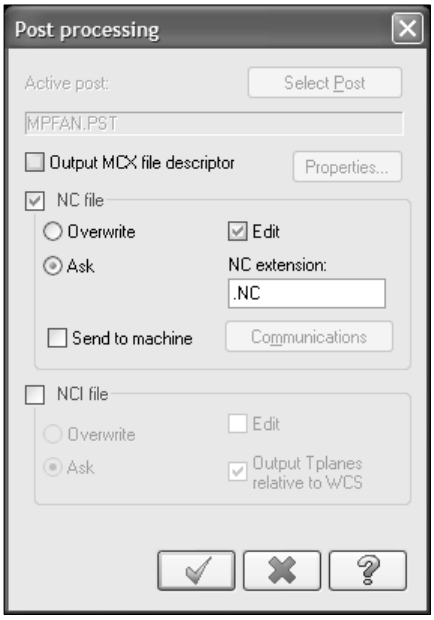


- Click on **Select all operations** icon to select all the toolpaths.
  - Note that because we toggle the postion off for the pocket operation, even though the operation is selected, the G-code for these operations are not going to be generated. Remember that these toolpaths were only used to generate the project toolpaths.



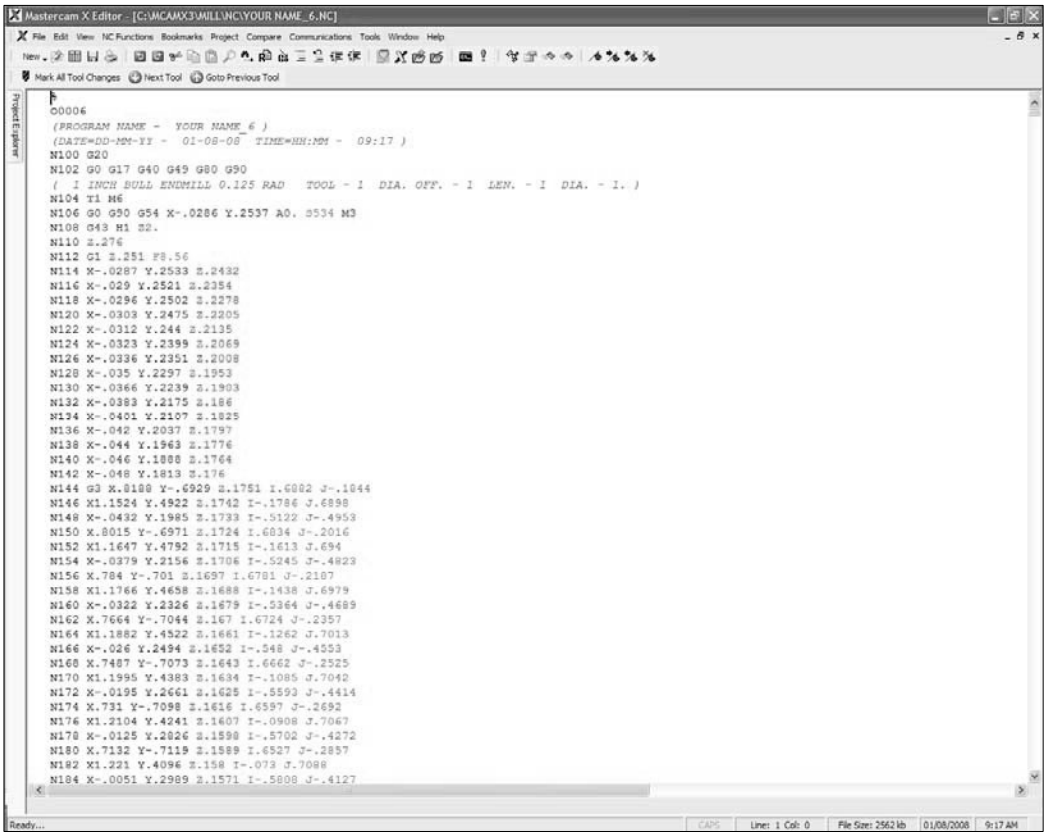
- Select the **Post selected operations** button from **Toolpath Manager**.

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



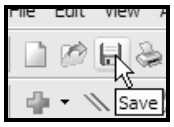
- Select the **OK** button to continue.

- Enter the same name as the geometry name in the NC File name field. (Your Name _6)
- Select the **OK** button. 

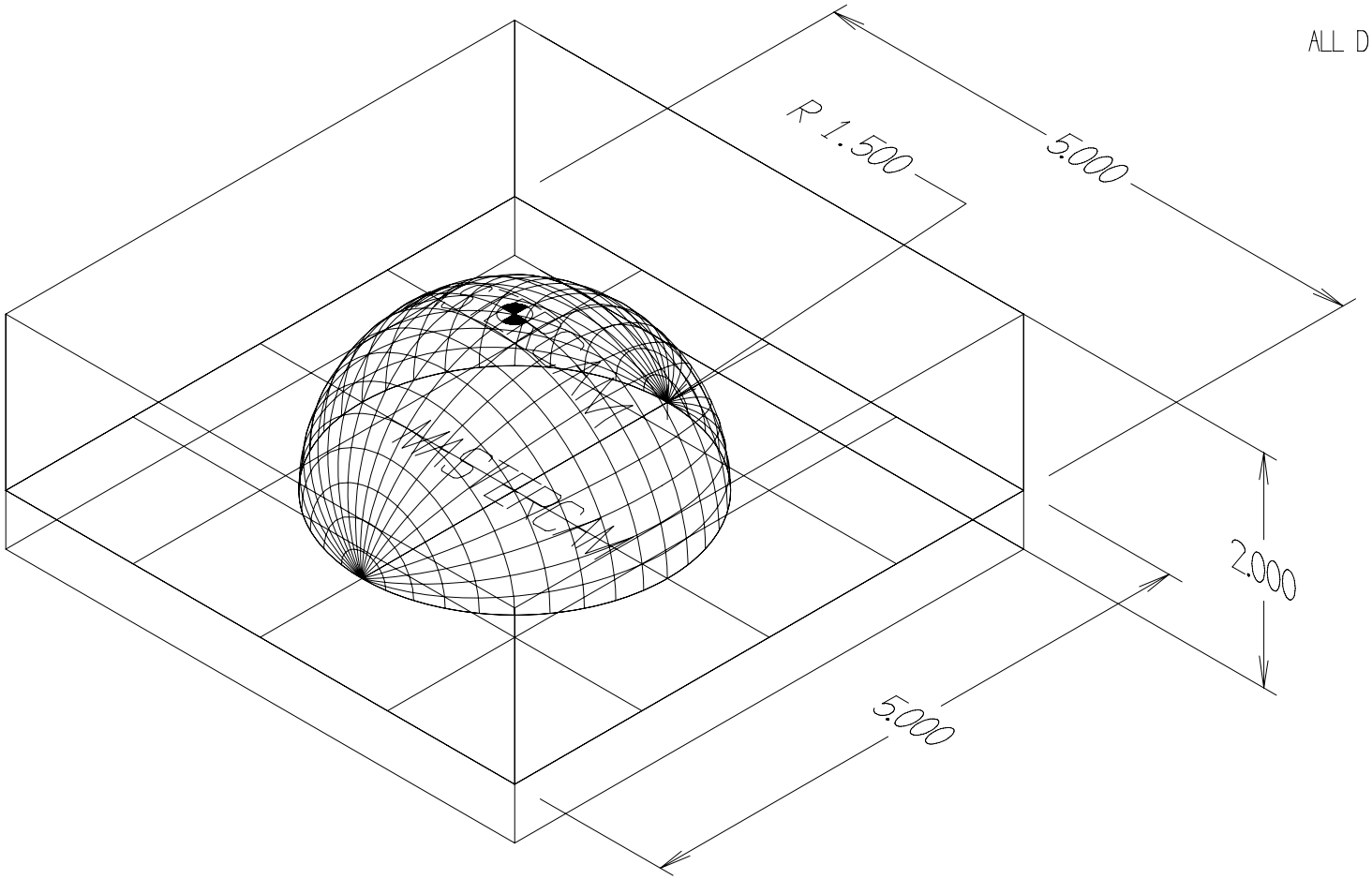


- Select the red **X** box at the upper right corner to exit the **Editor**.

STEP 26: SAVE THE UPDATED MCX FILE.

- Select the **Save** icon. 

ALL DIMENSIONS IN INCHES

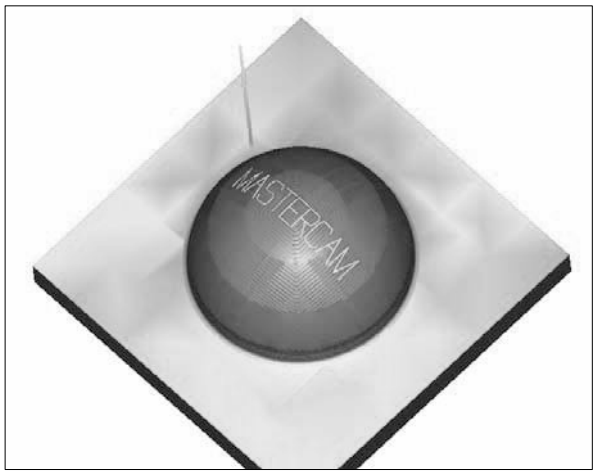


Create MASTERCAM using Drafting font letters.
Height = 0.375
Spacing between the letters = 0.0375
Center the letters

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

REVIEW EXERCISES.

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



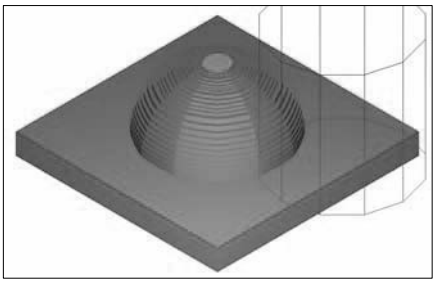
Tips: You need only flat letters!

1. Establish the **Stock size** Y = 5, X = 5, Z = 2

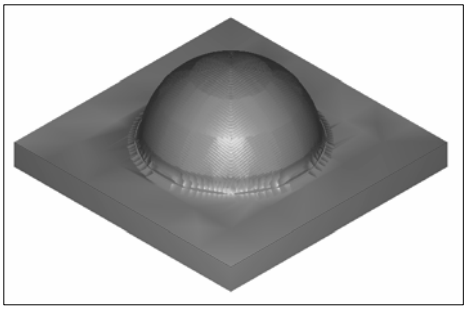
Stock origin X = 0, Y = 0, Z = 1.505

2. Create a rectangle with surface option at Z0, the same size as the stock (5 X 5)

3. Surface Rough Contour

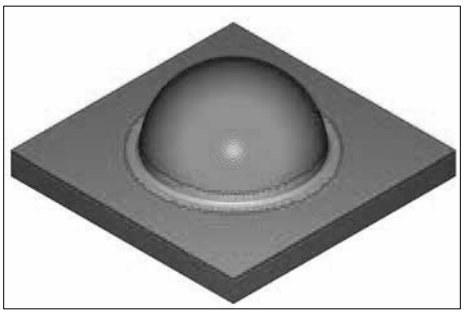


Select all surfaces
Use 3“ Face Mill (change the library to Big.Tools and edit the tool: Taper angle=0)
Clearance = 2.5
Retract = 2
Stock to leave on drive surfaces = 0.02
Total tolerance = .005
Max. stepdown = 0.1
Enable Entry/exit Arc Radius = 0.25, Arc Sweep = 90
Enable Shallow
Add cut to shallow areas
Min stepdown = .005
Limiting stepover = .05
Disable Allow partial cuts
One way cutting



4. Surface Finish Scallop

Use 1/2“ Ball End Mill
Total tolerance = .001
Max. stepover = 0.05
Expand inside to outside



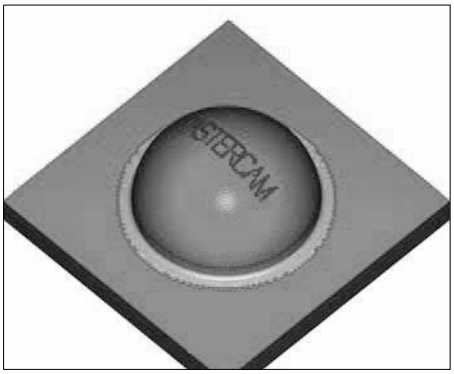
5. Surface Finish Leftover

Use 1/8“ Ball End Mill
Total tolerance = .001
Max. stepover = 0.02
Cutting method 3D Collapse
Roughing tool diam = .5

Roughing corner radius =.25

6. Surface Finish Project

Use 1/32“ Ball End Mill
Stock to leave on drive surfaces = -0.01
Projection type Curves
Enable Retract between cuts



7. Backplot and Verify the toolpaths.

8. Post process the file.

NOTES:

TUTORIAL 6 QUIZ

- What is a surface normal?

- When would you use the “surface high speed area mill” toolpath?

- When would you use a “surface finish leftover” toolpath?

- Why did we turn off the post function on the pocket operations?