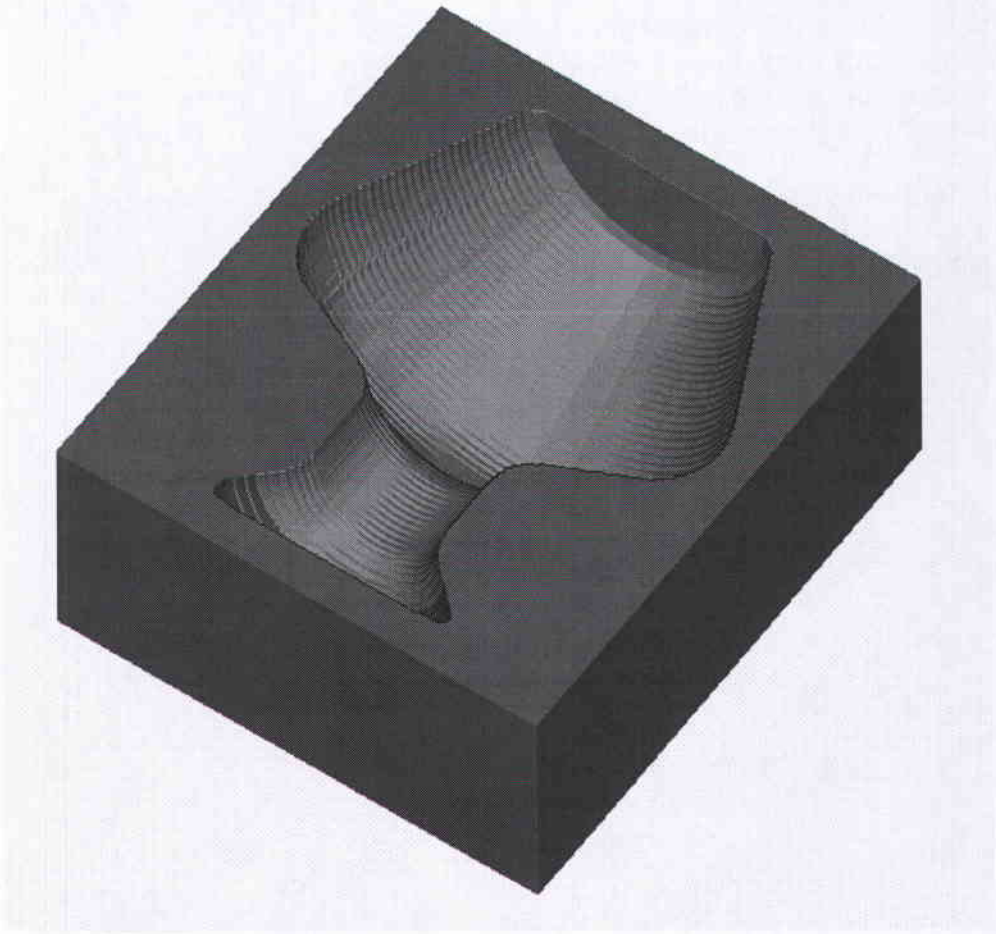
TUTORIAL SERIES FOR

Mastercam X²

TUTORIAL 8
REVOLVED SURFACE, TRIM-FLAT BOUNDARY SURFACE, ROUGH POCKET AND FINISH PARALLEL TOOLPATHS.



Objectives:

The Student will design a 3-dimensional drawing by:

- Creating lines using polar positioning.
- Creating parallel lines by defining the offset direction and distance.
- Trimming 1 entity to another existing entity.
- Creating fillet radii.
- Creating a revolved surface.
- Changing the view of the part for better visualization.
- Creating edge curves.
- Creating a trim-flat boundary surface.

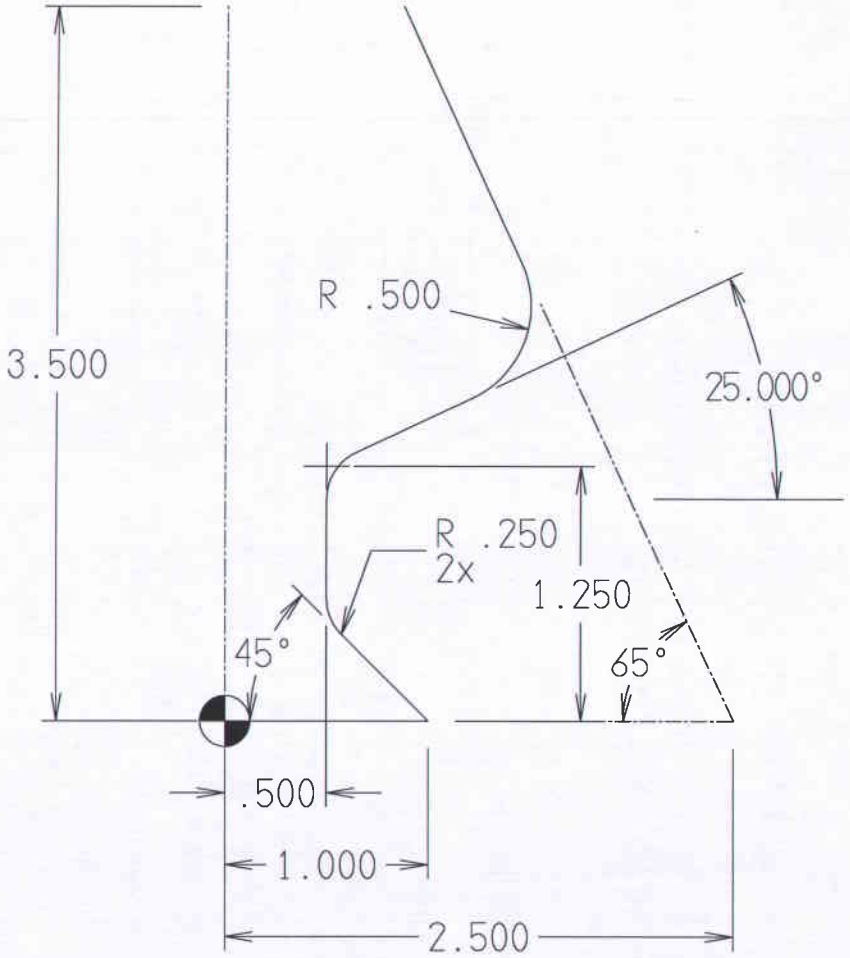
The Student will create a 3-dimensional milling toolpath consisting of:

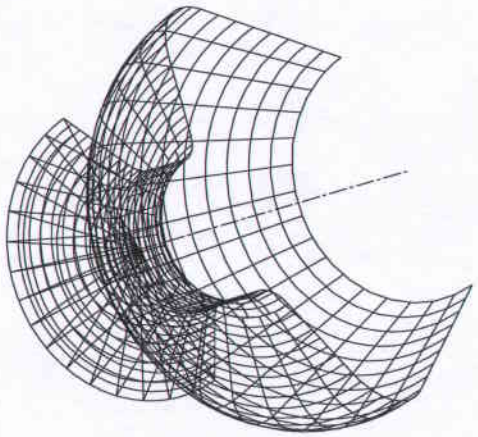
- Defining a 3-dimensional block the size of the workpiece.
- A 3-dimensional rough pocket toolpath.
- A 3-dimensional finish parallel toolpath.
- A 3-dimensional finish contour toolpath.

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

- Running the Verify function to machine the part on the screen.

ALL DIMENSIONS IN INCHES





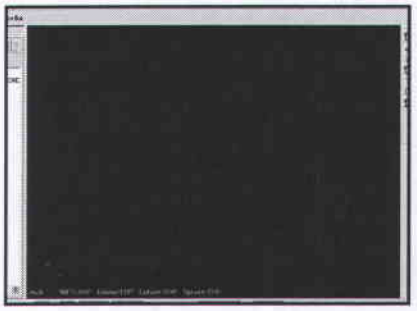
TITLE	TUTORIAL 8
MATERIAL	ALUMINUM T6061
DATE: JUNE 12, 2000	eMastercam.com

GEOMETRY CREATION
Setting the toolbar states

To start a new file from Mastercam:

File
➢ **New**

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

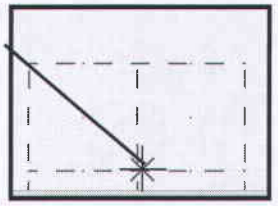


STEP 1:
CREATE THE PROFILE (THE SECTIONAL SHAPE).
1.1 Create a vertical line and a horizontal line knowing the starting point and the length of the lines.

Create
➢ **Line**
➢ **Create Line Endpoint**

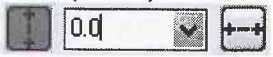
➢ Select the **Vertical** icon. 

Select the first point



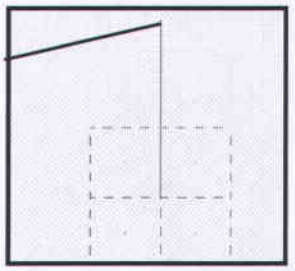
➢ [Specify the first endpoint]: Select the **Origin** (the center of the grid).
➢ [Specify the second endpoint]: Select a point above.
  - Note that the color of the geometry is cyan which means that the entity is "alive" and you can still change it.

➢ Enter the line **Length**  3.5 (Enter).

➢ [Enter the X coordinate]: 

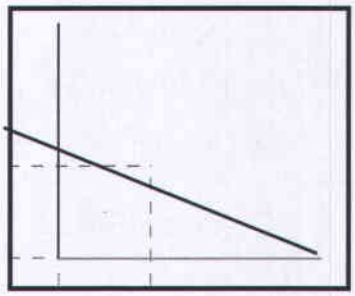
➢ Select the **Apply** button. 

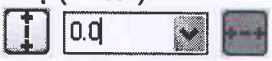
Select this Point



➢ Select the **Horizontal** icon. 

➢ [Specify the first endpoint]: Select the **Origin** again
➢ [Specify the second endpoint]: Select a point to the right as shown.



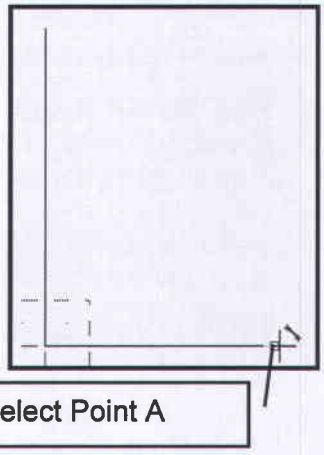
➢ Enter the line **Length** 2.5 (Enter).
➢ [Enter the y coordinate]: 

➢ Select the **Apply** button to continue with the same command.
➢ Disable **Horizontal** icon by select it again. 

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

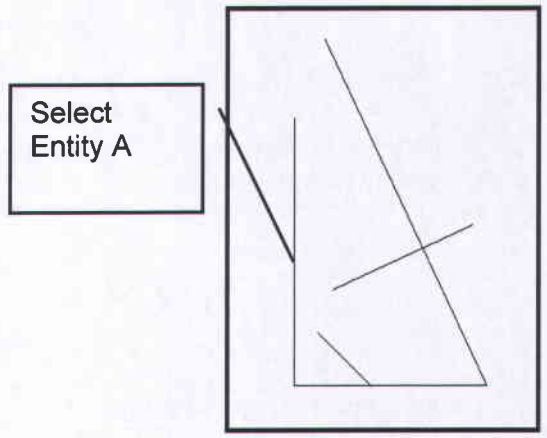
1.2 Create polar lines knowing the starting point, the length of the lines and the angle.

➢ Enter the line **Length** 5.0 (Tab).
➢ Enter the **Angle** in degrees 180-65 (Enter).
➢ [Specify the first endpoint]: Select Endpoint A.
➢ Select the **Apply** button to continue.
➢ Select the **Fit** button to fit the geometry to the screen.
➢ Enter the line **Length** 1.0 (Tab).
➢ Enter the **Angle** in degrees 180-45 (Enter).
➢ [Specify the first endpoint]: Select the **Fast point** icon.
➢ Enter the values: 1,0 (Enter).
➢ Select the **Apply** button to continue.
➢ Enter the line **Length** 2.0 (Tab).
➢ Enter the **Angle** in degrees 25 (Enter).
➢ [Specify the first endpoint]: Select the **Fast point** icon.
➢ Enter the values: 0.5, 1.25 (Enter).
➢ To exit the command you can either start a new command or select the **OK** button.



1.3 Create a parallel line.

Create
- **Line**
- **Create Line Parallel**
- [Select a line]: Select Entity A.
- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Type the **Distance** 0.5 (Enter).
- Select the **OK** button to exit the command.



1.4 Create fillets.

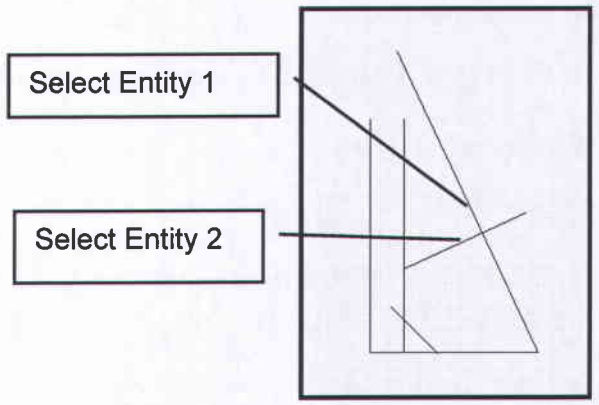
Create
- **Fillet**
- **Fillet Entities**
- Enter the fillet **Radius** 0.5 (Enter).
- [Select an entity]: Select Entity 1.
  - Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities.
- [Select another entity]: Select Entity 2.

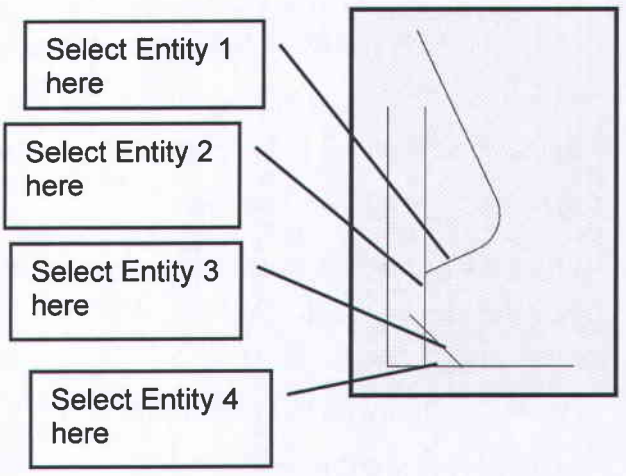
Select the **Apply** button to continue.

- Enter the fillet **Radius** 0.25 (Enter).
- [Select an entity]: Select Entity 1.
- [Select another entity]: Select Entity 2.
- [Select an entity]: Select Entity 2.
- [Select another entity]: Select Entity 3.

Select the **Apply** button to continue.

- Enter the fillet **Radius** 0.0625 (Enter).
- [Select an entity]: Select Entity 3.
- [Select another entity]: Select Entity 4.
- Select the **OK** button.

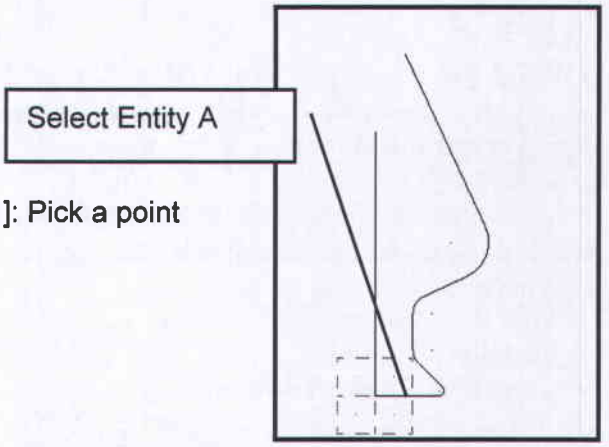




1.5 CREATE A PARALLEL LINE.

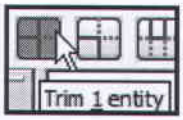
Create

- **Line**
- **Create Line Parallel**
- [Select a line]: Select Entity A.
- [Select the point to place a parallel line through]: Pick a point above the selected line.
- Type the **Distance** 3.5 (Enter).
- Select the **OK** button to exit the command.

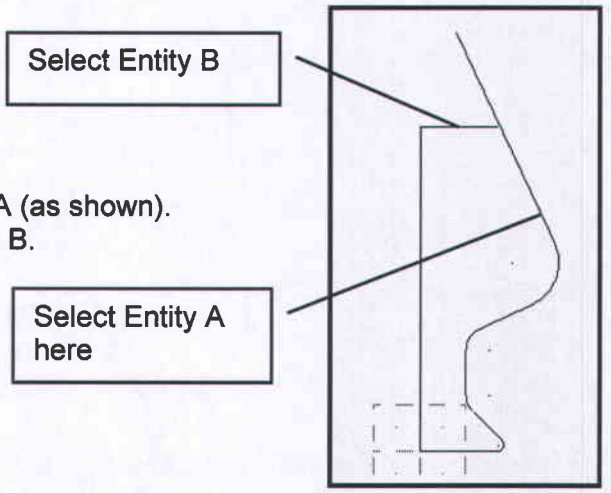


1.6 Trim one entity.

Edit

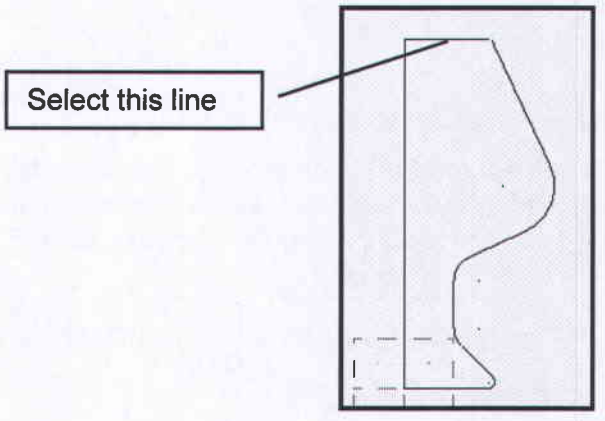
- **Trim/Break**
- **Trim/Break/Extend** 

- Select **Trim 1 entity**.
- [Select the entity to trim/extend]: Select Entity A (as shown).
- [Select the entity to trim/extend to]: Select Entity B.
- Select the **OK** button to exit the command.



1.6 Delete the construction line.

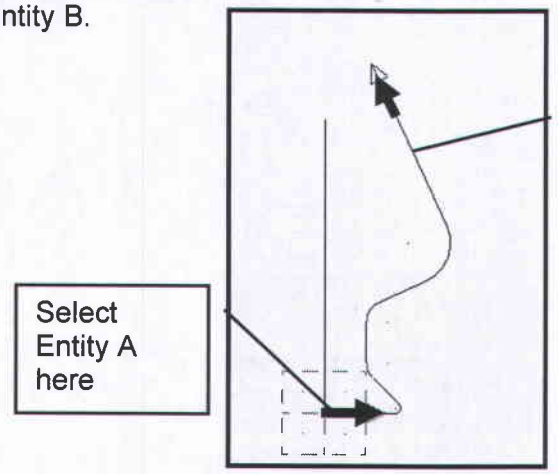
- Preselect the line as shown.
- Select the **Delete** entity icon.

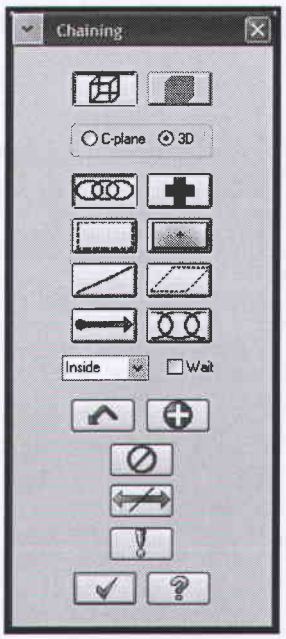


STEP 2:
CREATE THE REVOLVED SURFACE.

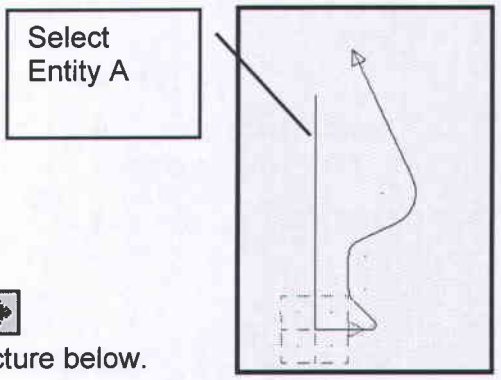
Revolved Surface: Is a geometrical surface generated by rotating a sectional shape around an axis or a line.
Applications: On parts that require arc or circular cross sections.

Create
- **Surface**
- **Create Revolved Surfaces**
- [Select profile curve(s)]: Select Entity A as shown (in CCW direction).
- Select Entity B.



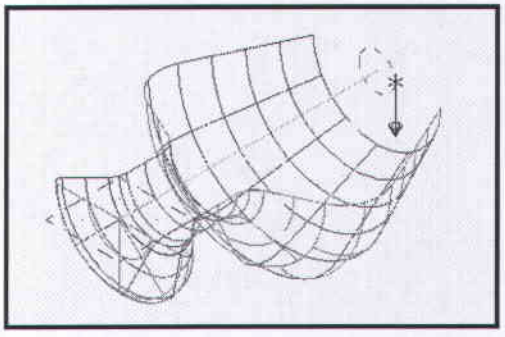


- Select the **OK** button to exit **Chaining**.
- [Select the axis of rotation]: Select Entity A.

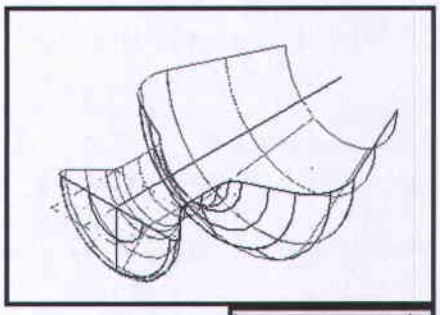


- Change the **Gview** to **Isometric**.
- Select the **Fit** button to fit the geometry to the screen.
  - Make sure that the arrow points as shown in the picture below.
  - Otherwise select the **Reverse** icon.

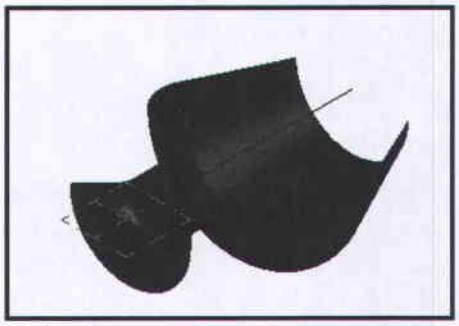
- Change the **End Angle** to 180. (Enter)
- Select the **OK** button.



➢ The surface should look as shown to the right.



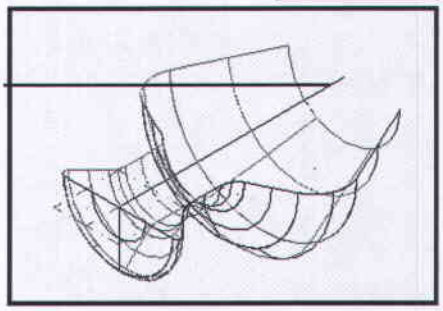
➢ To display the part in a shaded mode, select the **Shaded** icon.



➢ To return to the wireframe display, select the **Wireframe** icon.

➢ Delete the axis of rotation.
➢ Preselect the line as shown.
➢ Select the **Delete** entity icon.

Select the axis

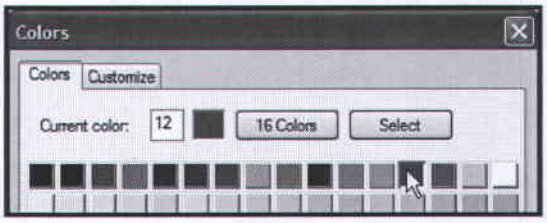


STEP 3:
CREATE THE EDGE CURVES.
3.1 Change the current color.

To be able to see the curves change the main color to red.

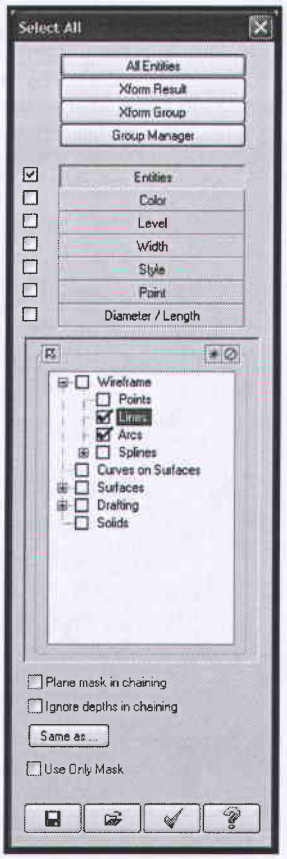
➢ Select **Color** from the **Status bar**.

➢ In **Current color** type 12 or click on the red color.



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

3.2 Mirror the initial wireframe.



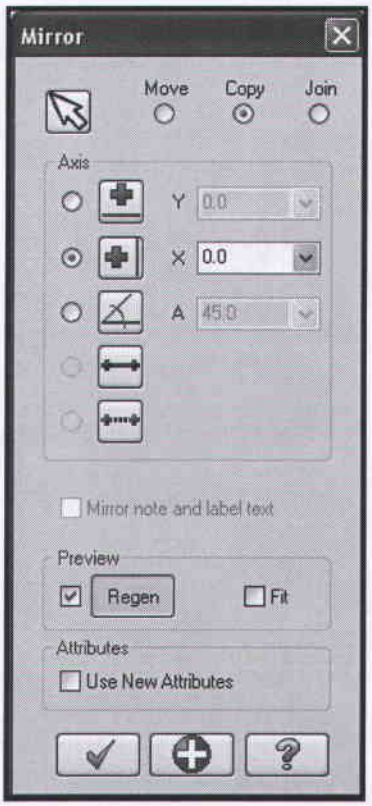
Xform
➢ **Xform Mirror**
➢ [Select entities to mirror]:
➢ Select **All** button. 

➢ Make sure that you enable only **Entities, Lines** and **Arcs**.
- Hold down the **Ctrl** key to be able to select only the entities that you need. To disable entities that were already selected click on them.

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

➢ Select the **End Selection** button. 

➢ Make the changes as shown in the **Mirror** dialog box.

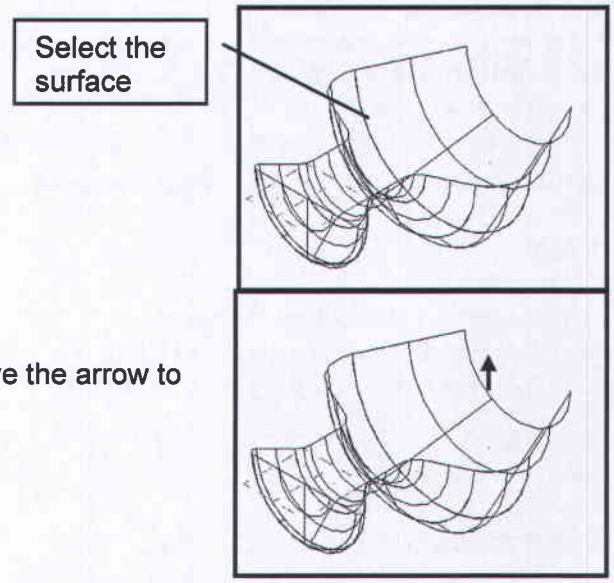


➢ Select the **OK** button to exit. 

Screen
➢ **Clear Colors**

3.3 Create curve on the edge of a surface.

Create
➢ **Curve**
➢ **Create Curve on One Edge**
➢ [Select a surface]: Select the revolve surface.



➢ [Move arrow to Desired Edge of Surface]: Move the arrow to the edge and select it as shown.

➢ Select the **OK** button

➢ Select the **Repaint** button from the toolbar, to be able to see the generated curve better.

STEP 4:
CREATE A LINE TO CLOSE THE BOUNDARY

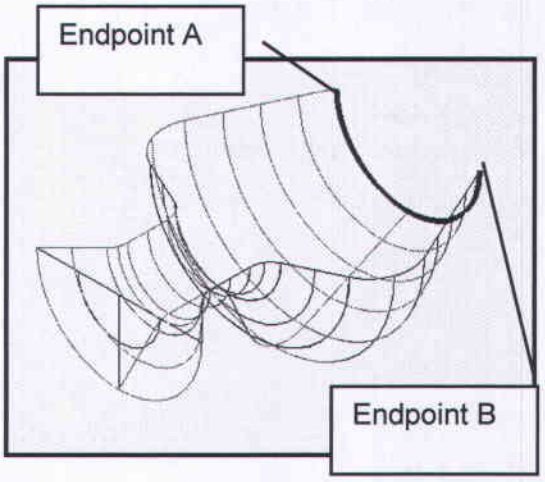
Create
➢ **Line**
➢ **Create Line Endpoint**

[Specify the first endpoint]: Select Endpoint A.

- Make sure that endpoint icon appears before you select the points.

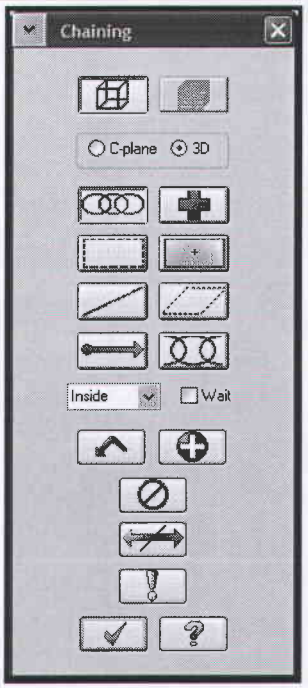
➢ [Specify the second endpoint]: Select Endpoint B.

➢ Select the **OK** button

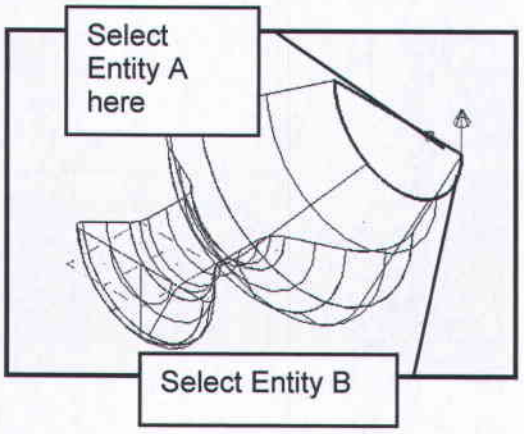


STEP 5:
CREATE A TRIM-FLAT BOUNDARY SURFACE.

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



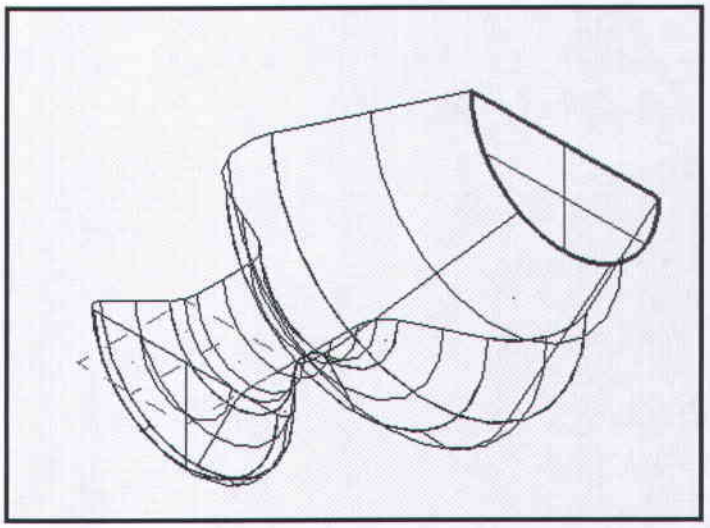
Create
- **Surface**
- **Create Flat Boundary Surface**
- [Select the first entity]: Select Entity A (in CCW direction).
- [Select the last entity]: Select Entity B.



- Select the **OK** button to exit **Chaining**.
- Select the **OK** button to exit surface flat boundary command.

STEP 6:
SAVE THE FILE.

File
- **Save As**
- **File name**: "Your Name_8"
- Select the **OK** button.

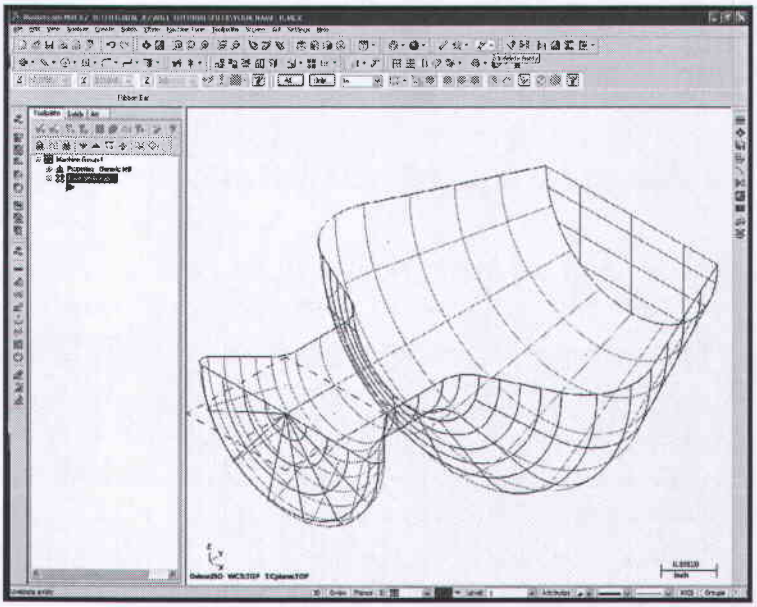


TOOLPATH CREATION
STEP 7:
SET UP THE STOCK TO BE MACHINED.

Machine type

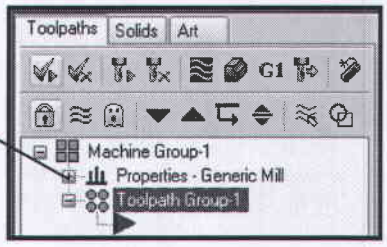
- **Mill**
- **Default**

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



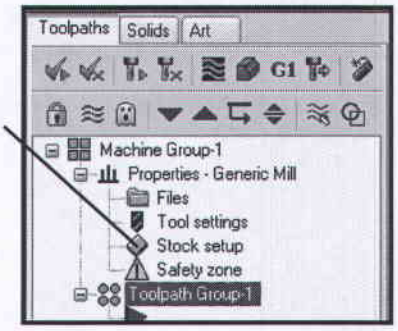
- Use the **Fit** icon to fit the drawing to the screen. 
- Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus



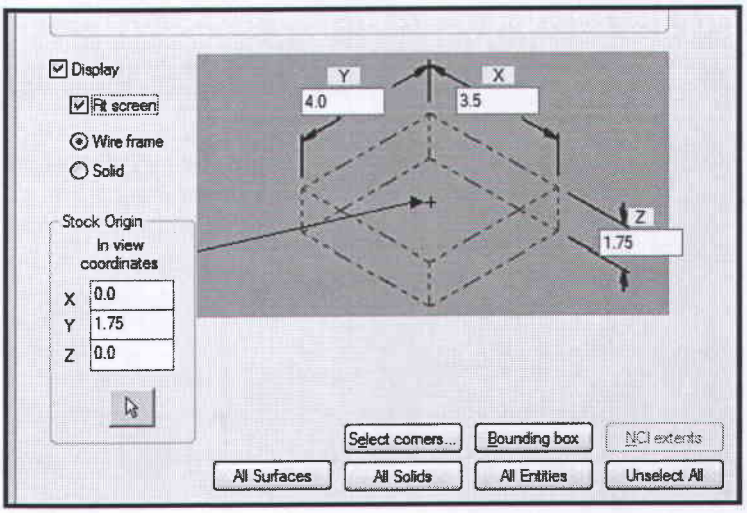
- Select **Stock setup**.

Select Stock setup



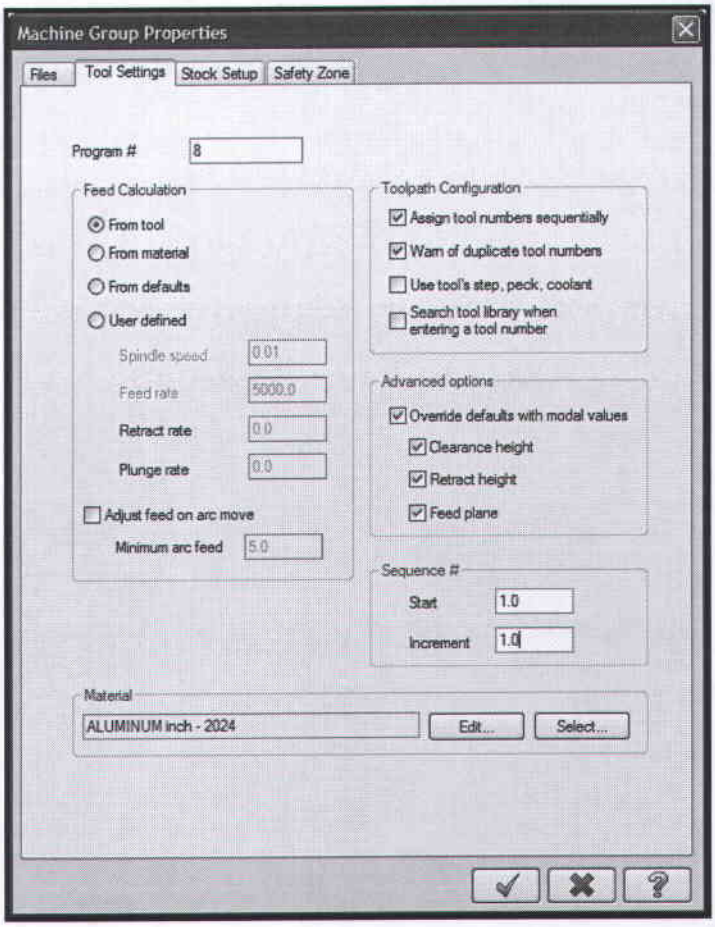
➢ Change the parameters to match the screenshot to the right.

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



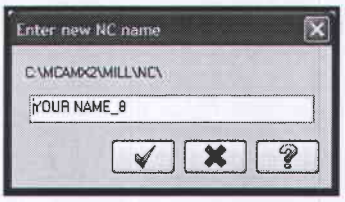
➢ Select the **Tool Settings** tab to set the tool parameters.
➢ Change the parameters to match the following screenshot.

Assign tool numbers sequentially allows you to overwrite the tool number from the library with the next available tool number. (First operation →tool number 1; Second operation →tool number 2, etc.)
Warn of duplicate tool numbers allows you to get a warning if you enter two tools with the same number.
Override defaults with modal values enables the system to keep the values that you enter.
Feed Calculation set **From tool** uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.



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

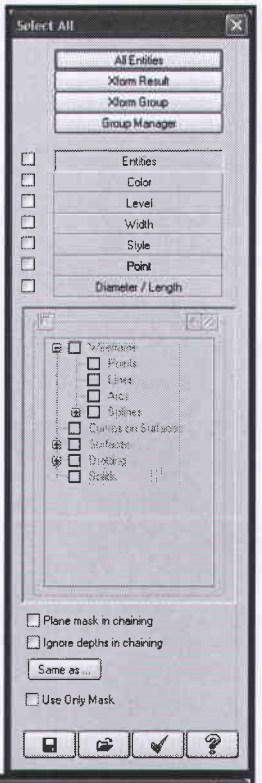
STEP 8:
ROUGH OUT THE SURFACE USING SURFACE POCKET.



Toolpaths
- **Surface Rough**
- **Rough Pocket Toolpath**
- Select the **OK** button to accept the NC name.

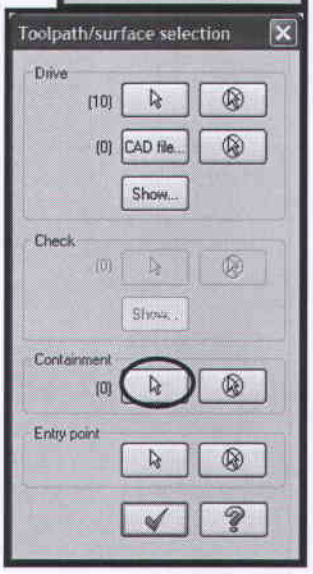
- [Select Drive Surfaces]: Select the **All** button. 

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

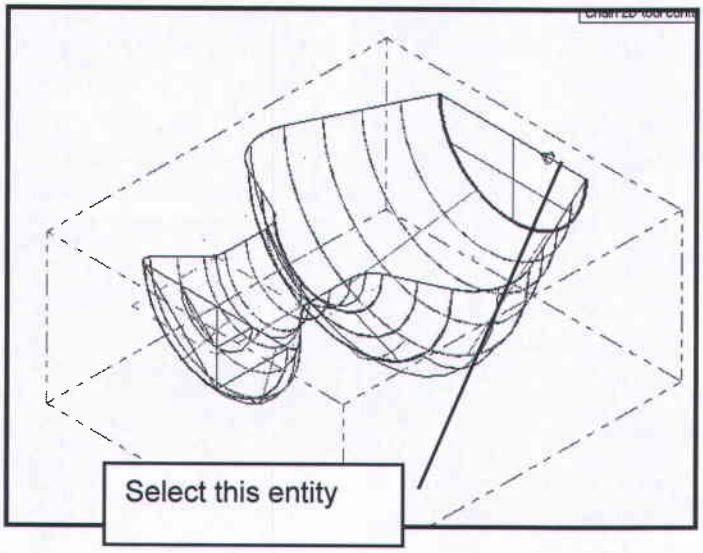


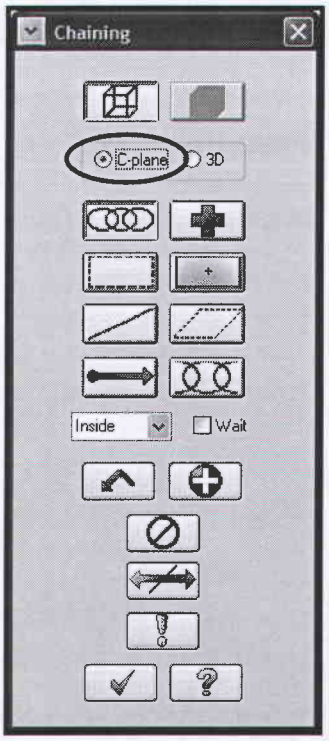
- Select the **End Selection** button. 

- Select the **Containment** button.

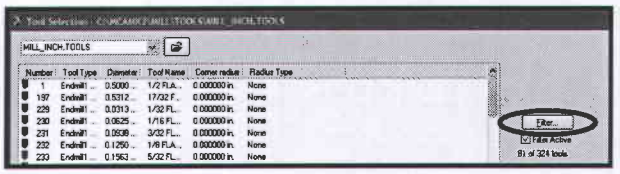


- Make sure that **C-plane** mode is enabled.
- Select an entity of the chain.



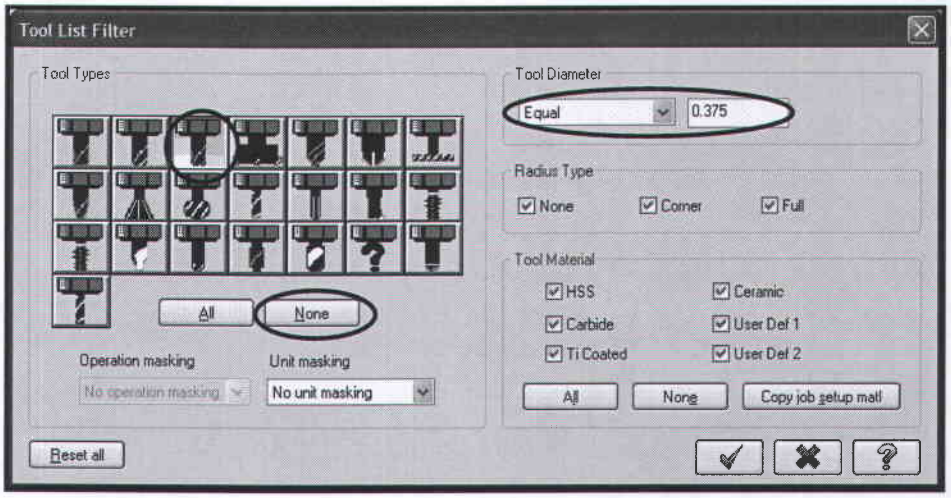


- Select the **OK** button to exit **Chaining**.
- Select the **OK** button to exit **Toolpath/surface selection**.
- Click on **Select library tool** in the **Toolpaths parameters** dialog box.
- Select the **Filter** button in the **Tool Selection** dialog box.



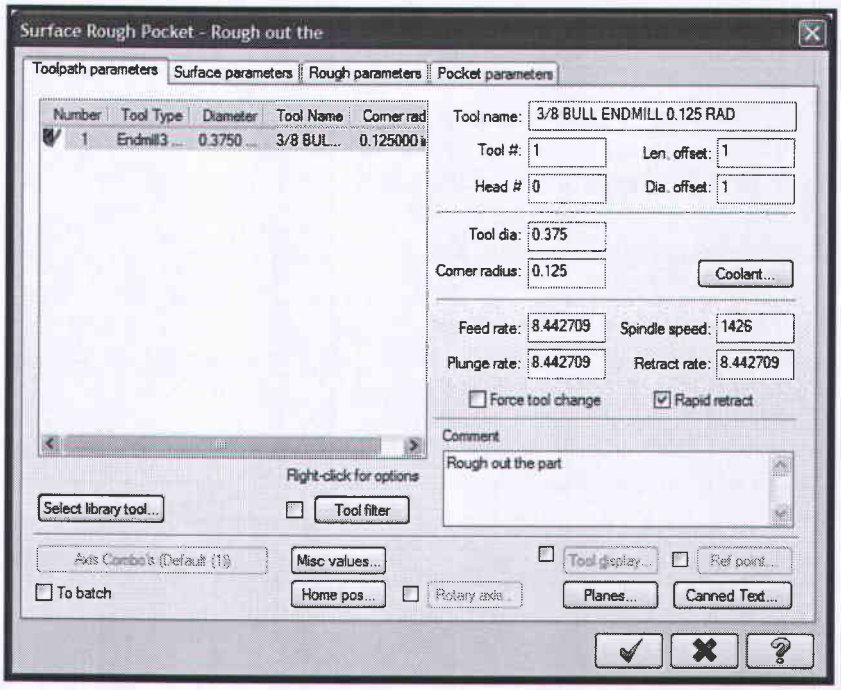
- Select the **None** button in the **Tool Types** area.
- Click on the **Endmill bull** icon to select it.

- Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.
- Enter the **Tool Diameter** 0.375.
- Select the **OK** button to exit.
- Select 0.375" Bull Endmill with the 0.125" corner radius in the **Tool Selection** page.



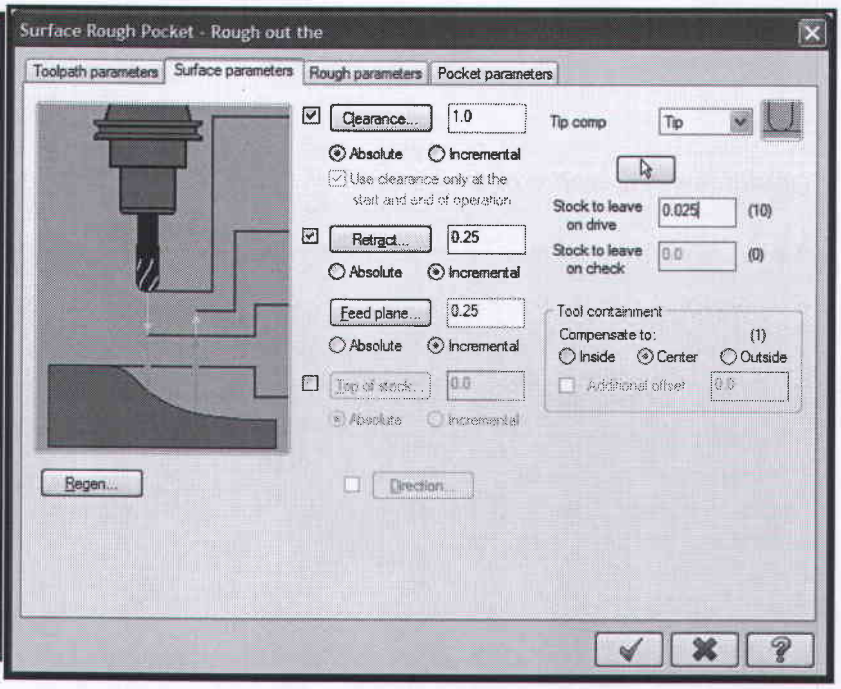
➢ Select the **OK** button to exit.

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



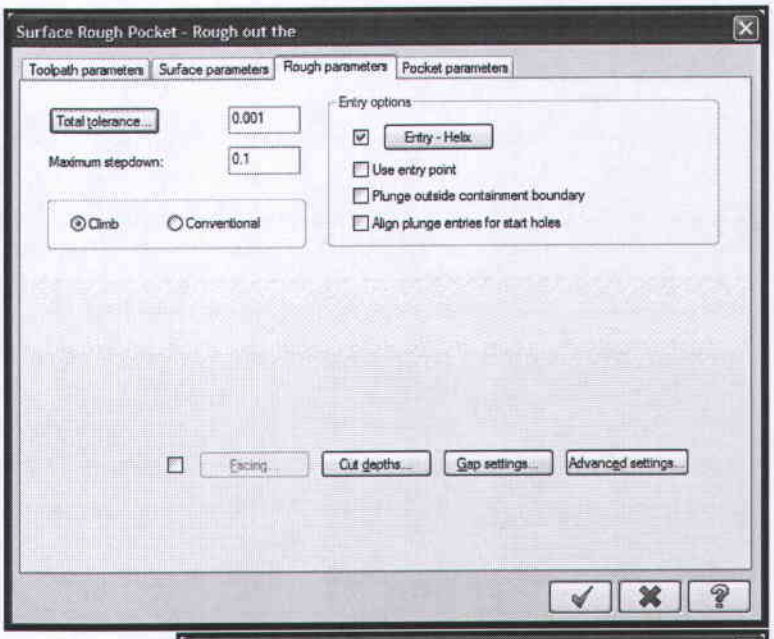
- The **Feed rate, Plunge rate, Retract rate** and **Spindle speed** are based on the tool definition. Change them as needed.

➢ Select the **Surface parameters** tab and make the changes 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 finish operation as a constant value all the way around the drive surfaces.

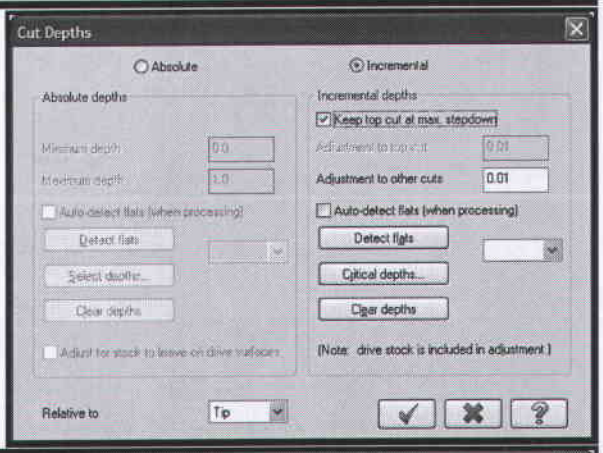
➢ Select the **Rough parameters** page and make the changes to match the following screenshot.



Total tolerance is the sum of the arc filter tolerance and cut tolerance. The cut tolerance determines the accuracy of the surface toolpath using chordal deviation.
Maximum stepdown value sets maximum distance (along the z-axis) between adjacent cuts in the surface toolpath.

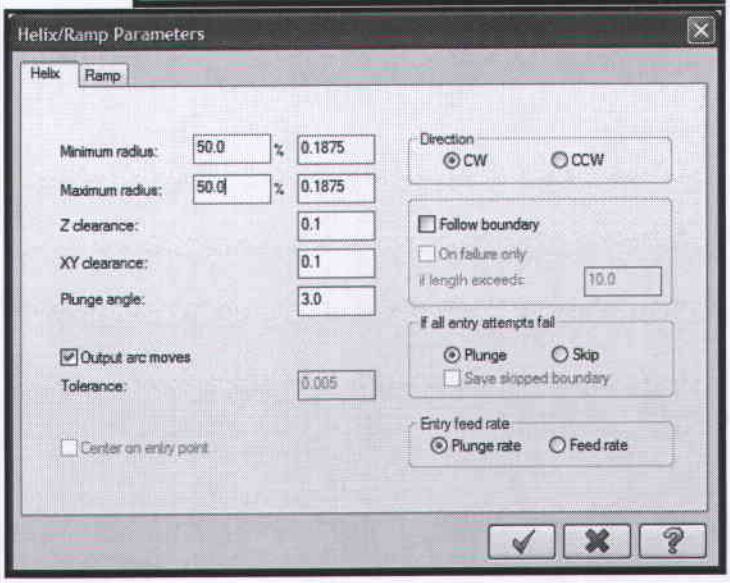
➢ Select the **Cut depths** button and enable Keep top cut at max stepdown to force the system to cut 0.1 deep at the first pass.

➢ Select the **OK** button to exit.

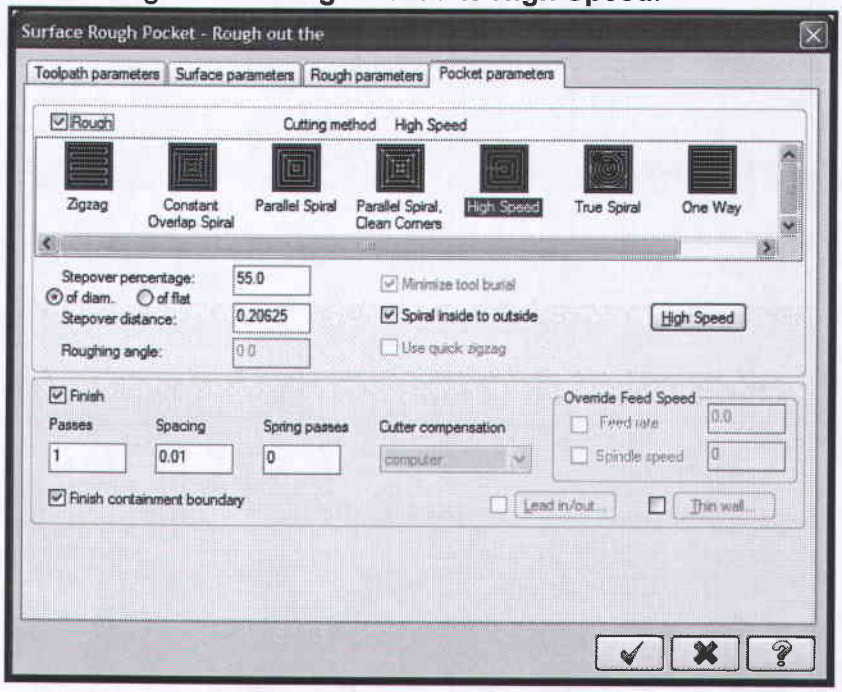


➢ Enable and select the **Entry-Helix** button and make any necessary changes.

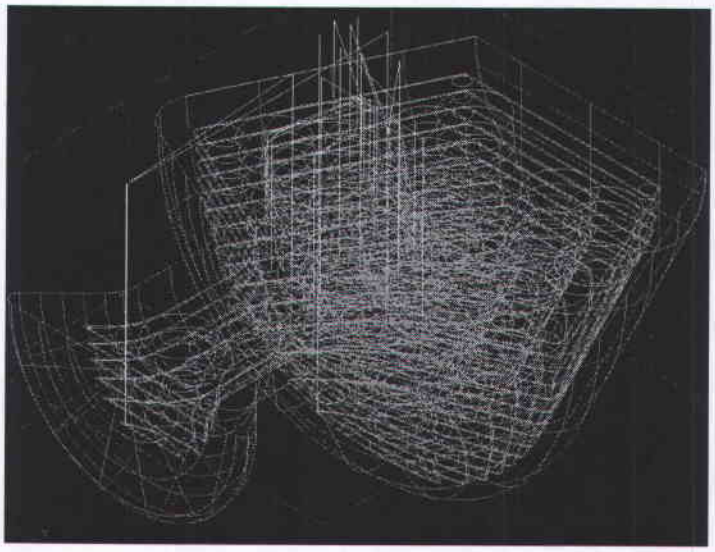
Entry-Helix sets the parameters for a helical entry into the part.
Minimum/Maximum radius value sets the minimum/maximum radius of the helix as a percentage of the tool diameter or as an absolute value.
Z clearance value sets the distance in the Z-axis above the top of the stock where the helix entry begins.
XY clearance sets the minimum distance in the XY axis between the helix and the pocket walls.
Plunge angle sets the helix angle.
Output arc moves instructs the system to output circular movements instead of linear movements.



- Select the **OK** button to exit **Entry-Helix**.
- Select the **Pocket parameters** and change the **Cutting method** to **High Speed**.



- Make sure that **Spiral inside to outside** is selected.
- Select the **OK** button to exit.

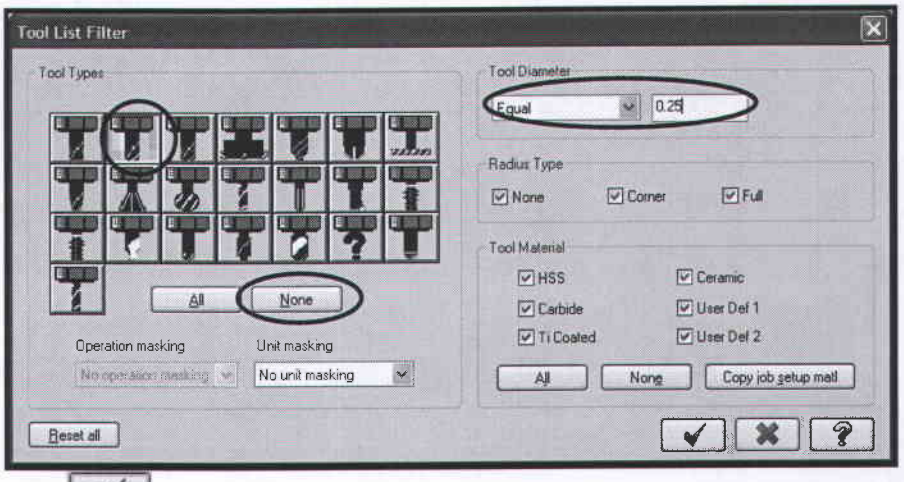


STEP 10:
FINISH THE SURFACE USING SURFACE FINISH PARALLEL.

Toolpaths
- **Surface Finish**
- **Finish Parallel Toolpath**
- [Select Drive Surface]: Select the **All** button. 

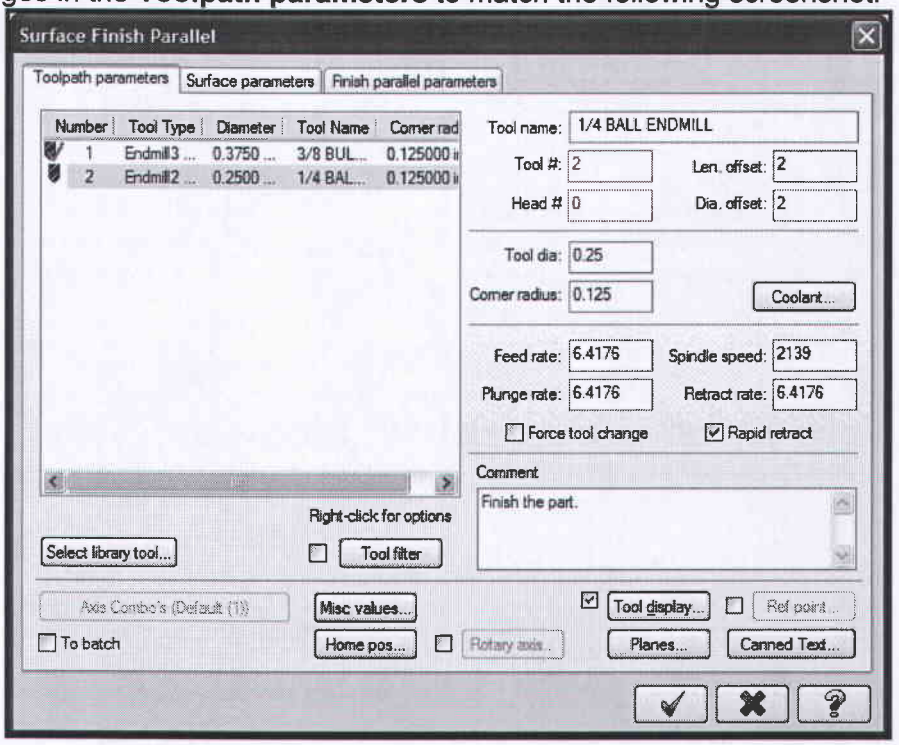
- Select the **OK** button to exit. 
- Select the **End Selection** button. 

- Select the **OK** button in the **Toolpath/surface selection** dialog box as we don't need any chain as a boundary for this toolpath. 
- Click on **Select library tool** in the **Toolpaths parameters** dialog box.
- Select the **Filter** button in the **Tool Selection** dialog box.
- Select the **None** button in the **Tool Types** area.
- Click on the **Endmill Sphere** icon to select the tool type.
- Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.
- Enter the **Tool Diameter** 0.250.

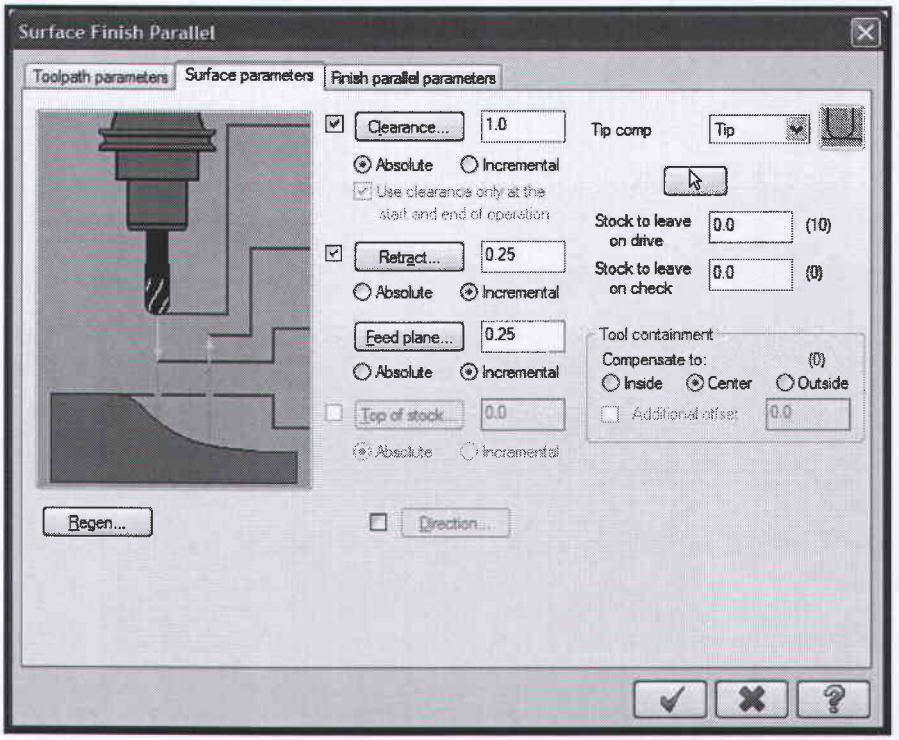


- Select the **OK** button to exit. 
- Select the **OK** button to exit the **Tool Selection** page. 

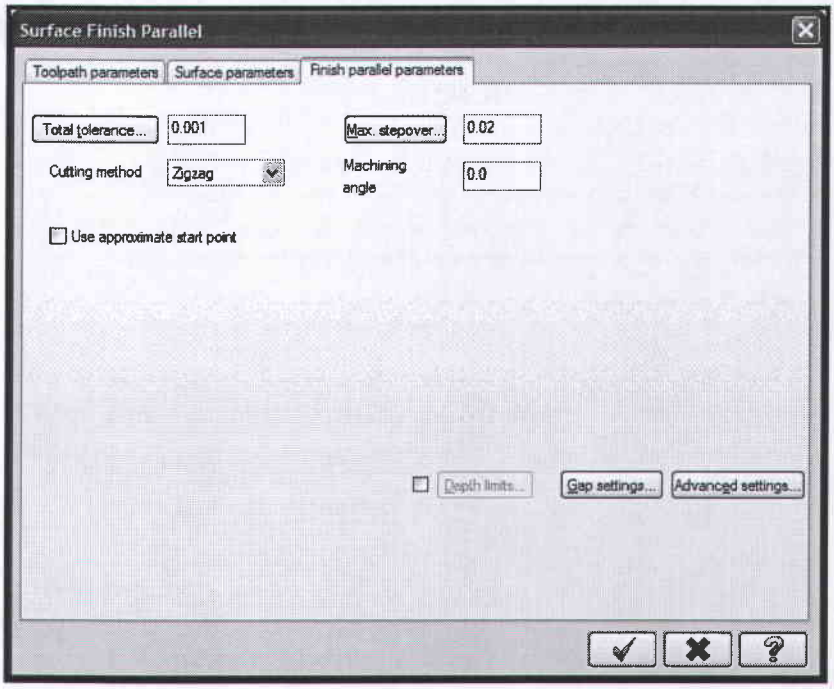
➢ Make the necessary changes in the **Toolpath parameters** to match the following screenshot.



➢ Change the parameters as shown in the following screenshots.



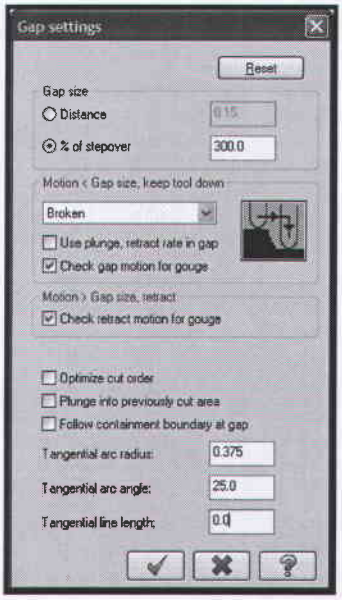
➢ Select **Finish parallel parameters** and make any necessary changes.



Total tolerance is the sum of the arc filter tolerance and cut tolerance. The cut tolerance determines the accuracy of the surface toolpath using chordal deviation.
Maximum stepover value sets the size of the step between XY cuts in a surface toolpath. A smaller value creates a more accurate toolpath, but it also creates a longer NC program.

➢ Select the **Gap settings...** button.

Gap parameters set the way the tool moves between gaps or spaces in a surface toolpath.
Smooth parameter enables a smooth transition between gaps if the tool motion is smaller than the gap size.
Each surface toolpath gap has an entry and an exit point. To create a smoother tool motion between gaps, place **tangential arcs** at these points.



➢ Select the **OK** button to exit **Gap Settings**.

➢ Select the **OK** button to exit the parameter screen.

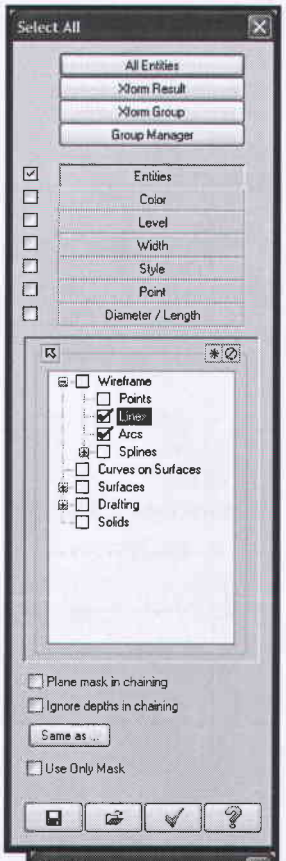
STEP 11:
FINISH THE STEEP WALLS USING SURFACE CONTOUR TOOLPATH.

- Before starting the toolpath we should create two contours inside of which the contour toolpath will be contained.
- Select all operations and enter Alt +T to remove the toolpath display.

11.1 Project the existing contour 1" above the part.

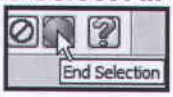
Xform
➢ **Xform Project**

➢ [Select entities to project]: Select the **All** button. 



➢ Make sure that you enable only **Entities, Lines** and **Arcs**.
- Hold down the **Ctrl** key to be able to select only the entities that you need. To disable entities that were already selected click on them.

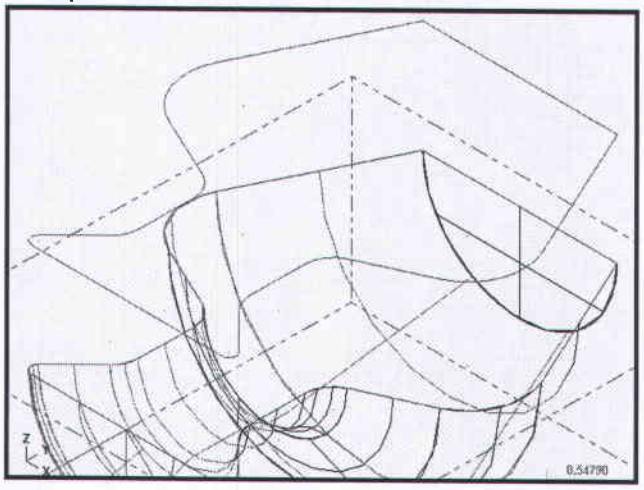
➢ Select the **OK** button to exit the **Select All** dialog box. 

➢ Select **End Selection** button. 

- Make sure that the lines are selected, not the surfaces.

➢ Make the changes in the **Project** dialog box to match the screenshot to the left.

➢ The part should look as shown below.

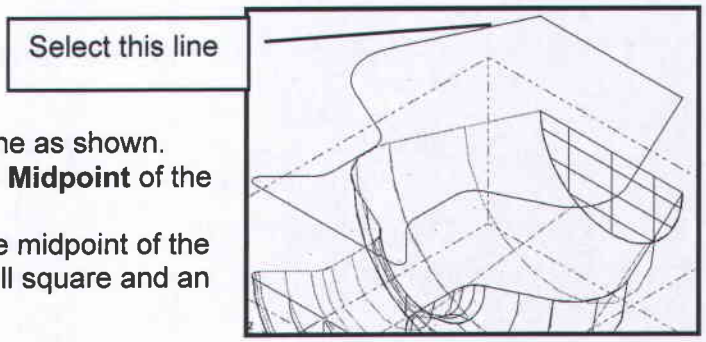


Select the **OK** button to exit **Project**.

➢ Select **Fit** icon.

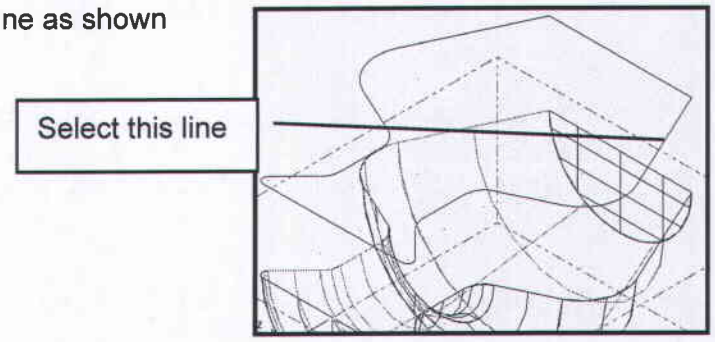
11.2 BREAK TWO LINES FROM THE PROJECTED CURVE.

Edit
➢ **Trim/Break**
➢ **Break Two Pieces**



➢ [Select an entity to break]: Select the line as shown.
➢ [Indicate the break position]: Select the **Midpoint** of the line.
- By moving the cursor exactly on the midpoint of the entity the system will display a small square and an icon.

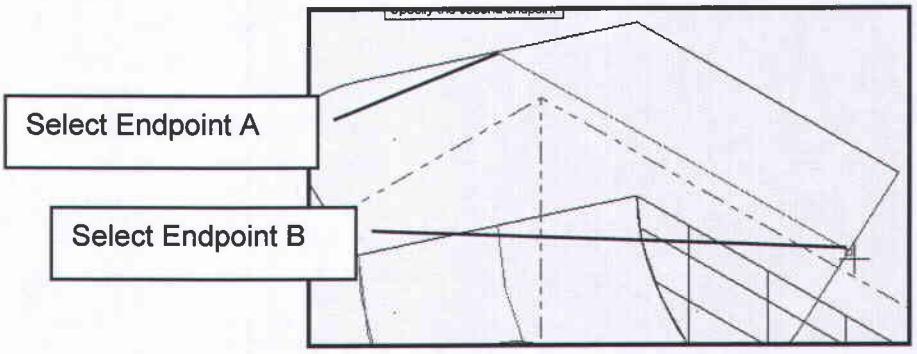
➢ [Select an entity to break]: Select the line as shown below
➢ [Indicate the break position]: Select the **Midpoint** of the line.
➢ Select the **OK** button.



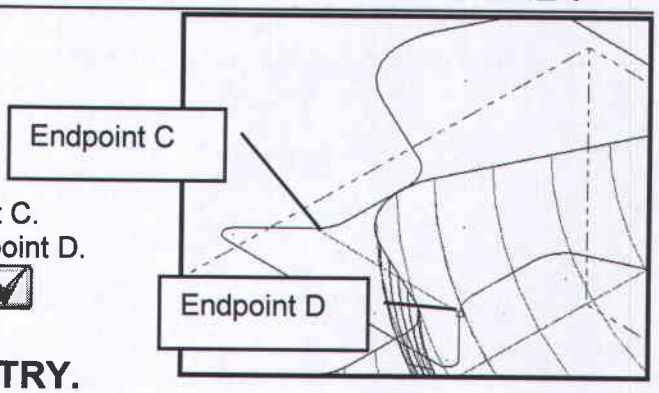
11.2 Create two lines.

Create
➢ **Line**
➢ **Create Line Endpoint**
➢ [Specify the first endpoint]: Select Endpoint A.
➢ [Specify the second endpoint]: Select Endpoint B.

By moving the cursor exactly on the endpoint of the entity the system will display a small square and an icon.



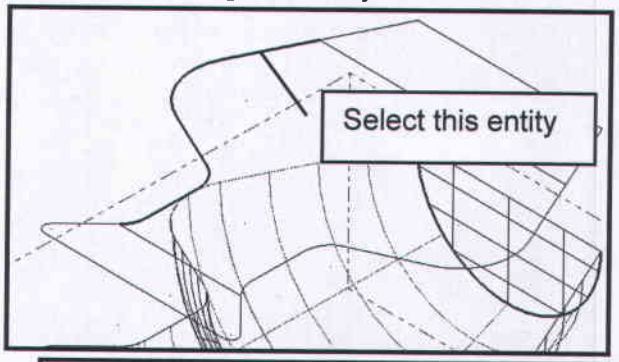
➢ Select the **Apply** button to continue.



➢ [Specify the first endpoint]: Select Endpoint C.
➢ [Specify the second endpoint]: Select Endpoint D.
➢ Select the **OK** button to exit the command.

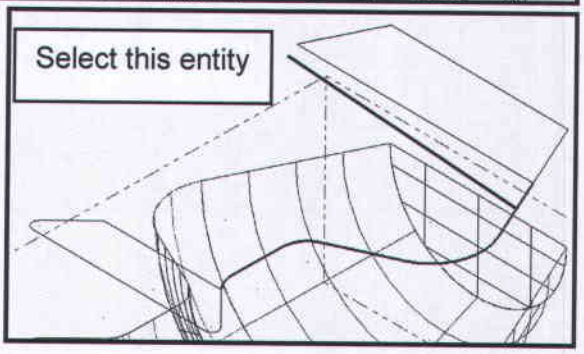
11.3 DELETE THE EXTRA GEOMETRY.

➢ Preselect the entities by holding down the **Shift** key and selecting one entity as shown.
  - The system will highlight the entities selected as shown in the picture to the right.



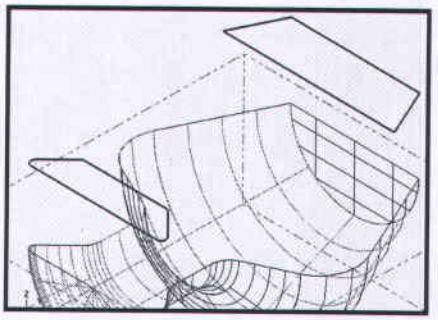
➢ Select the **Delete** entity icon.

➢ Preselect the entities by holding down the **Shift** key and selecting one entity as shown.
  - The system will highlight the entities selected, as shown in the picture to the right.



➢ Select the **Delete** entity icon.

➢ The contours should look as shown to the right.



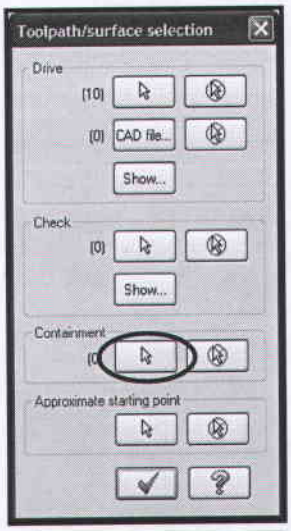
Toolpaths
➢ **Surface Finish**

➢ **Finish Contour Toolpath**

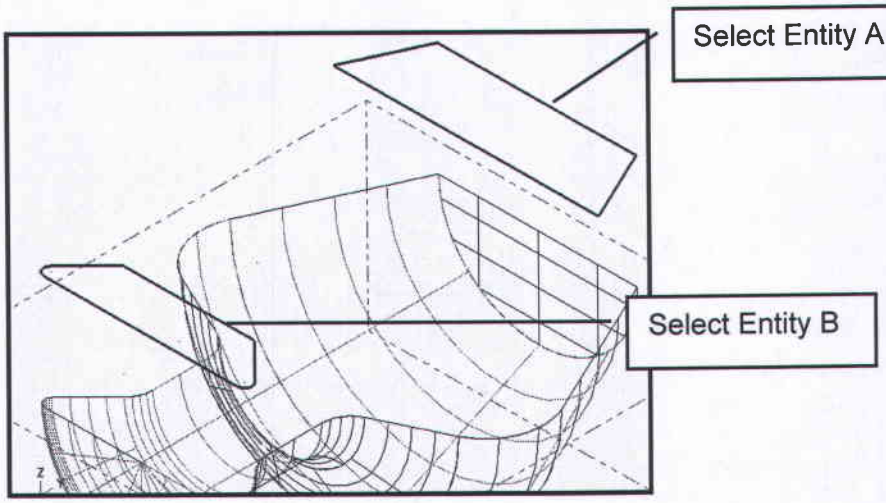
➢ [Select Drive Surface]: Select the **All** button.
➢ Select the **OK** button to exit.

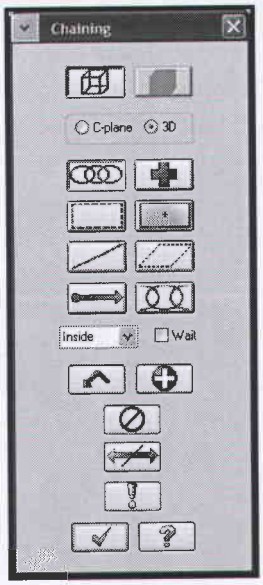
Select the **End Selection** button.

➢ Select the **Containment** button as shown to the right.



➢ [Chain 2D tool containment boundary #1]: Select Entity A as shown.
➢ [Chain 2D tool containment boundary #2]: Select Entity B as shown.





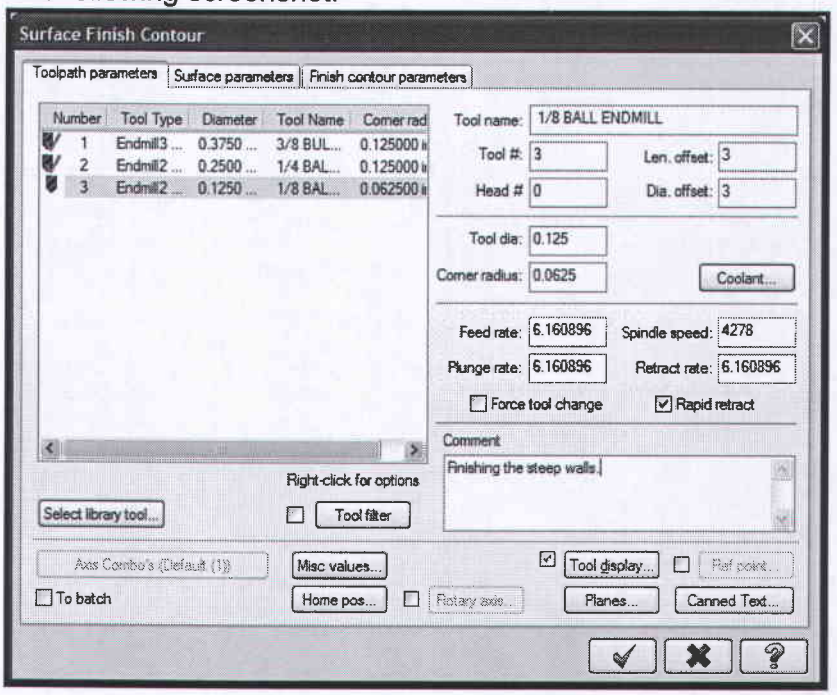
➢ Select the **OK** button to exit the **Chaining** dialog box.

➢ Select the **OK** button to exit the **Toolpath/surface selection** dialog box.

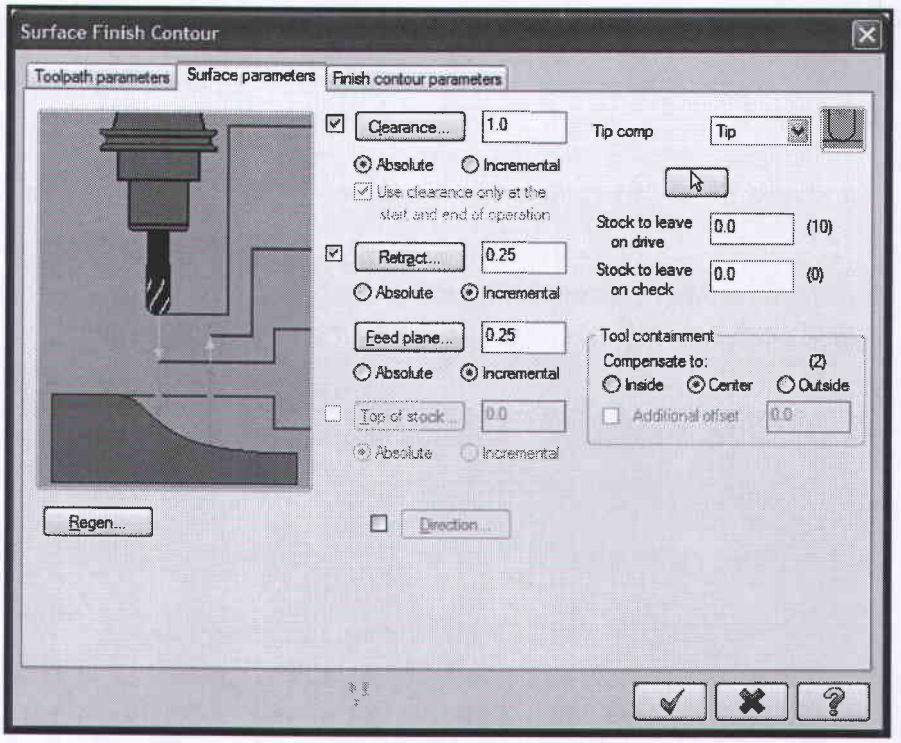
➢ Click on **Select library tool** in the **Toolpaths parameters** dialog box.

➢ Following the steps outlined earlier, using the **Filter** button in the **Tool Selection** dialog box, select the **1/8 Ball Endmill.**

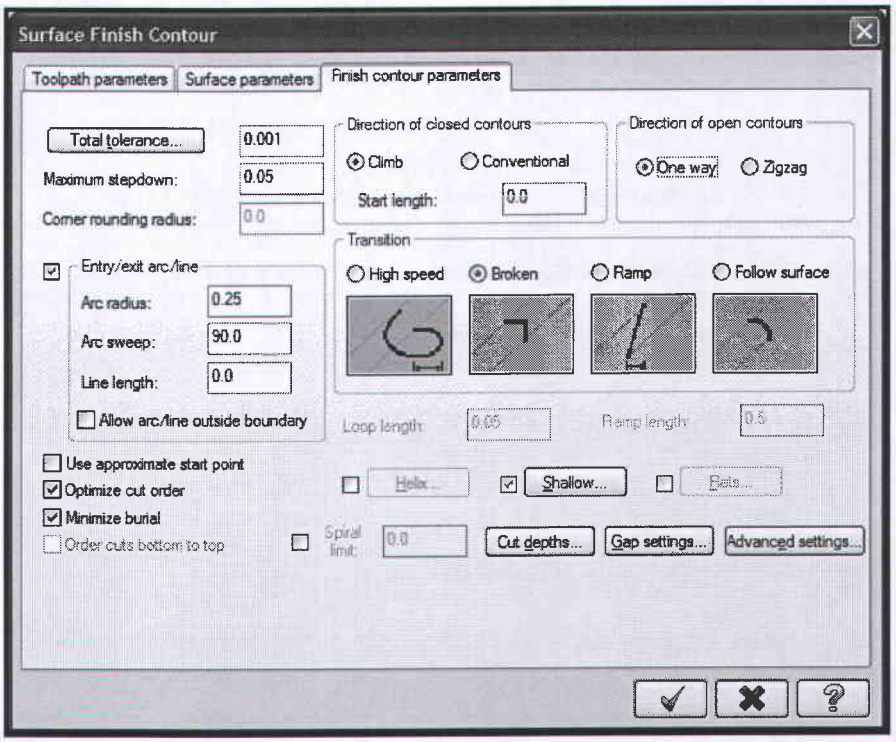
➢ Make the changes as shown in the following screenshot.



➢ Select the **Surface parameters** page and match the parameters with the following screenshot.



➢ Select the **Finish contour parameters** page to set the cutting method, the total tolerance and the maximum stepdown.



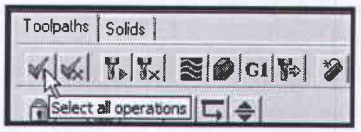
Total tolerance is the sum of the arc filter tolerance and cut tolerance. The cut tolerance determines the accuracy of the surface toolpath using chordal deviation.
Maximum stedown value sets the size of the step along the Z-axis in a surface toolpath. A smaller value creates a more accurate toolpath, but it also creates a longer NC program.

➢ Select the **Shallow** button to set the parameters necessary for the system to add cuts to shallow areas in a surface finish contour (constant Z) toolpath.

➢ Select the **OK** button to exit the **Contour Shallow** dialog box.

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

STEP 12:
BACKPLOT THE TOOLPATH.

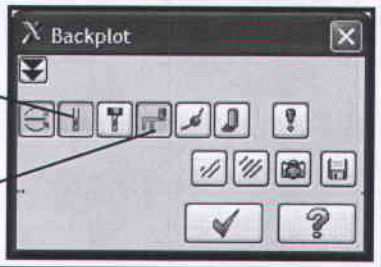


➢ Click on the **Toolpaths** tab to enable **Toolpaths Manager**.
➢ Select the **Select all operations** icon to select all operations.

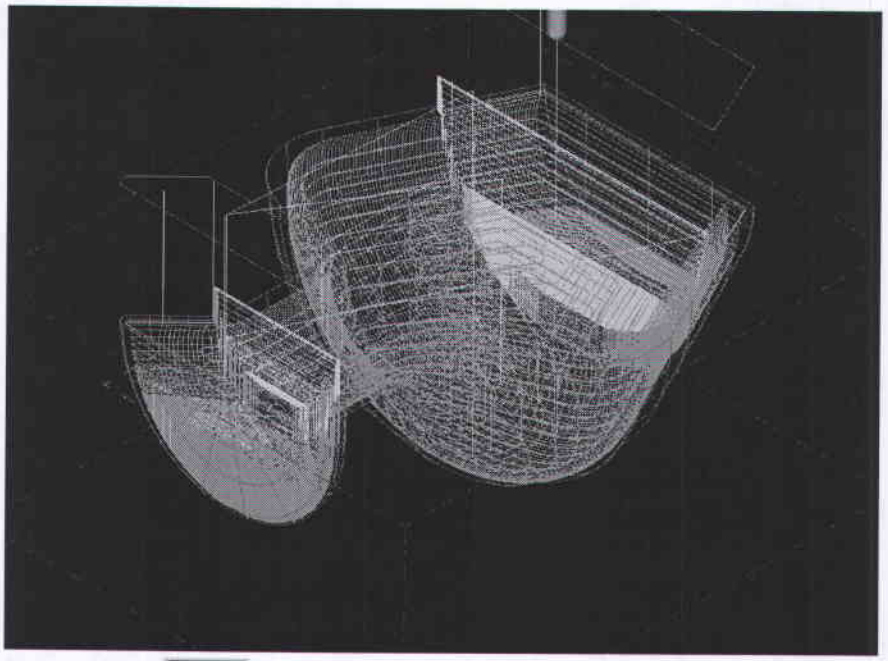
➢ Select the **Backplot** selected operations button.

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

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



➢ Select the **Play** button.



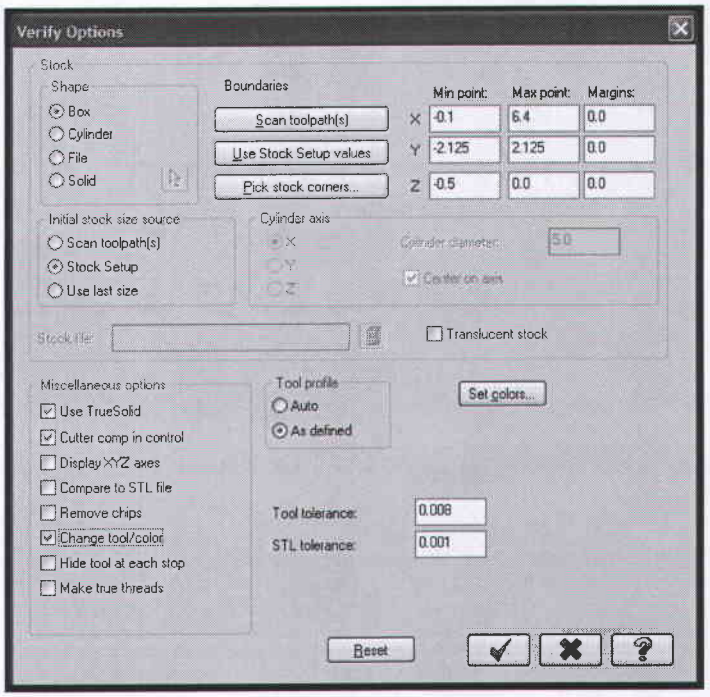
➢ Select the **OK** button to exit **Backplot**. 

VERIFY-TOOLPATH VERIFICATION
STEP 13:
VERIFY.

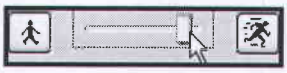
- Expand the **Toolpaths Manager** if necessary by dragging the right side.
- Select the **Verify selected operations** button.

- Select the **Configure** button.
- Make the changes as shown.

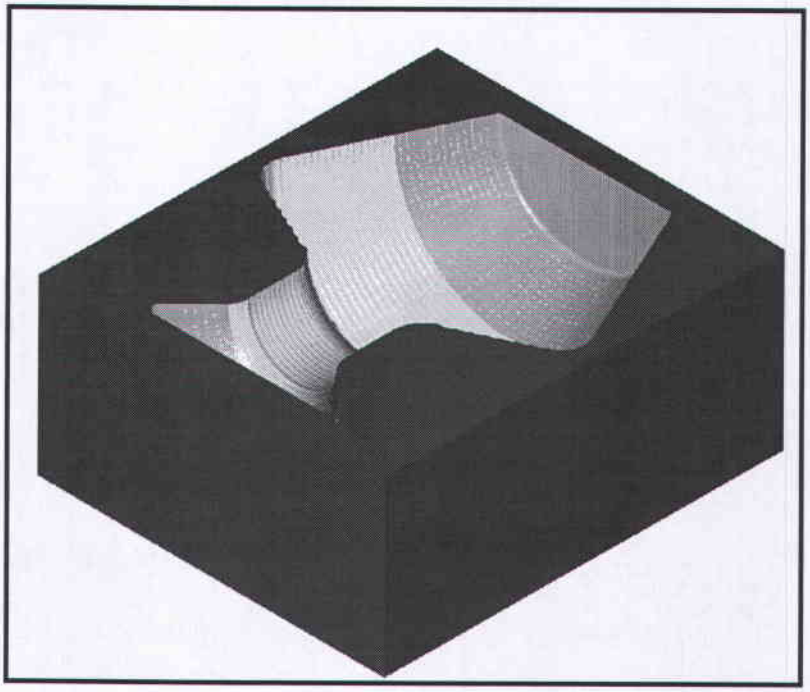
Initial stock size source should be set to **Job Setup** to use the stock information from Stock Setup.
Use True Solid allows you, after verifying the part, to rotate and magnify it to more closely check features, surface finish, or scallops.
Cutter comp in control allows Verify to use the information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



- Select the **OK** button to exit **Verify Options**.
- Set the **Verify speed** by moving the slider bar in the speed control bar.



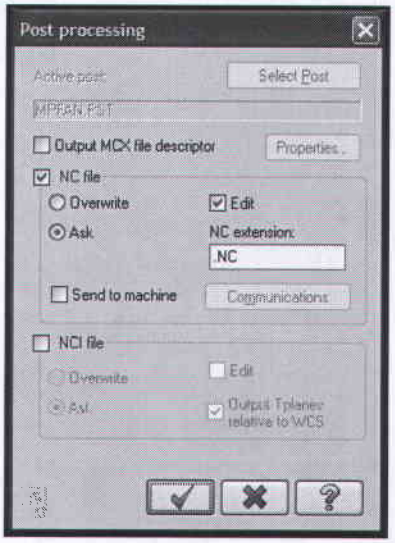
- Select the **Play** button to start simulation.
- The finished part should appear as shown in the picture to the right.



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

STEP 14:
POST PROCESS THE FILE.

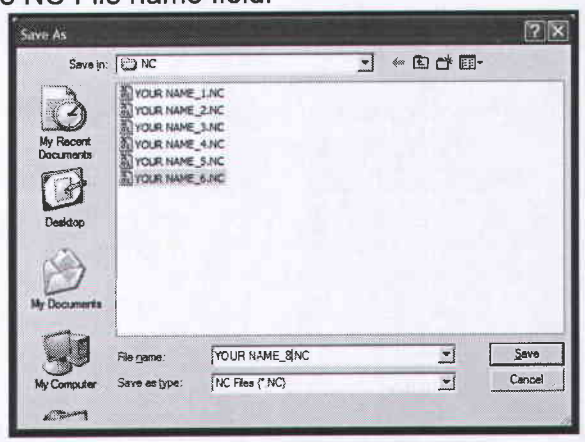
- Select the **Post selected operations** button from **Toolpath Manager.**
- In the **Post processing** window, make all the necessary changes as shown to the left.

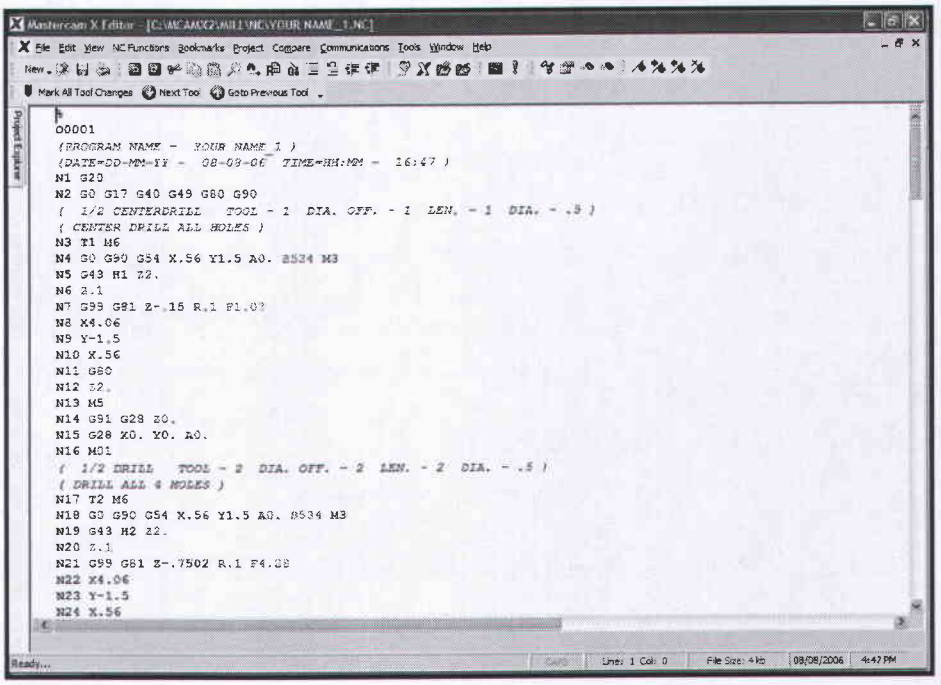


> **NC file** enabled allows you to keep the NC file and to assign the same name as the MCX file
> **Edit** enabled allows you to automatically launch the default editor.

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

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



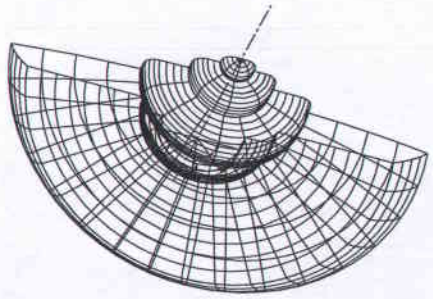


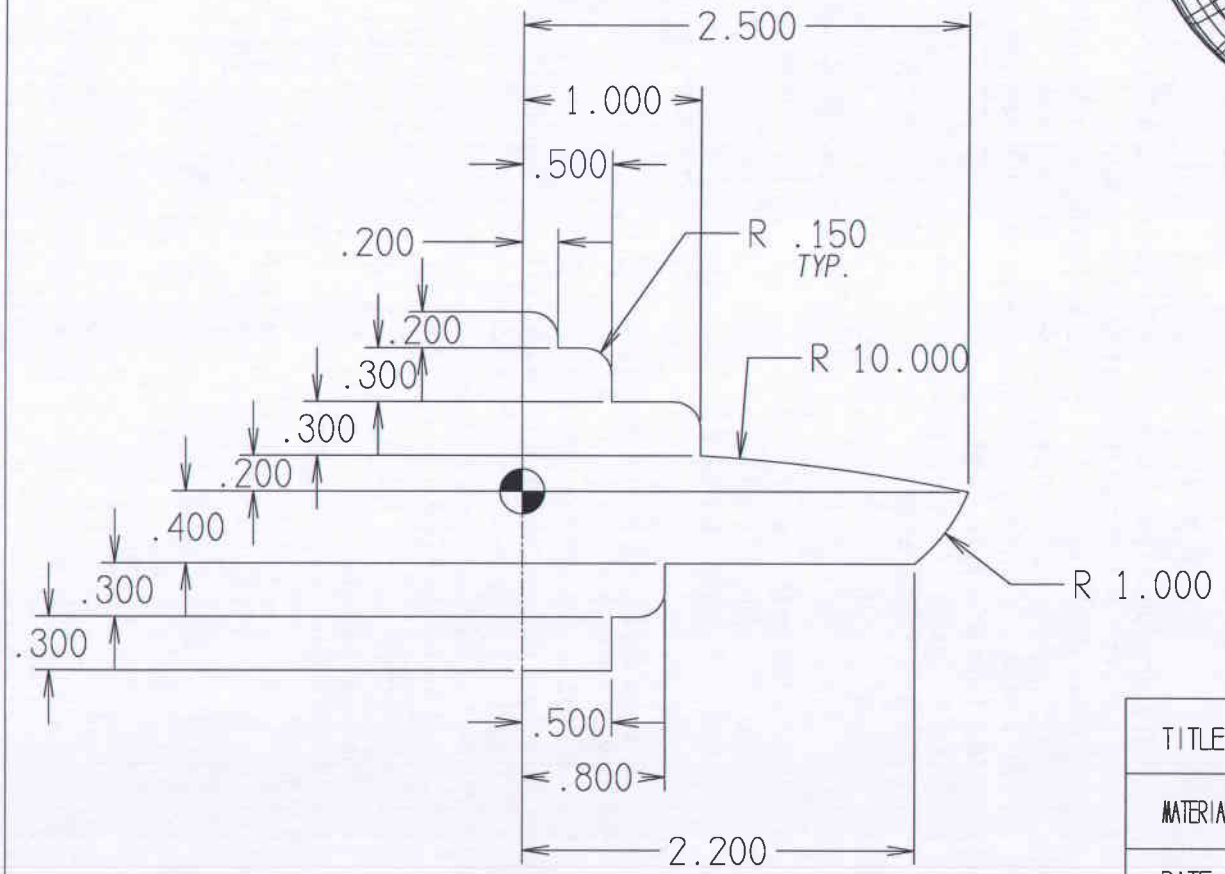
➢ Select the red **X** box at the upper right corner to exit the **Editor**.

STEP 15:
SAVE THE UPDATED MCX FILE.

➢ Select the **Save** icon.

ALL DIMENSIONS IN INCHES

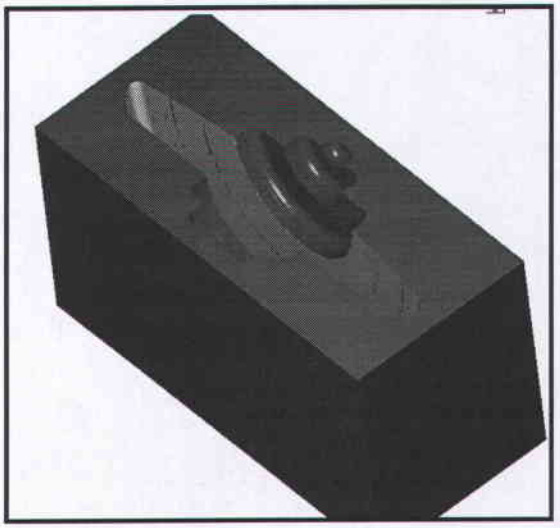




TITLE	TUTORIAL 8 - EXERCISE
MATERIAL	ALUMINUM T6061
DATE: JUNE 12, 2000	eMastercam.com

REVIEW EXERCISE.

Student practise. Create the Toolpath for Exercise-Tutorial 8 as per the instructions below;



Tips:

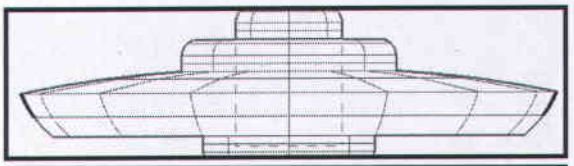
Mirror the top 2d geometry about the axis of rotation to have a closed contour for tool containment in **Surface Rough Pocket** toolpath

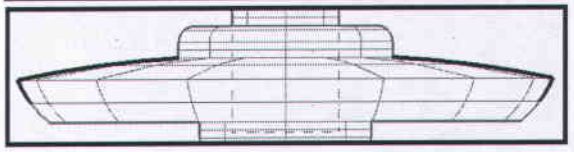
1. To establish the **stock** size select Bounding box/All/Surfaces/ and change X =5.5", Y = 2.5", give z = 3" (The Stock Origin X= 0, Y=0, Z= 0)

2. Create a boundary around the area that will need extra finish.

Edit/Trim/Break/Trim Break

Enable **Break** button and break the arcs as shown at the midpoint.





Xform/Xform Project

Select the geometry as shown

Set the Project parameters to **Copy** at a

Height =0.5

Create/Line/Create Line Endpoints

Close the contour

3. Surface Rough Pocket

Select the 2d geometry at the edges of the surfaces.

Use 1/2" Bull Nose Mill with a 0.125 corner radius

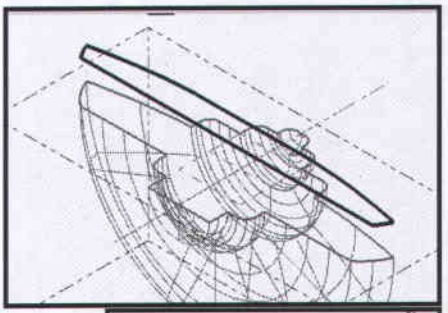
Stock to leave on drive surfaces= 0.03"

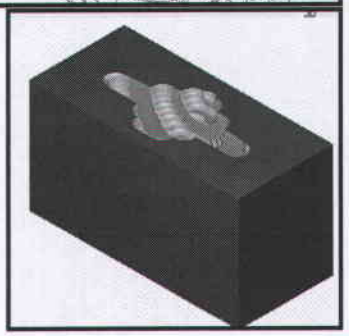
Total tolerance = 0.005

Entry-ramp

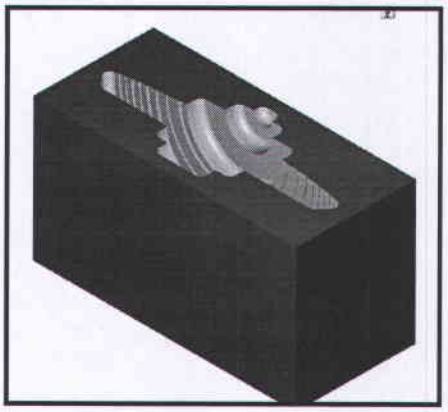
Use quick zigzag

Finish containment boundary

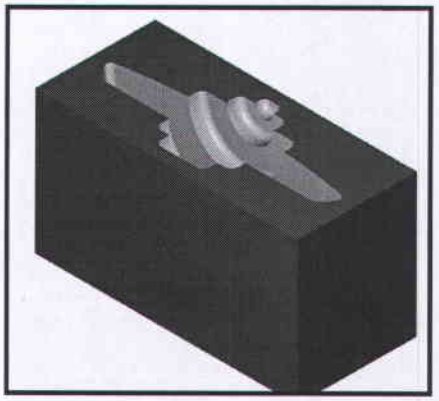




4. Surface Finish Parallel
Use 1/4" Ball End Mill
Stock to leave on drive surfaces= 0
Feed plane = 0.25
Total tolerance = 0.001
Max stepover = 0.02



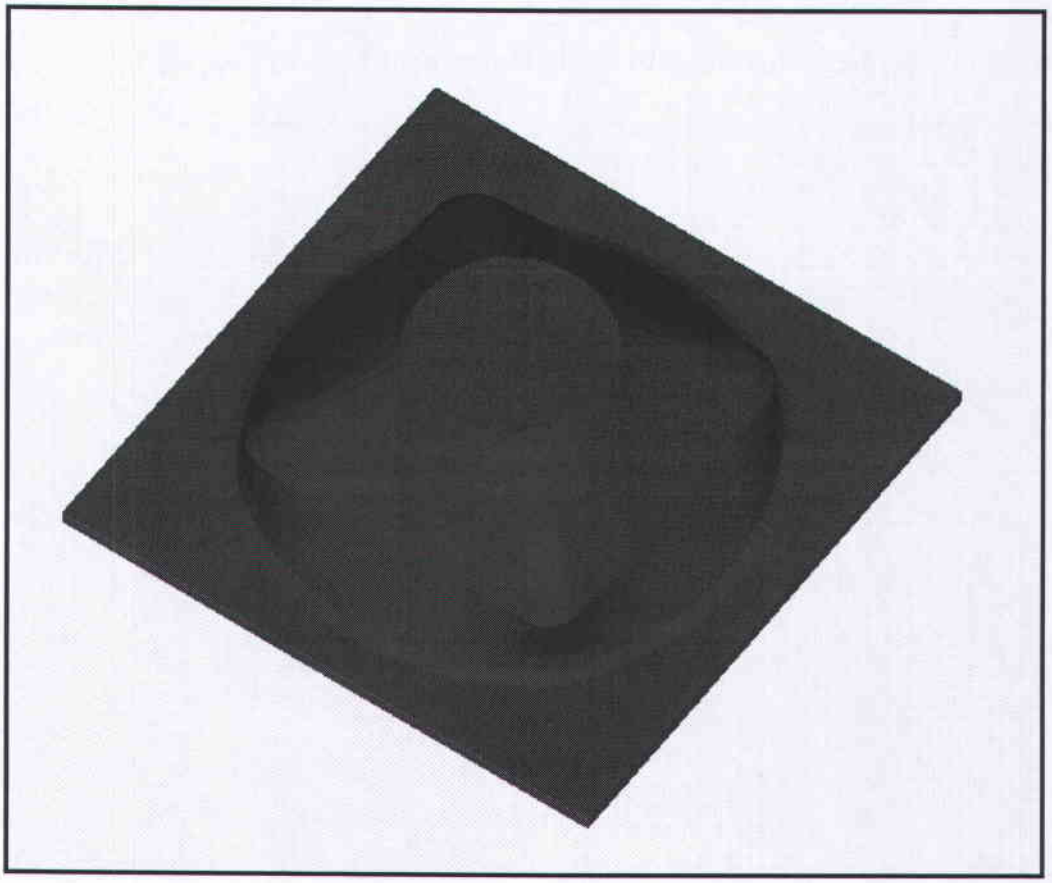
5. Surface Finish Contour
Select projected boundary as **Containment Boundary**
Use 3/16" Ball End Mill
Stock to leave on drive surfaces= 0
Total tolerance = .001
Max stepdown = .025
Enable **Shallow** and **Add cuts to shallow areas** and disable **Allow partial cuts.**
6. Backplot and Verify the toolpaths.
7. Post process the file.



TUTORIAL SERIES FOR

Mastercam X²

TUTORIAL 9
RULED SURFACE, TRIM-FLAT BOUNDARY SURFACE, RADIAL & SCALLOP TOOLPATHS.



Objectives:

The Student will design a 3-dimensional drawing by:

- Creating arcs using co-ordinate positioning.
- Creating rectangles using co-ordinate positioning.
- Creating fillet radii.
- Changing the construction depth of the drawing entities.
- Creating a ruled surface.
- Creating a trimmed flat boundary surface.
- Changing the view of the part for better visualisation.

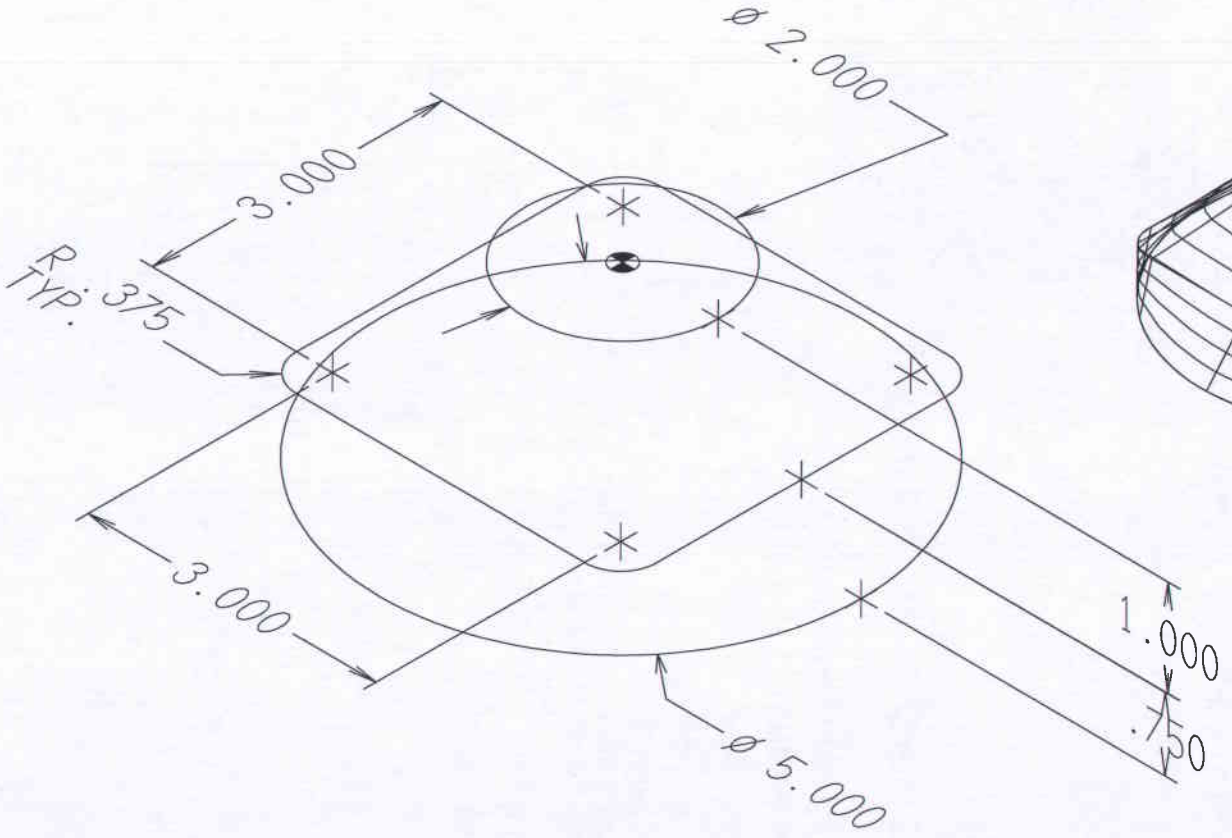
The Student will create a 3-dimensional milling toolpath consisting of:

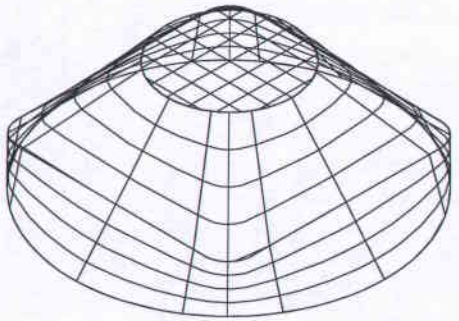
- A 3-dimensional rough toolpath.
- A 3-dimensional finish toolpath.

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

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

ALL DIMENSIONS IN INCHES





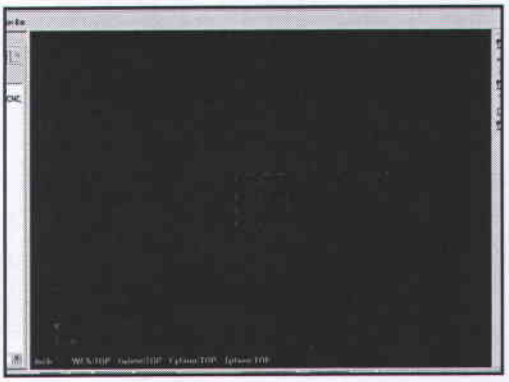
TITLE	TUTORIAL 9
MATERIAL	ALUMINUM T2024
DATE: JUNE 12, 2000	eMastercam.com

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

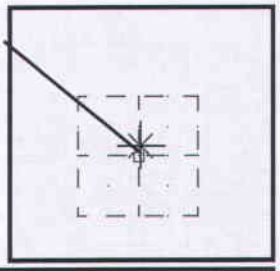


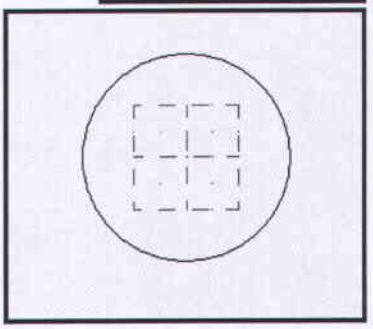
STEP 1:
CREATE THE WIREFRAME GEOMETRY.

1.1 Create an arc knowing the center point and the radius.

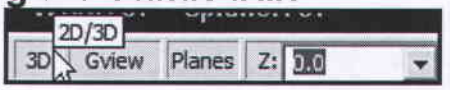
Create
➢ **Arc**
➢ **Create Circle Center Point**
➢ [Enter the center point]: Select the **Origin** by moving the cursor at the center of the grid.
➢ Enter the **Radius** value 1.0. (Enter)
➢ Select the **OK** button.

Select the Origin





1.2 Change the C-plane to 2D using the Status bar.



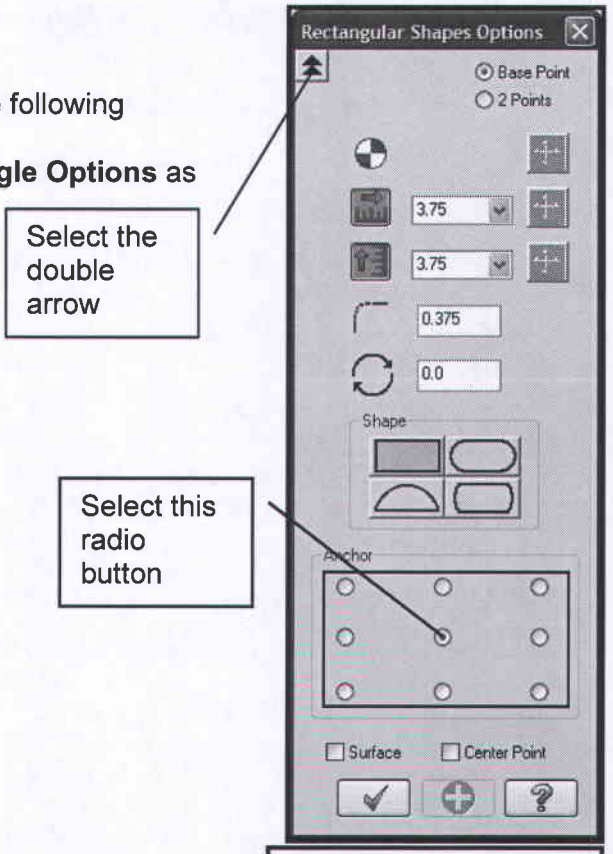
➢ Select **3D** in the **Status bar** to toggle to **2D**.

1.3 Change the current Z Depth to –1.0 using the Status bar.

➢ Click on **Z**: value in the **Status bar**.
➢ Type -1.0 (Enter).

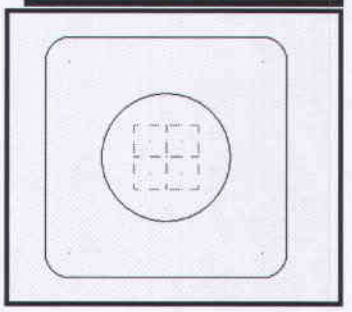
1.4 Create a rectangle with fillets at the corners.

Create
➢ **Create Rectangular Shapes**
➢ Select the radio button in front of **Base Point**.
➢ Type the **Width** and the **Height** as shown in the following screenshot.
➢ Select the **double arrow** to expand the **Rectangle Options** as shown.



➢ Enter the fillet **radius** 0.375.

➢ Select the middle radio button as the anchor.



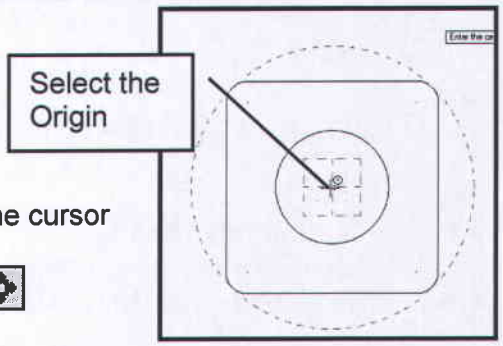
➢ [Select position for the base point]: Select the center location of the grid (the origin).
➢ Select the **OK** button to exit the rectangle dialog box.
➢ Select the **Fit** button to fit the geometry to the screen.

1.5 Change the current Z Depth to –1.75 using the Status bar.

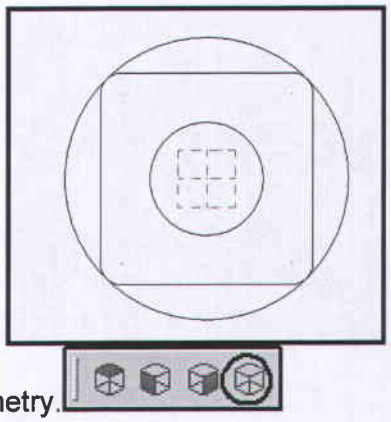
- In the **Status bar**.
- Type -1.75 (Enter). 

1.6 Create an arc knowing the center point and the radius.

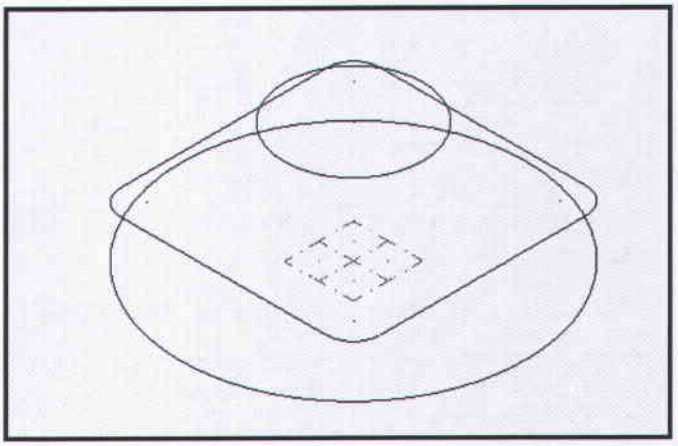
Create
- **Arc**
- **Create Circle Center Point**
- Enter the **Radius** value 2.5.
- [Enter the center point]: Select the **Origin** by moving the cursor at the center of the grid.
- Select the **Fit** button to fit the geometry to the screen.
- Select the **OK** button to exit the command.



- The wireframe should look as in the following picture.



- Select the **Isometric View** from the view toolbar to see the geometry.
- Select the **Fit** button to fit the geometry to the screen.



STEP 2:
CREATE A RULED SURFACE.

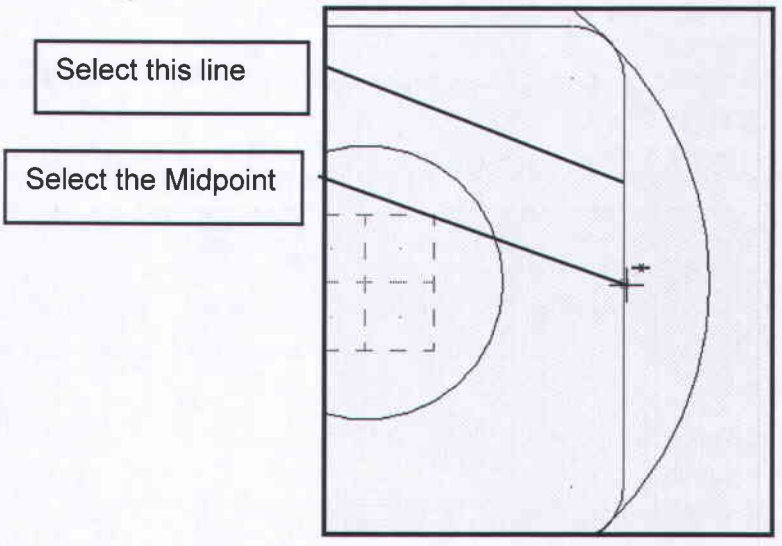
Ruled Surface: Is a surface generated by making a linear transition between two or more open or closed contours. As a result the surface has a linear blending at the intermediate contours.
Applications: Any time a surface must be fit between two or more open or closed contours.

To properly define a surface:
- all of the start points must be lined up, if necessary by breaking an entity of the contour in two pieces;
- the contours should be selected sequentially;
- the contours should be chained in the same direction, or the surface will become twisted and therefore be incorrect.

2.1 Break the right-side vertical line of the rectangle into two pieces.

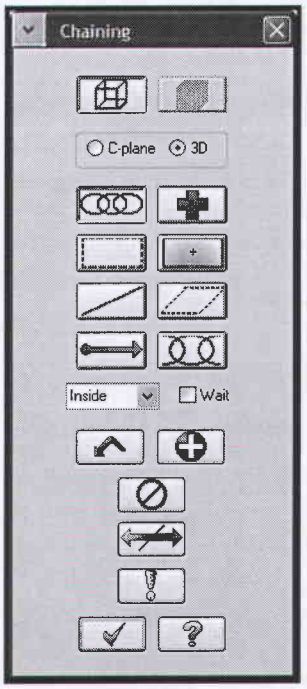
- Select the **Top View** from the view toolbar to see the geometry. 

Edit
- **Trim/Break**
- **Break Two Pieces**
- [Select an entity to break]: Select the line as shown below.
- [Indicate the break position]: Select the Midpoint of the line.

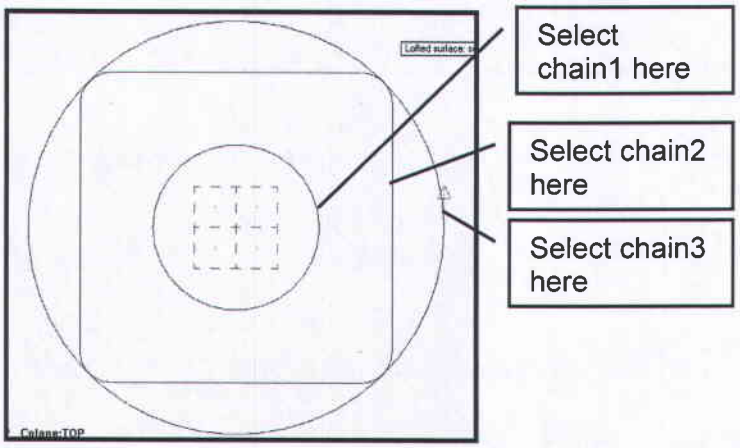


- Select the **OK** button to exit.

2.2 Create the ruled surface.



Create
- **Surface**
- **Create Ruled/Lofted Surfaces**
- [Select chains1]: Select the 1.0" radius arc, just above 0 degrees.
- [Select chains2]: Select the broken line on the rectangle just above 0 degrees.
- [Select chains3]: Select the 2.5" radius arc, just above 0 degrees.

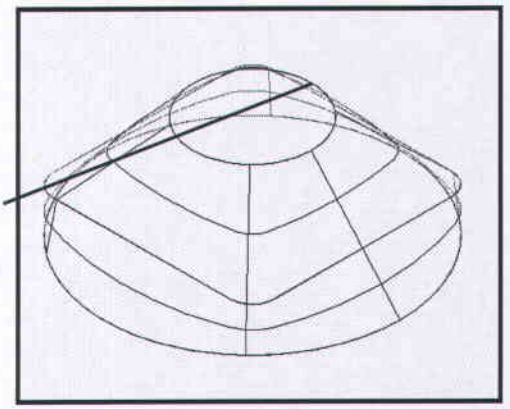


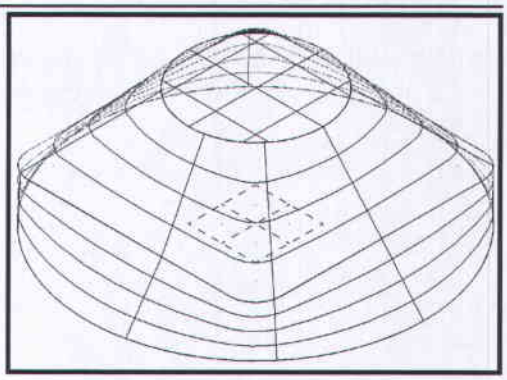
- Select the **OK** button to exit **Chaining**.
- Change the surface type to **Ruled**.
- Select the **OK** button to exit the command.
- Select the **Isometric View** from the view toolbar to see the geometry.

STEP 3:
CREATE A TRIM-FLAT BOUNDARY SURFACE.

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

Create
- **Surface**
- **Create Flat Boundary Surface**
- [Select chains to define flat boundary]: Select the arc as shown.





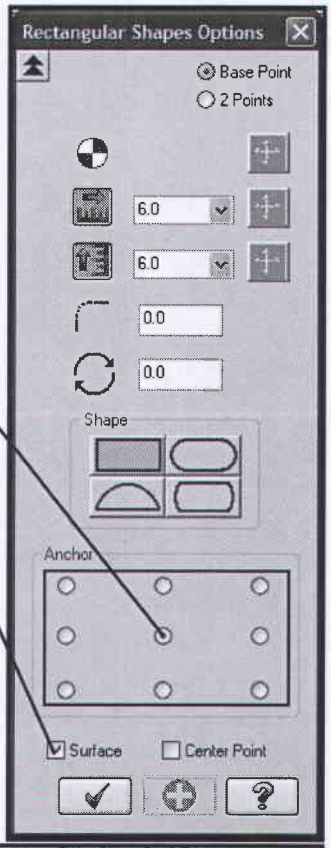
- Select the **OK** button to exit **Chaining**.
- Select the **OK** button to exit the surface flat boundary command.

STEP 4: CREATE A SURFACE USING THE RECTANGLE OPTION.

- The surface type will be ruled.

Create

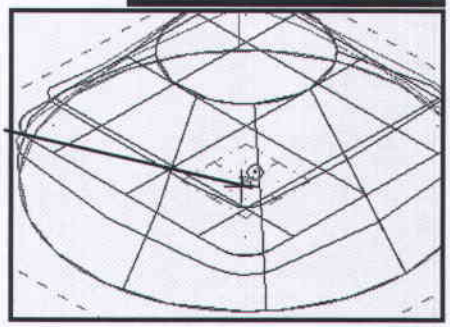
- **Create Rectangular Shapes**
- Select the radio button in front of **Base Point**.
- Type the **Width** and the **Height** as shown in the screenshot to the right.
- Select the double arrow to expand the **Rectangle Options** if you need to.
- Select the middle radio button as the anchor.
- Enable **Surface**.



Select this radio button as anchor

Enable surface

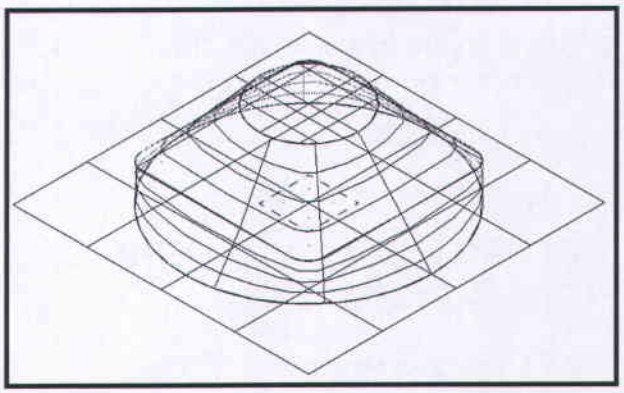
- [Select position for the base point]: Select the center location of the grid (the origin).

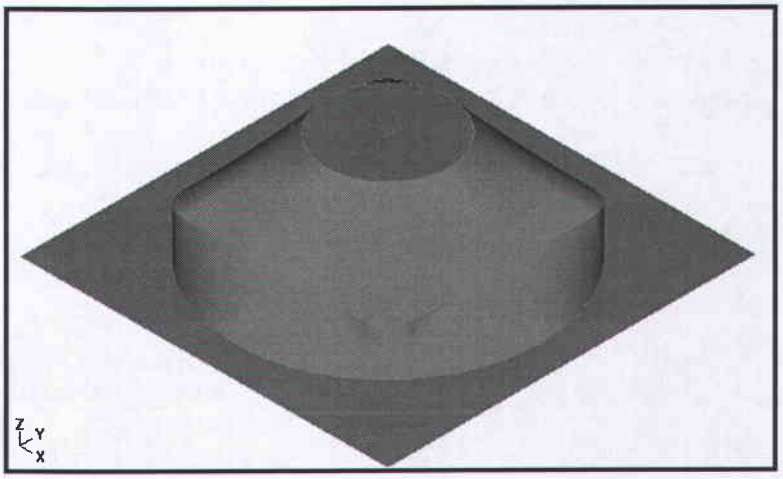


Select Origin

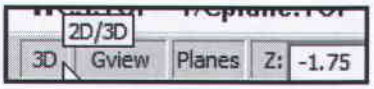
- Select the **OK** button to exit the rectangle dialog box.
- Use the **Fit** icon to fit the drawing to the screen.

- The final part should look as shown below.
- To shade the part hold down the **Alt** key and type **S**.





- **Alt + S** again will unshade the part.

- Toggle back to **3D**. 

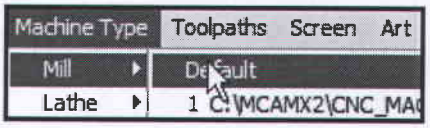
STEP 5:
SAVE THE FILE.

File
- **Save as**
- **File name**: "Your Name_9"
- Select the **OK** button. 

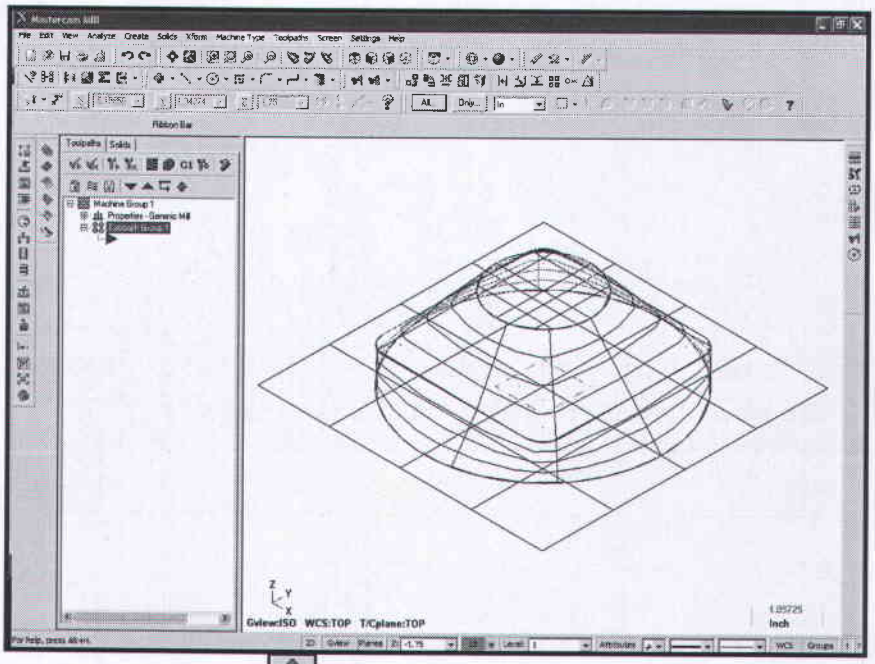
TOOLPATH CREATION

STEP 6:
SET UP THE STOCK TO BE MACHINED.

Machine type

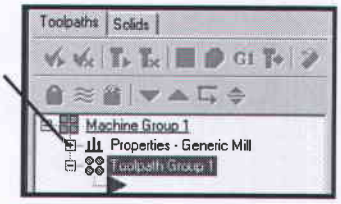


- **Mill**
- Select **Default**.
- To display the **Toolpaths Manager** press **Alt + O**.



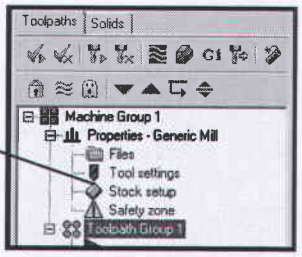
- Use the **Fit** icon to fit the drawing to the screen. 
- Select the plus in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus

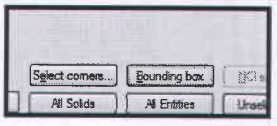


- Select **Stock setup**.

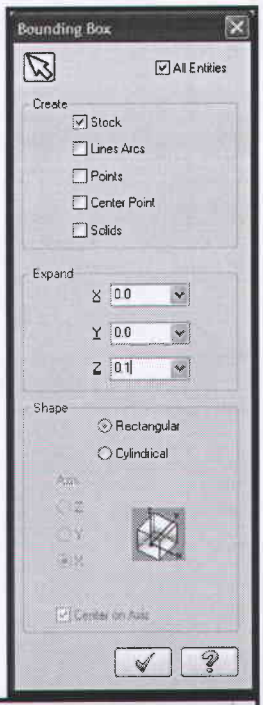
Select the Stock



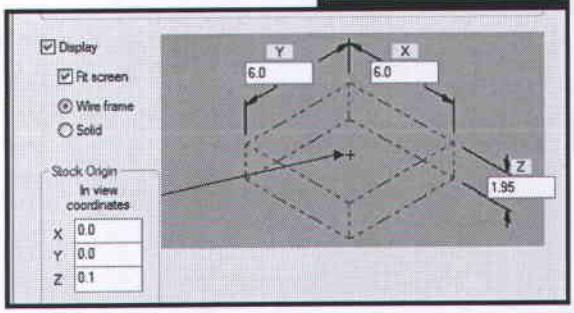
- Change the parameters to match the following screenshot.
- Select **Bounding box** button to automatically find the part extents.



Expand the Z to 0.1.

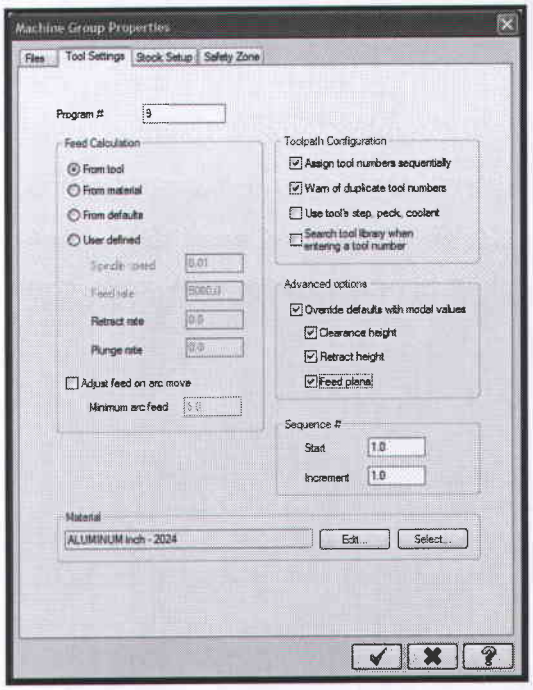


➢ Select the **OK** button to exit the bounding box dialog box.



➢ The stock dimensions should be as shown in the screenshot to the right.

➢ Select the **Tool Settings** tab to set the tool parameters.
➢ Change the parameters to match the screenshot to the right.



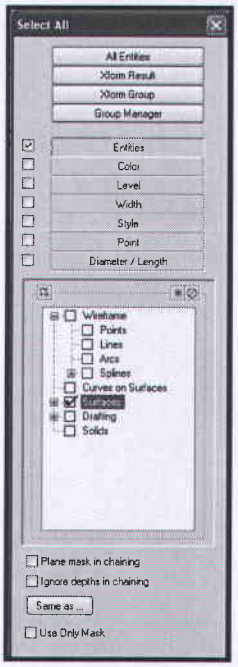
➢ Select the **OK** button to exit **Toolpath Group Properties**.

STEP 7:
ROUGH OUT THE SURFACE (RADIAL).

Surface rough and finish radial toolpaths are used mainly for rounded shape areas or parts. The toolpath requires a rotation point (usually at the center of the part) from which the tool moves along the radius. The distance between the passes is determined by the incremental angle. The part's surface will be better finished close to the rotation point and less finished towards the edges.

Toolpaths
- **Surface Rough**
- **Rough Radial Toolpath**
- Select **Boss**.

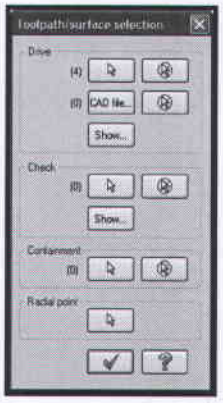
- Select the **OK** button to exit.
- Select the **OK** button to accept the NC name.
- [Select Drive Surface]: Select the **All** button.



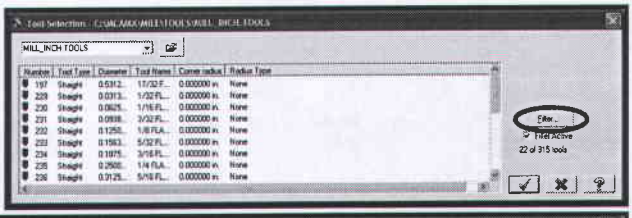
- Make sure that **Entities** and **Surfaces** are enabled.
- Select the **OK** button to exit.

- Select the **End Selection** button.

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

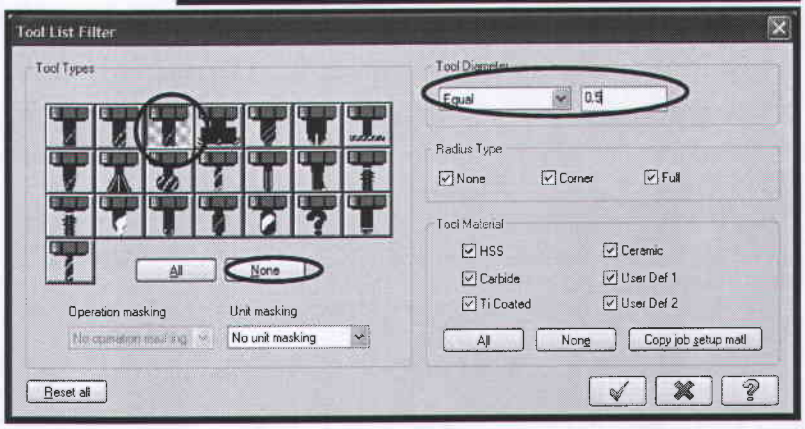


- Click on **Select library tool** in the **Toolpaths parameters** dialog box.
- Select the **Filter** button in the **Tool Selection** dialog box.



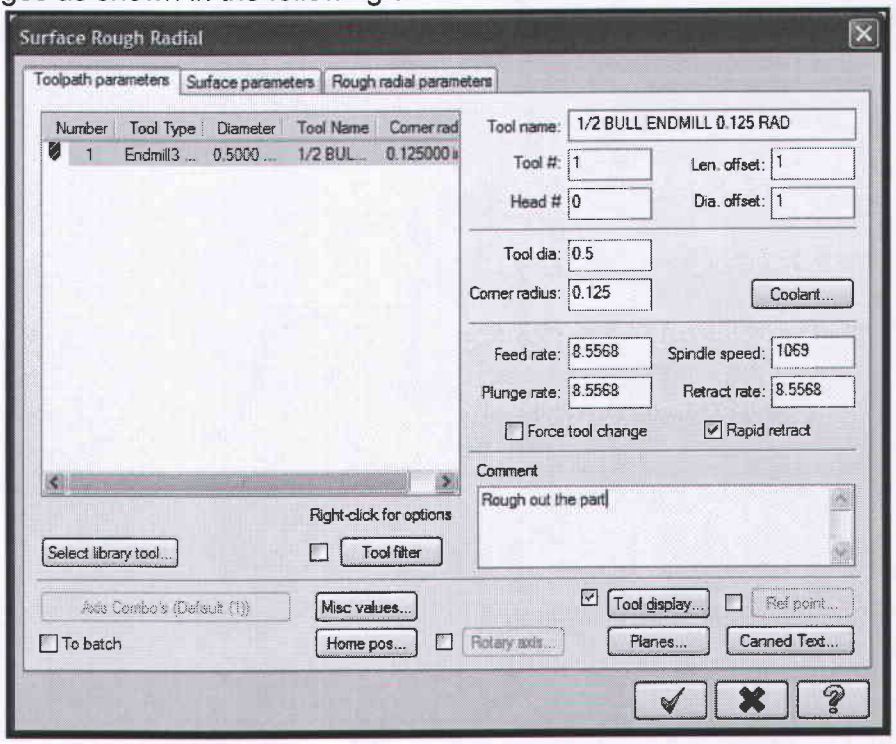
- Select the **None** button in the **Tool Types** area.
- Select the **Bullnose** icon.

- Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.
- Enter the **Tool Diameter** 0.5.
- Select the **OK** button to exit.



- Select 0.5" Bull Nose with 0.125" corner radius in the **Tool Selection** page.
- Select the **OK** button to exit.

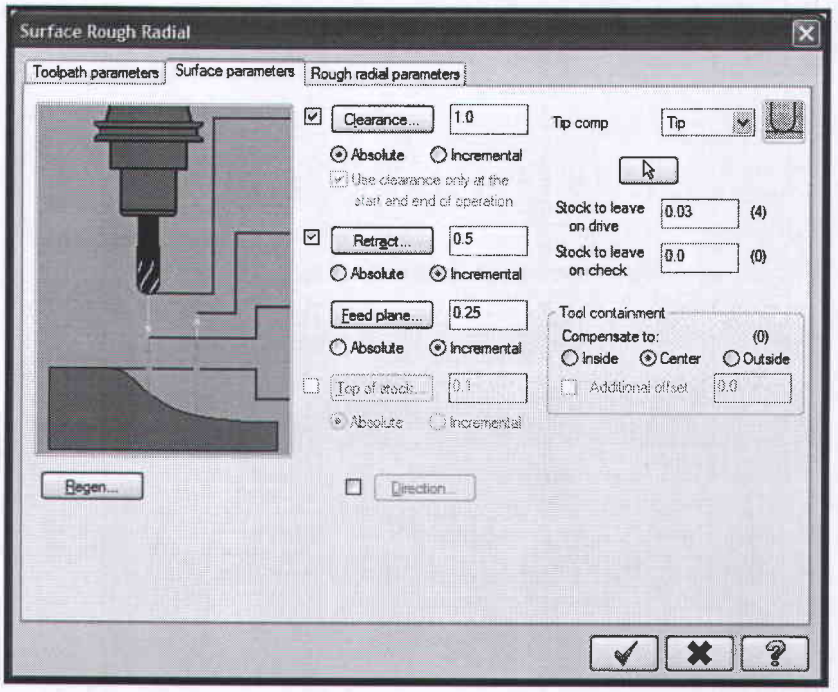
- Make the necessary changes as shown in the following screenshots.



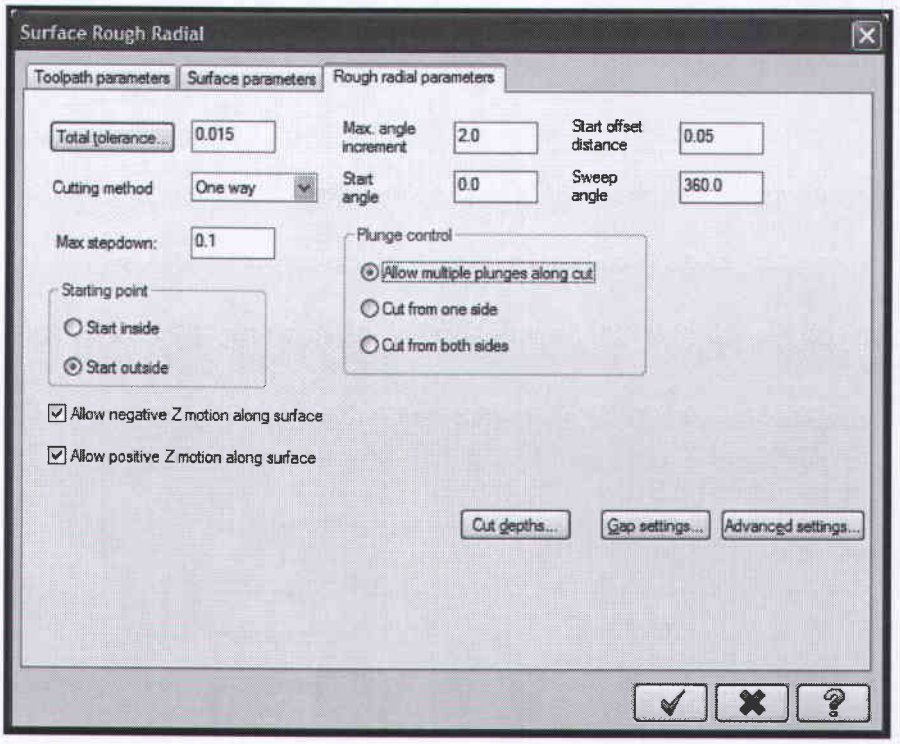
- The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition. Change them as desired.

➢ Select the **Surface parameters** tab.
➢ Change the parameters to match the following screenshot and leave stock on drive surfaces.

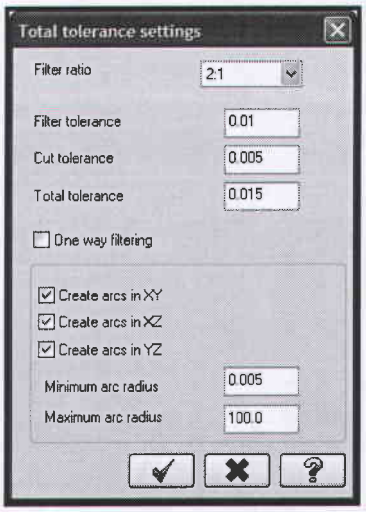
- For more info on the parameters please check the previous tutorial.



➢ Select the **Rough radial parameters** page.

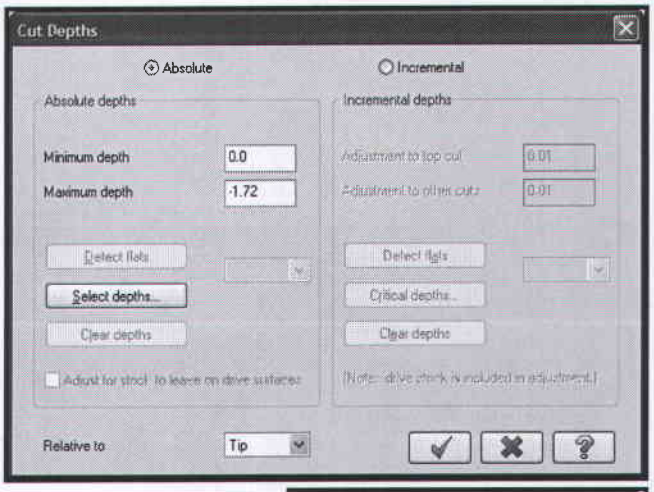


- Select the **Total tolerance** button.
- Change **Filter ratio** to 2:1 as shown in the picture to the right.
- Change the **Cut tolerance** to .005. The system will automatically calculate the filter tolerance.

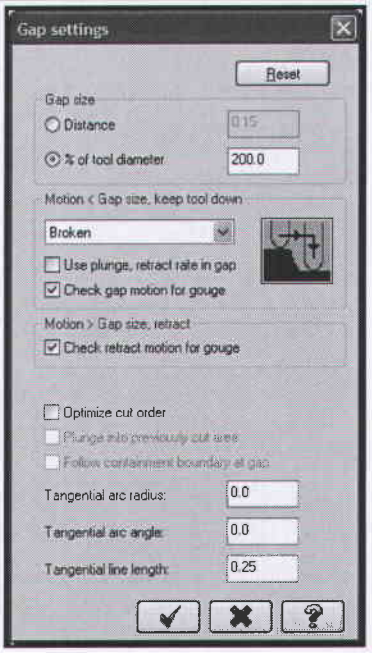


> **Filter** eliminates unnecessary tool moves that lie in a straight line generating a single tool move, within a specified tolerance.
> Filter can also replace multiple linear moves with an arc move with the radius established between a maximum and a minimum radius in different construction planes.

- Select the **OK** button to exit.
- Select the **Cut depths** button and make all the necessary changes as shown in the picture to the right.
- Select the **OK** button to exit **Cut Depths**.



- Select the **Gap settings** button and change the settings as shown in the picture.



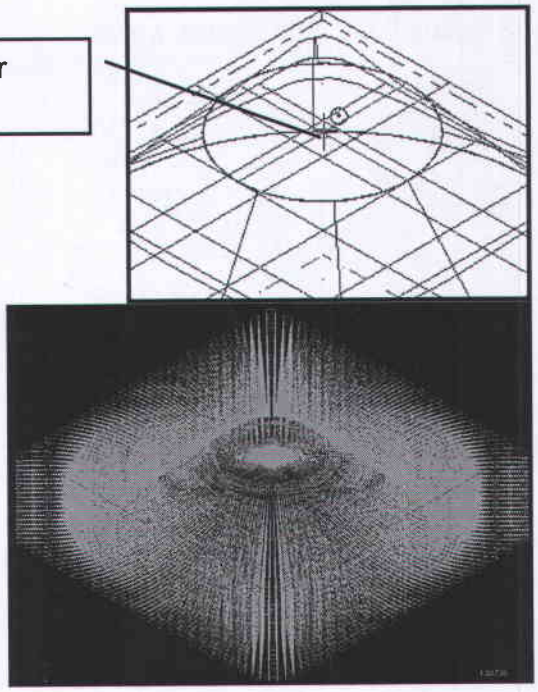
> **Gap** parameters set the way the tool moves between gaps or spaces in a surface toolpath.
> **Direct** parameter enables a linear transition between gaps if the tool motion is smaller than the gap size.
> To create a smoother tool motion between gaps, and extend the area machined by the tool, place **Tangential lines** at each entry/exit from the part.

Refer to **Tutorial # 8** page 8-21 for more info regarding gap settings parameters.

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

- Select the **OK** button from the parameter screen.
- [Enter rotation point]: Select the center of the 1.0" radius arc.

Select the center point here



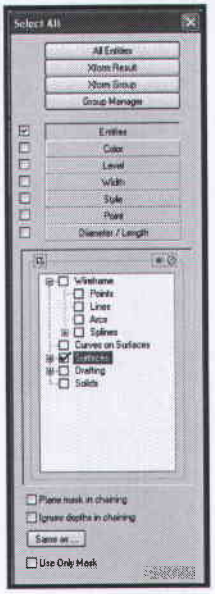
- Select **Toggle toolpath display on selected operations** to remove the toolpath display.

STEP 8:
FINISH THE SURFACE (SCALLOP).

Scallop finish toolpaths create consistent scallop heights over an entire set of surfaces. The toolpath consistently touches the surfaces and minimizes retract motion. It is used for different type of shapes, both cavities and bosses.

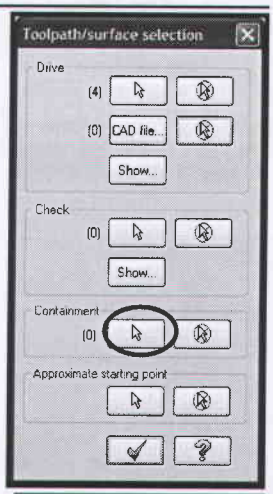
Toolpaths
- **Surface Finish**
- **Finish Scallop Toolpath**

- [Select Drive Surface]: Select the **All** button.

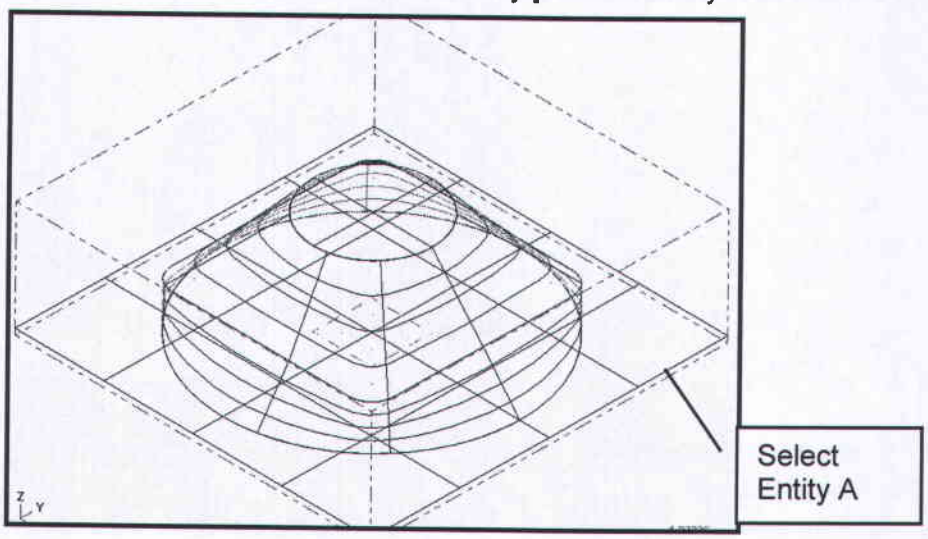


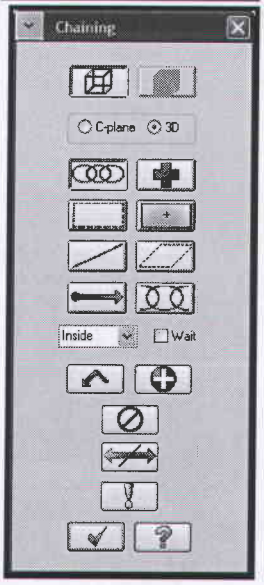
- Make sure that **Entities** and **Surfaces** are enabled.
- Select the **OK** button to exit.
- Select the **End Selection** button.

➢ Select the **Containment** button.



➢ [Chain 2D tool containment boundary]: Select Entity A as shown.





➢ Select the **OK** button to close the **Toolpath/surface selection** dialog box.

➢ Click on **Select library tool** in the **Toolpaths parameters** dialog box.

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

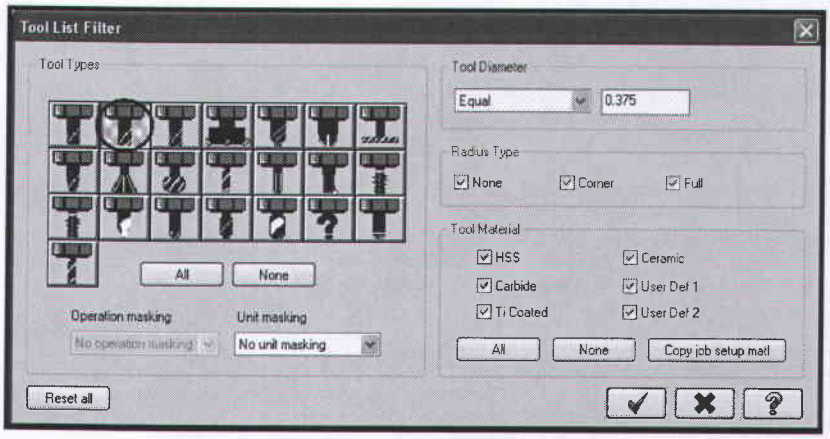
➢ Select the **None** button in the **Tool Types** area.

➢ Select the **Spherical** icon in the **Tool Types**.

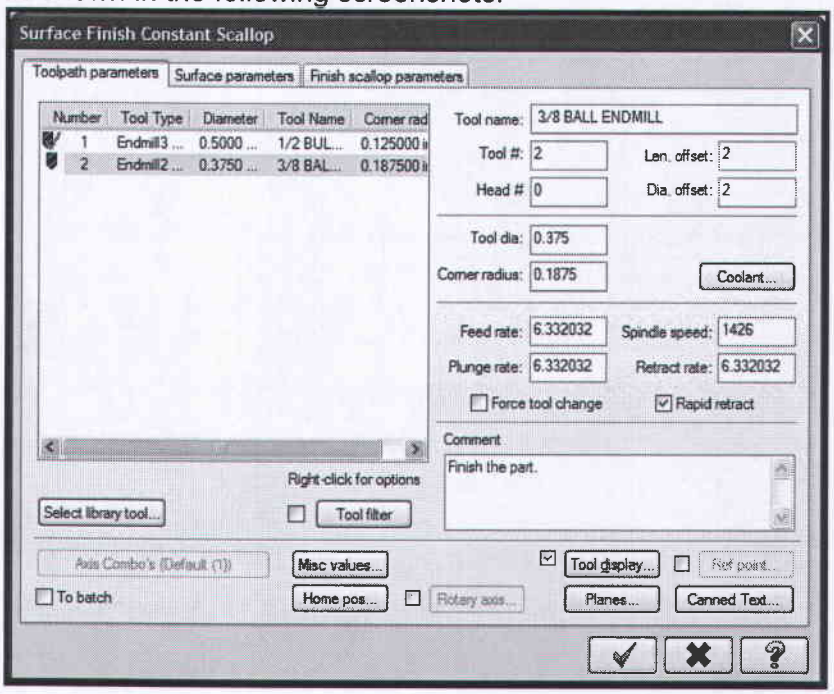
➢ Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.

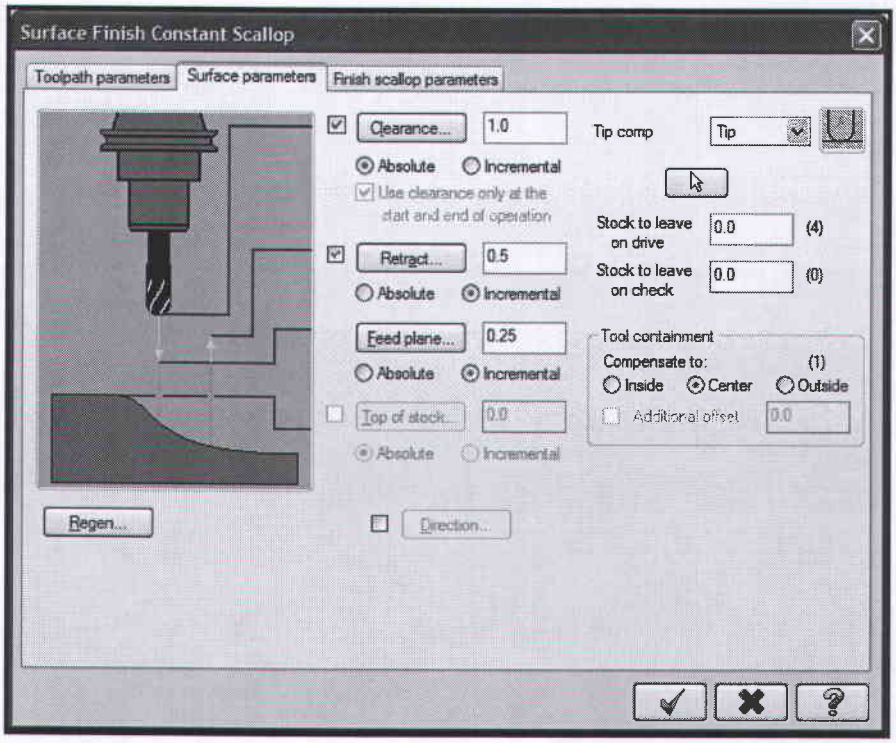
➢ Enter the **Tool Diameter** 0.375.

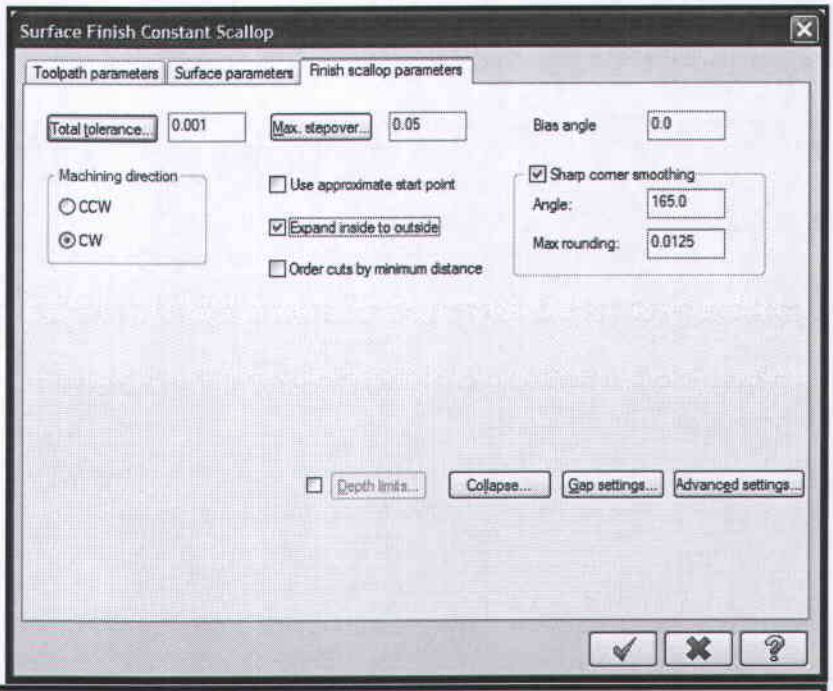
➢ Select the **OK** button to exit.



- Make sure that the tool is selected and click on the **OK** button to exit the **Tool Selection** page.
- Make any necessary changes as shown in the following screenshots.

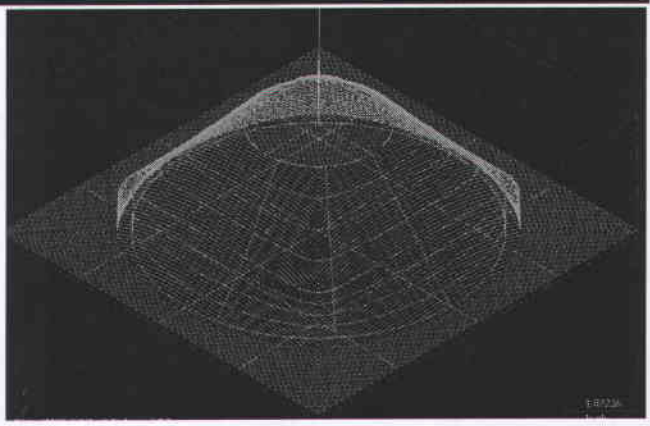






Scallop finish toolpath creates consistent scallop heights over an entire set of surfaces. **3D Collapse** cutting method creates a zone around the cutting area, cuts the boundary of the zone, and then steps in to create a cut that offsets the outer boundary by the maximum stepover.

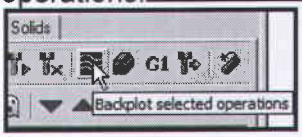
➢ Select the **OK** button to exit scallop parameters.



STEP 9:
BACKPLOT THE TOOLPATH.

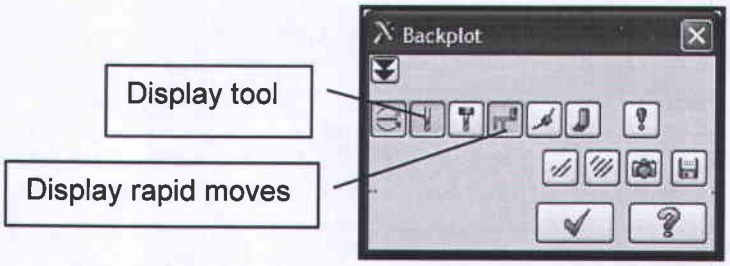
➢ Select the **Select all operations** icon to select all operations.

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

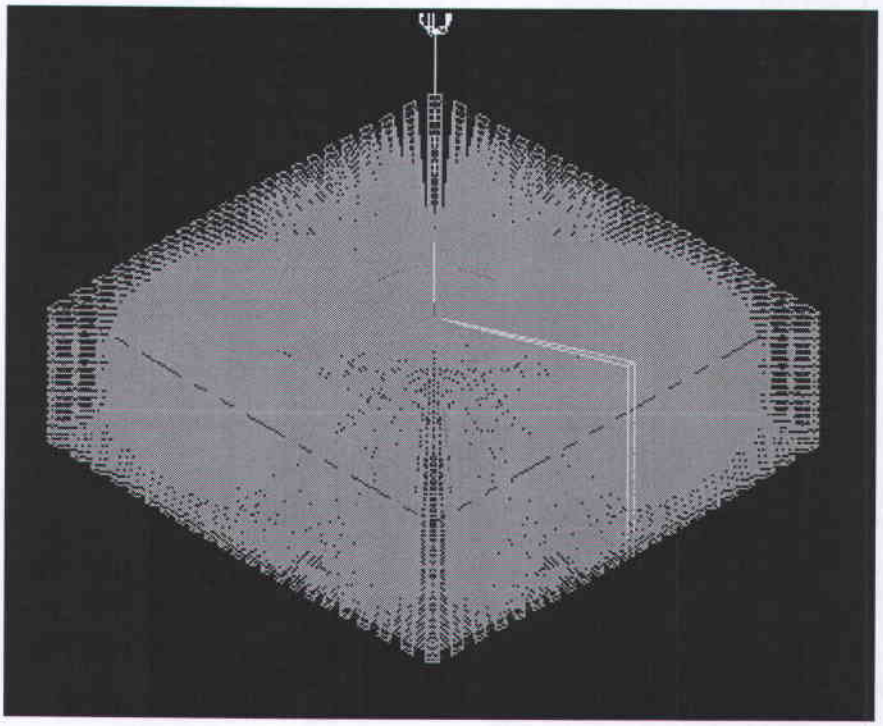


➢ Make sure that you have the following buttons turned on (they will appear pushed down).

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



➢ Select the **Play** button.



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

VERIFY-TOOLPATH VERIFICATION

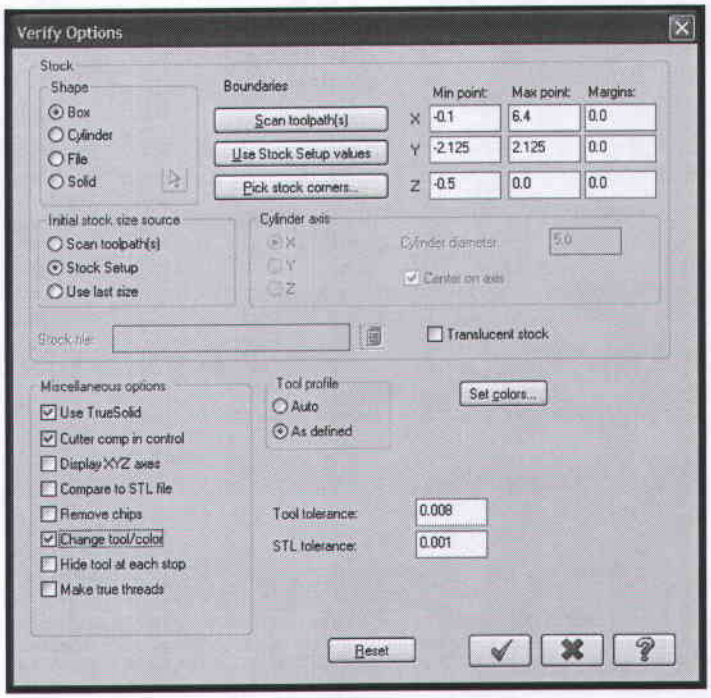
STEP 10:
VERIFY.

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

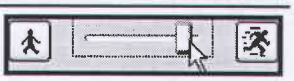
➢ Enable **Turbo** button to verify the part faster.
➢ Enable **Stop on collision.**

➢ 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.



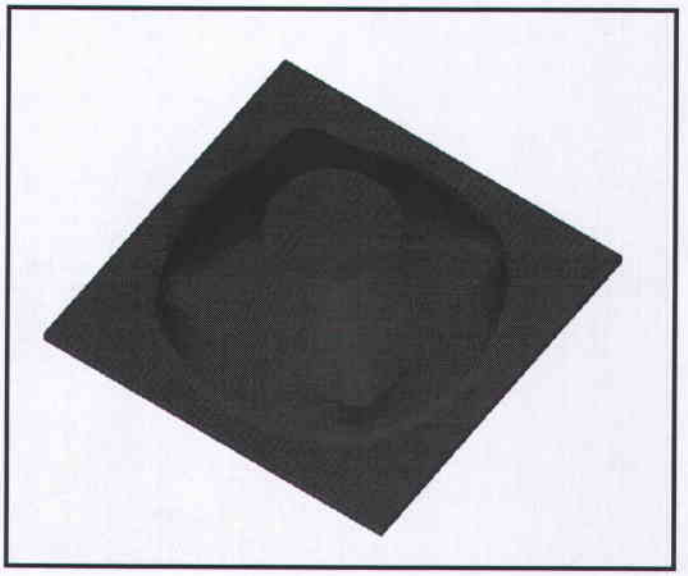
➢ Select the **OK** button to exit **Verify Options**.



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

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

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



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

STEP 11:
POST PROCESS THE FILE.

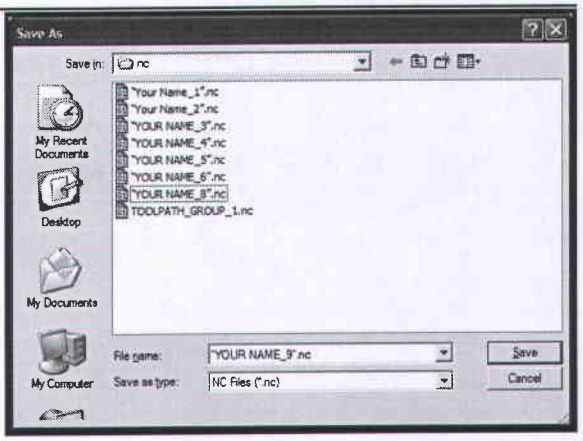
➢ Select the **Post selected operations** button from **Toolpath Manager.**

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

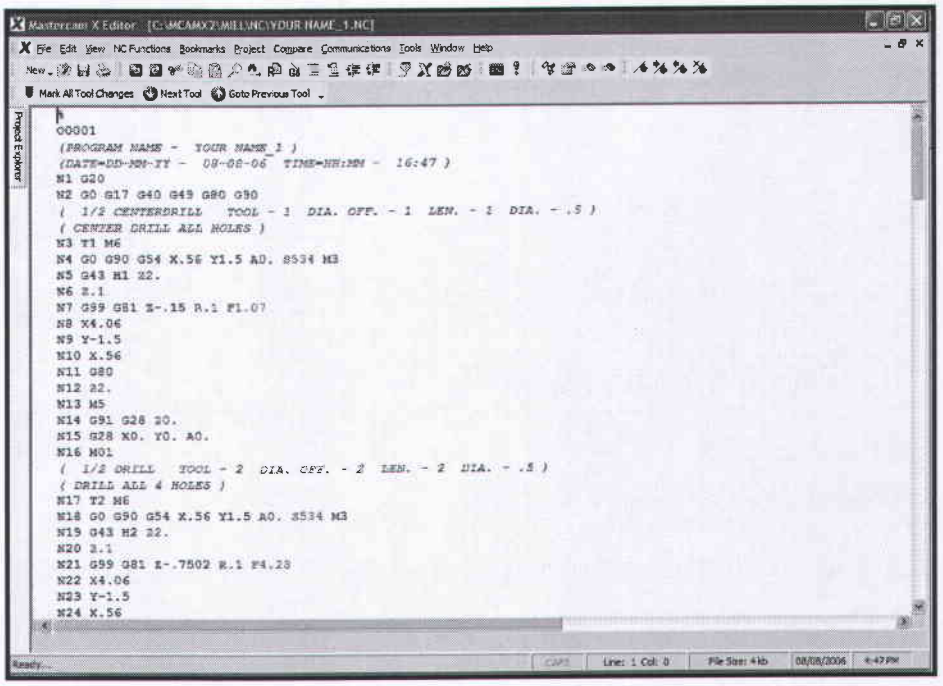
NC file enabled allows you to keep the NC file and to assign the same name as the MCX file
Edit enabled allows you to automatically launch the default editor.

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

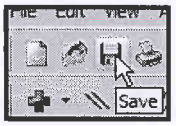
➢ Select the **Save** button.



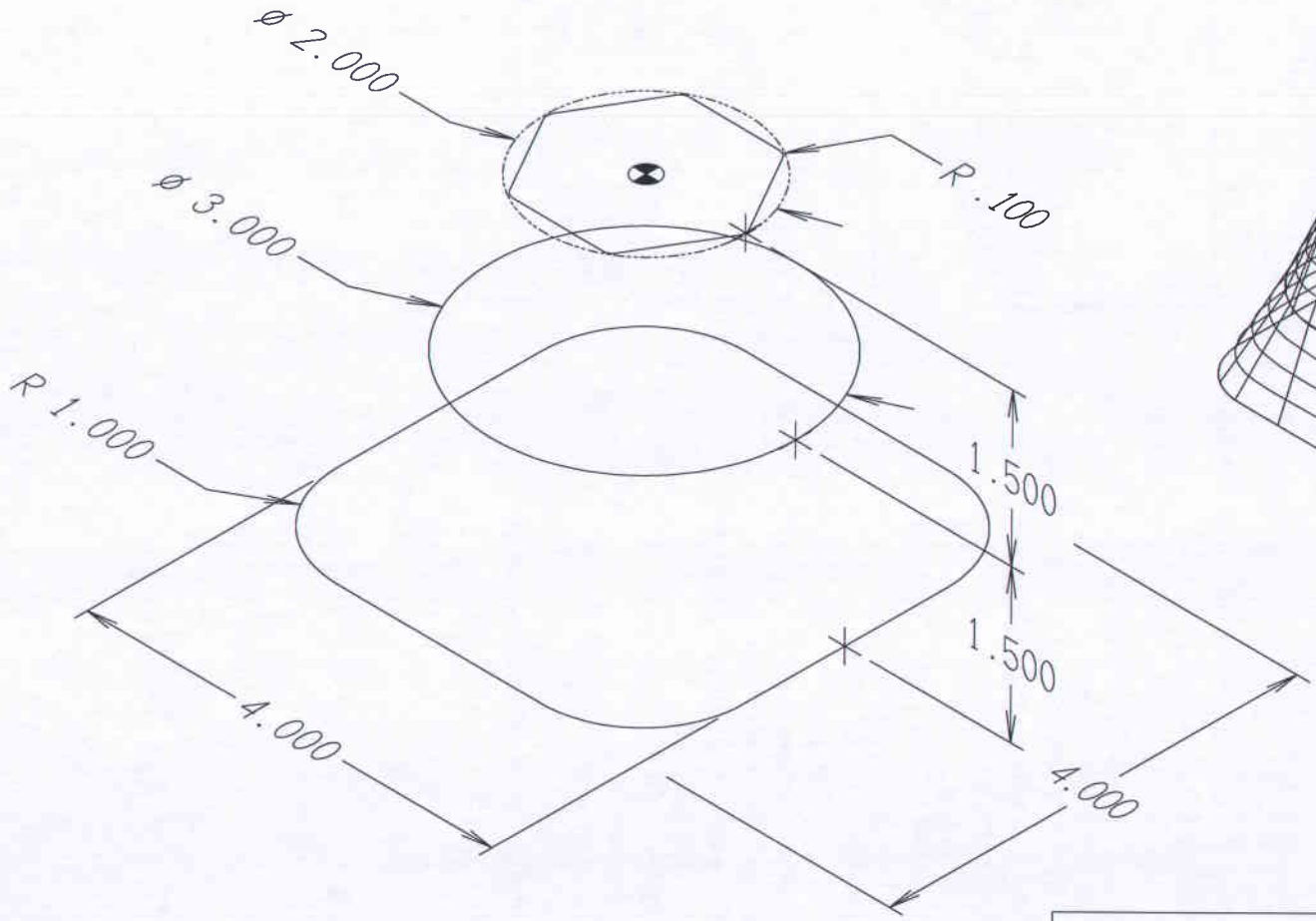
➢ Select the red **X** box at the upper right corner to exit the **Editor**.

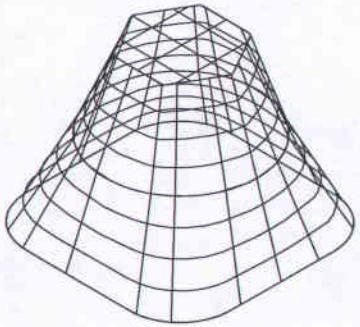


STEP 12:
SAVE THE UPDATED MCX FILE.

➢ Select the **Save** icon. 

ALL DIMENSIONS IN INCHES

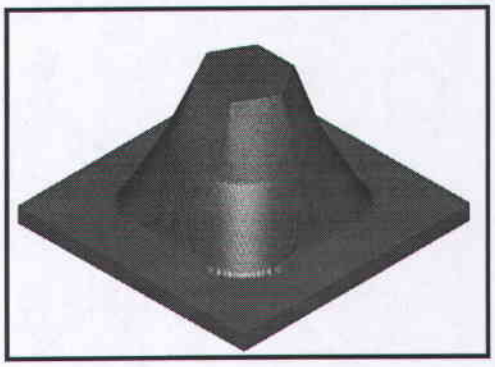


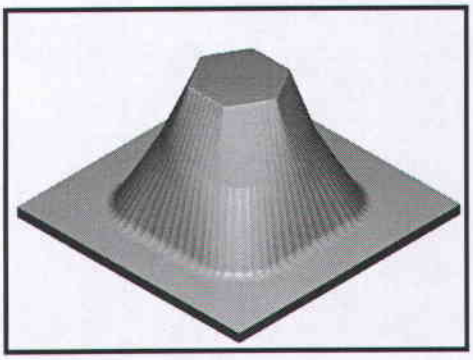


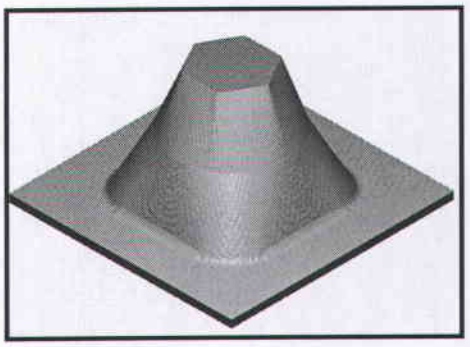
TITLE	TUTORIAL 9 - EXERCISE
MATERIAL	ALUMINUM T2024
DATE: JUNE 12, 2000	eMastercam.com

REVIEW EXERCISES.

Student practise. Create the Toolpath for Exercise-Tutorial 9 as per the instructions below;







☞ **Tips:**

1.Create a flat surface at the bottom of the part using **Create Rectangle** size 6 X 6
2. Establish the **stock** size giving X =6", Y = 6", give z = 3.3" and z stock origin =0.1
3. Surface Rough Radial
Use 1 " Bull End Mill rad 1/4
Stock to leave on drive surfaces= .03"
Total tolerance = 0.01
Filter ratio 2:1
Max stepdown =0.15
Cutting method One way
Start outside
Allow positive Z motion
Max angle increment 5 deg.
Rotation point select **Origin**
4. Surface Finish Scallop Toolpath
Use 3/8" Ball End Mill
Stock to leave on drive surfaces= 0
Total tolerance = .001
Max stepover = .05
Expand inside to outside
5. Backplot and Verify the toolpaths.
6. Post process the file.