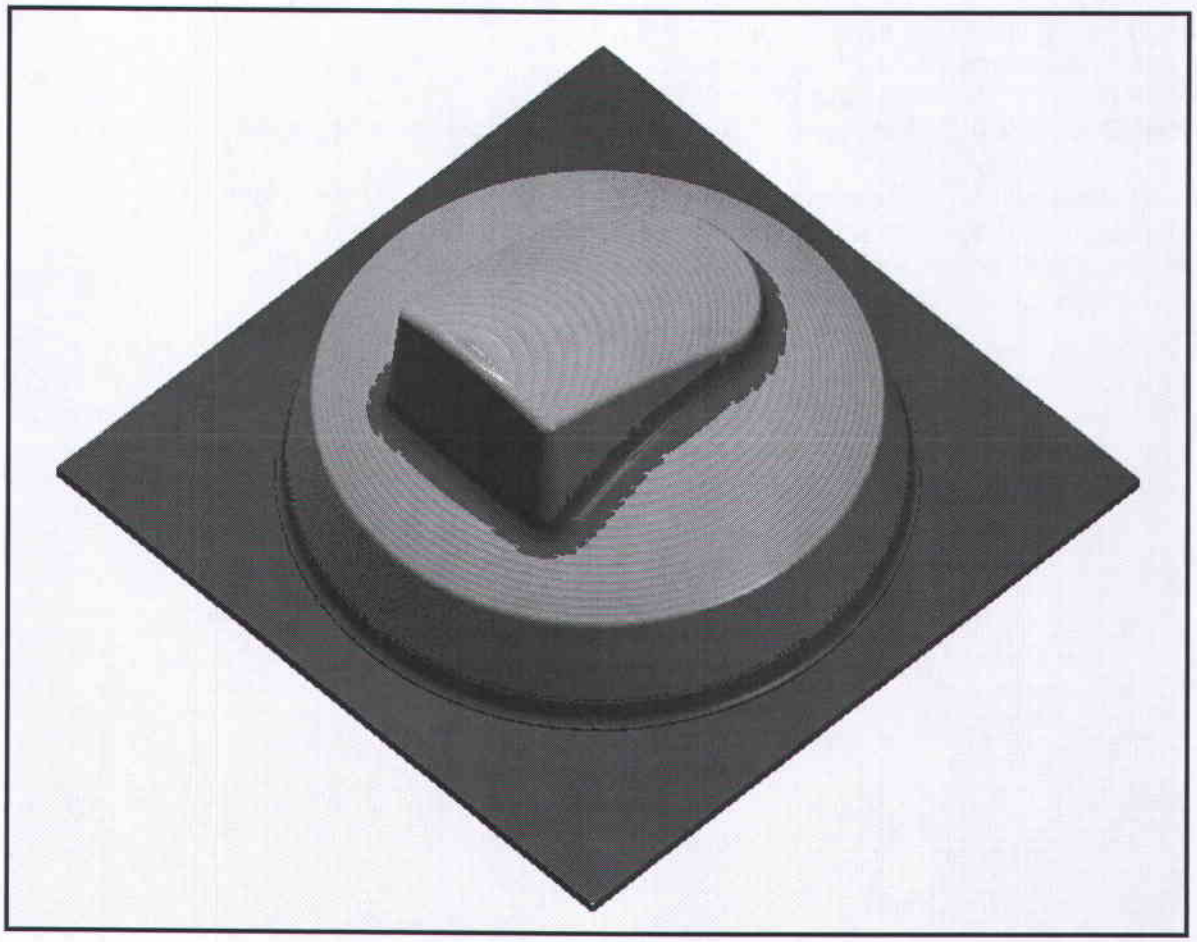
TUTORIAL SERIES FOR

Mastercam X2

TUTORIAL 14
DRAFT, SWEEP, REVOLVE AND FILLET SURFACES, HIGH SPEED TOOLPATHS: CORE ROUGHING, HORIZONTAL AREA, WATERLINE & SCALLOP REST PASSES.



Objectives:

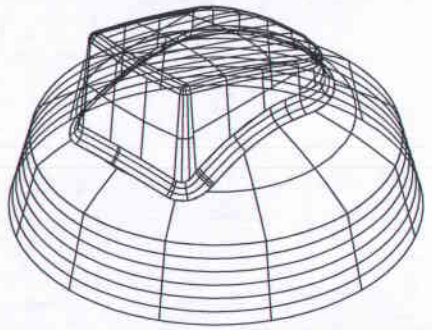
The Student will design a 3-dimensional drawing by:
- Creating the 3D wireframe in different construction planes.
- Creating draft surfaces at a height of 2" with a draft angle of 5 degrees.
- Creating a sweep surface.
- Creating fillet surfaces at the top of the part.
- Creating revolved surfaces.
- Creating fillet surfaces between the draft surfaces and the revolved surfaces.

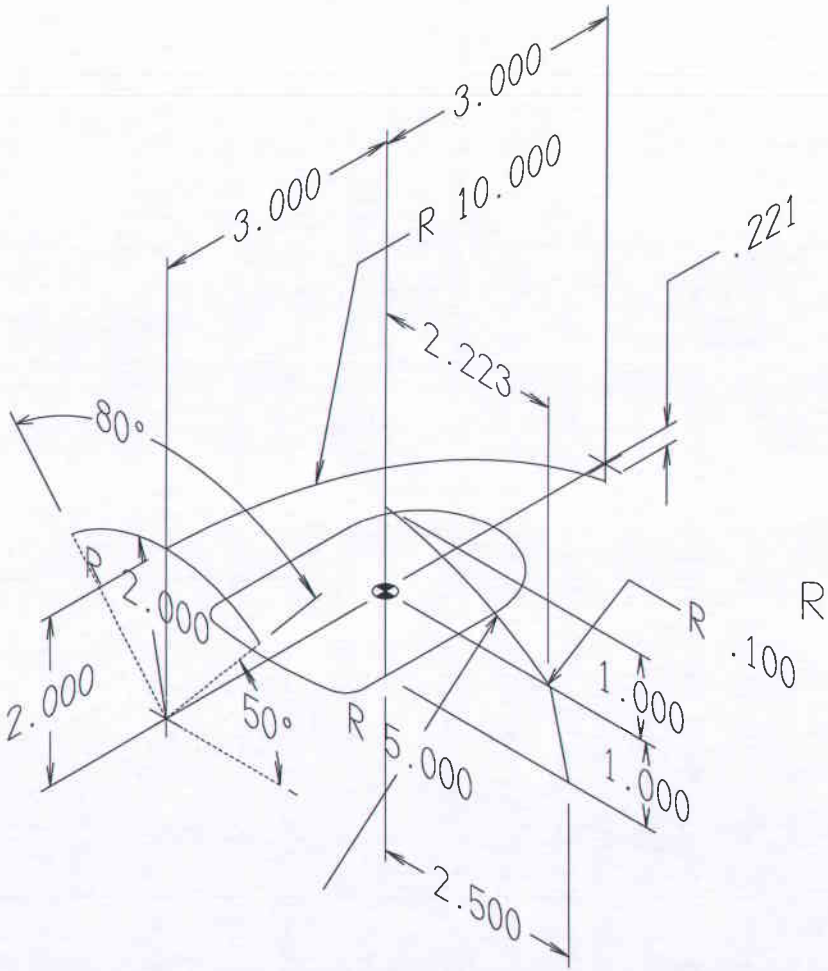
The Student will create a 3-dimensional milling toolpath consisting of:
- High Speed Core Roughing toolpath.
- High Speed Horizontal finish toolpath.
- High Speed Waterline finish toolpath.
- High Speed Scallop rest passes finish toolpath to clean areas where previous tool did not fit.

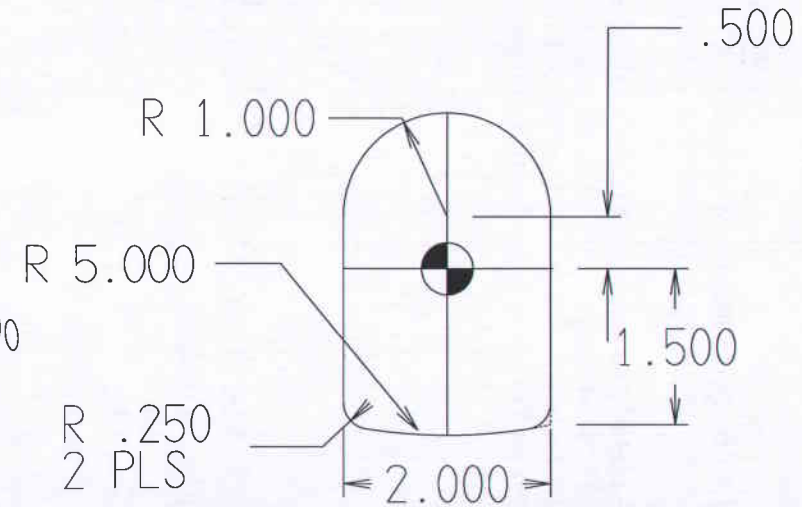
The Student will check the toolpath using Mastercam's Verify module by:
- Defining a 3-dimensional block, the size of the workpiece.
- Running the Verify program to create an intermediate stock.
- Running the Verify program to machine the part on the screen.

The draft surface height is 2.000 " and has a draft angle of -5 deg.
The top fillet radius is .100
The second fillet radius is .250

ALL DIMENSIONS IN INCHES





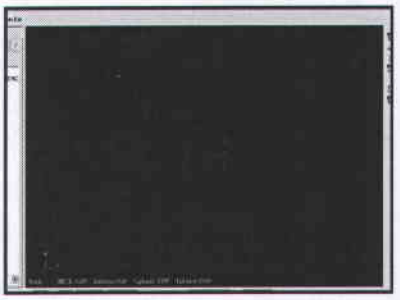


TITLE	TUTORIAL 14
MATERIAL	ALUMINUM T2024
DATE: JUNE 12, 2000	eMastercam.com

GEOMETRY CREATION

Setting the toolbar states:

- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-5 in the **User Notes**.
- Toolpaths/Solids operations manager to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to 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 3D WIREFRAME IN DIFFERENT CONSTRUCTION PLANES.

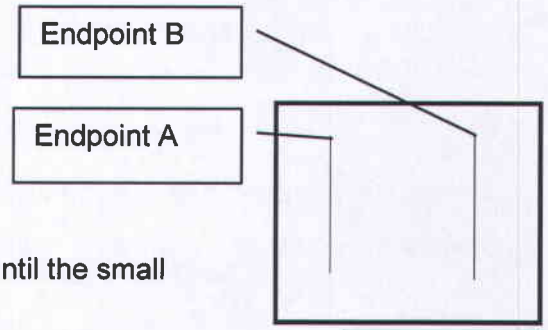
1.1 Create polar lines.

Create
- **Line**
- **Create Line Endpoint**

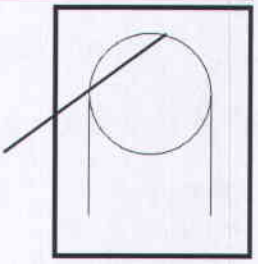
- [Specify the first endpoint]: Select the **Fast Point** icon.
- Enter the coordinates: -1.0, -1.5 (Enter).
- Enter the line **Length** 2.0 (Tab).
- To set the other parameters of the line use the Tab key.
- Enter the **Angle** in degrees 90 (Enter).
- To keep the same length and angle for the next line click on the **Length** icon and on the **Angle** icon
- The values will be highlighted in red. 90.0
- Select the **Apply** button.
- [Specify the first endpoint]: Select the **Fast Point** icon.
- Enter the coordinates: 1.0, -1.5 (Enter).
- Select the **OK** button to exit.

1.2 Create the arcs knowing the endpoints and the radius.

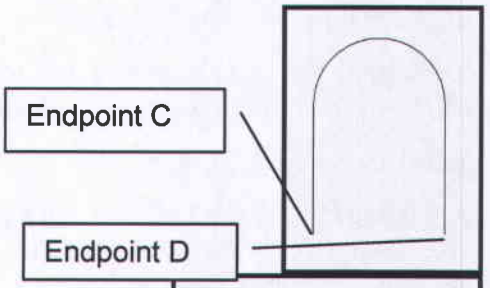
Create
➢ **Arc**
➢ **Create Arc Endpoints**
➢ Enter the **Radius** 1.0 (Enter).
➢ [Enter the first point]: Select Endpoint A.
➢ [Enter the second point]: Select Endpoint B.
To select the points move the cursor on the entity until the small square appears , and then click on the point. Using the autocursor option you can select endpoints, midpoints, center points, quadrants, etc.



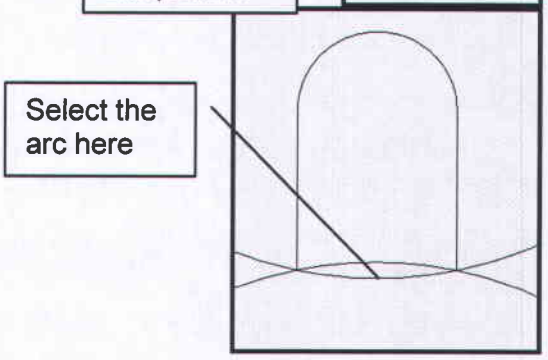
➢ [Select an arc]: Select the arc as shown.
➢ Select the **Apply** button to continue.
➢ Use the **Fit** icon to fit the drawing to the screen.



➢ Enter the **Radius** 5.0 (Enter).
➢ [Enter the first point]: Select Endpoint C.
➢ [Enter the second point]: Select Endpoint D.
➢ Use the **Fit** icon to fit the drawing to the screen.



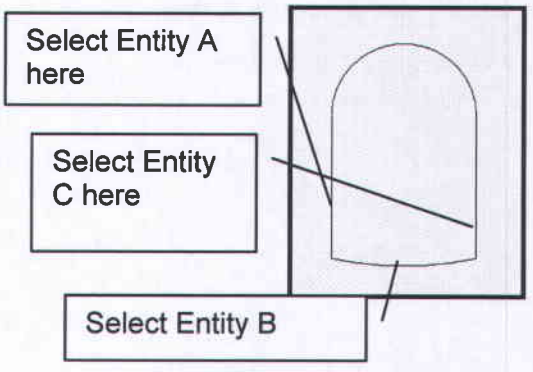
➢ [Select an arc]: Select the lower arc.



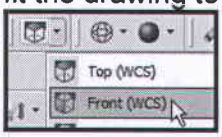
➢ Select the **OK** button to exit.

1.3 Create the fillets with radius 0.25".

Create
➢ **Fillet**
➢ **Fillet Entities**
➢ Enter the fillet **Radius** 0.25.
➢ [Select an entity]: Select Entity A.
➢ [Select another entity]: Select Entity B.

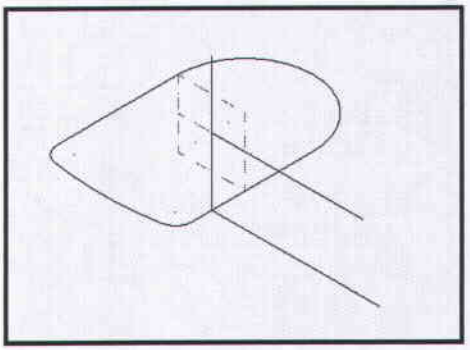


- Note that a fillet option will be automatically drawn depending on where you move the cursor around the entities.
- [Select an entity]: Select Entity B.
- [Select another entity]: Select Entity C.
- Select the **OK** button to exit.
- Change the **Graphic view** to **Isometric**.
- Use the **Fit** icon to fit the drawing to the screen.
- Set plane to **Front**



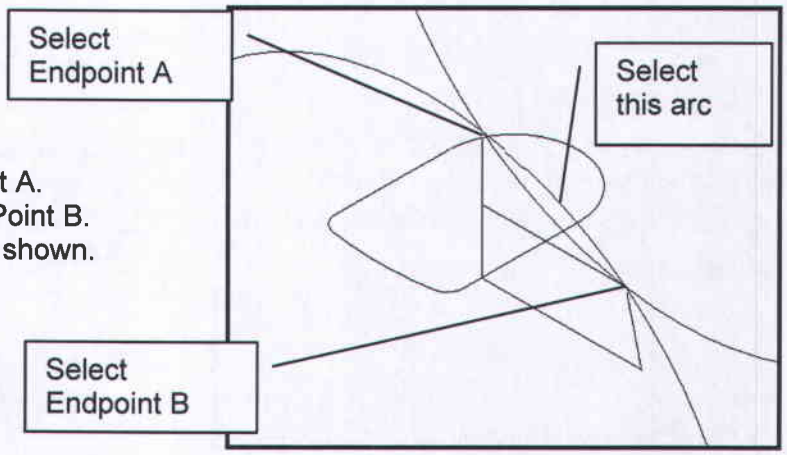
1.4 Create a line knowing the first endpoint, the length and the angle.

Create
- **Line**
- **Create Line Endpoints**
- Note that the Length and the Angle are locked. (highlighted in red)
- [Specify the first endpoint]: Select the **Fast Point** icon.
- Enter the coordinates: 0, -1.0 (Enter).
- Select the **Apply** button.
- [Specify the first endpoint]: Select the **Fast Point** icon
- Enter the coordinates: 0, 0 (Enter).
- Enter the line **Length** 2.25 (Tab).
- Enter the **Angle** in degrees 0 (Enter).
- Select the **Apply** button.
- Leave the values locked. (highlighted in red)
- [Specify the first endpoint]: Select the **Fast Point** icon.
- Enter the coordinates: 0, -1.0 (Enter).
- Enter the line **Length** 2.5 (Enter).
- Use the **Fit** icon to fit the drawing to the screen.
- Select the **OK** button to exit.
- The geometry should look as shown to the right.



1.5 Create the arc with radius 5".

Create
➢ **Arc**
➢ **Create Arc Endpoints**
➢ Enter the **Radius** 5.0 (Enter).
➢ [Enter the first point]: Select Point A.
➢ [Enter the second point]: Select Point B.
➢ [Select an arc]: Select the arc as shown.

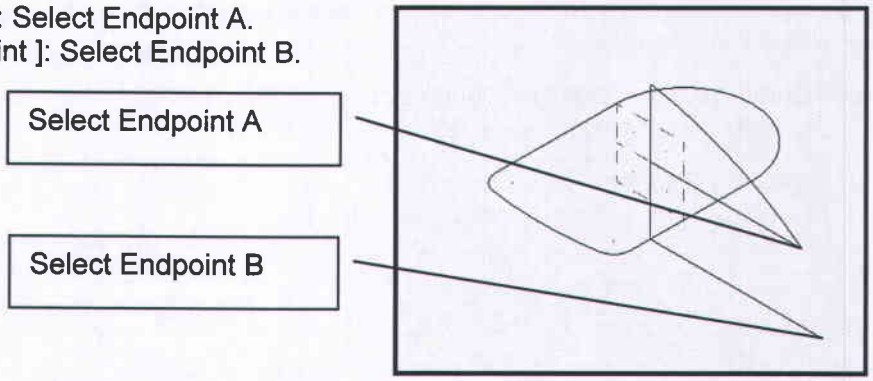


➢ Select the **OK** button to exit.

1.6 Create a line knowing the endpoints.

Create
➢ **Line**
➢ **Create Line Endpoints**
Unlock the length and the angle values by selecting the icons in front of them.

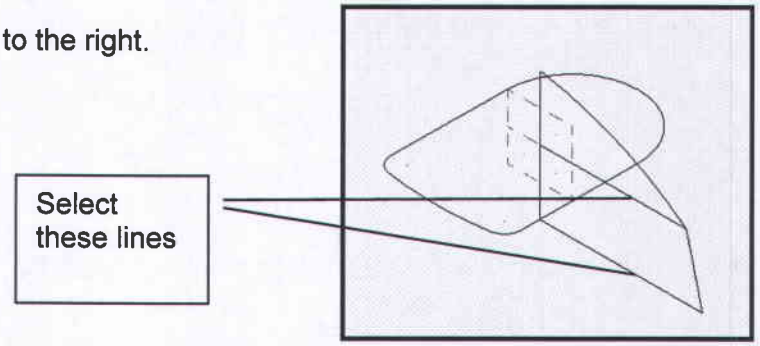
➢ [Specify the first endpoint]: Select Endpoint A.
➢ [Specify the second endpoint]: Select Endpoint B.



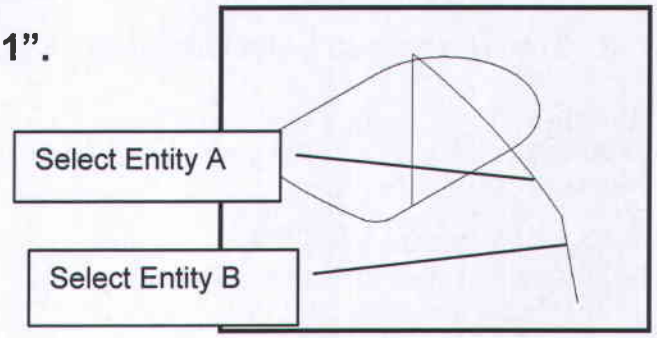
➢ Select the **OK** button to exit.

1.7 DELETING TWO CONSTRUCTION LINES.

➢ Select the horizontal lines as shown to the right.
➢ Select the **Delete** entity icon.

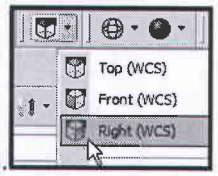


1.8 Create the fillets with radius 0.1".



Create
- **Fillet**
- **Fillet Entities**

- Enter the fillet **Radius** 0.1.
- [Select an entity]: Select Entity A.
- [Select another entity]: Select Entity B.
- Select the **OK** button to exit.

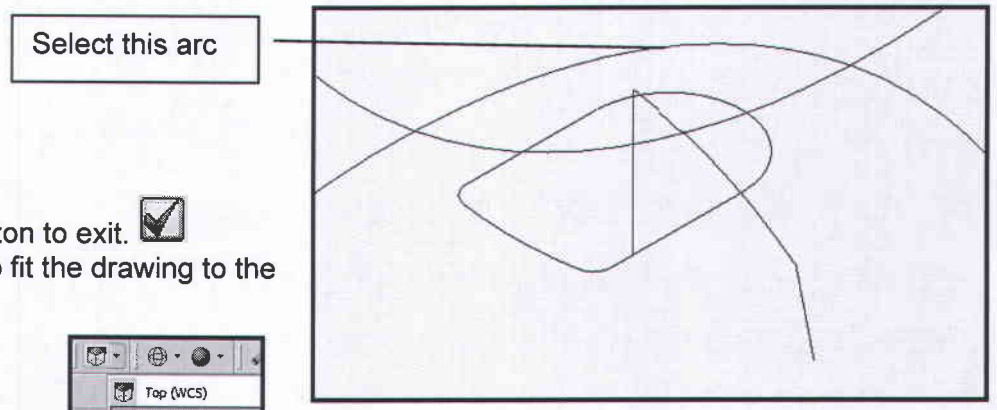


- Set planes to **Right** side.

1.9 Create the arc with radius 5".

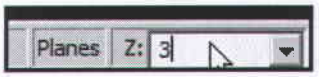
Create
- **Arc**
- **Create Arc Endpoints**

- [Specify the first endpoint]: Select the **Fast Point** icon.
- Enter the coordinates: 3, -0.221 (Enter).
- [Enter the second point]: Select the **Fast Point** icon.
- Enter the coordinates: -3, 2 (Enter).
- Enter the **Radius** 10.0 (Enter).
- [Select an arc]: Select the arc as shown.



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

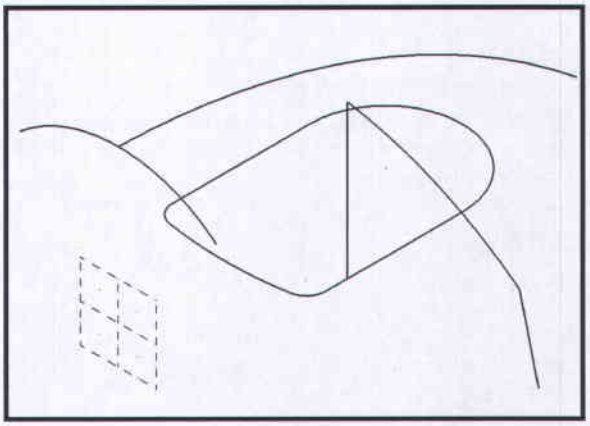
- Set the plane to **Front**.
- Change the construction depth **Z** to 3 in the **Status Bar.**



1.10 Create a polar arc.

Create
- **Arc**
- **Create Arc Polar**

- [Enter the center point]: Select the **Fast Point** icon.
- Enter the center point coordinates 0,0 (Enter).
- Enter the **Radius** 2.0.
- Enter the **Start Angle** 50 (Tab).
- Enter the **End Angle** 130 (Enter).
- Use the **Fit** icon to fit the drawing to the screen.
- Select the **OK** button to exit.
- The final 3D wireframe should look like the picture to the right.



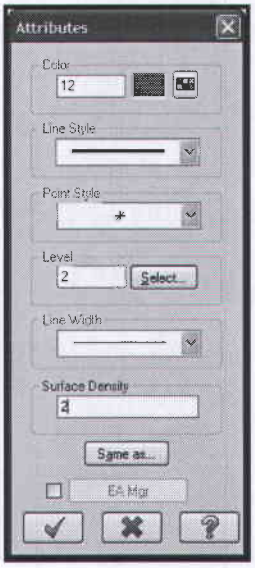
STEP 2:
CREATE THE DRAFT SURFACES.

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

2.1 Change the Main color to red (#12) and the Main Level to 2 from the Status Bar.

- Select **Attributes** from the Status Bar.

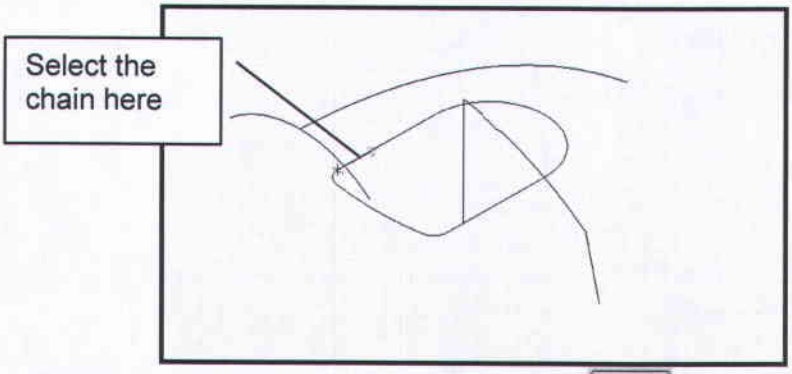
- Changes the **Color** to **#12**, the **Level** to **2** and the **Surface Density** to **2** as shown.
- Select the **OK** button to exit the **Attributes** dialog box.

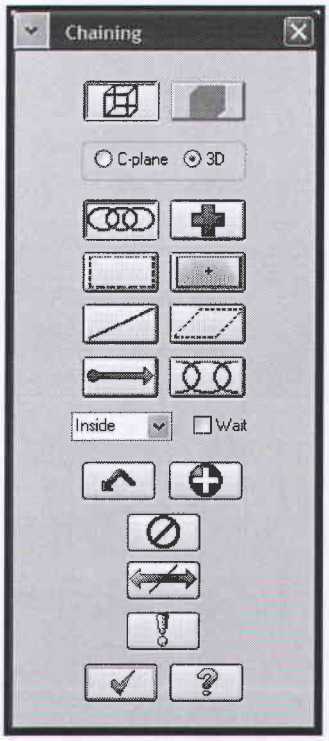


➢ Change the **Construction plane** to **Top**.

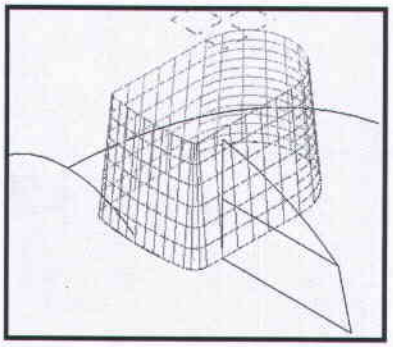
2.2 Create the draft surface.

Create
➢ **Surface**
➢ **Create Draft Surface**
➢ Select the chain as shown in the picture.





➢ Select the **OK** button to exit **Chaining**.
➢ Change the **Length** and the **Angle** as shown in the screenshot.
➢ The drawing should look like the following picture.

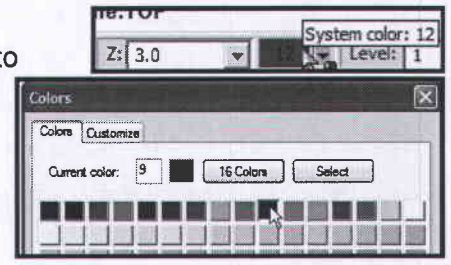


➢ Select the **OK** button to exit the command.
➢ Use the **Fit** icon to fit the drawing to the screen.

STEP 3:
CREATE THE SWEPT SURFACE.

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

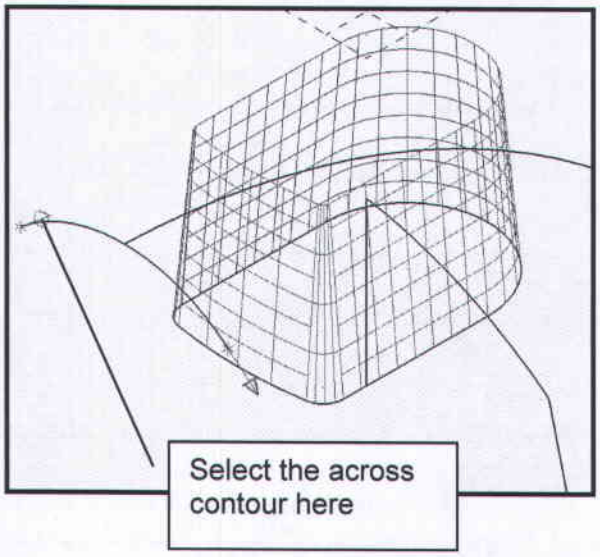
3.1 Change the Main color to blue (#9).

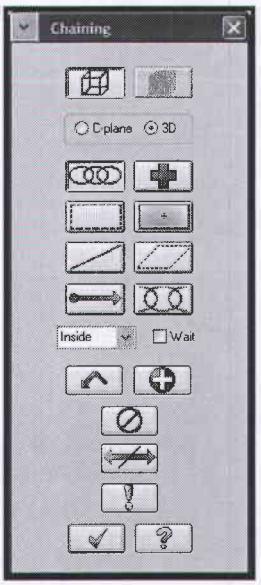


- Select **Color** from the **Status Bar** to change the main color to blue (#9).
- Select the color blue (**#9**).
- Select the **OK** button to exit **Colors**.

3.2 Create the Sweep surface.

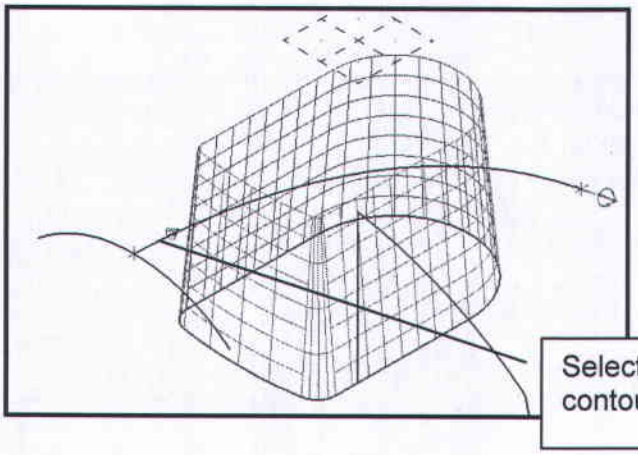
Create
- **Surface**
- **Create Swept Surfaces**
- [Define the across contour(s) 1]: Select Chain 1 as shown.

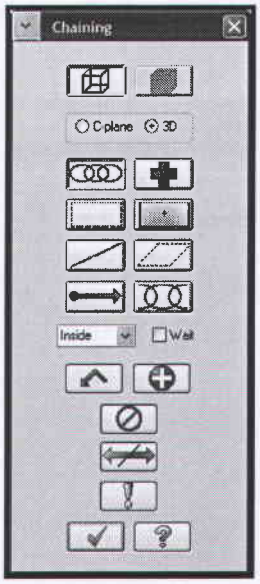




- Select the chains in the same directions as shown in the pictures. Please use the **Reverse** button from the **Chaining** dialog box if you need it.
- Select the **OK** button to exit Across contour chaining.

➢ [Define the along contour(s) 1]: Select Chain 1 as shown.

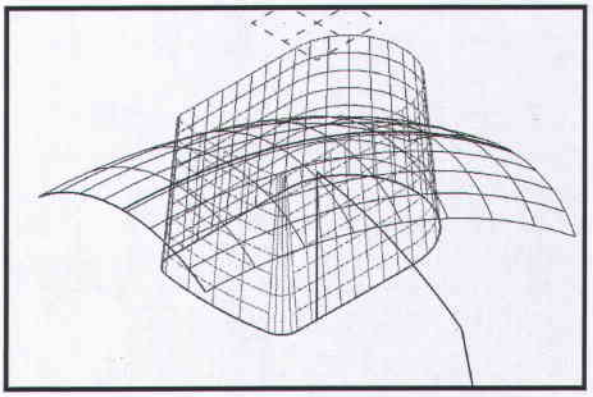




➢ Select the **OK** button to exit Along contour chaining.

➢ Select the **OK** button from the ribbon bar to complete surface swept creation.

➢ The geometry should look as shown to the right.



STEP 4:
CREATE THE FILLET SURFACES BETWEEN THE DRAFT AND THE SWEEP SURFACES.

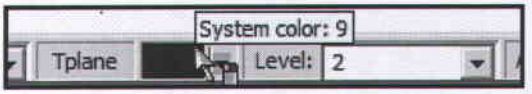
Fillet Surface: a surface generated by creating fillets (radius) that are tangent to the original surfaces. You can also create a fillet surface between a surface and a plane, and between a surface and a curve.
Application: used to smooth sharp edges.

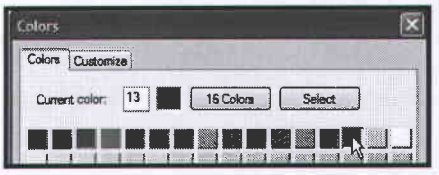
4.1 Change the Main color to magenta (#13).

➢ Select **Color** from the **Status Bar** to change the main color to magenta (**#13**).

➢ Select color magenta (**#13**).

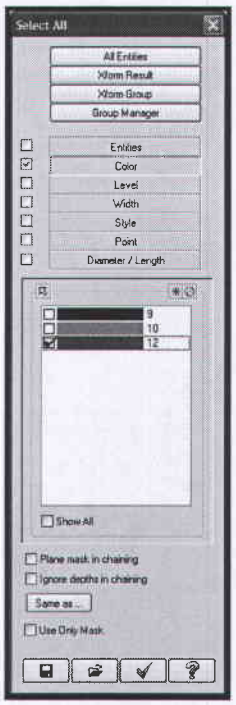
➢ Select the **OK** button to exit **Colors**.





4.2 Create fillet surfaces.

Create
- **Surface**
- **Fillet Surface**
- **Fillet Surface to Surface**



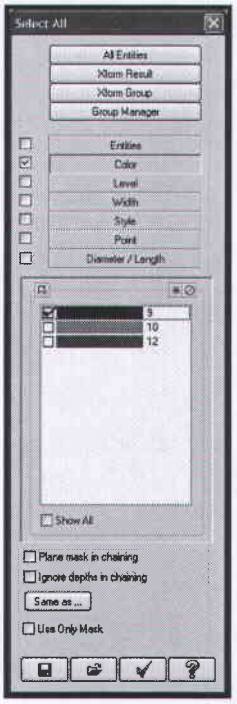
- [Select first set of surfaces]: Select the **All** button. 

- Change the parameter as shown, click on the **Color** button.
- Select the color **red.**
- Select the **OK** button. 

- Select **End Selection**. 

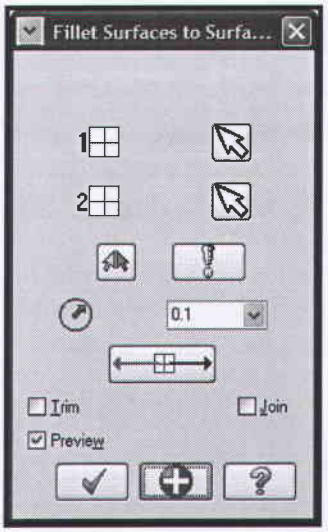
- [Select second set of surfaces]: Select the **All** button. 

- Select **color blue**.

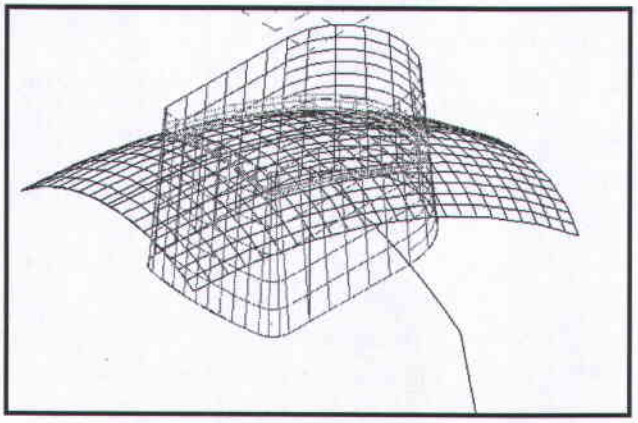


- Select the **OK** button. 
- Select **End Selection**. 

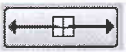
➢ Change the parameters as shown in the screenshot to the right.



- If the part looks as shown below, you have to change the surface normals.

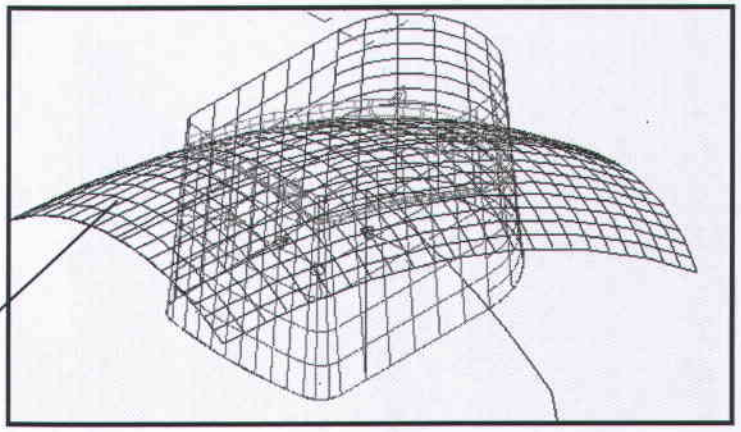


- The surface normals should always point toward the center curve of the resulting fillet surface.

➢ Select the **Flip Normal** button to change the surface normal. 

➢ The drawing should look as in the following picture.

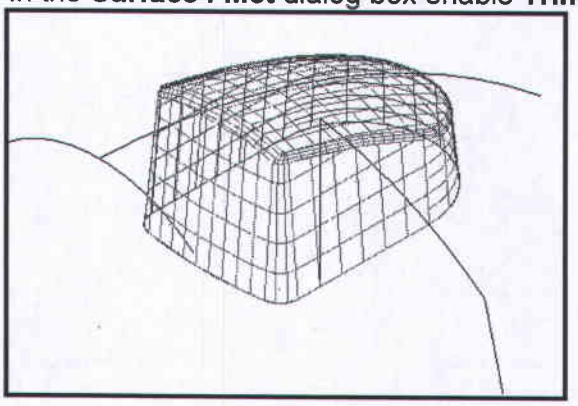
- Note that the surface normals for the red color surfaces are all orientated correctly towards the center of the part. The draft surface (blue) normal is orientated upwards. We will need to change it.

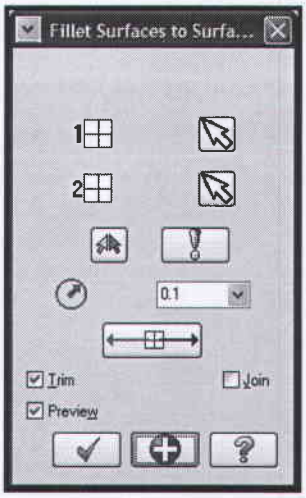


➢ [Click surface to flip normal]: Select Surface 1.

➢ Press **Enter**.

➢ In the **Surface Fillet** dialog box enable **Trim**.



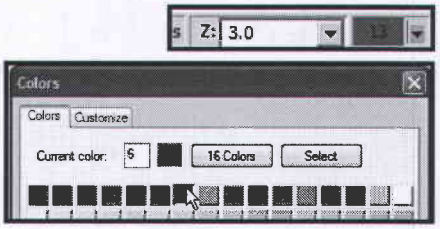


➢ Select the **OK** button to exit.

STEP 5:
CREATE THE REVOLVED SURFACES.

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

5.1 Change the Main color to brown (#6).



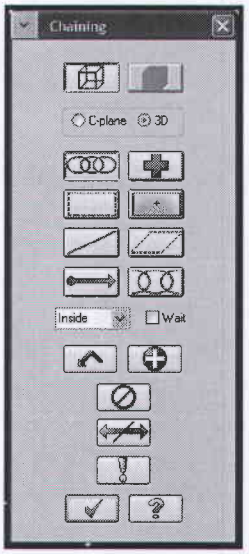
➢ Select **Color** from the **Status Bar** to change the main color to brown (#6).

➢ Select the color brown (**#6**).

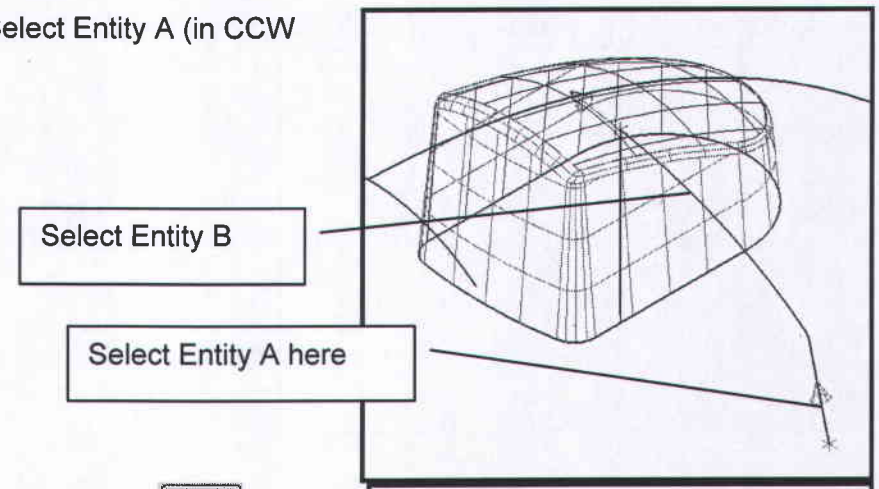
➢ Select the **OK** button to exit **Colors**.

5.2 Create the revolved surface.

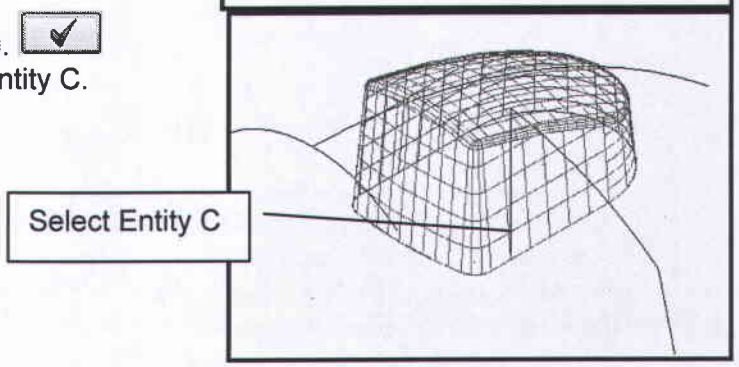
Create
➢ **Surface**
➢ **Create Revolved Surfaces**



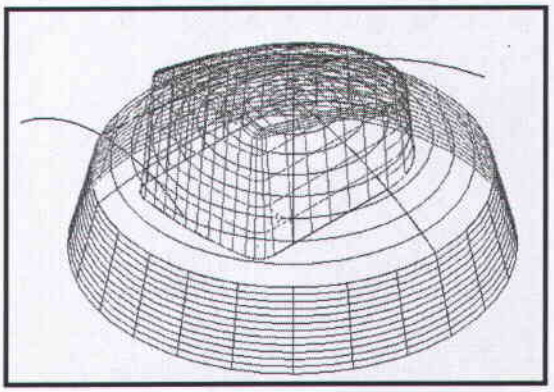
- [Select profile curve(s)]: Select Entity A (in CCW direction).
- Select Entity B.



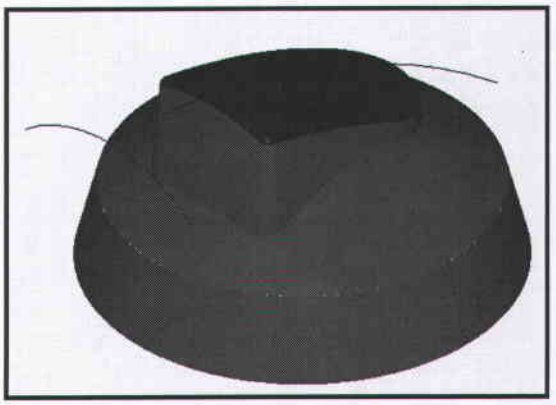
- Select the **OK** button to exit **Chaining**.
- [Select the axis of rotation]: Select Entity C.



- Make sure that the **End angle** is 360.
- Select the **OK** button.
- Use the **Fit** icon to fit the drawing to the screen.
- The surface should look as shown to the right.



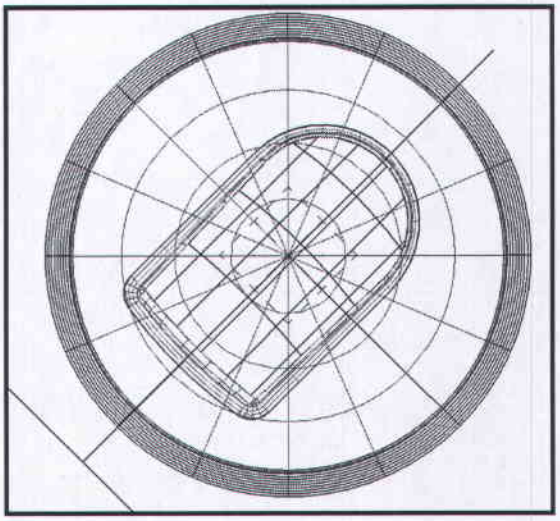
- To display the part in a shaded mode, **Alt +S**.
- To display the part back in wireframe mode, **Alt +S** again.



STEP 6:
CREATE THE FILLET SURFACES BETWEEN THE DRAFT SURFACES AND THE REVOLVED SURFACES.

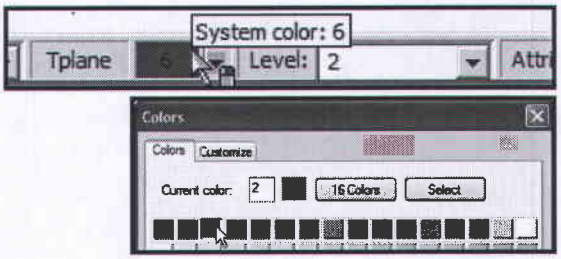
6.1 Change the graphic view by using Alt key and down arrow cursor.

- Hold down the **Alt** key and select the down arrow cursor key ↓ several times until the part looks as shown in the following picture.
- Screen fit .



6.2 Change the Main color to green (#2).

- Select **Color** from the **Status Bar** to change the main color to green (**#2**).

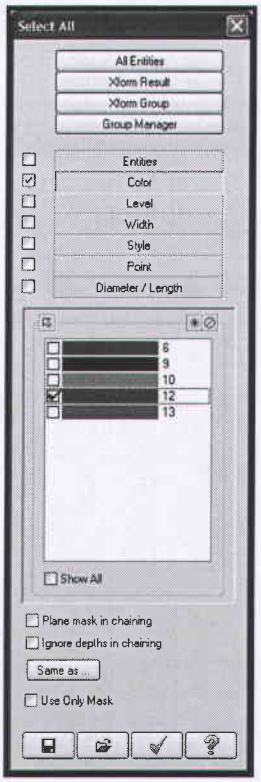


- Select the color dark green (**#2**).
- Select the **OK** button to exit **Colors**.

6.3 Create fillet surfaces.

Create
- **Surface**
- **Fillet Surface**
- **Fillet Surface to Surface**

- [Select first set of surfaces]: Select the **All** button.



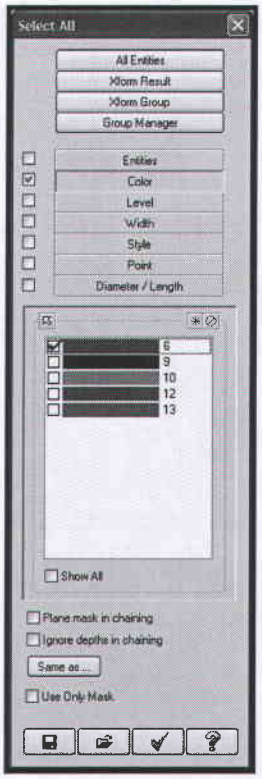
➢ Select the color red.

➢ Select the **OK** button.

➢ Press **Enter** to continue.

➢ [Select second set of surfaces]: Select the **All** button.

➢ Select the color brown.

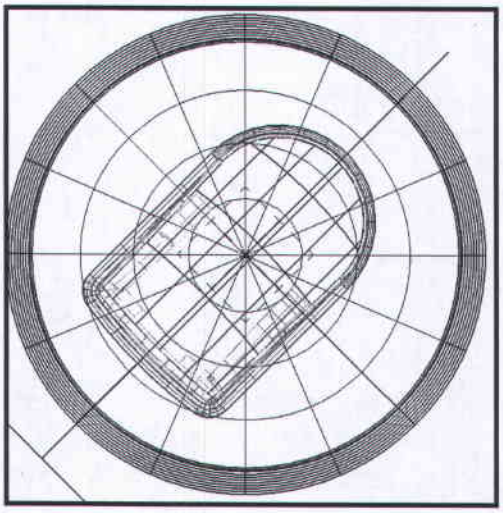


➢ Select the **OK** button.

➢ Press **Enter** to continue.

➢ Change the parameters as shown in the screenshot to the right.

- If the part looks as shown below, you have to change the surface normals.

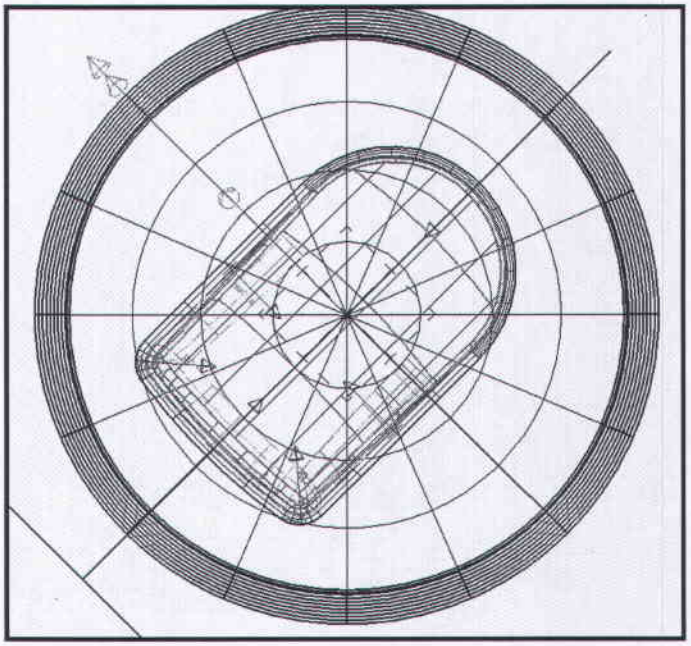


- The surface normals should always point toward the center curve of the resulting fillet surface.

➢ Select the **Flip Normal** button to change the surface normal. 

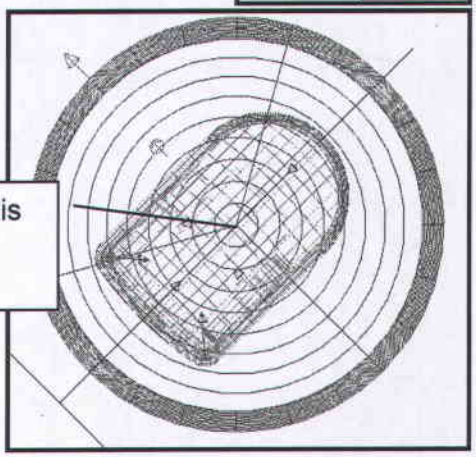
➢ The drawing should look like the following picture.

- Note that the surface normals for the red color surfaces are all orientated towards the center of the part. The revolved surfaces (brown) normals are orientated correctly outwards from the center of the part. We will need to change the normals of the six red color surfaces.

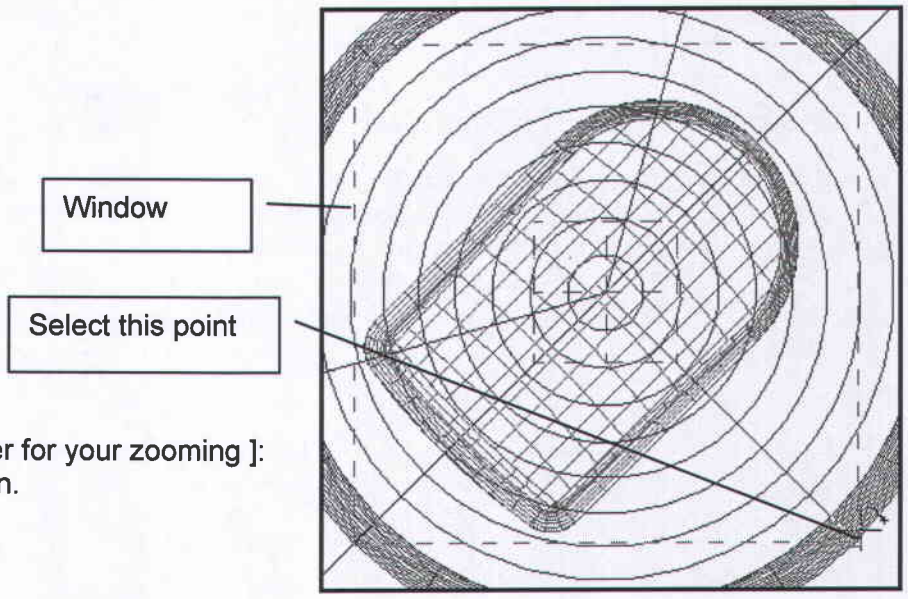


➢ Select the **Zoom Target** icon.

➢ [Pick point to zoom from]: Select the center of the part.

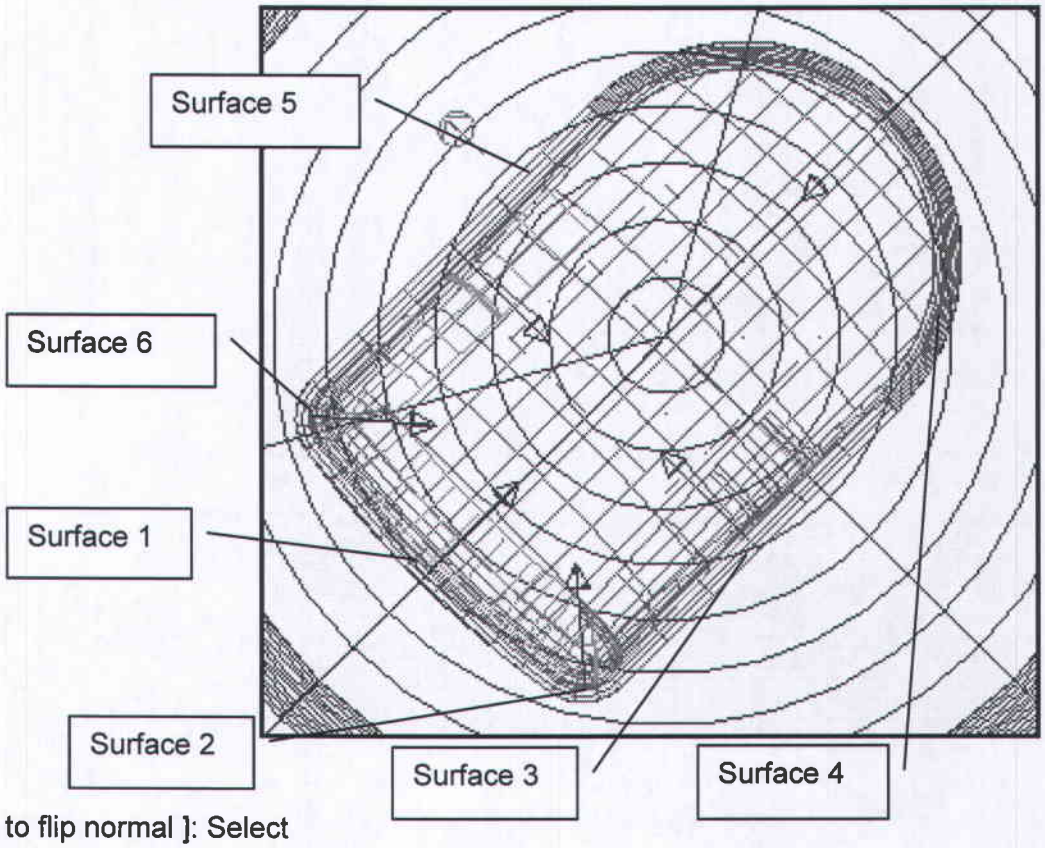


➢ Drag the cursor to make a window around the part as shown with hidden lines.

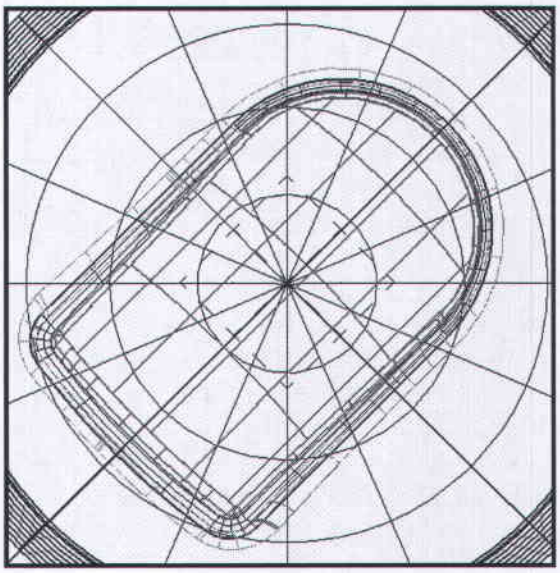


➢ [Choose a second corner for your zooming]: Select the point as shown.

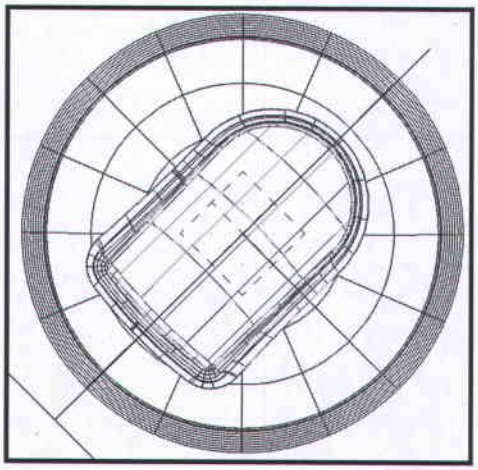
- [Click surface to flip normal]: Select Surface 1.
- Make sure that you select the surface on the "flowlines" used to display it. You might need to zoom the area too
- [Click surface to flip normal]: Select Surface 2.
- [Click surface to flip normal]: Select Surface 3.

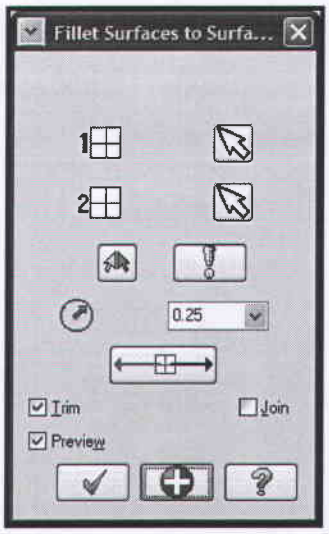


- [Click surface to flip normal]: Select Surface 4.
- [Click surface to flip normal]: Select Surface 5.
- [Click surface to flip normal]: Select Surface 6.
- Press **Enter**.
- The part should look as shown to the right.



➢ In the **Surface Fillet** dialog box enable **Trim**.
➢ The part should look as shown below.

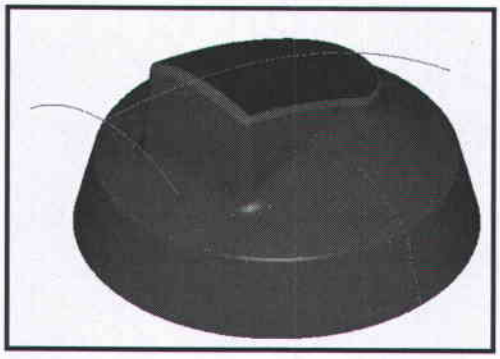


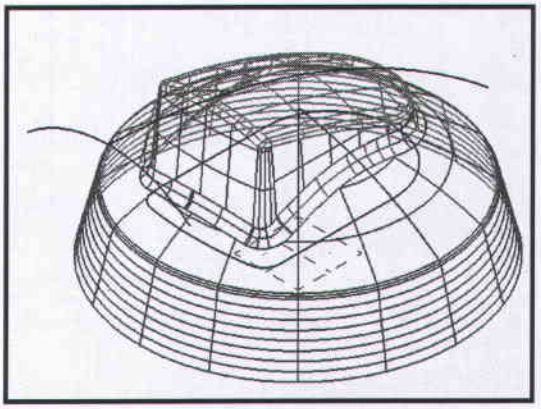


➢ Select the **OK** button to exit.

➢ Change the **Graphic view** to **Isometric**.

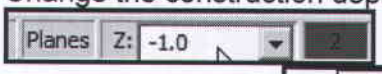
➢ The drawing should look as shown in the pictures below.



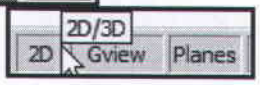


STEP 7:
CREATE A RECTANGLE.

➢ Change the construction depth to **Z-1.0**.



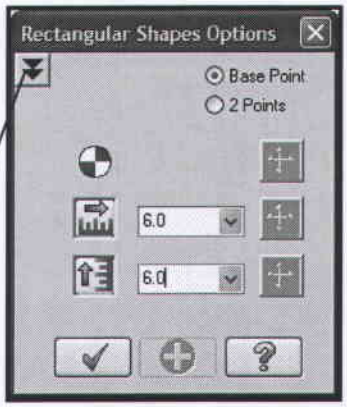
➢ Change to **2D** plane.



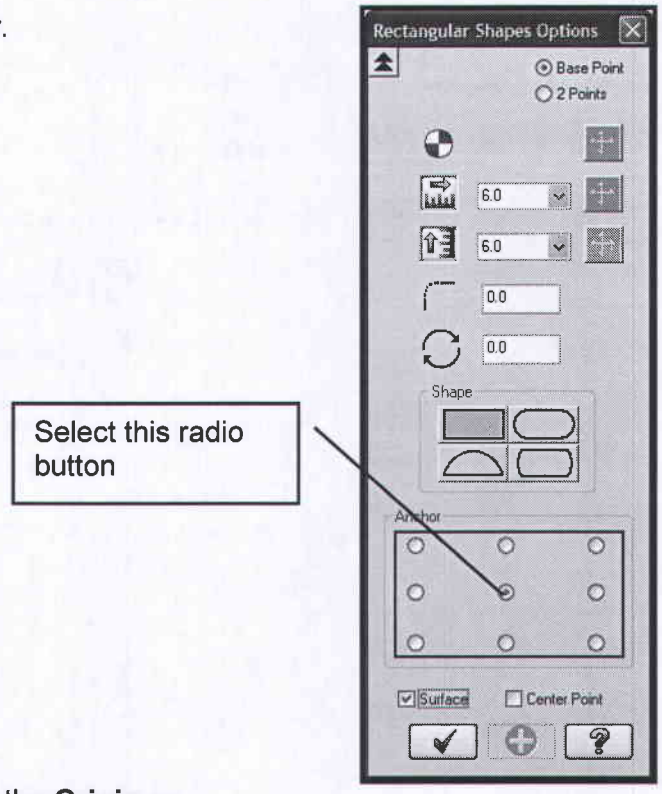
Create

➢ **Create Rectangular Shapes**
➢ Type the **Width** and the **Height** as shown in the screenshot to the right.
➢ Select the **double arrow** to expand the **Rectangle Options** if needed.
➢ Enable the **Surface** option.

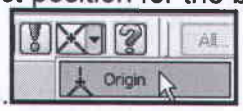
Select the double arrow



➢ Select the center radio button as the anchor.



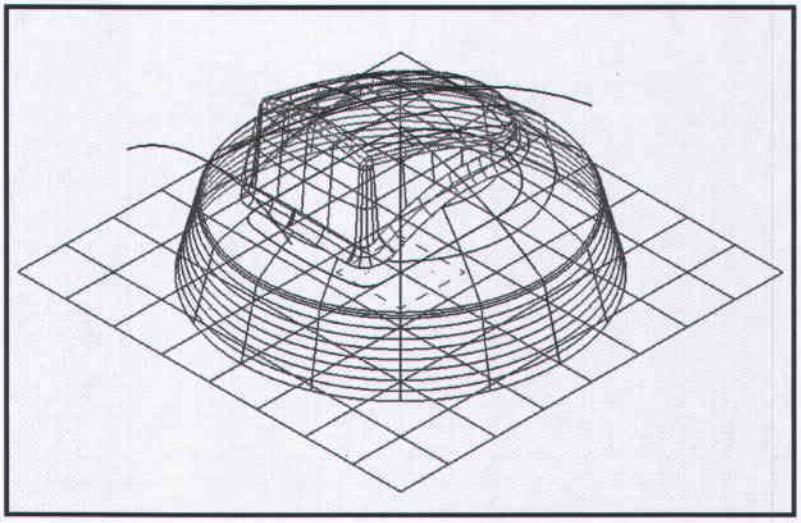
➢ [Select position for the base point]: Select the **Origin** as shown.



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

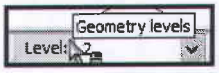
➢ Select the **Fit** button to fit the geometry to the screen.

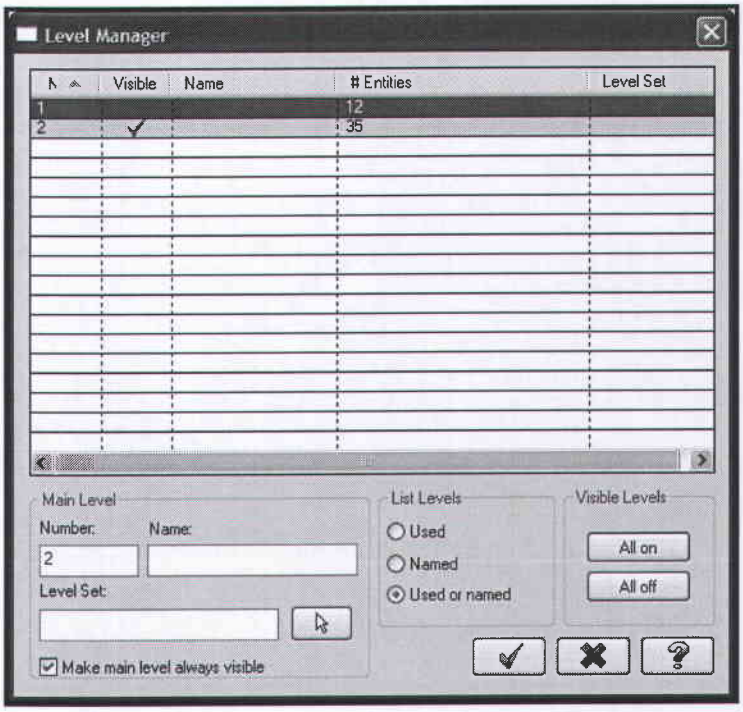
➢ The drawing should look as shown in the following picture.



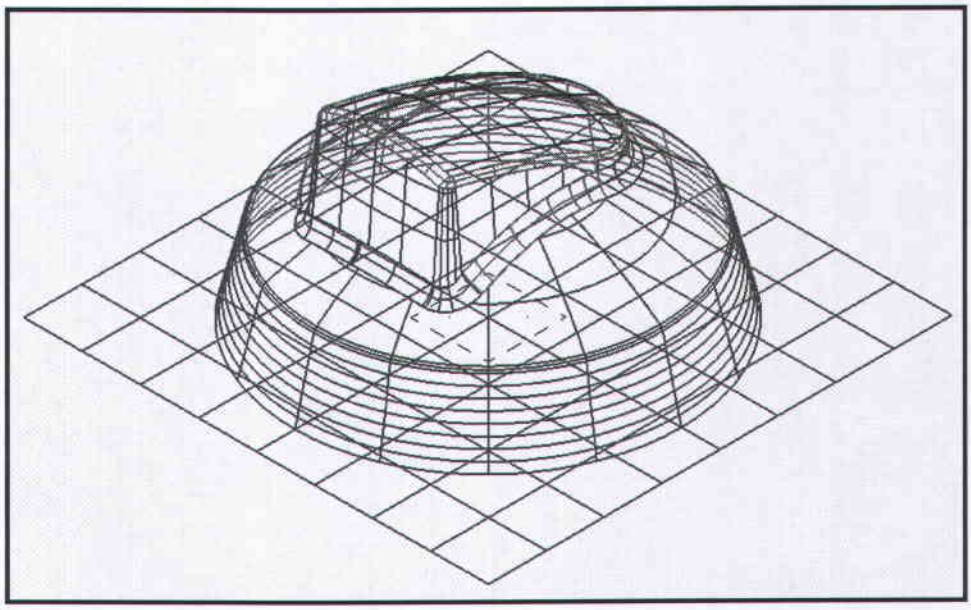
STEP 8:
HIDE THE 3D WIREFRAME.

➢ Select **Level** in the **Status Bar**.

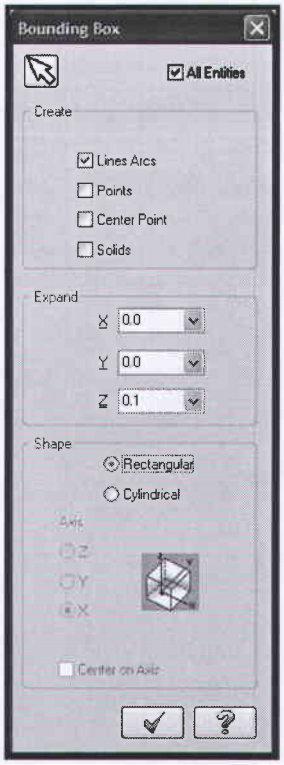
➢ Click on the check mark next to **Level 1** to disable it. 



➢ Select the **OK** button to exit. 



STEP 8:
CREATE A BOX TO REPRESENT THE STOCK.



➢ Set the plane to **3D**. 

Create

➢ **Create Bounding Box**

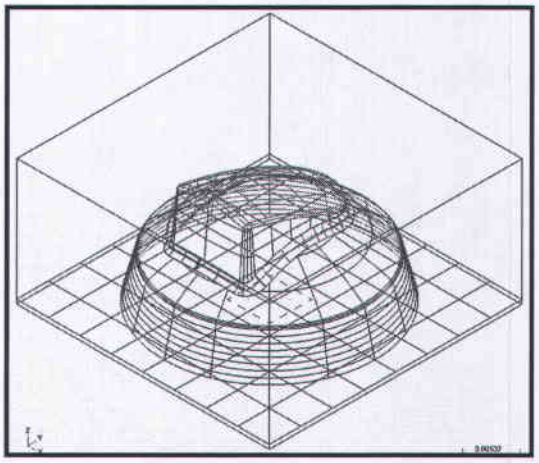
➢ Make the changes as shown in the screenshot to the right.

➢ **Expand** value along **Z**-axis is 0.1.

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

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

➢ The drawing should look as shown in the following picture.



STEP 9:
SET UP THE DATUM (WORK ZERO) AT THE TOP CENTER OF THE BOX.

Xform

➢ **Xform Translate**

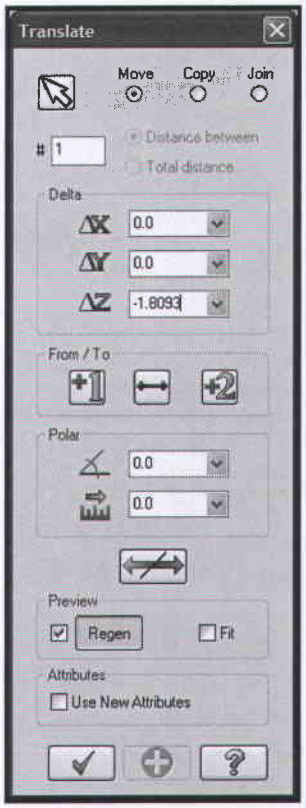
➢ Select the **All** button. 

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

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

➢ Enable **Move**.

➢ Enter Delta Z -1.8093.



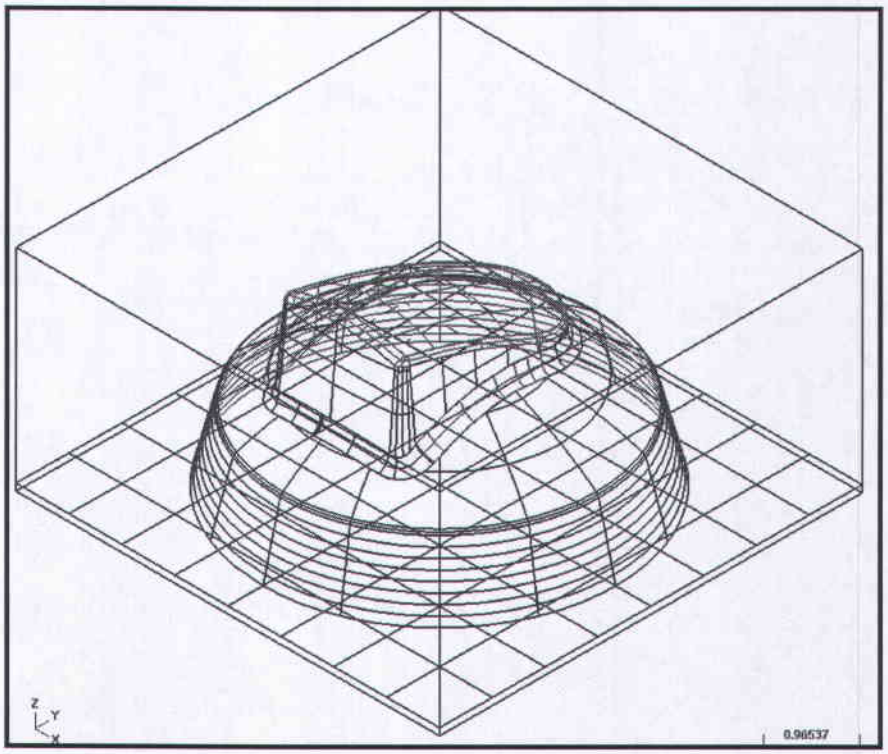
➢ Select the **OK** button to exit the **Translate** dialog box.

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

Screen

➢ **Clear Colors**

➢ The final geometry should look as shown below.



STEP 10:
SAVE THE DRAWING.

File
➢ **Save as**
➢ **File name**: "Your Name_14"
➢ Select the **OK** button.

TOOLPATH CREATION

STEP 11:
SET UP THE STOCK TO BE MACHINED.

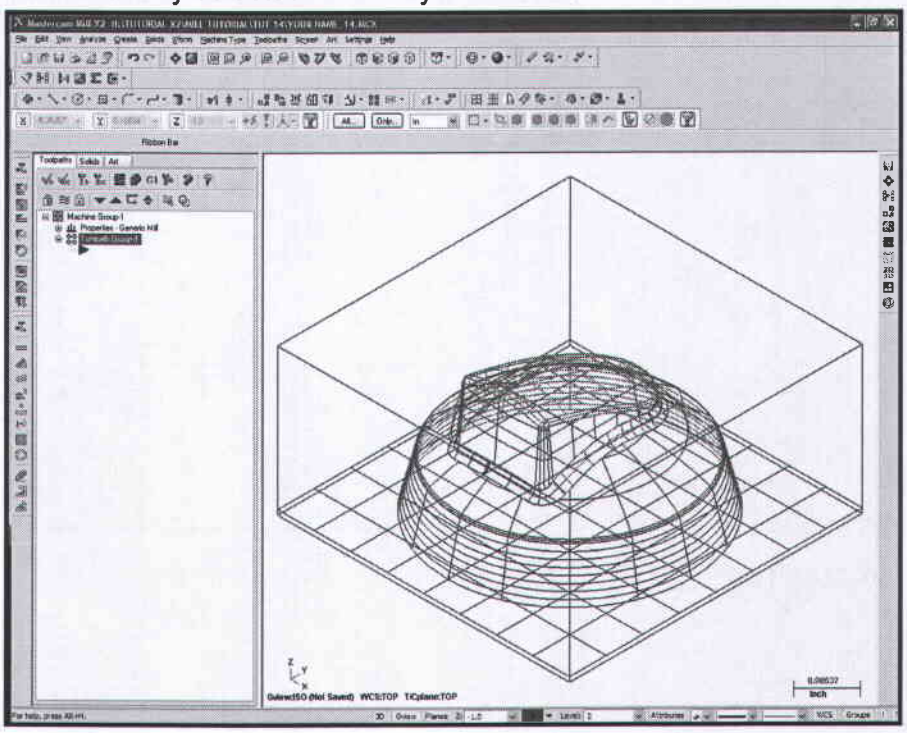
➢ To display the **Toolpaths Manager** press **Alt + O**.

Make sure that no machine is already selected. You only need one.

Machine Type

➢ **Mill**

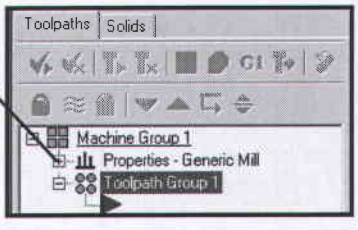
➢ Select **Default**.



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

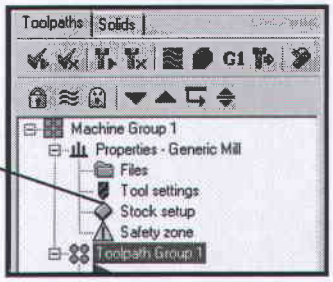
➢ Select the plus in front of **Properties** to expand the **Toolpaths Group Properties.**

Select the plus



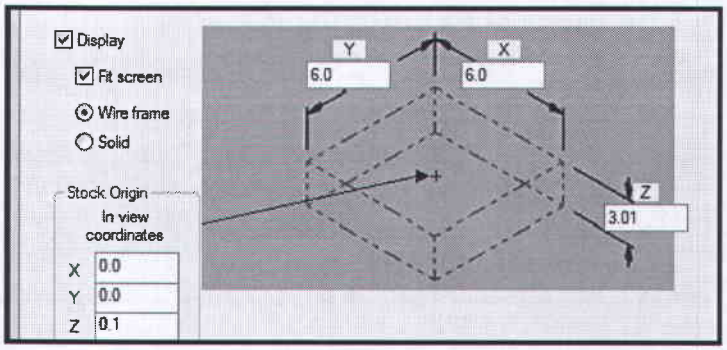
➢ Select **Stock setup**.

Select the Stock



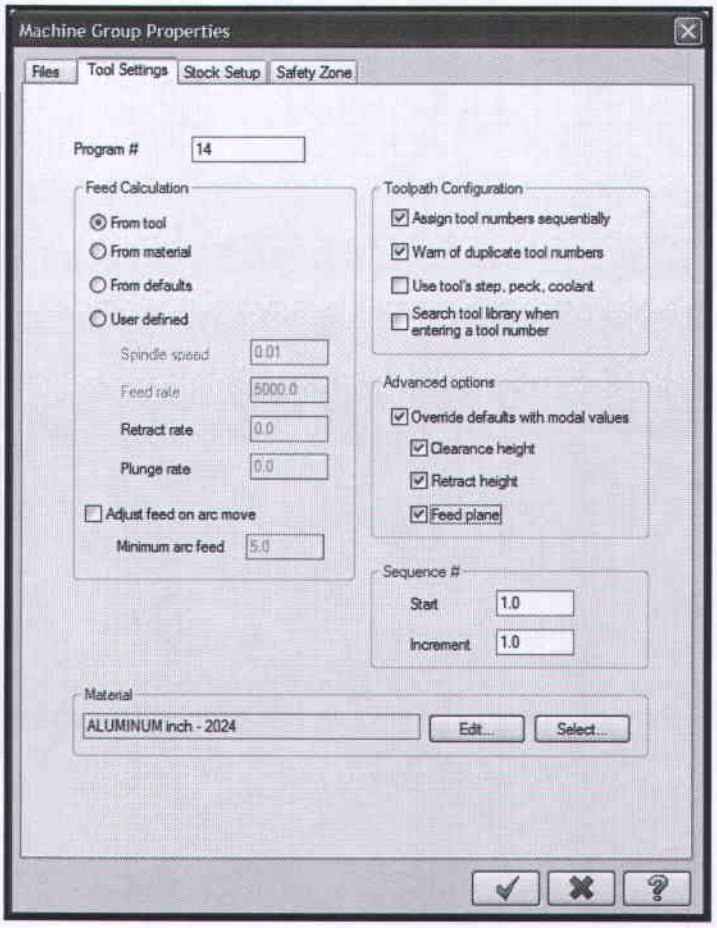
➢ Change the parameters to match the following screenshot.

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



➢ Select the **Tool Settings** tab to set the tool parameters and the part material.

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.



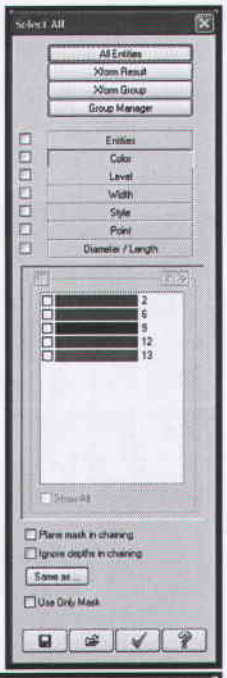
➢ Select the **OK** button to exit **Toolpath Group Properties**.

STEP 12:
HIGH SPEED - CORE ROUGHING.

Core roughing toolpaths are designed for machining cores which can be approached from the outside. An important feature of core roughing is that Mastercam can change the machining strategy within the same operation if your part has, for example, a mixture of bosses and cavities. In these cases, Mastercam will cut the cavities inside to out (like an area clearance cutting pass), and machine the bosses from the outside.

Toolpaths

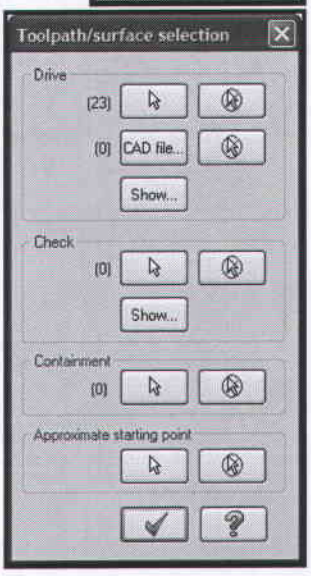
- **Surface High Speed Toolpath**
- **Select the OK button to accept the NC name**
- [Select Drive Surface]: Select the **All** button.



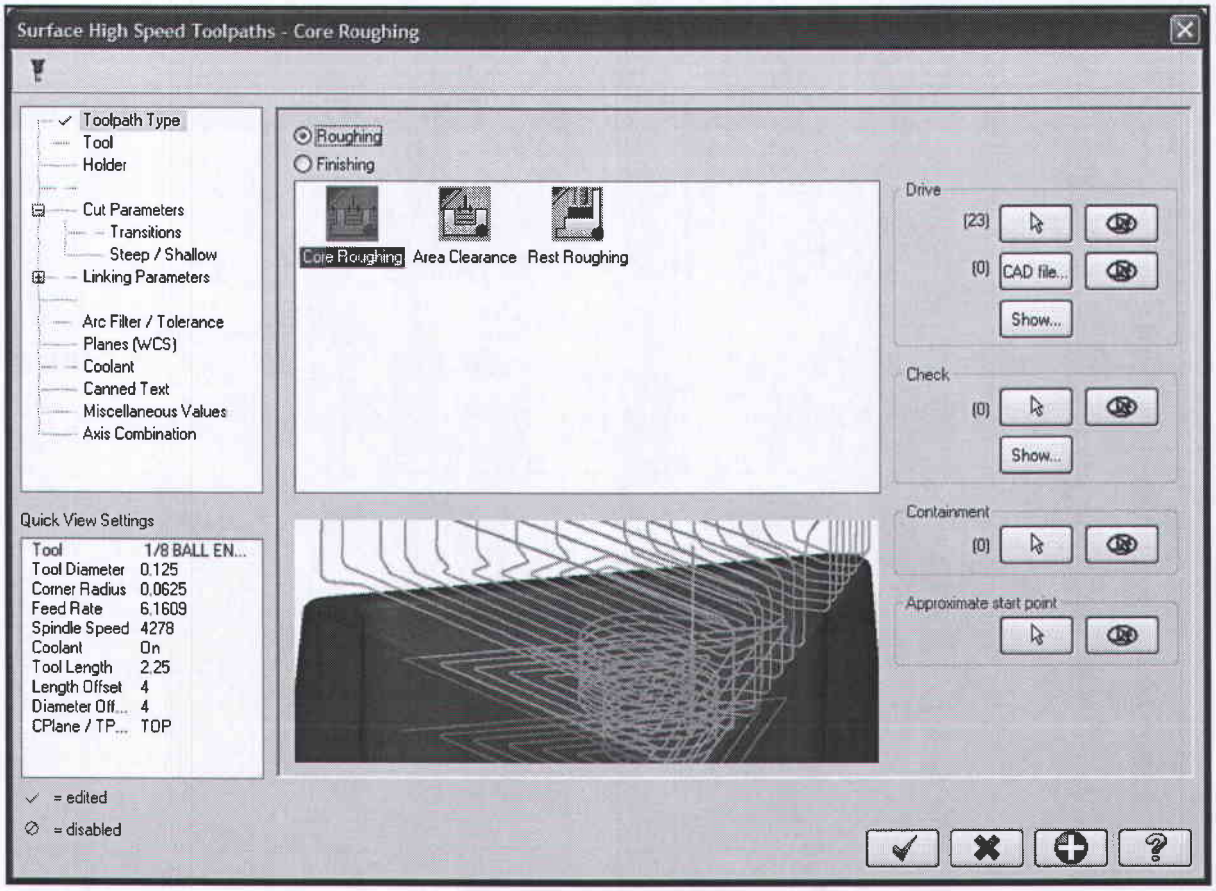
- Select the **OK** button to exit.

- Press **Enter** key.

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

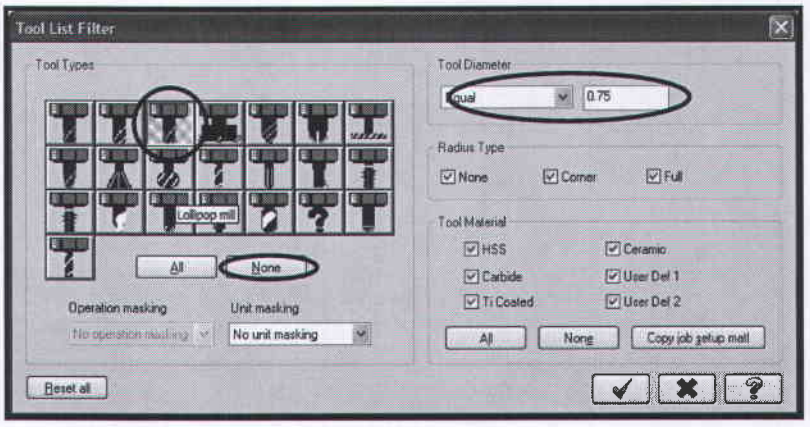


- Select **Toolpath Type** and enable **Roughing**
- Select **Core Roughing**

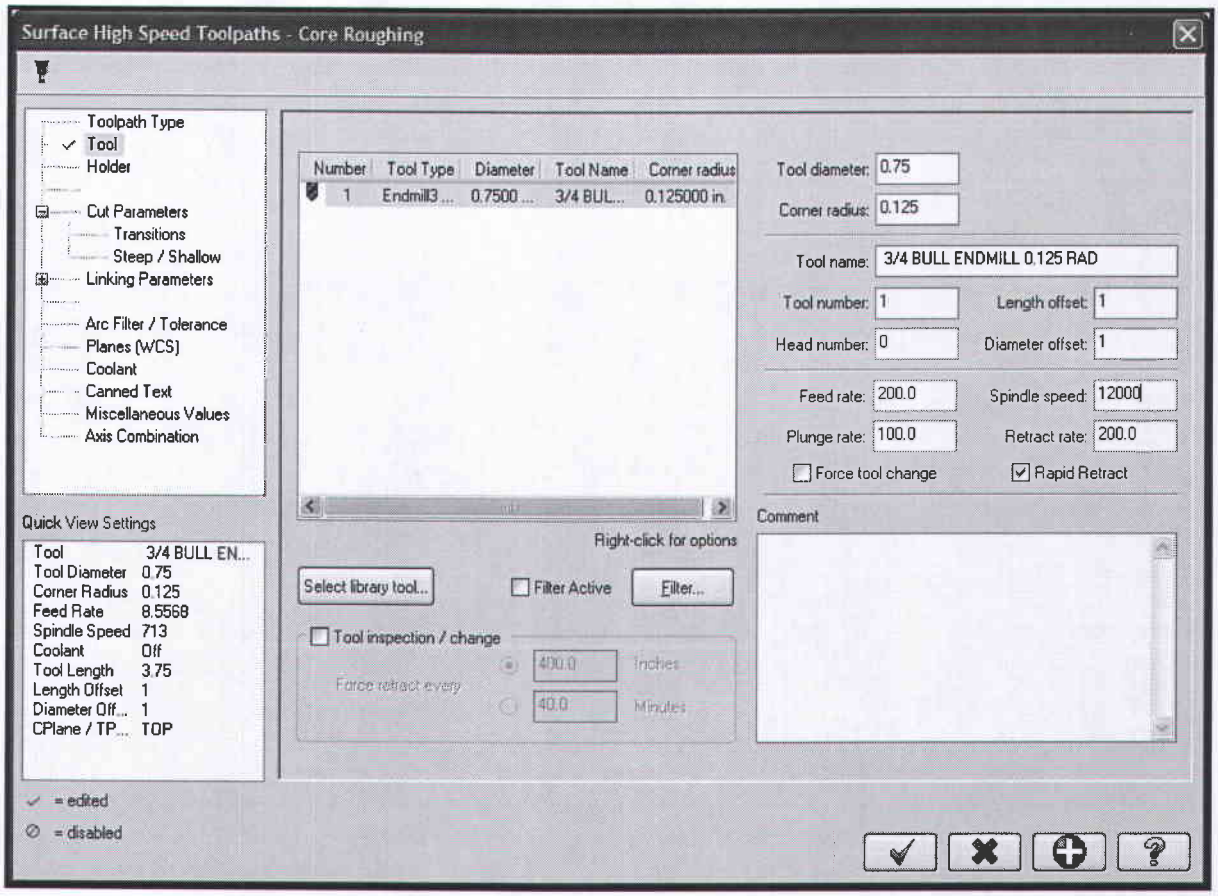


- Select **Tool**.
- Click on the **Select library tool** button.

- Select the **Filter** button in the **Tool Selection** dialog box.
- Select the **None** button in the **Tool Types** area.
- Click on the **Bullnose** icon to select the tool type.
- Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.
- Enter the diameter 0.75.
- Select the **OK** button to exit.

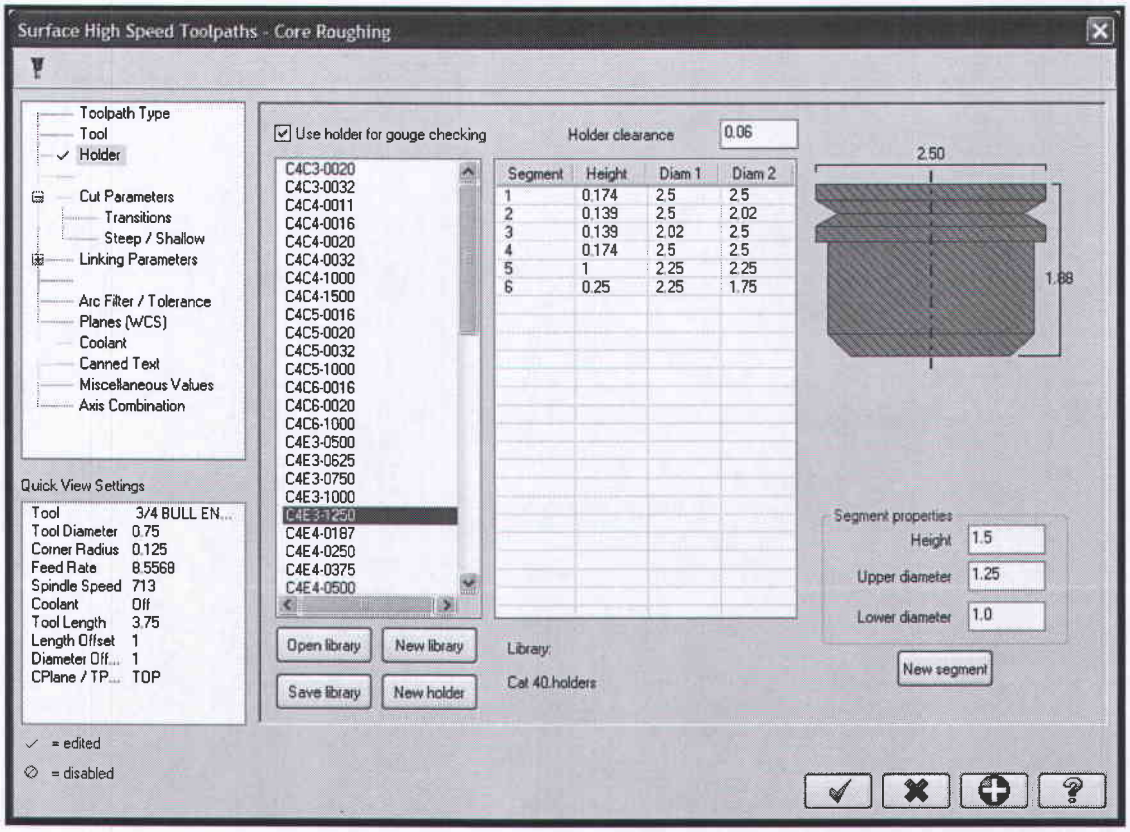


- Select **0.75" Bull Nose** with a **0.125" corner radius**.
- Select the **OK** button to exit the **Tool Selection** dialog box.
- Make the necessary changes in the **Toolpath parameters** to match the following screenshot.



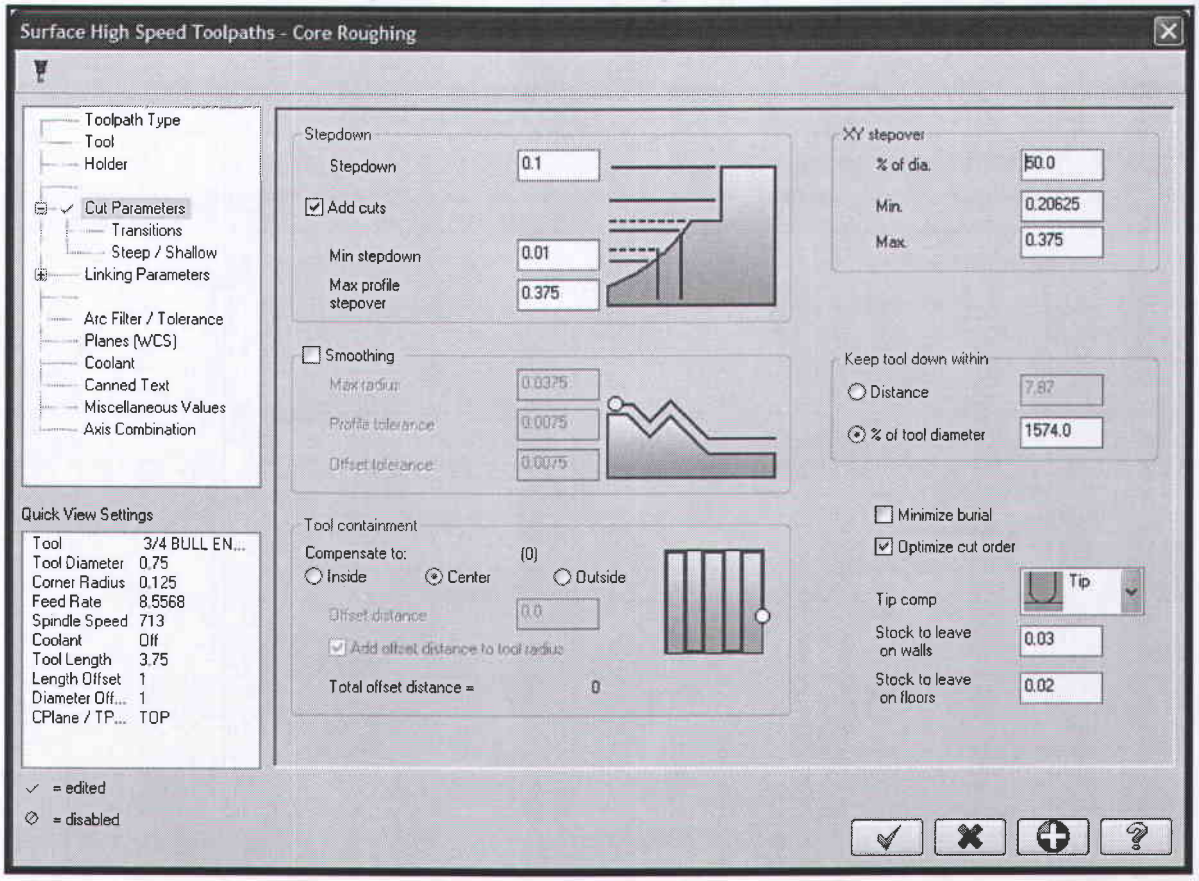
➢ Select the **Holder**.

➢ Select **Open library button**.
➢ Select the **Cat 40.holders** and **Open** 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 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 to establish the minimum separation between the holder and your surface model. Set the clearance bigger than the stock to leave on the walls.

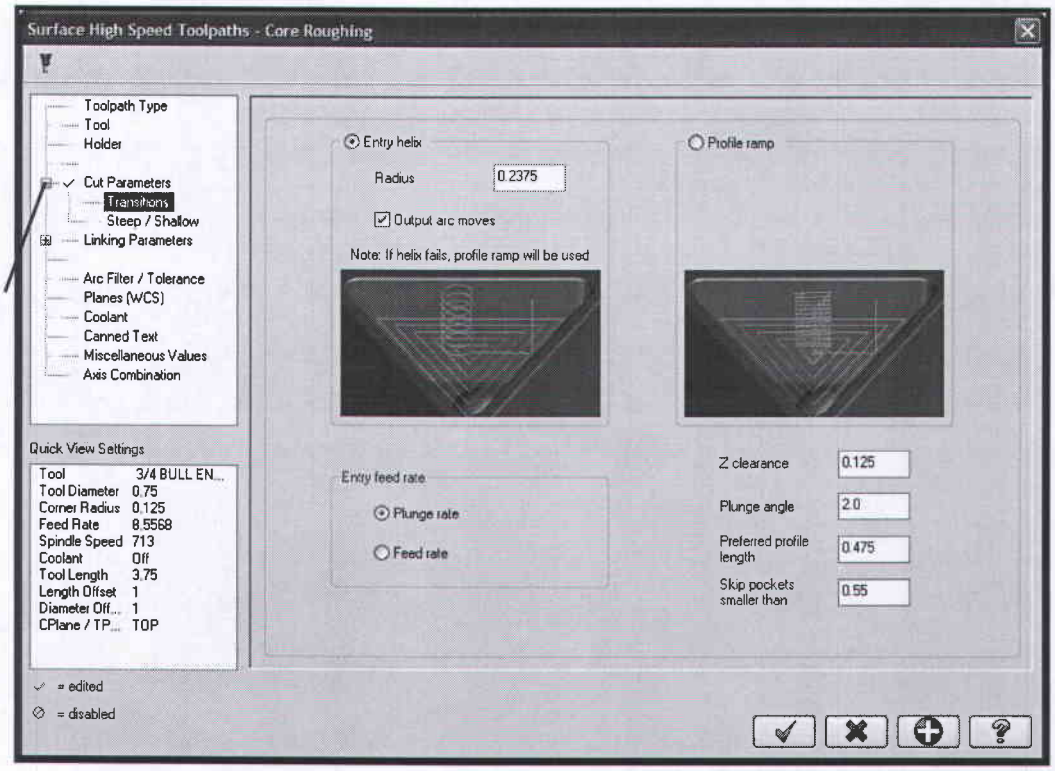
➢ Select the **Cut Parameters** page and make the changes to match the following screenshot.



Stepdown options allows you to configure how Mastercam spaces the cuts in Z.
Stepdown 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 stepdown.
XY stepover settings allows 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 doesn't allow the tool to retract if the distance between the end of one pass and the start of the next pass is less than the distance.

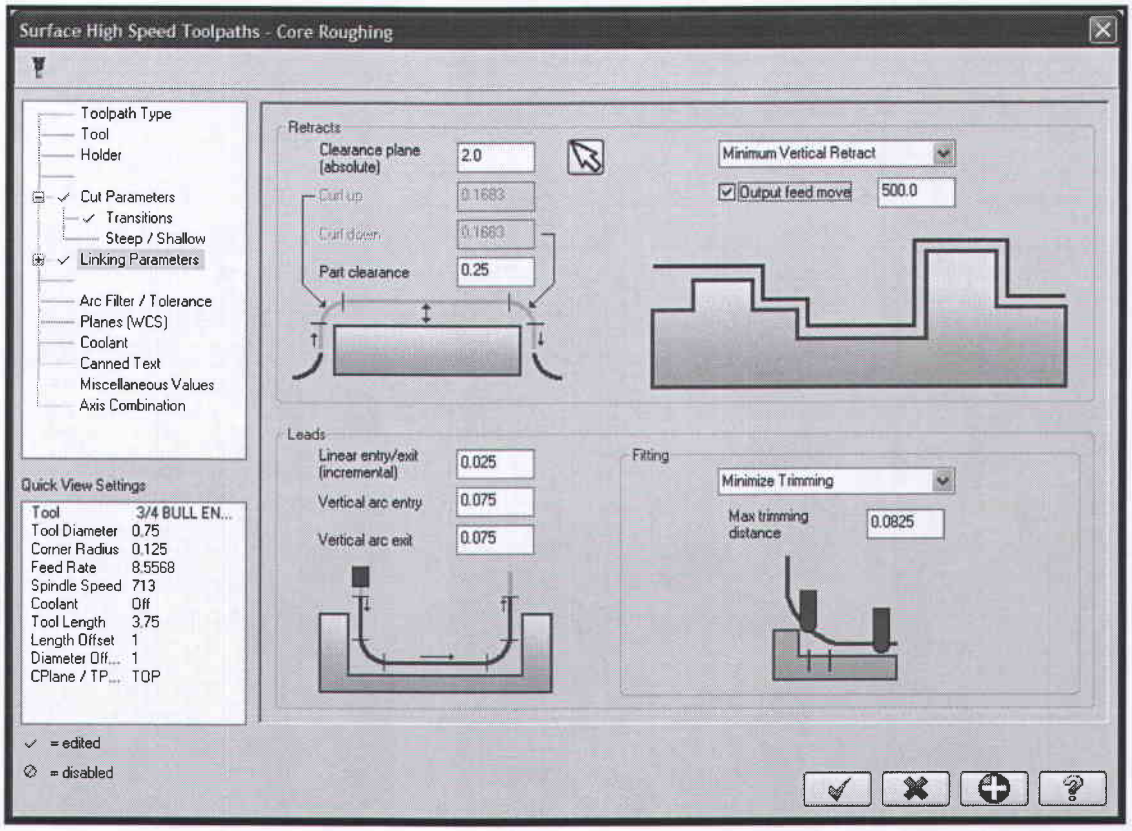
➢ Expand **Cut Parameters** and select **Transitions** to set the **Entry helix**.

Select the plus to expand



Transiton allows 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 allows 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.

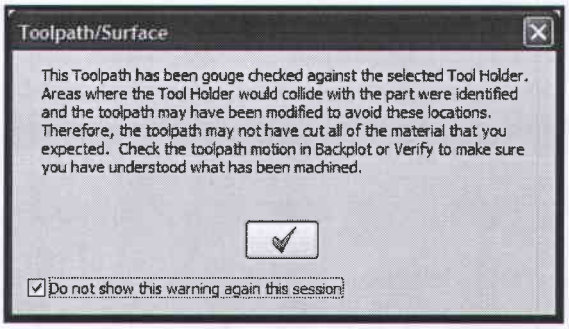
Fitting settings allows you to choose how the entry and exit arcs fit to the ends of the cutting passes.

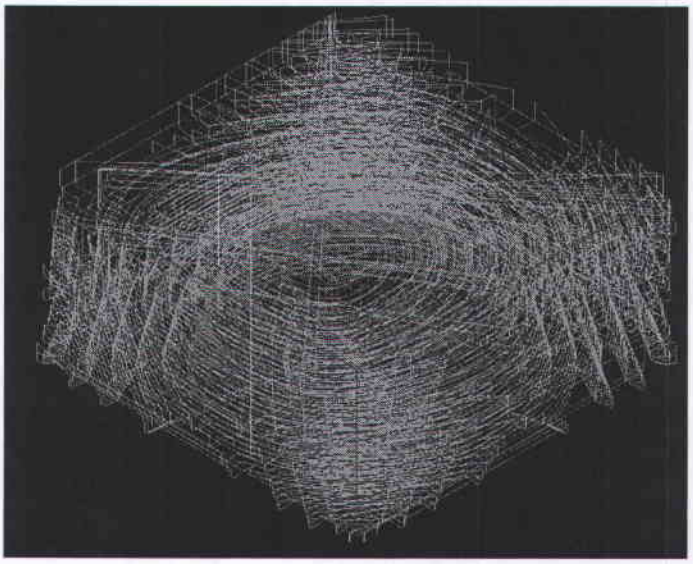
Minimize trimming sets the path of the retract to be as close to the surface as possible, maintaining a minimum distance from the surface to fit the arc.

Max trimming distance parameter limits the amount of trimming applied to non-horizontal passes.

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

- Enable **Do not show this warning again this session**
-
- Select the **OK** button to accept the warning.

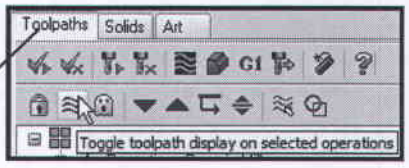




STEP 13:
VERIFY THE ROUGH TOOLPATH AND SAVE IT AS A STL FILE TO BE USED AS AN INTERMEDIATE STOCK.

- Select the **Toolpaths manger** tab to enable it.
- Select **Toggle toolpath display on** selected operations to remove the toolpath display.

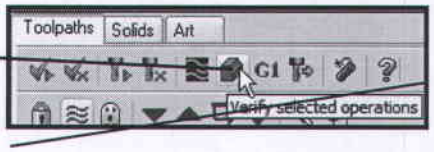
Select Toolpaths manager



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

Select Verify

Drag the right side to expand it

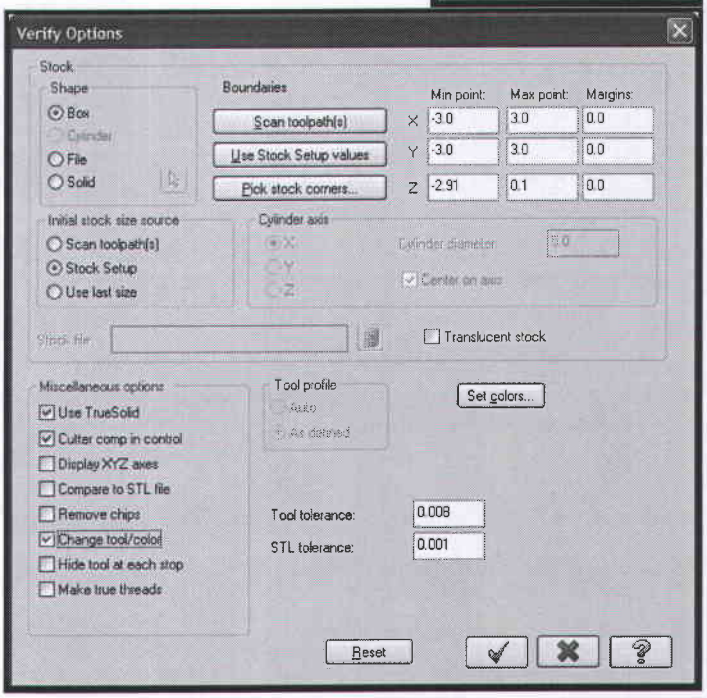


Update after each toolpath updates the stock after each operation.

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

- Enable **Turbo** mode.
- Select the **Configure** button.

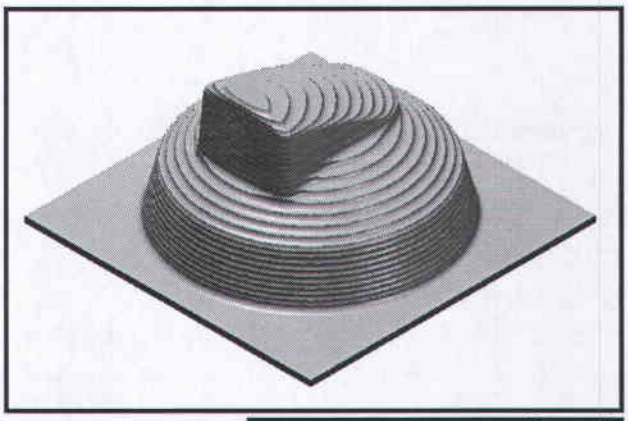
Initial stock size source should be set to **Job Setup** to use the stock information from Stock Setup.
Use True Solid allows you, after verifying the part, to rotate and magnify it to more closely check features, surface finish, or scallops.
Change tool/color to change the color of the cut stock to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



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

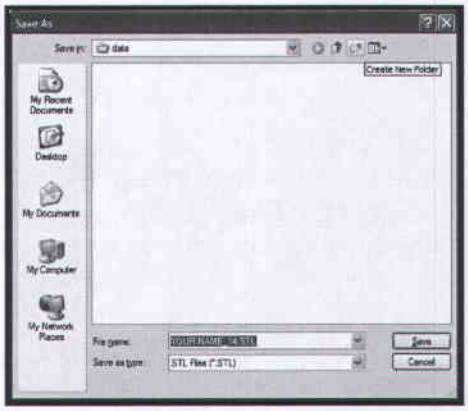
➢ Select **Machine** button to start simulation.

➢ The finished part should appear as shown to the right.



➢ Select the **Save stock as a file** button.

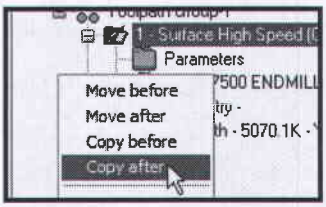
➢ Save the **STL file** with the same name as the geometry file.
➢ Select the **Save** button.
➢ Select the **OK** button to exit **Verify**.



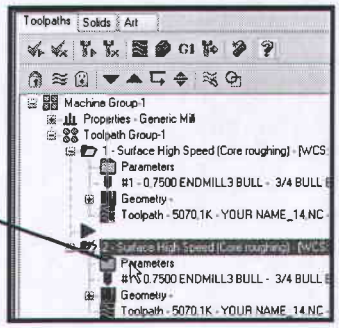
STEP 14:
HIGH SPEED – FINISH HORIZONTAL AREA.

Horizontal Area toolpaths are designed for machining the flat areas of your surface model. Mastercam will create cutting passes at the Z height of each area. Even if the drive surface as a whole is not flat, Mastercam will identify and only machine the flat areas. Mastercam will automatically calculate the toolpath in such a way that the tool does not exit on a sidewall.

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

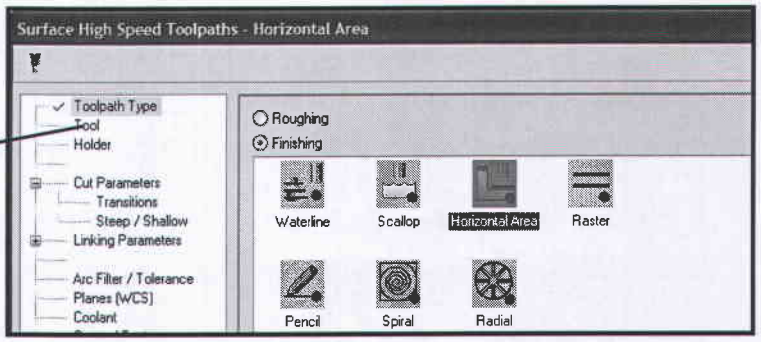


- You should now have two operations.
- Left-click on the second operation **Parameters**.



Select Parameters

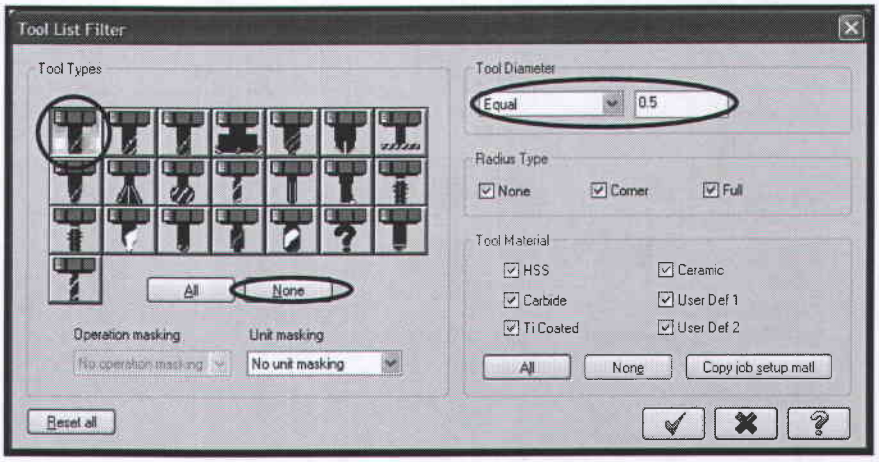
- Select **Toolpath Type**
- Enable **Finishing** and choose **Horizontal Area**



Select Tool page

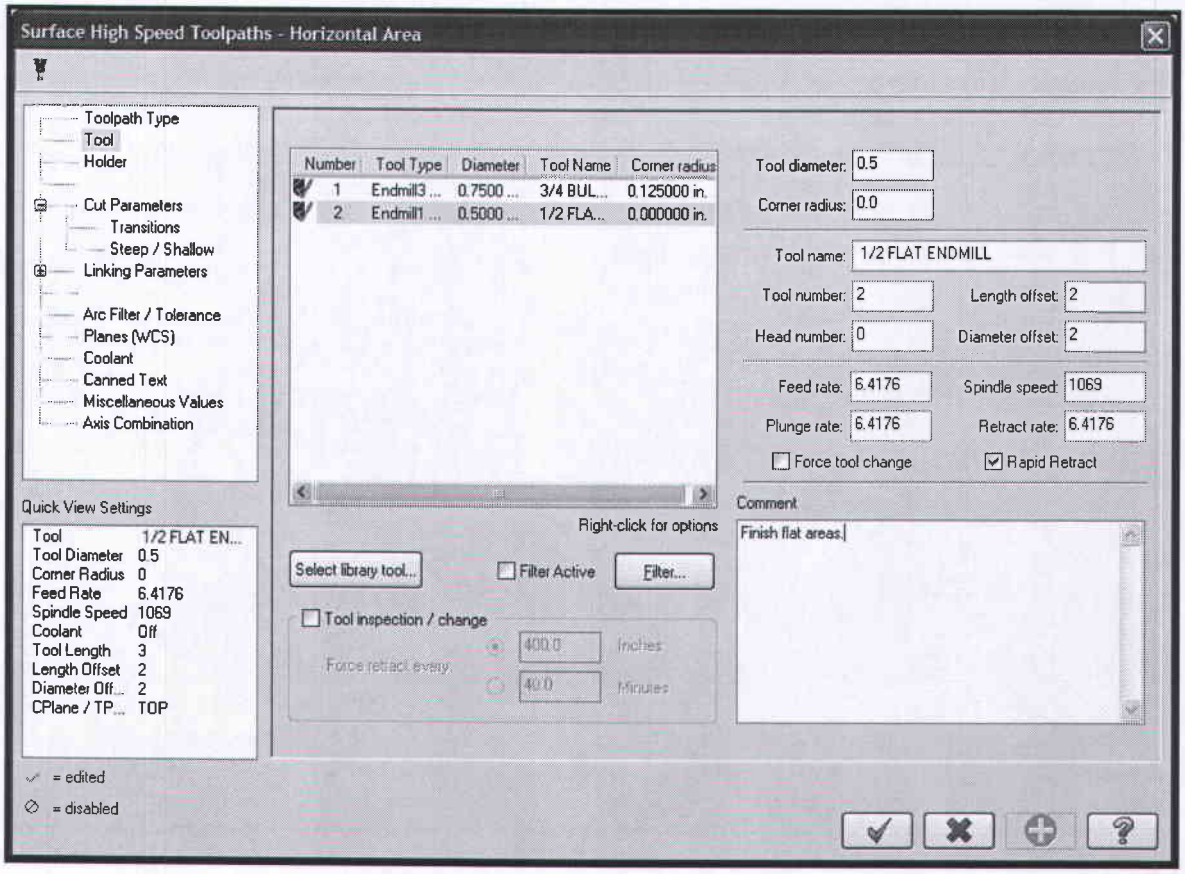
- Select **Tool** page and click on the **Select library tool** button. 

- Select the **Filter** button.
- Make sure that you select the **None** button first and then the **Endmill Flat** as the tool type.



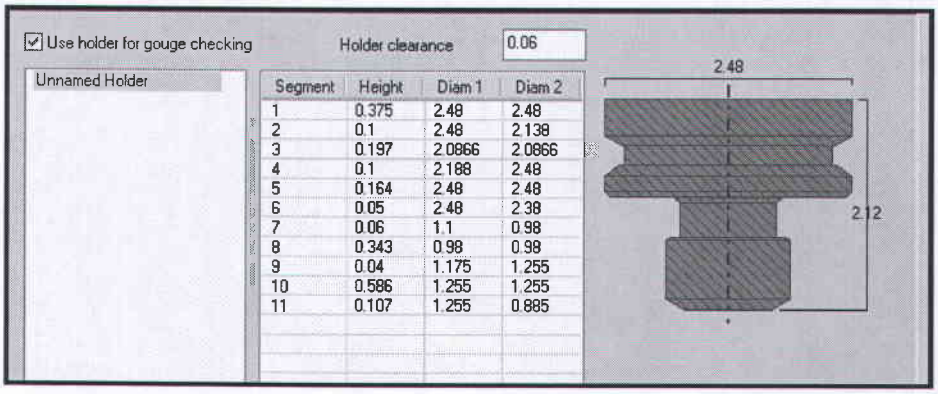
- Change the **Tool Diameter** to **Equal** and 0.5".
- Select the **OK** button to exit **Filter**. 
- Make sure that the tool is selected and select the **OK** button to exit **Tool Selection**. 

➢ Change the parameters to match the following screenshot.

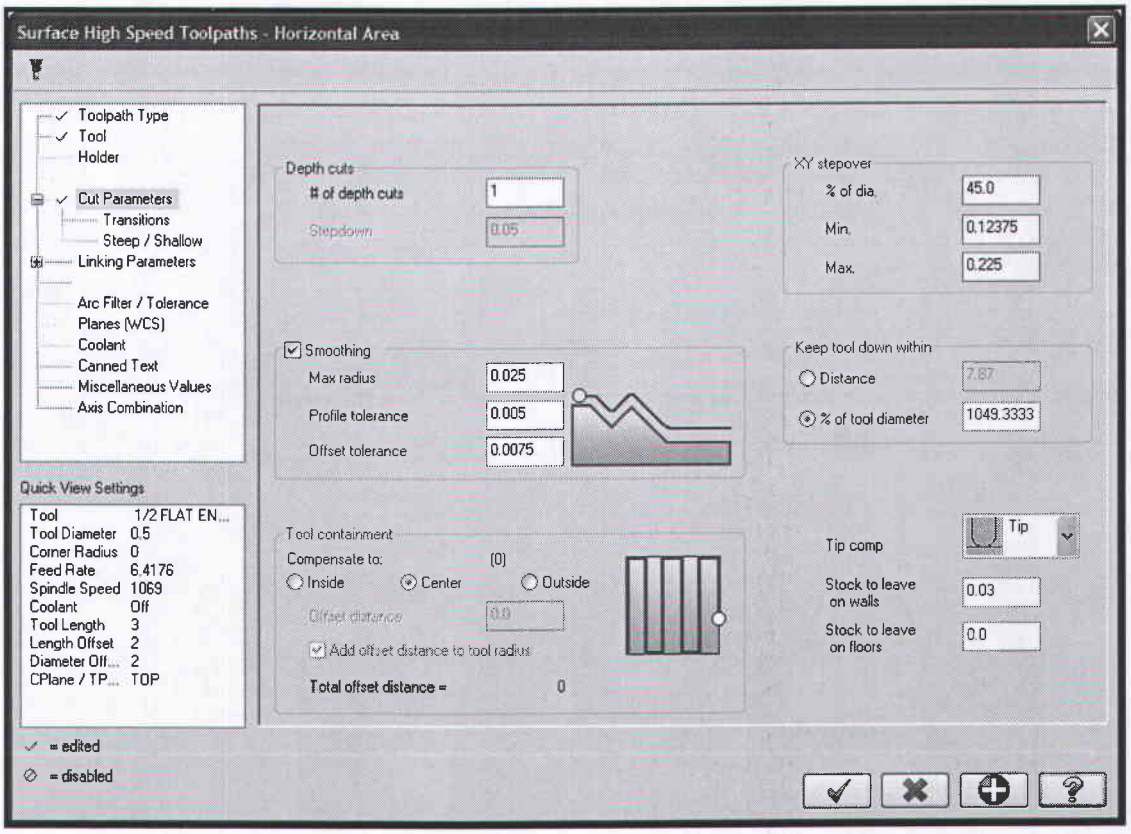


➢ Select the **Holder** page

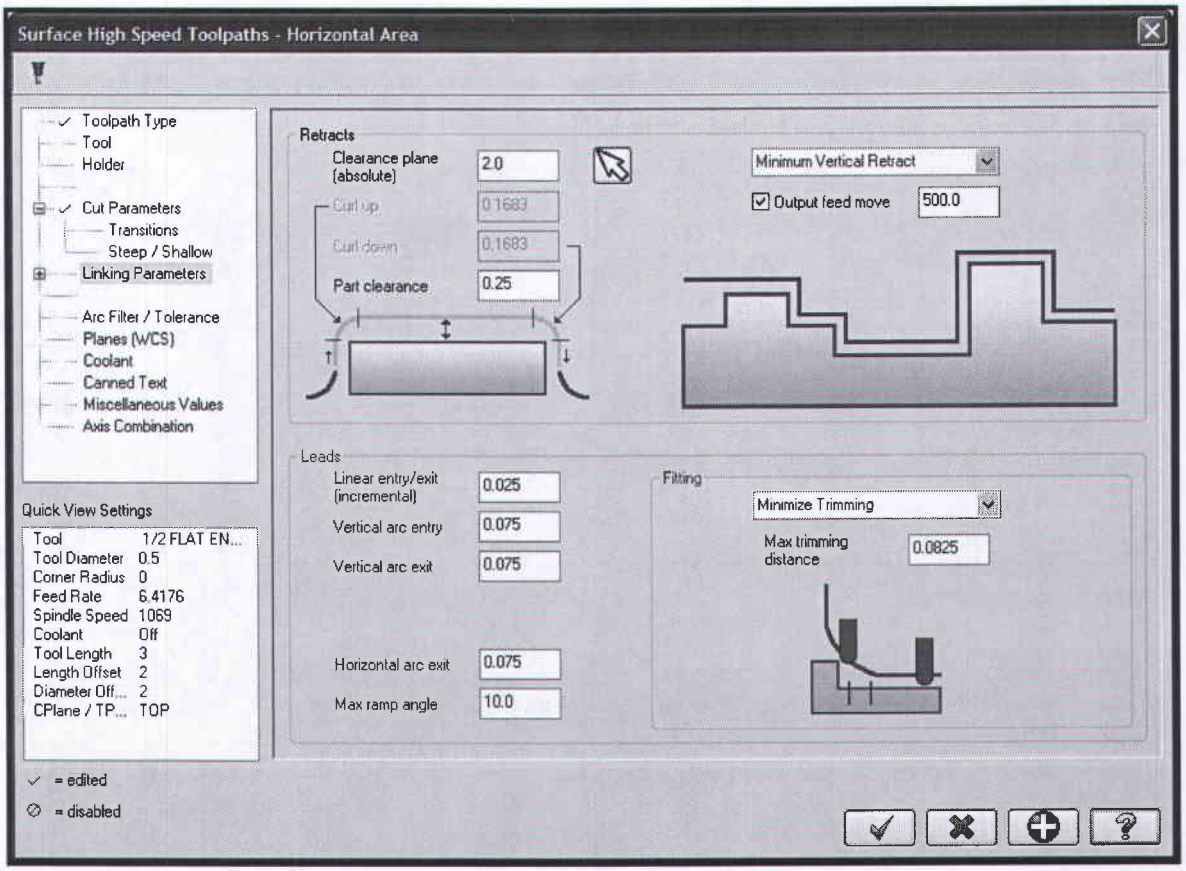
- Note the holder is already selected from the previous operation that we copied.



➢ Select **Cut Parameters** and change the parameters as shown.

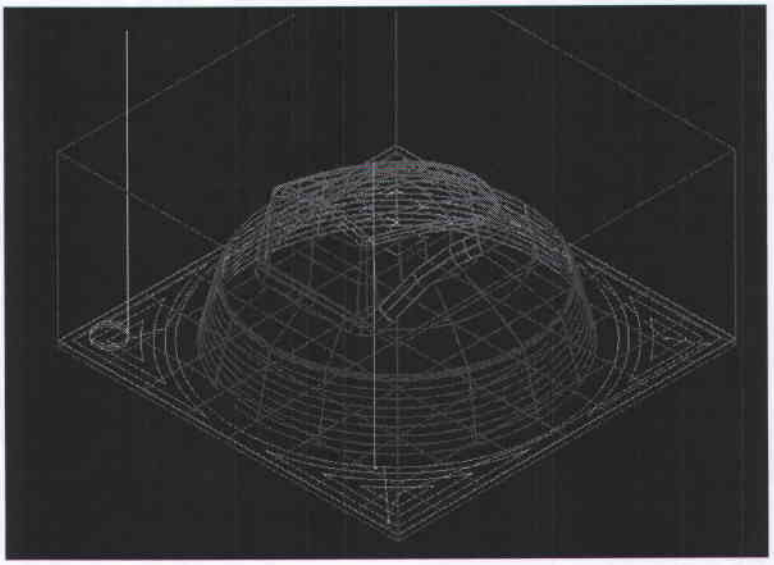


➢ Select **Linking Parameters** page.



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

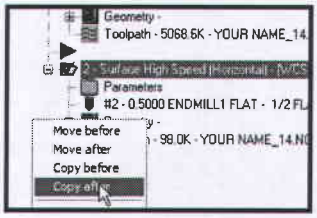
➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.



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

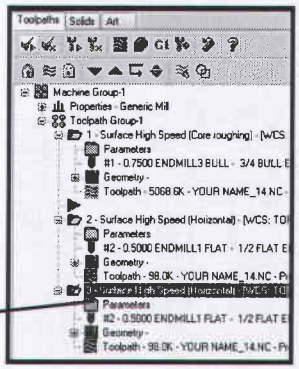
STEP 15:
HIGH SPEED – FINISH WATERLINE.

High speed waterline toolpath is a finish toolpath that performs multiple cuts at constant z depths. Waterline toolpaths are best suited for surfaces whose angles are between 30 and 90 degrees (steep walls).



➢ Select **Toolpath Manager.**
➢ **Right-mouse click** (hold it down) on the folder icon in front of the **Surface High Speed (Horizontal)**

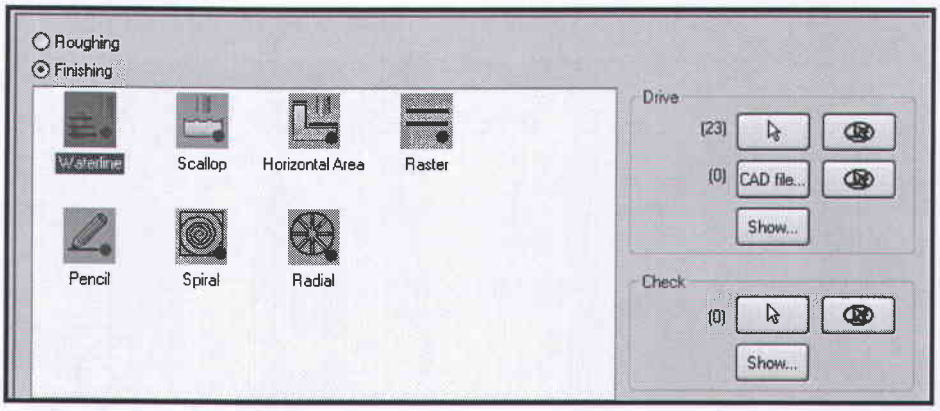
➢ Drag the mouse down and release it.
➢ Select **Copy after.**



➢ You should now have two **Surface High Speed (Horizontal)** operations.
➢ **Left-click** on the third operation **Parameters**.

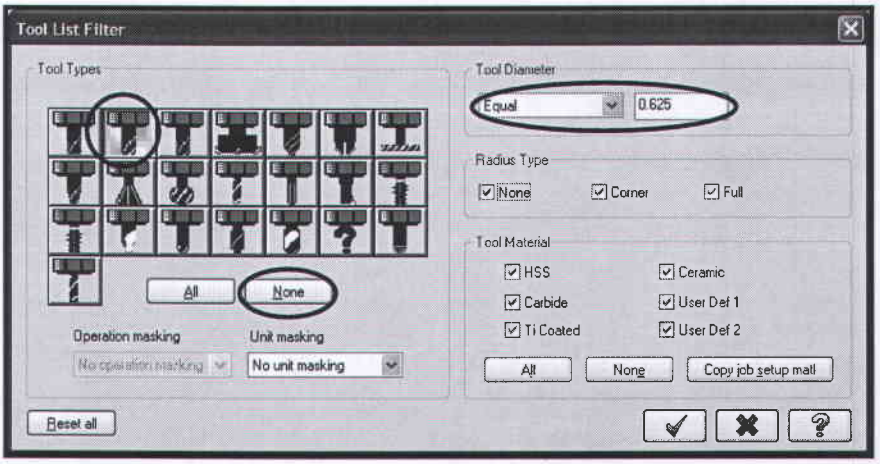
Select Parameters

➢ Select **Toolpath Type**
➢ Enable **Finishing** and choose **Waterline**



➢ Select **Tool** page.
➢ Click on the **Select library tool** button. Select library tool...
➢ Use the **Filter** option to select the **0.625 Ball Endmill**.

➢ Make sure that you select the **None** button first and then the **Spherical** as the tool type.

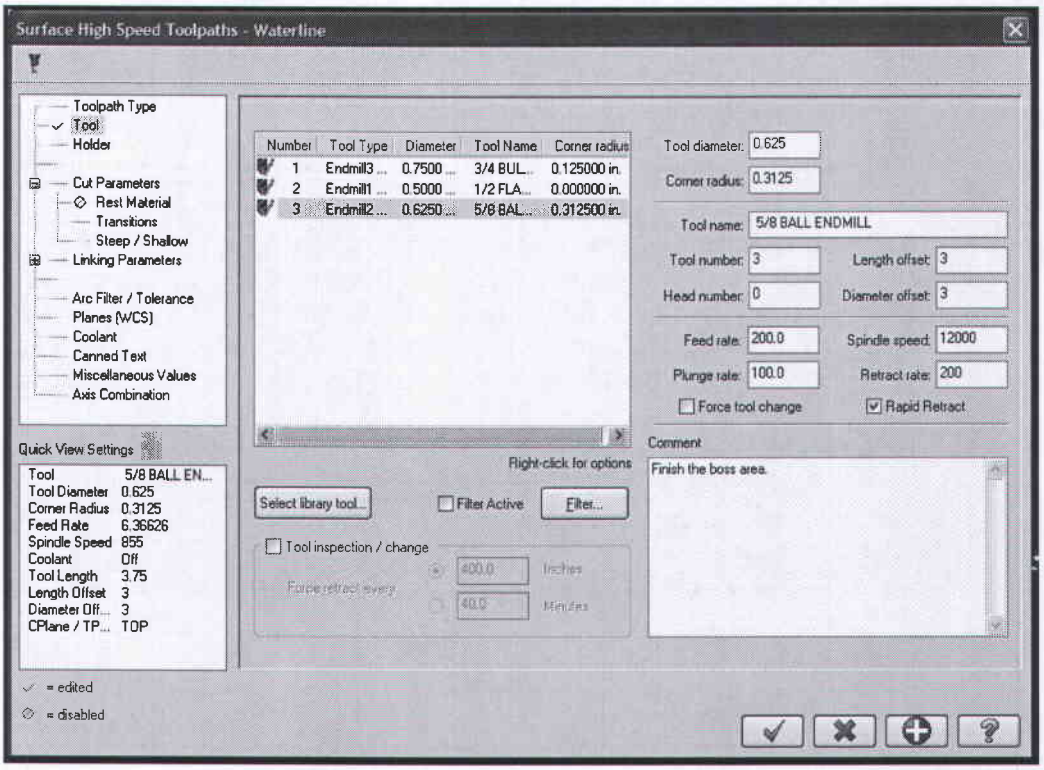


➢ Change the **Tool Diameter** to **Equal** and 0.625".

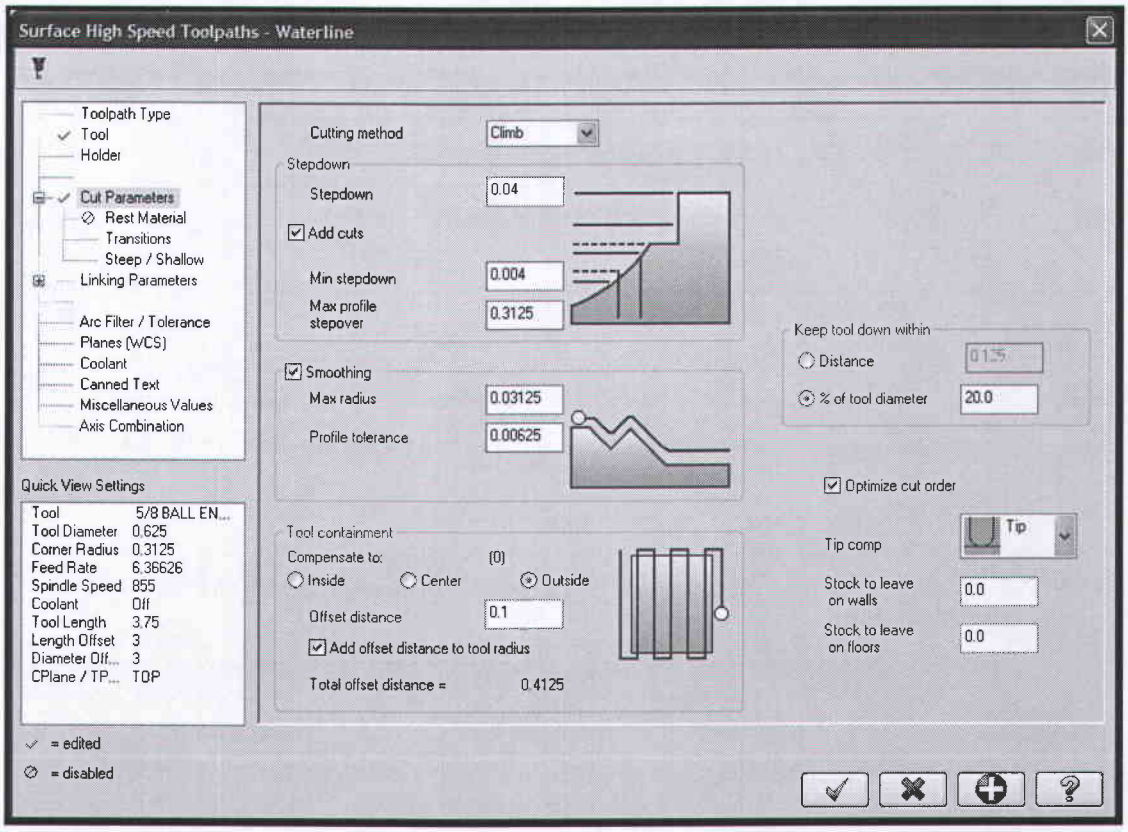
➢ Select the **OK** button to exit **Filter**.

➢ Make sure that the tool is selected and select the **OK** button to exit **Tool Selection**.

➢ Change the parameters as shown in the following screenshots.



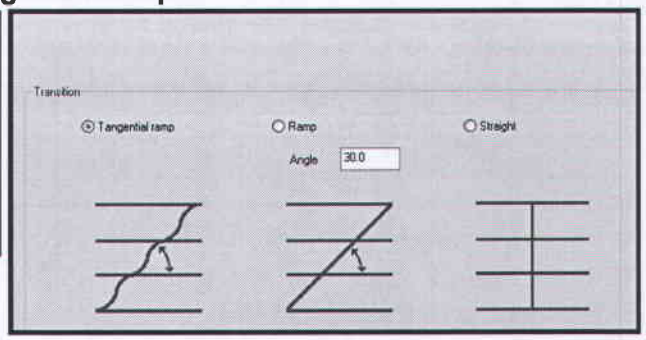
➢ Select the **Cut parameters** page and make the changes as shown.



Stepdown is the spacing between cutting passes and is measured along z axis.
Add cuts option allows you to configure adaptive stepdown strategies. Use the **Add cuts** feature to insert additional cutting passes in close to flat areas.
Smoothing settings allows you to insert arcs at corners to create smoother tool motion. The arc radii will depend on the maximum radius that you enter, and the profile/offset tolerances.
Profile tolerance determines the maximum deviation between the smoothed and unsmoothed toolpaths and is only applied on the outermost cutting pass.
Offset tolerance is defined in the same way as the **Profile tolerance**, but it is applied to all the inner passes.

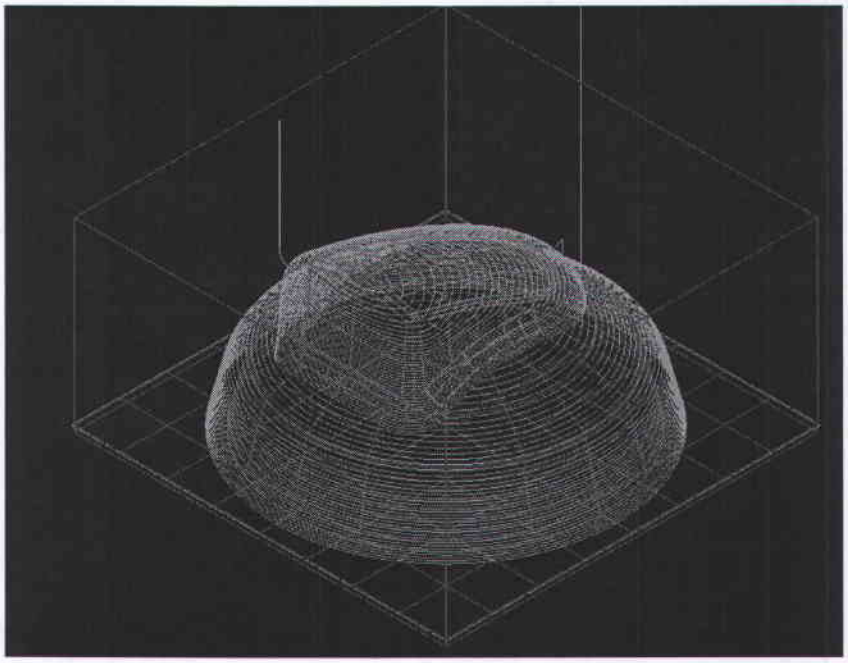
➢ Select the **Transitions** page and enable **Tangential ramp.**

Tangential ramp creates a true high speed transition between the cutting passes. Mastercam inserts arcs at the beginning and end of the ramp for the smoothest tool motion into and out of the move.



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

➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.

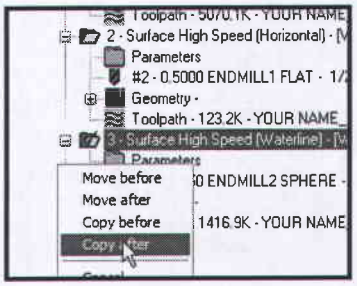


➢ Select **Toggle toolpath display on** selected operations to remove the toolpath display.

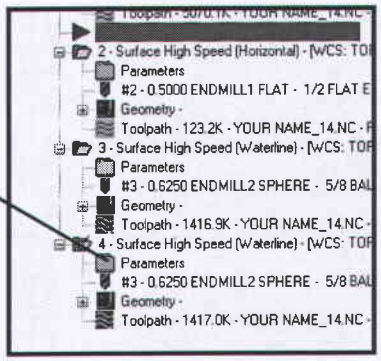
STEP 16:
REMACHINE THE SURFACE (SCALLOP REST PASSES).

Rest passes is a finish toolpath that removes material in areas where the previous tool could not fit.

- Select **Toolpath Manager.**
- **Right-mouse click** (hold it down) on the folder icon in front of the **Surface High Speed (Waterline)**

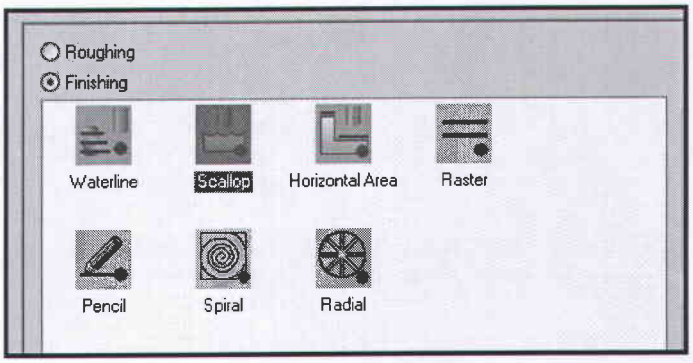


- Drag the mouse down and release it.
- Select **Copy after.**
- Select **Parameters** in the second **Surface High Speed (Waterline)** operation.

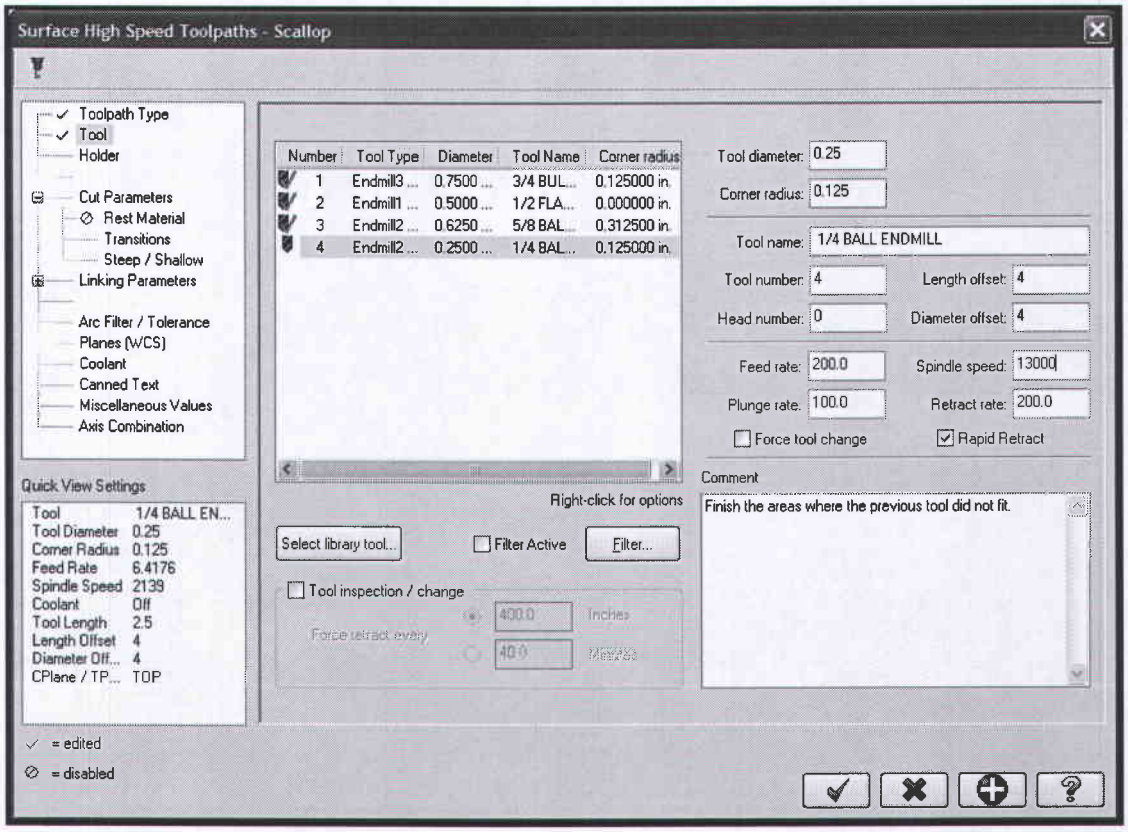


Select Parameters

- Select **Scallop** toolpaths



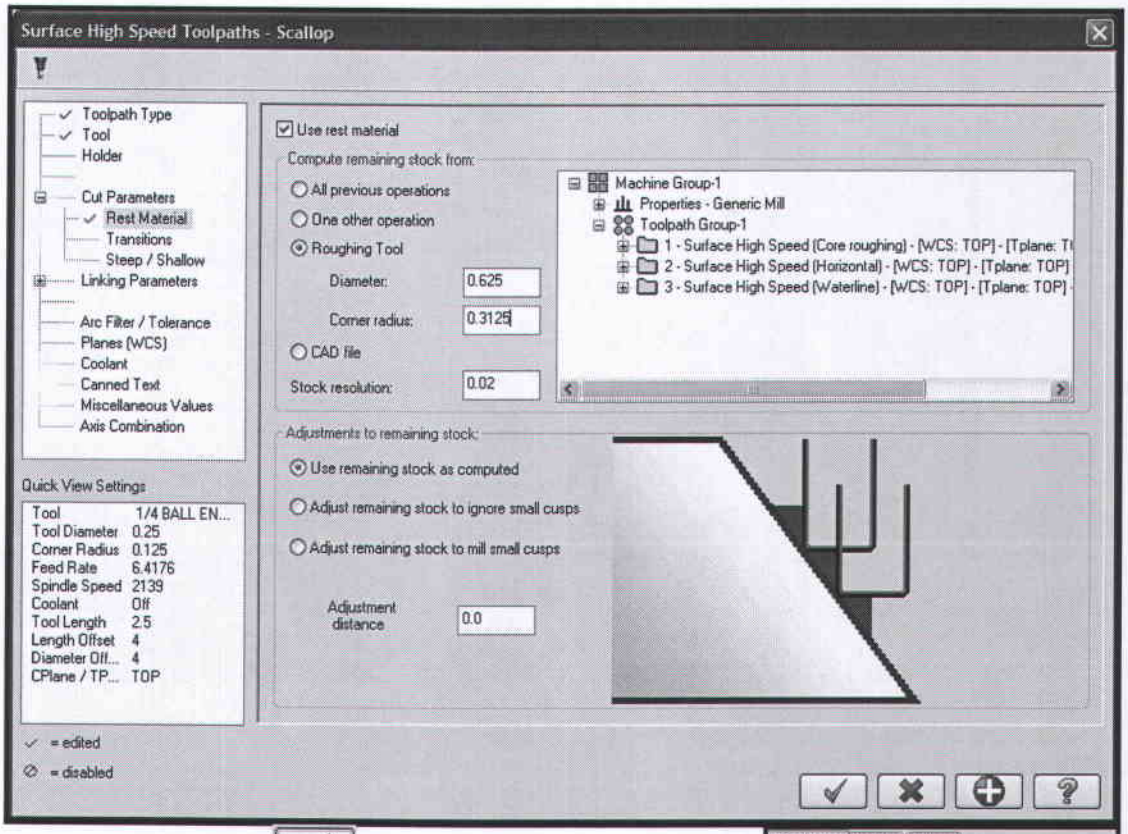
- Select **Tool** page
- Click on **Select library tool** and then on the **Filter** to select the 0.25 diameter End mill ball.
- Change the parameters in the **Tool** page as shown.



- Select the **Rest Material** page.
- Enable **Use rest material.**

➢ Enable **Compute the remaining stock from** the **Roughing Tool Diameter** and change the parameters as shown.

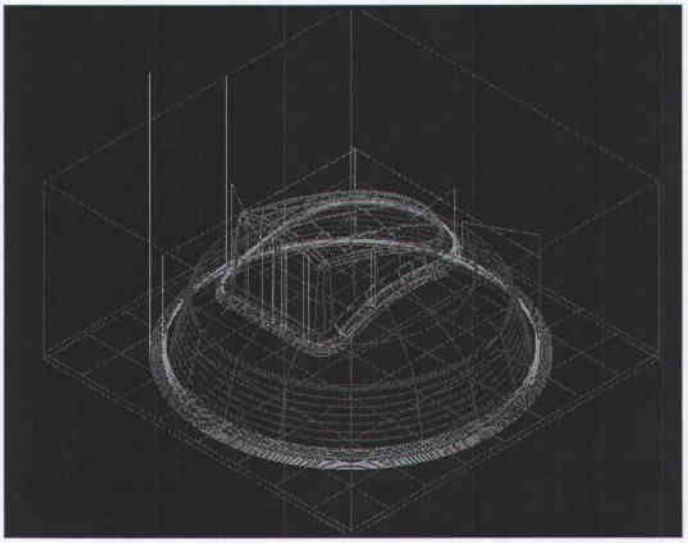
Compute remaining stock from Roughing Tool enable Mastercam to calculate the stock based on the areas that could not be cut by a previous tool defined by its diameter and corner radius.



➢ Select the **OK** button to exit.

➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.

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



STEP 17: BACKPLOT THE TOOLPATH.

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

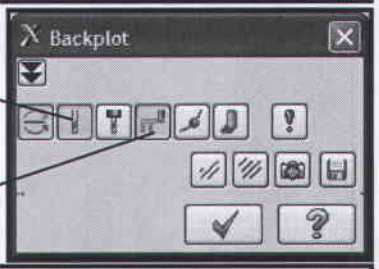
Toolpaths tab

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

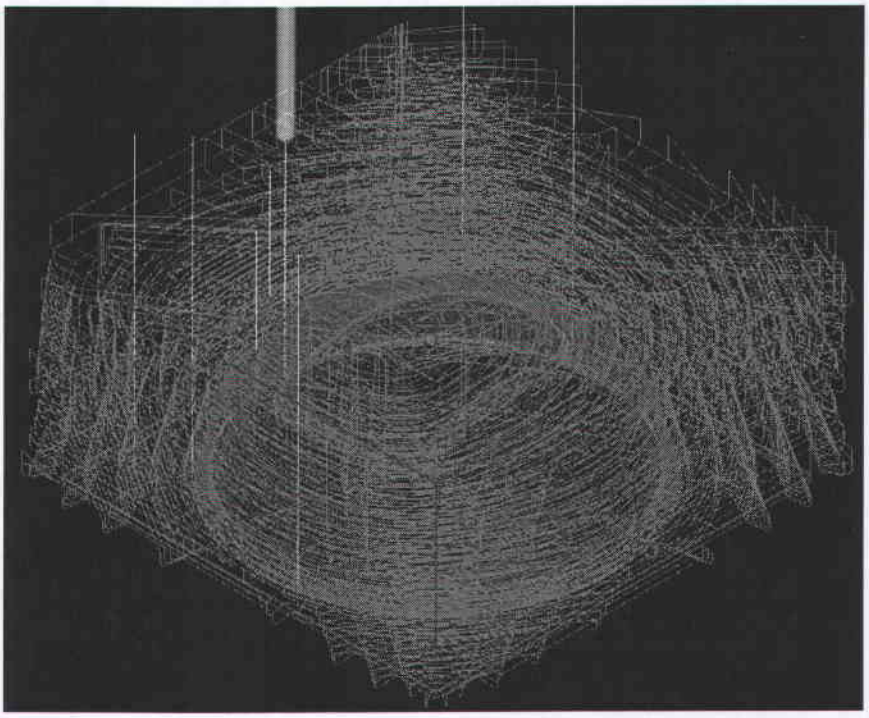
➢ Make sure that you have the following buttons turned on (they will appear pushed down).
➢ **Display tool**
➢ **Display rapid moves**

Display tool

Display rapid moves



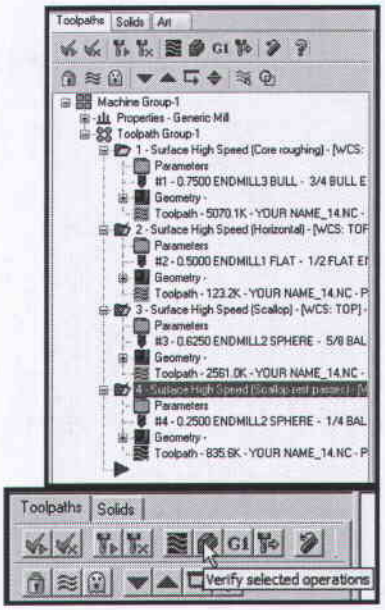
➢ Select the **Play** button



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

STEP 18:
VERIFY-TOOLPATH VERIFICATION.

➢ Select only the finish operations by selecting first the **Surface High Speed (Horizontal)**, and then **holding down** the **Shift** key select **Surface High Speed (Scallop rest passes)**.



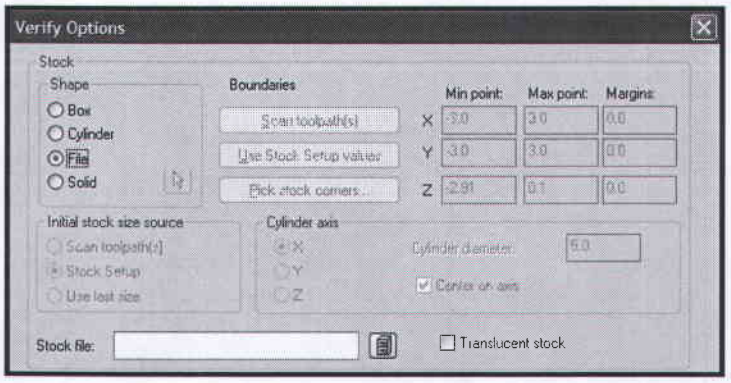
➢ Select the **Verify selected operations** button.

Update after each toolpath updates the stock after each operation.

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

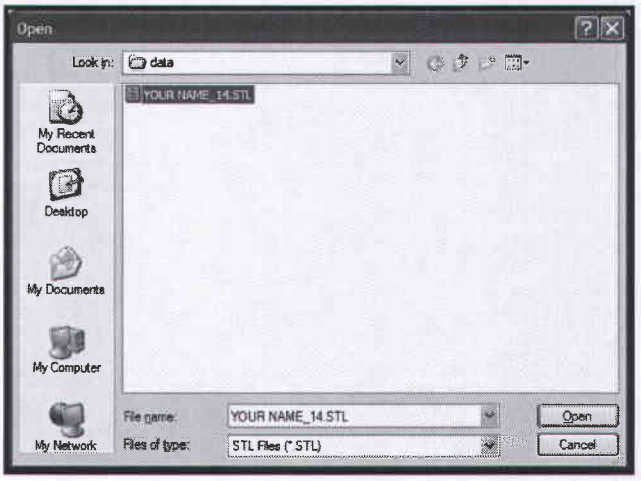
- Select **Turbo** mode.
- Select the **Configure** button.

- Enable **File** in the **Stock** field.
- Select the **File Manager** icon.

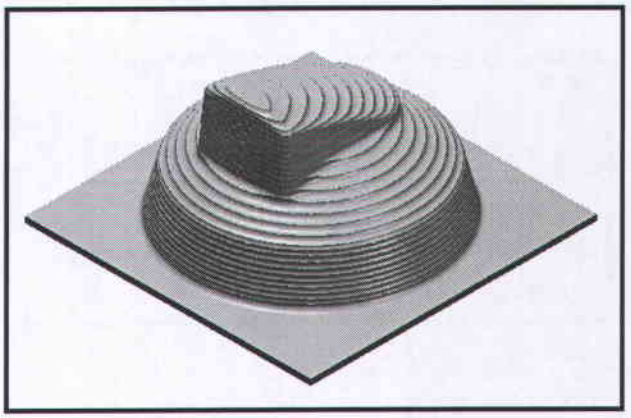


- Select the **Stl file** that you saved earlier **Your Name_14.STL**
- Select the **Open** button.

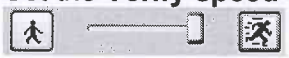
- Select the **OK** button to exit **Verify Options**.



➢ The stock should look as shown to the right.

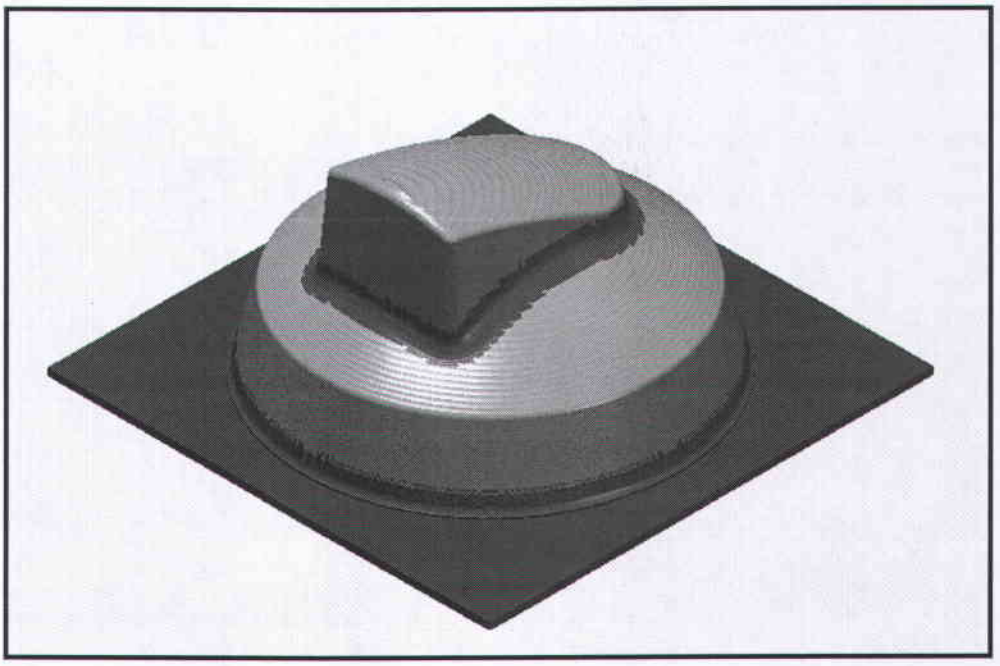


➢ Set the **Verify speed** to maximum by moving the slider bar in the speed control bar as shown.



➢ 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 19:
POST PROCESS THE FILE.

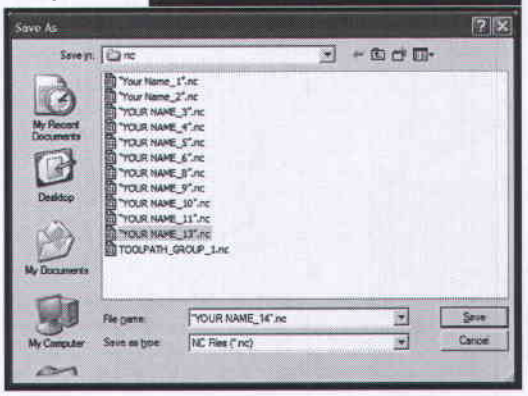
- Make sure that all operations are selected, otherwise:
- Click on the **Select all visible operations** button.

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

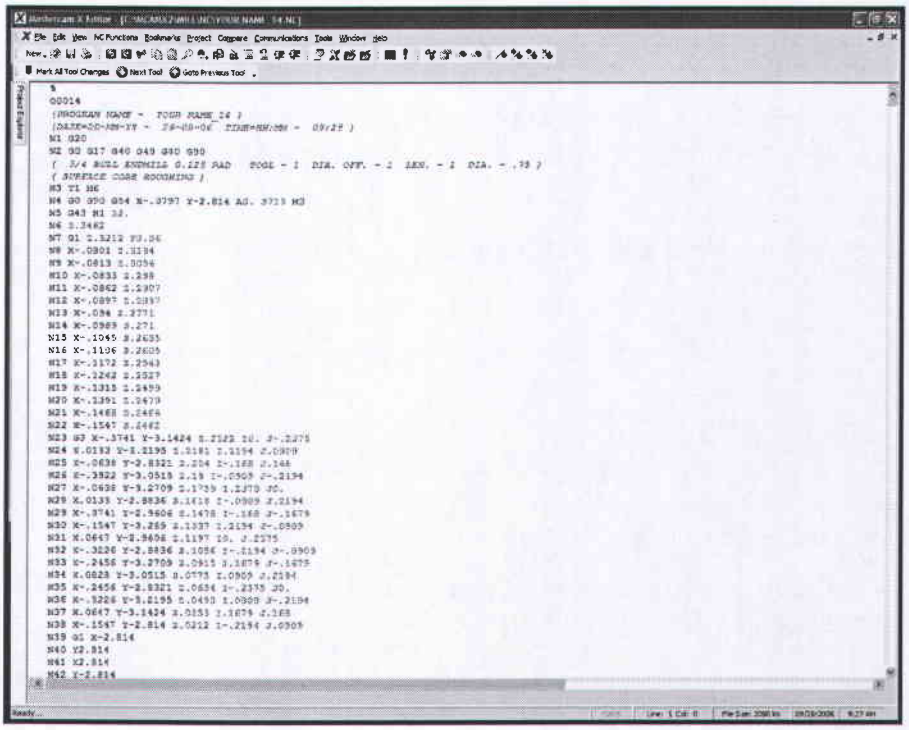
- In the **Post processing** window, make all the necessary changes as shown to the right.

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

- Enter the same name as the geometry name in the NC File name field.



- Select the **Save** button.



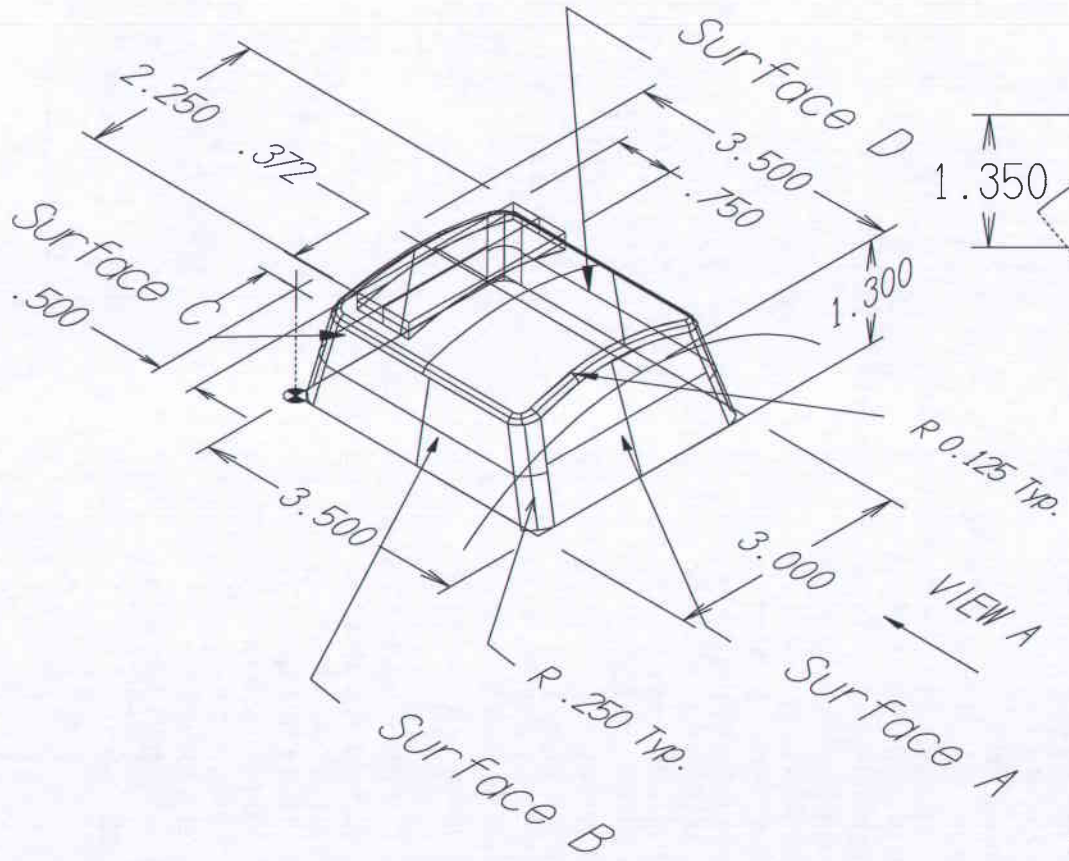
➢ Select the red **X** box at the upper right corner to exit the **Editor**.

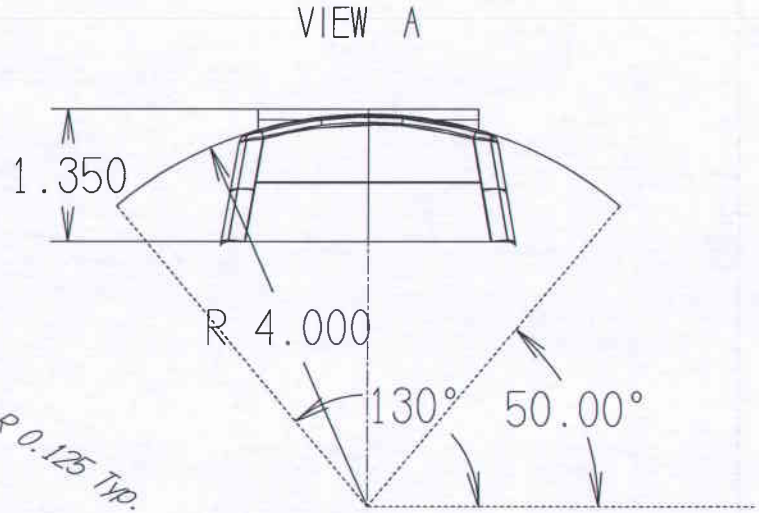
STEP 20:
SAVE THE UPDATED MCX FILE.

➢ Select the **Save** icon. 

Surface A has a draft angle of 20 deg.
Surfaces B,C and D have a draft angle of 10 deg.

ALL DIMENSIONS IN INCHES



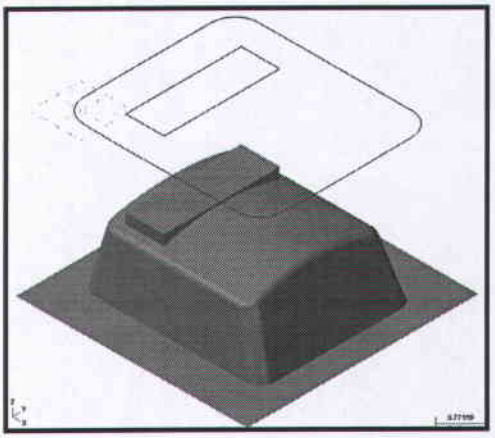


TITLE TUTORIAL 14 - EXERCISE	
MATERIAL ALUMINUM T2024	
DATE: JUNE 12, 2000	eMastercam.com

REVIEW EXERCISE

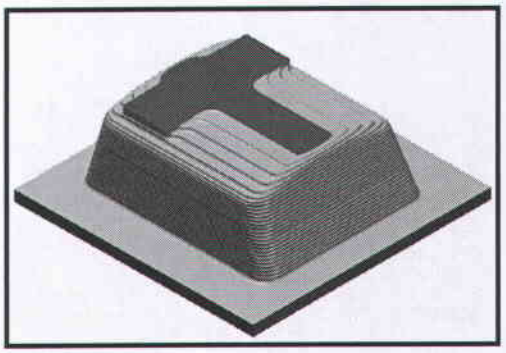
Student practise. Create the Toolpath for **Exercise-Tutorial 14** as per the instructions below;
Downloaded from www.emastercam.com/files

- Note that the URL address is written with lowercase letters only.

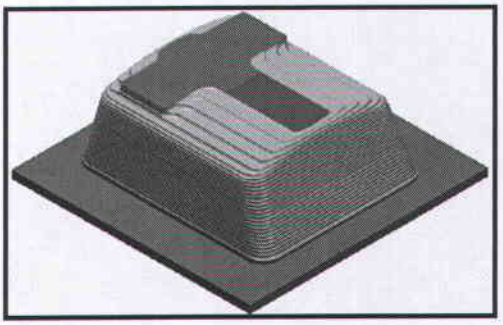


☞ **Tips:**

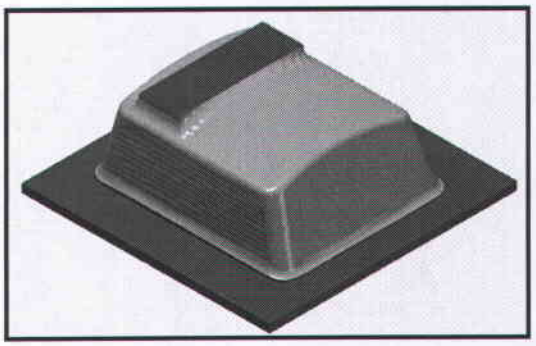
1. Establish the **Stock size** Y = 4.5, X = 4.5, Z = 1.5
   Stock origin X = 1.75, Y = 1.5, Z = 1.35
2. **High speed Core Roughing**
   Select All
   Use 7/16" Bull End Mill with radius 0.0625
   Stepdown =0.05
   Enable add cuts
   XY stepover 45 % of diam
   Stock to leave on walls and on floors = 0.05
   Transitions –Entry helix Radius 0.2
   Linking parameters set to Minimum Vertical Retract; Part clearance =0.25
   Keep default Leads values
   Minimize Trimming; distance =0.0825



3. **High Speed - Finish Horizontal Area**
   Copy previous operation and change the following parameters;
   Use 3/8" Flat End Mill
   # of depth cuts =1
   Enable Smoothing
   Stock to leave on walls = 0.05
   Stock to leave on floors = 0
   XY stepover = 45% of dia.

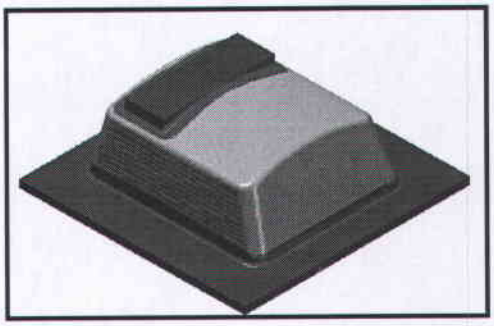


4. **High Speed - Finish Scallop (check Tutorial 15)**
   Copy previous operation and change the following parameters;
   Use 3/8" Ball End Mill
   Select the top red boundary as containments
   Stock to leave on walls = 0
   Stepover = 0.0375
   Disable Expand inside to out



5. High Speed - Finish Scallop rest passes

Copy previous operation and change the following parameters;
Use 1/8" Ball End Mill
In Rest material page enable Use rest material
Enable Roughing Tool
Diameter 0.375
Corner radius 0.1875



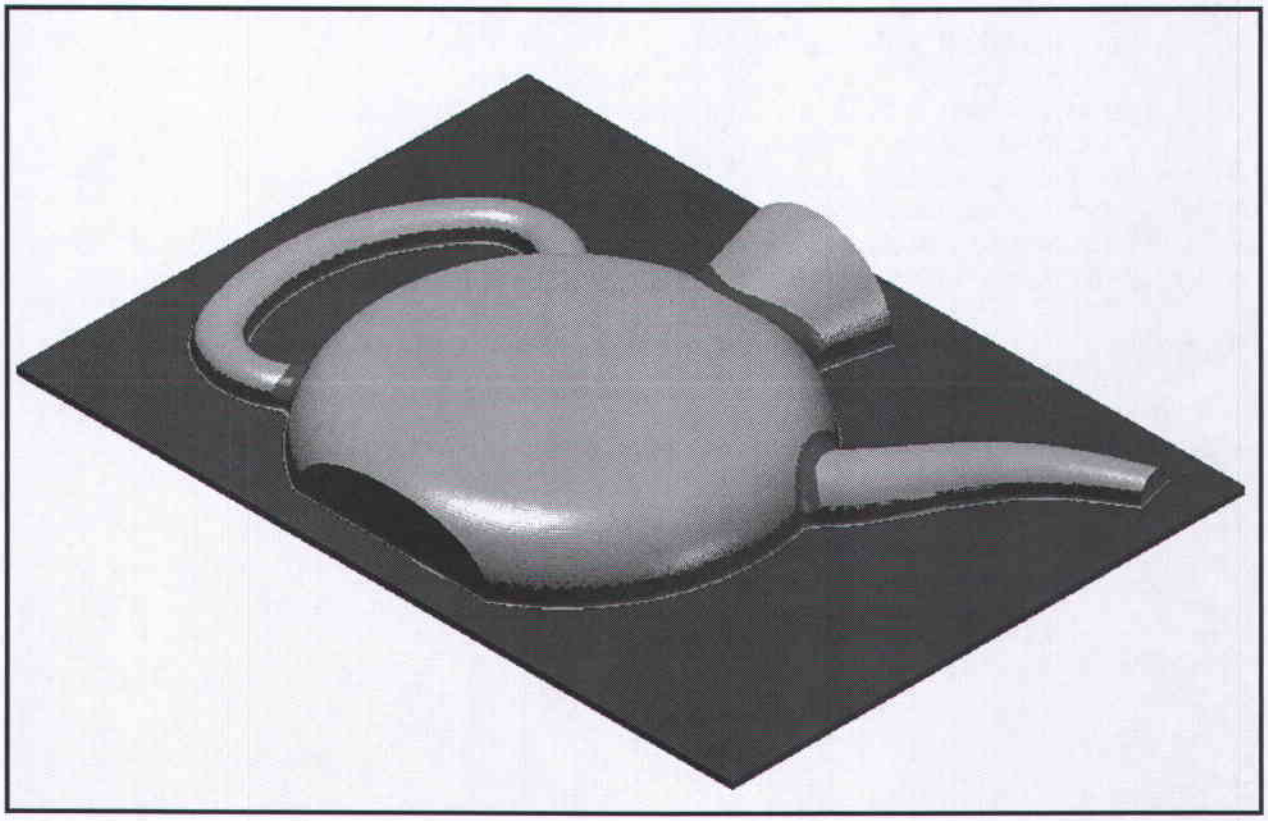
6. Backplot and Verify the toolpaths.

7. Post process the file.

TUTORIAL SERIES FOR

Mastercam X²

TUTORIAL 15
RULED, SWEEP, REVOLVE AND FILLET SURFACES AND HIGH SPEED TOOLPATHS: CORE ROUGHING, HORIZONTAL AREA; SCALLOP, SCALLOP REST PASSES & PENCIL.



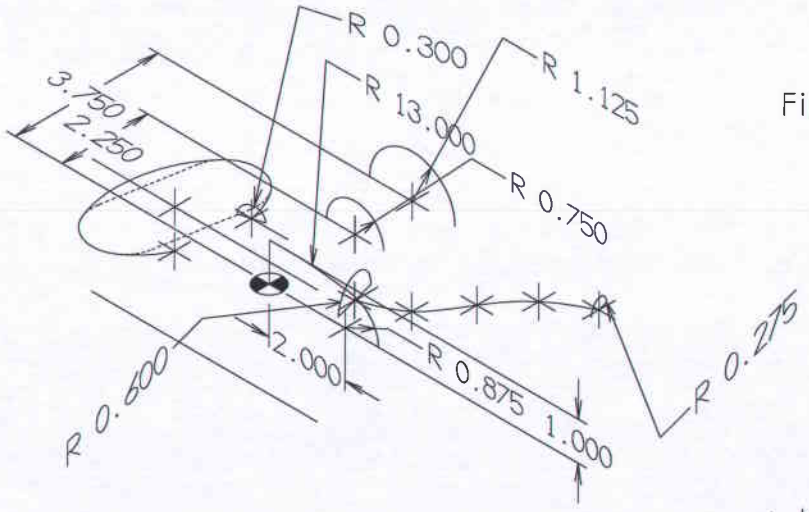
Objectives:

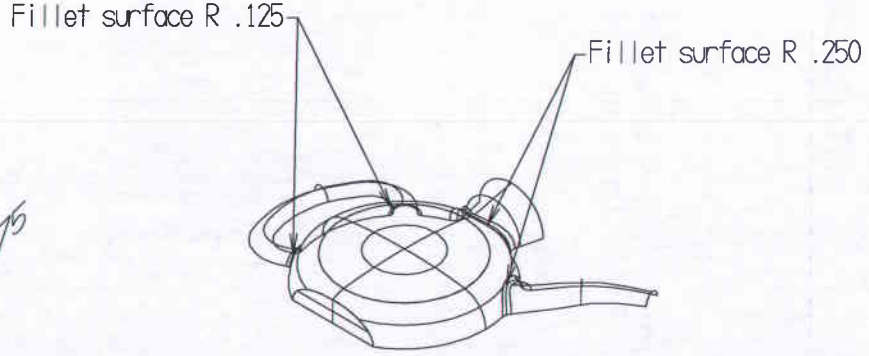
The Student will design a 3-dimensional drawing by:
- Opening an existing file with 3D wireframe in different construction planes.
- Creating a revolved surface.
- Creating a sweep surface from two across contours and one along contour.
- Creating a sweep surface from an across contour and an along contour.
- Creating two ruled surfaces.
- Creating fillet surfaces between the sweep surfaces and the revolved surfaces.
- Creating fillet surfaces between the ruled surface and the revolved surfaces.
- Trimming the revolved surface with the ruled surface.
- Creating flat boundary surfaces in the open areas.
- Creating the bounding box used for stock definition.
- Creating a rectangular surface at the bottom of the part.

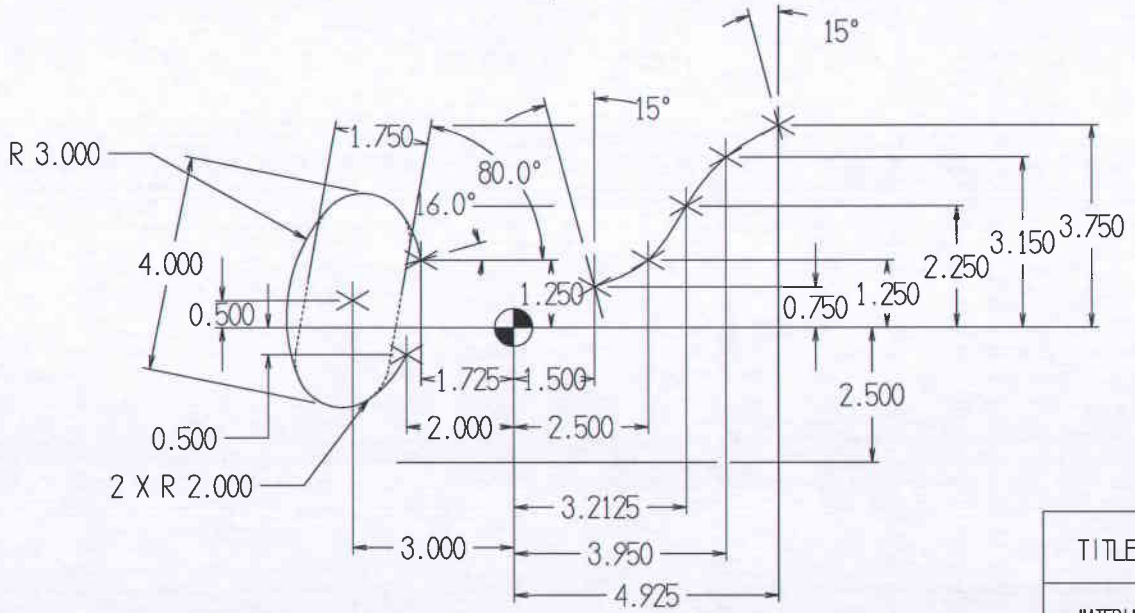
The Student will create a 3-dimensional milling toolpath consisting of:
- High Speed Core Roughing toolpath.
- High Speed Horizontal finish toolpath.
- High Speed Scallop finish toolpath.
- High Speed Scallop rest passes finish toolpath to clean areas where previous tool did not fit.
- High Speed Pencil finish toolpath to clean sharp corners.

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 program to create an intermediate stock.
- Running the Verify program to machine the part on the screen.

ALL DIMENSIONS IN INCHES







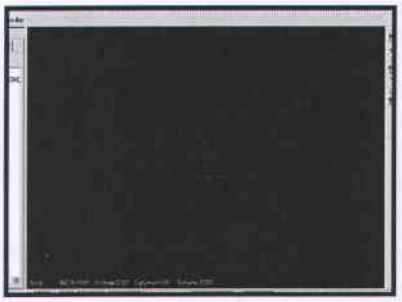
TITLE	TUTORIAL 15
MATERIAL	ALUMINUM T6061
DATE: OCT 26, 2004	eMastercam.com

GEOMETRY CREATION

Setting the toolbar states:

- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine the 3D part. See **Getting started** page A-5 in the **User Notes**.
- **Toolpaths/Solids operations manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to 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: OPEN THE FILE.



The wireframe geometry Tutorial15_wireframe.mcx can be downloaded from http://www.emastercam.com/files.

- Note that the URL address is written with lowercase letters only.

File

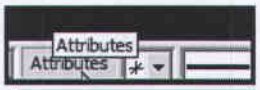
➢ **Open**

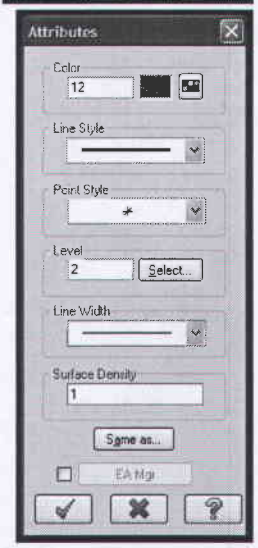
➢ Select the **Tutorial15_wireframe.mcx**.

STEP 2:
CREATE THE REVOLVED SURFACE.

Revolved Surface: 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.

2.1 Change the attributes to create the revolved surface on Level 2 with color red (12).

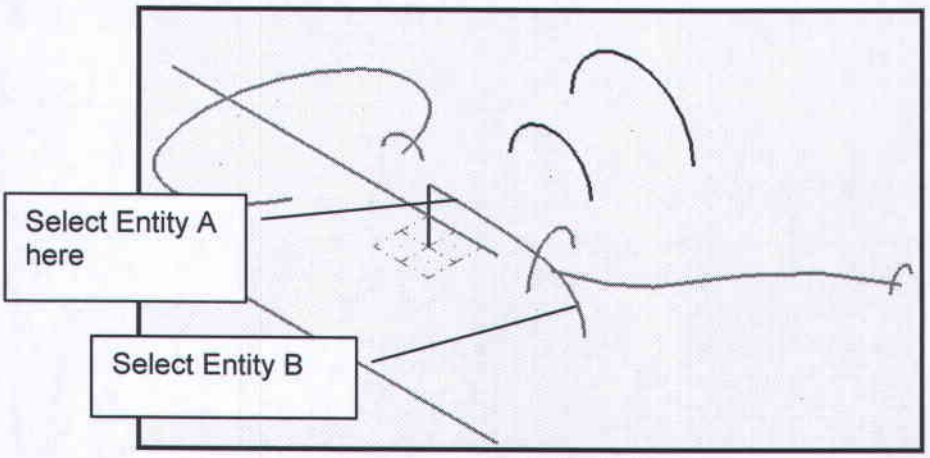


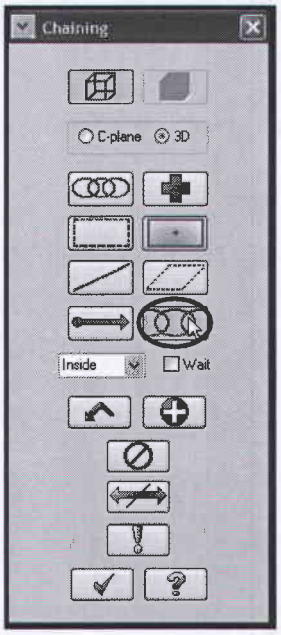


- Select **Attributes** in the **Status bar**.
- Change the **Color** number to **12** and the **Level** number to **2**.
- Select the **OK** button to exit.

2.2 Create the revolved surface.

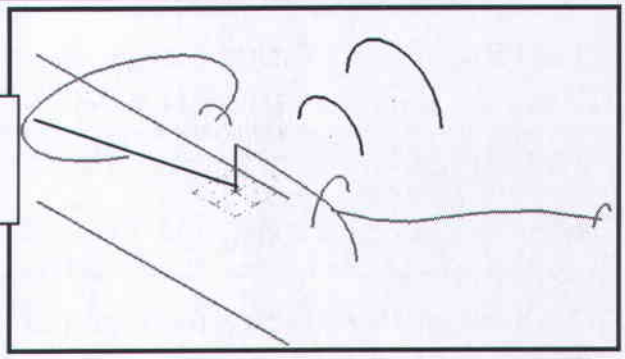
Create
- **Surface**
- **Create Revolved Surfaces**
- Enable the radio button in front of **3D** and **Partial** in the **Chaining** dialog box.
- [Select profile curve(s)]: Select Entity A (in CW direction).
- Select Entity B.





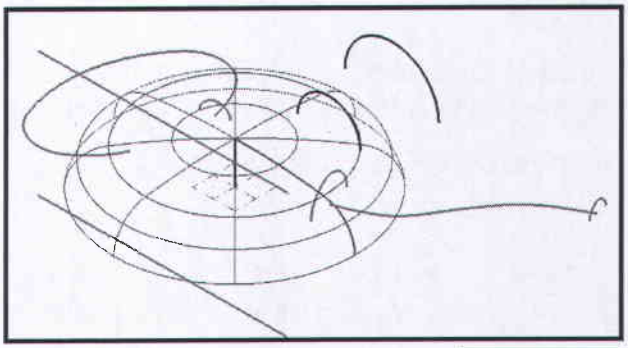
- Select the **OK** button to exit **Chaining**.

➢ [Select the axis of rotation]: Select Entity A.

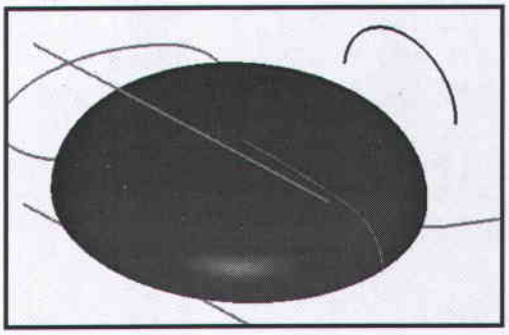


➢ Make sure that the **End angle** = 360.

➢ 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 or **Alt+S**.
Alt+S again to display the part back in wireframe mode.



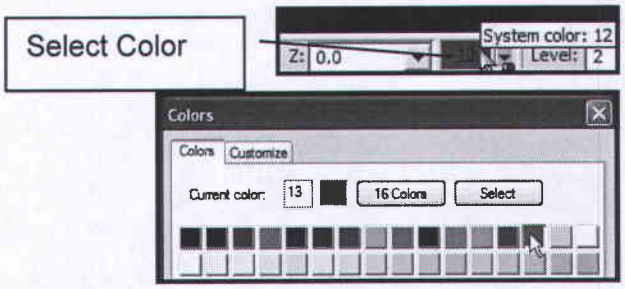
STEP 3:
CREATE THE SWEEP SURFACES.

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

3.1 Change the main color to magenta (13).

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

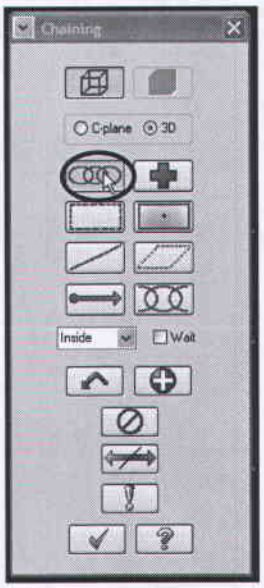
➢ Change the **Color** number to **13**.

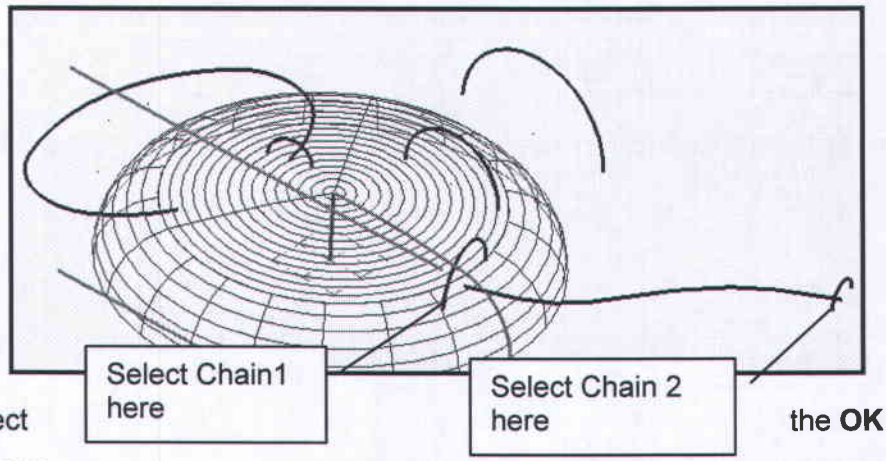


➢ Select the **OK** button to exit.

3.2 Create the swept surface with two across contours and one along contour.

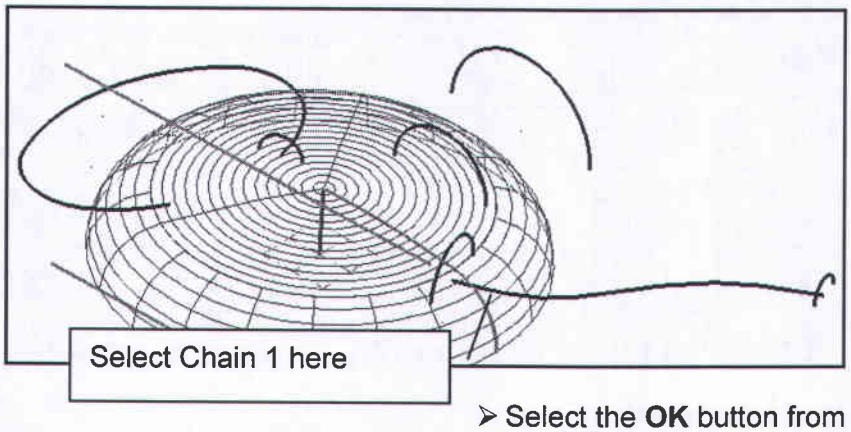
Create
➢ **Surface**
➢ **Create Swept Surfaces**
➢ Select **Chain** in the **Chaining** dialog box.
➢ [Define the across contour(s) 1]: Select Chain 1 as shown.
➢ [Define the across contour(s) 2]: Select Chain 2.
➢ Select all the chains in the same directions. Please use the **Reverse** button from the **Chaining** dialog box if you need it.

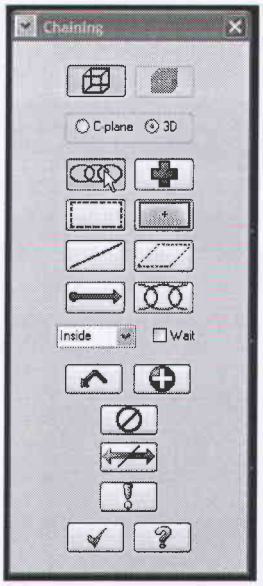




➢ Select the **OK** button to exit across contour chaining.

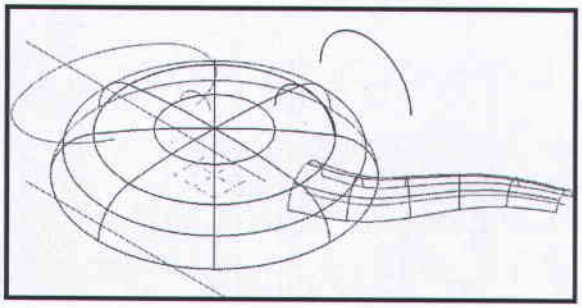
➢ [Define the along contour(s) 1]: Select Chain 1 as shown.





➢ Select the **OK** button from the ribbon bar to complete surface swept creation.

➢ The surface should look as shown below.



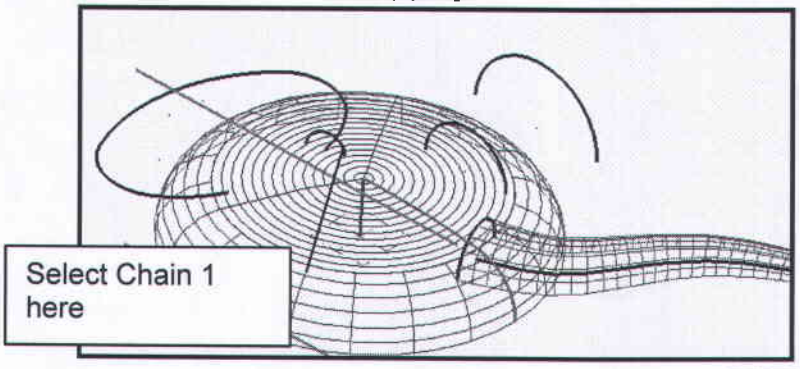
3.3 Create the swept surface with one across contours and one along contour.

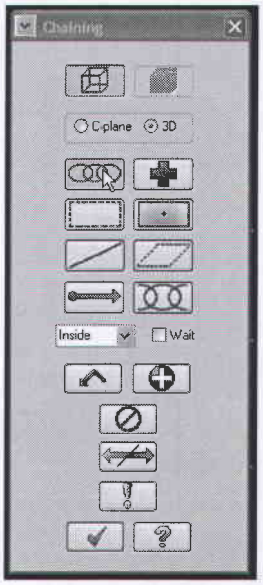
Create

➢ **Surface**

➢ **Create Swept Surfaces**

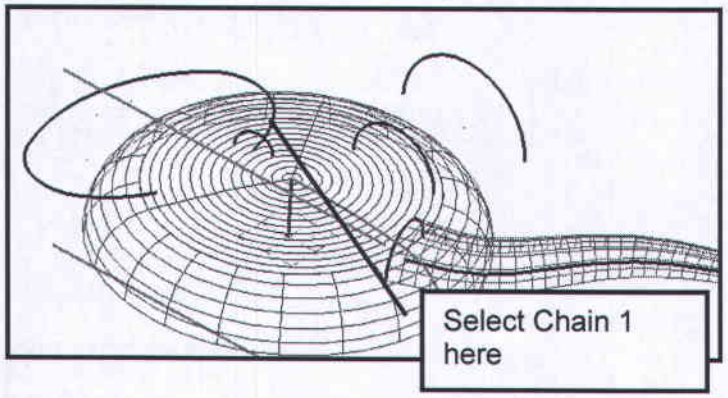
➢ [Define the across contour(s) 1]: Select Chain 1 as shown.

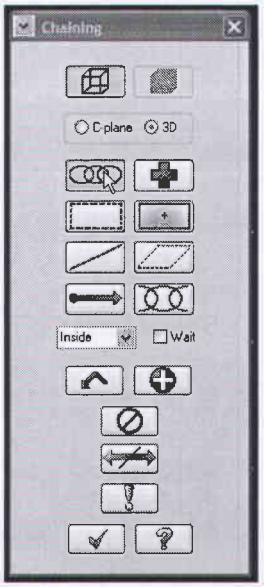




➢ Select the **OK** button to exit across contour chaining.

➢ [Define the along contour(s) 1]: Select Chain 1 as shown.

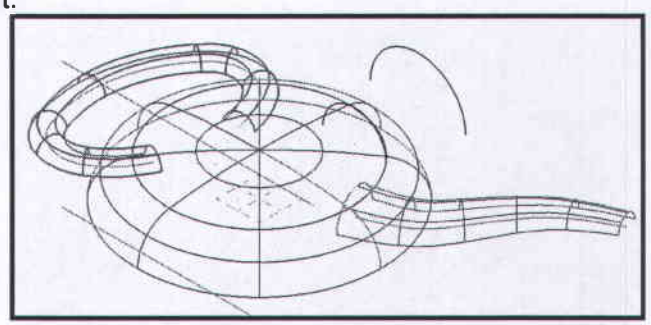




➢ Select the **OK** button to exit chaining along contour chaining.

➢ Select the **OK** button from the ribbon bar to complete surface swept creation.

➢ The surface should look as shown to the right.



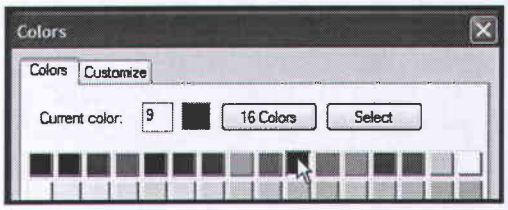
STEP 4:
CREATE THE RULED SURFACES.

Ruled Surface: 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; and
the contours should be chained in the same direction, or the surface will become twisted and therefore be incorrect.

4.1 Change the main color to blue (9).

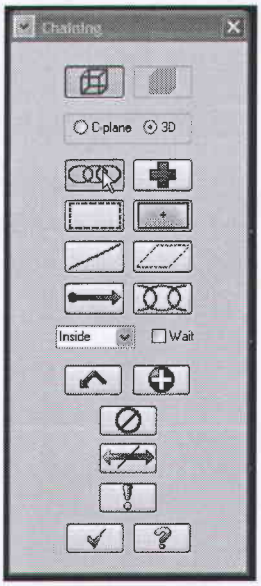
➢ Select the **Color** in the **Status bar**.



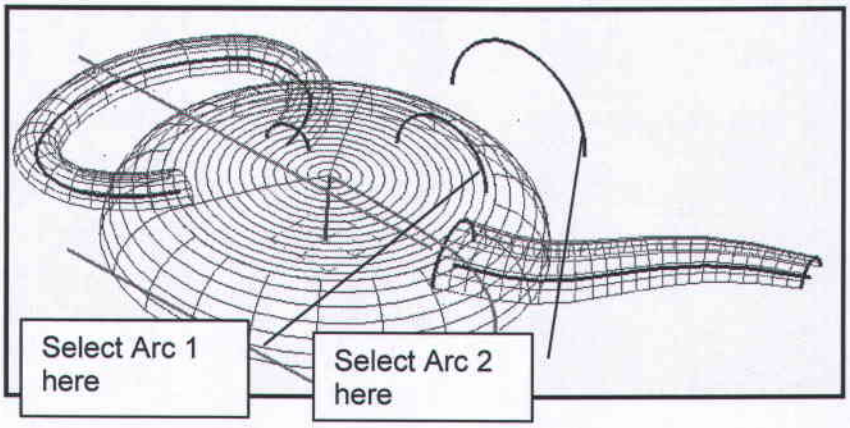
➢ Change the **Color** number to **9**.

➢ Select the **OK** button to exit.

4.2 Create the first ruled surface.

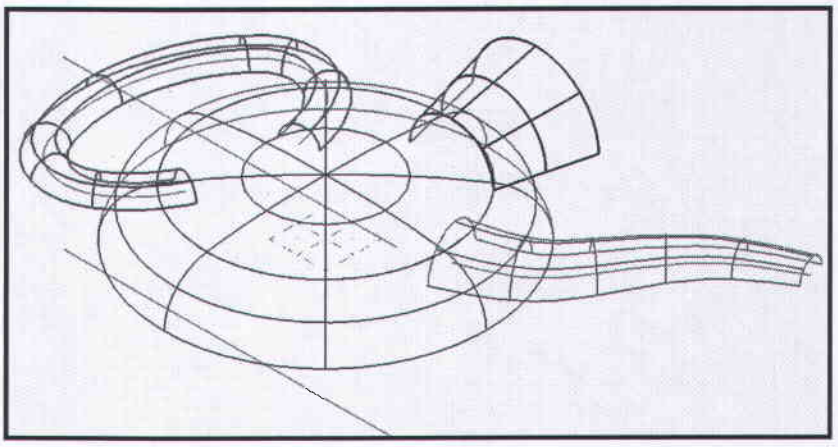


Create
➢ **Surface**
➢ **Create Ruled/Lofted Surfaces**
➢ [Select chains1]: Select the Arc 1 as shown.
➢ [Select chains2]: Select the Arc 2 as shown.

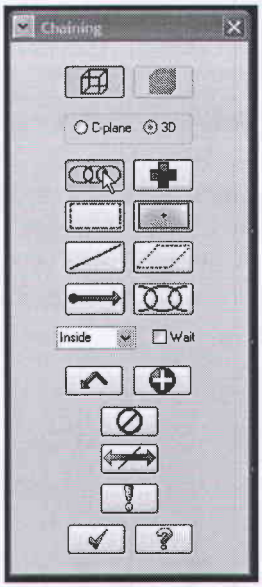


➢ Select the **OK** button to exit **Chaining**.

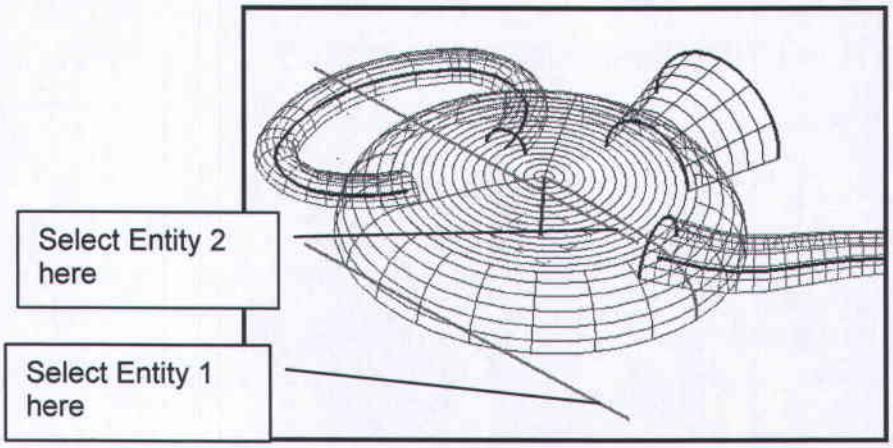
➢ Select the **OK** button to exit the command.



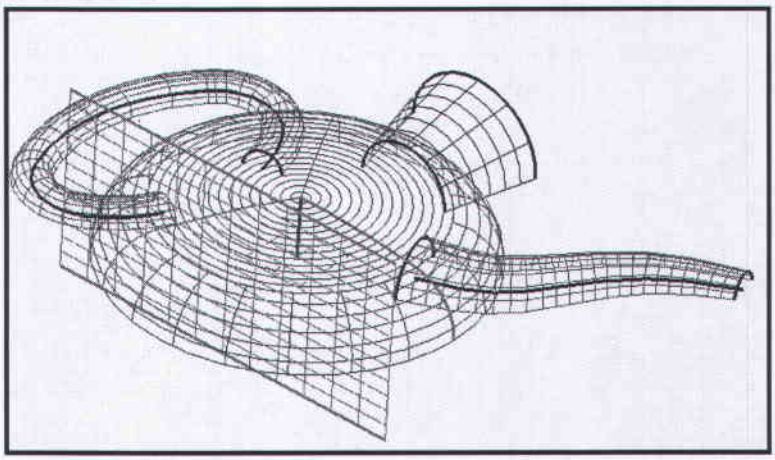
4.3 Create the second ruled surface.



Create
- **Surface**
- **Create Ruled/Lofted surfaces**
- [Select chains1]: Select the Entity 1 as shown.
- [Select chains2]: Select the Entity 2 as shown.



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

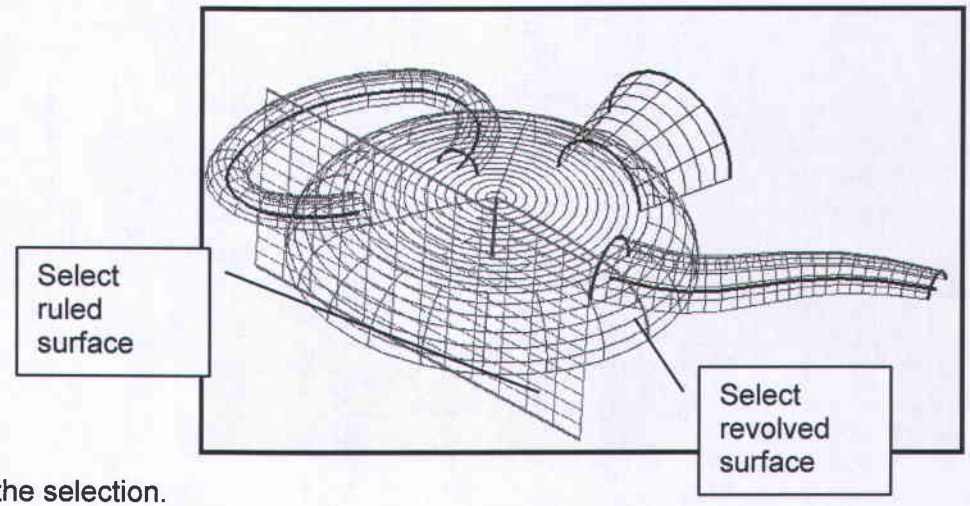


STEP 5:
TRIM THE RULED SURFACE TO THE REVOLVED SURFACE.

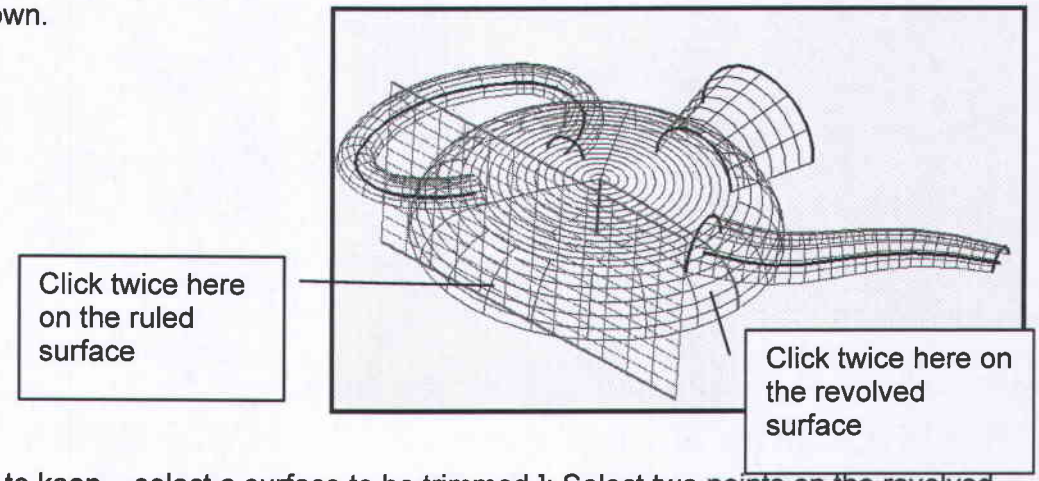
Trim Surface: a surface generated by trimming an existing surface to a specific boundary, to another surface, or to a given plane.

5.1 Create the trim surfaces.

Create
- **Surface**
- **Trim Surface**
- **Trim Surfaces to Surfaces**
- [Select first set of surfaces]: Select the ruled surface as shown.
- Press **Enter** to end the selection.
- [Select second set of surfaces]: Select the revolved surface as shown.

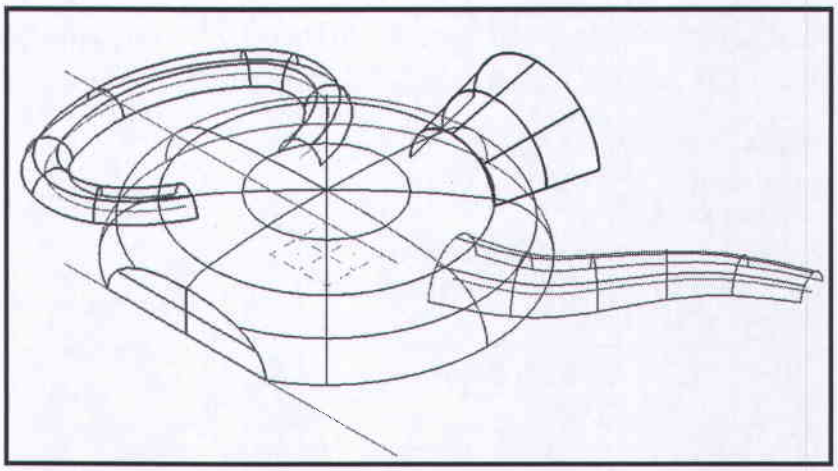


- Press **Enter** to end the selection.
- [Indicate area to keep – select a surface to be trimmed]: Select two points on the ruled surface as shown.



- [Indicate area to keep – select a surface to be trimmed]: Select two points on the revolved surface as shown above.
- Select the **OK** button to exit trim surface.

➢ The part should look as shown to the right.

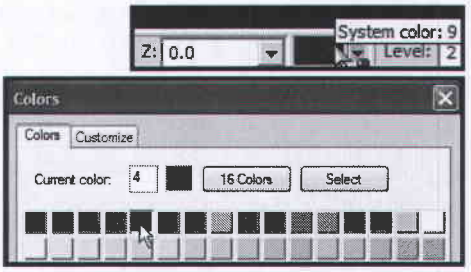


STEP 6:
CREATE THE FILLET SURFACES.

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

6.1 Change the main color to brown (4).

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



➢ Change the **Color** number to **4**.

➢ Select the **OK** button to exit.

6.2 Flip the normal of one surface.

Edit

➢ **Change Normal**

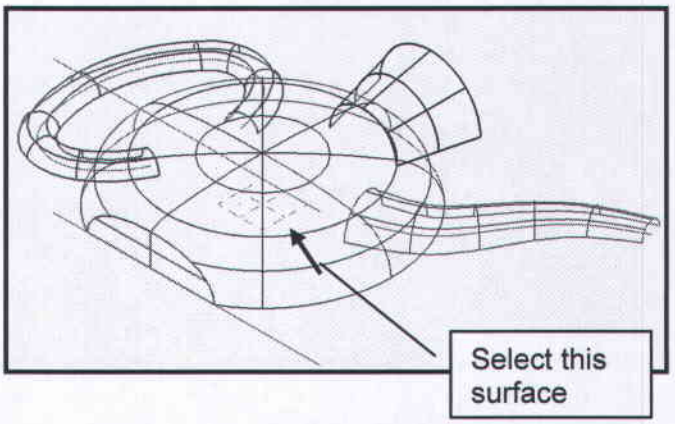
➢ [Select a surface]: Select this surface

➢ Move the arrow to a location where you can see the direction towards which is pointing and click.

➢ If the arrow points towards the inside of the part, as shown, select **Flip** button.

Press **Esc** if the arrow points towards the outside

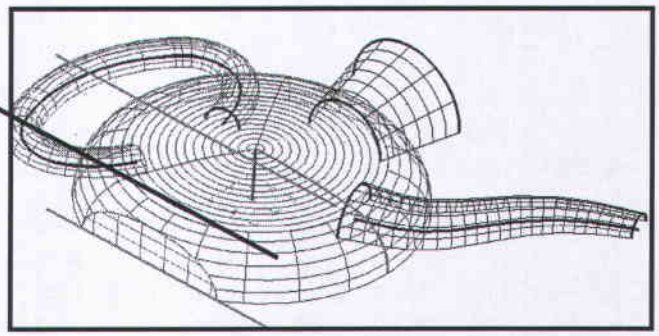
➢ Select the **OK** button to exit the command.



6.3 Create fillet surfaces with a 0.25" radius between the revolved surface and the sweep and ruled surfaces.

Create
- **Surface**
- **Fillet Surface**
- **Fillet Surfaces to Surfaces**
- [Select first set of surfaces]: Select the revolved surface.

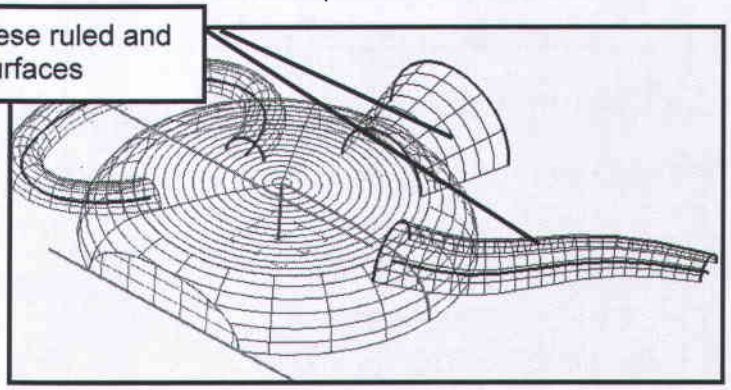
Select the Revolved surface



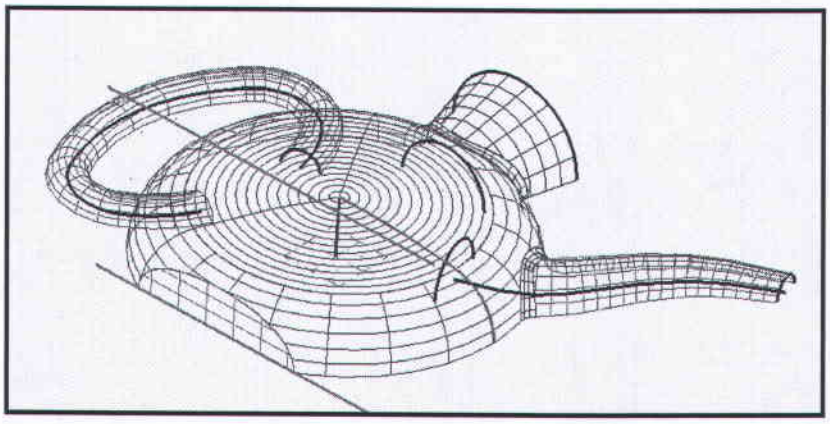
Note that there are two revolved surfaces.

- Press **Enter** to end the selection.
- [Select second set of surfaces]: Select the ruled and the sweep surfaces as shown.

Select these ruled and sweep surfaces



- Press **Enter** to end the selection.
- Change the parameters as shown in the screenshot to the right.
- The part should look as shown below.

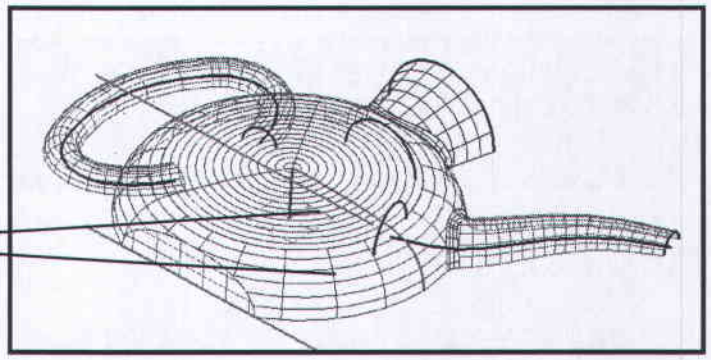


- Select the **OK** button to exit.

6.3 Create fillet surfaces with a 0.125" radius between the revolved surface and the sweep surface.

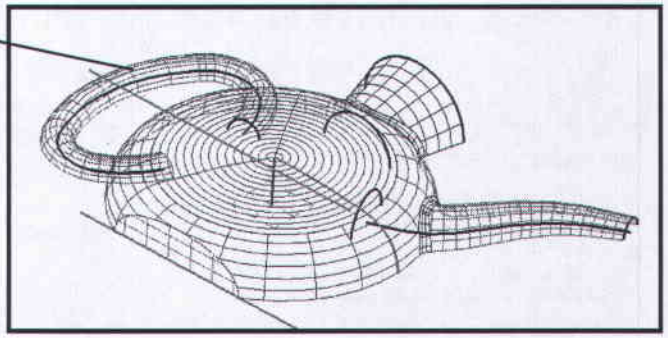
Create
- **Surface**
- **Fillet Surface**
- **Fillet Surfaces to Surfaces**
- [Select first set of surfaces]: Select the two revolved surfaces.

Select the two Revolved surfaces

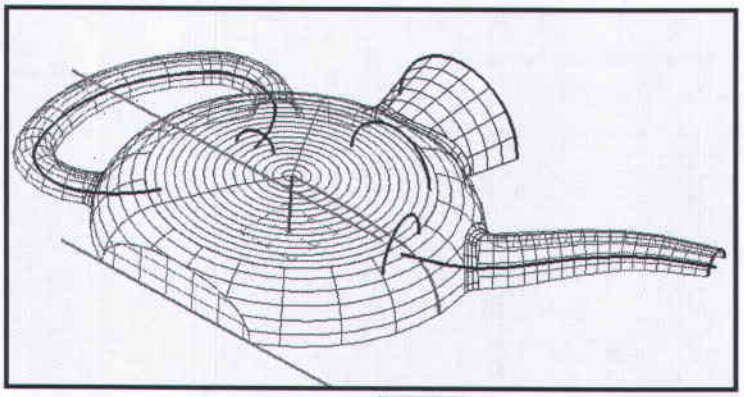


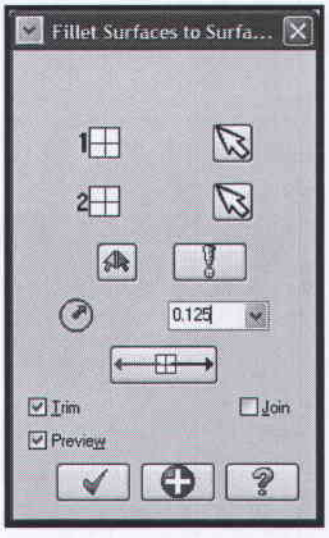
- Press **Enter** to end the selection.
- [Select second set of surfaces]: Select the sweep surface as shown.

Select the sweep surface



- Press **Enter** to end the selection.
- Change the parameters as shown in the screenshot to the right.
- The part should look as shown below.





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

STEP 7:
CREATE FLAT BOUNDARY SURFACES TO CLOSE THE OPEN AREAS.

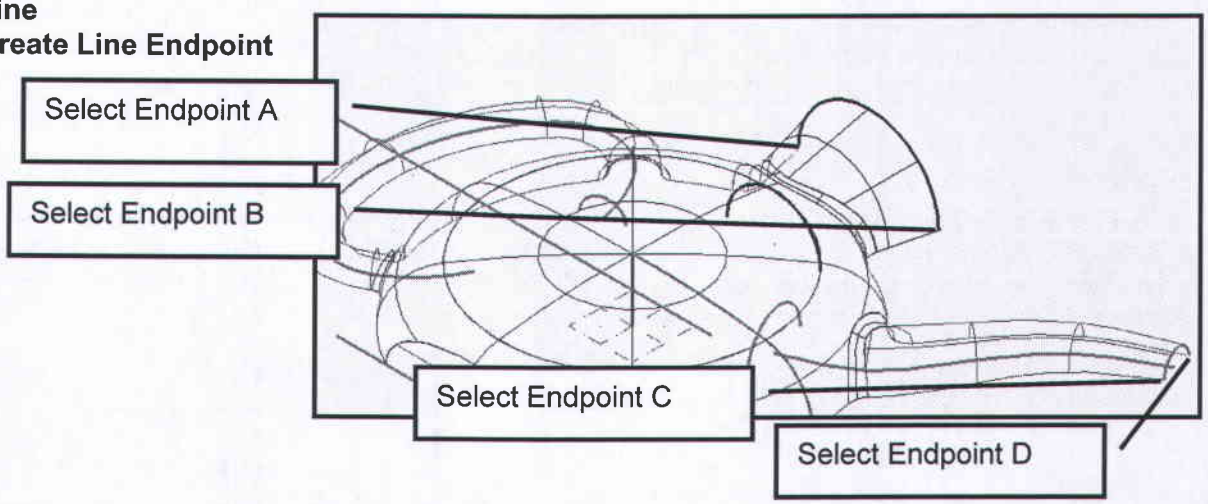
Flat Boundary Surface: a trimmed surface generated by trimming a flat surface to a specific boundary.
Applications: to create a flat surface on a part inside of a closed boundary.

7.1 CHANGE TO 2D.

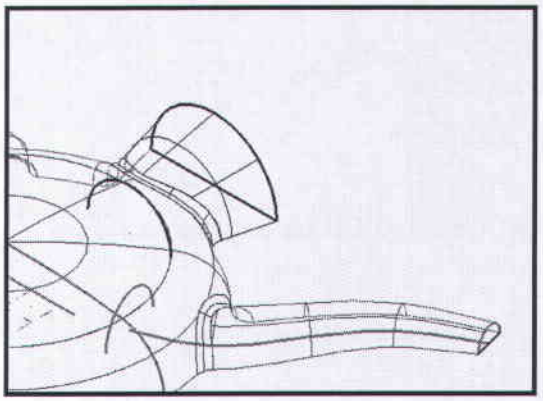
➢ Select the **3D** toggle function from the **Status bar**.
● Note that the system will automatically change the plane to **2D**.

7.2 Create the lines to close the contours.

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



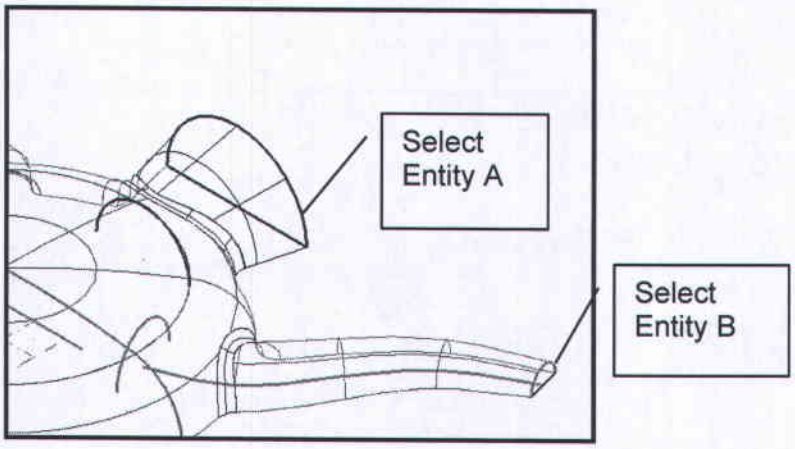
● When you move the cursor exactly on the endpoint of the entity the system will display a small square.

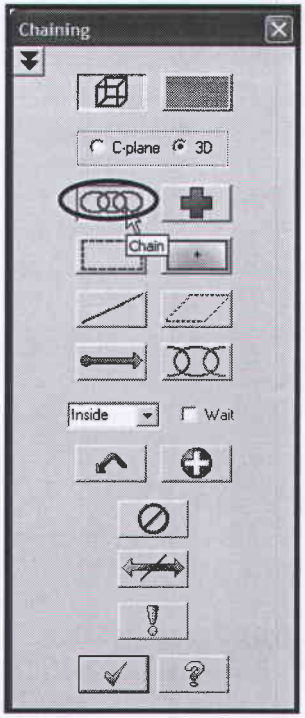


➢ [Specify the first endpoint]: Select Endpoint A.
➢ [Specify the second endpoint]: Select Endpoint B.
➢ [Specify the first endpoint]: Select Endpoint C.
➢ [Specify the second endpoint]: Select Endpoint D.
➢ Select the **OK** button to exit the command.

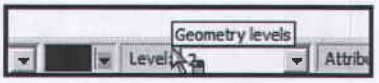
7.3 Create the flat surfaces

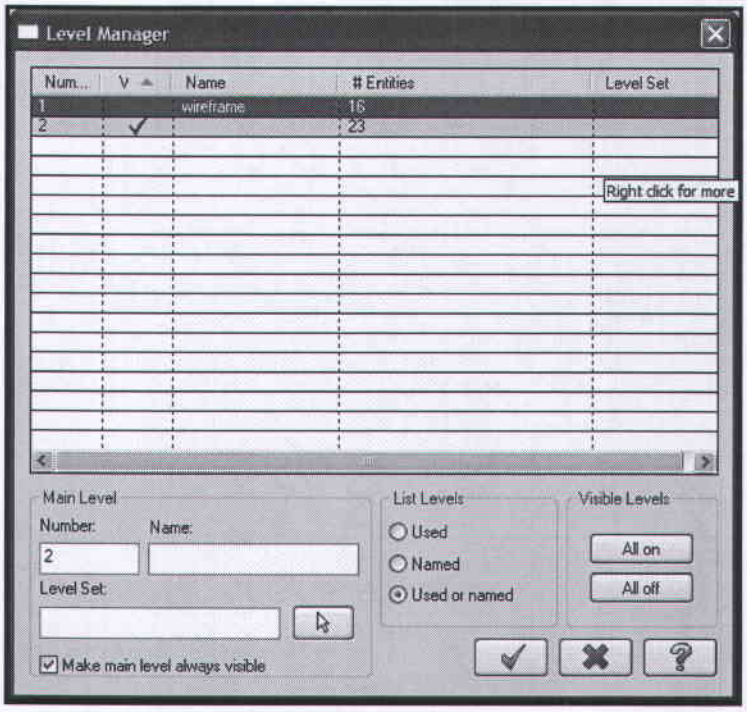
Create
- **Surface**
- **Create Flat Boundary Surface**
- [Select chains to define flat boundary 1]: Select Entity A.
- [Select chains to define flat boundary 2]: Select Entity B.



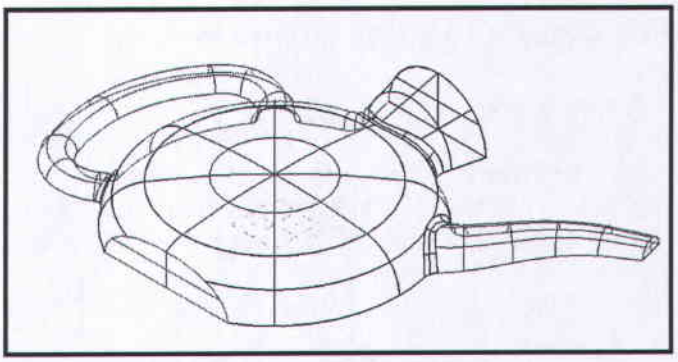


- Select the **OK** button to exit **Chaining**.
- Select the **OK** button to exit surface flat boundary command.
- Select the **Geometry Levels** from the **Status bar**.
- Enter "Surfaces" in the Name field and click on the check mark next to level 1 to disable it.
- The **Level Manager** should look as shown.



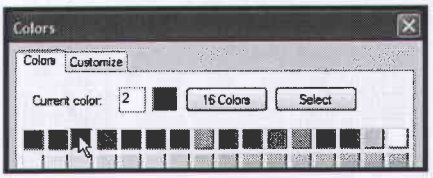


- Select the **OK** button to exit.



STEP 8:
CREATE THE BOUNDING BOX TO REPRESENT THE STOCK.

8.1 Change the Main color to green (#2)



- Select **Color** from the **Status Bar** to change the main color to green (**#2**).
- Select the color **green** (**#2**).
- Select the **OK** button to exit **Colors**. 

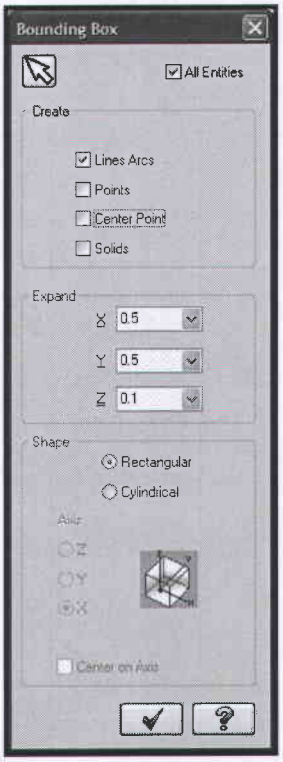
8.2 Change the plane to 3D.

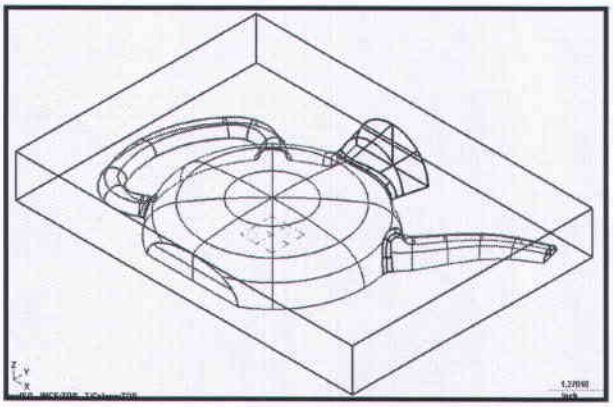
- Select the **2D/3D** toggle function from the **Status bar**.
- Note that the system will automatically change the plane to **3D**.

8.3 Create the bounding box.

Create

- **Create Bounding Box**
- Make sure that you have the same settings as shown in the screenshot to the right.
- The part should look as shown.





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

STEP 9:
CREATE A RECTANGLE WITH A SURFACE ON.

9.1 Change to 2D.

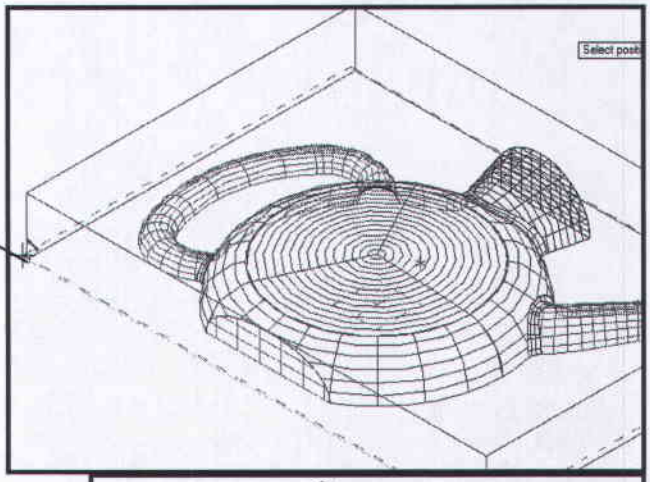
➢ Select the **2D/3D** toggle function from the **Status bar**.
✹ Note that the system will automatically change to **2D** mode.

9.2 Create the rectangle with surface on.

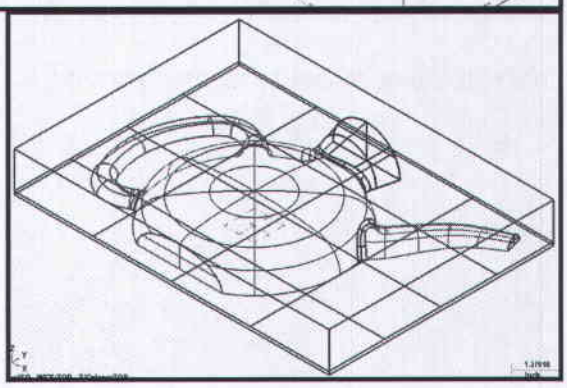
Create
➢ **Create Rectangle**

➢ Enter the **Width** 10.5 (Tab).
➢ Enter the **Height** 7.5 (Enter).
➢ Select the **Surface** button to enable it.
➢ [Select position of first corner]: Select the lower left corner on the box as shown.

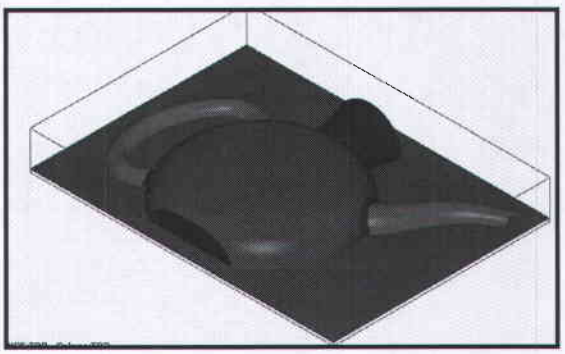
Select the lower left corner



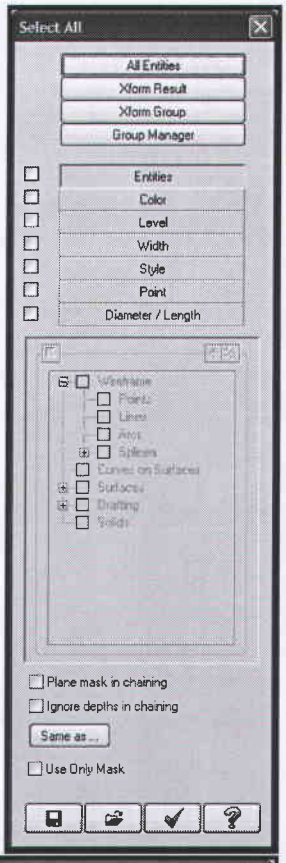
➢ Select the **OK** button.
➢ The part should look as shown.



➢ **Alt +S** to see the part in a shaded mode.
➢ **Alt+S** again to change it back to wireframe mode.



STEP 10:
SET UP THE DATUM (WORK ZERO) AT THE BOTTOM AND CENTER OF THE BOX.



➢ Set the plane back to **3D**. 

Xform

➢ **Xform Translate**

➢ Select the **All** button. 

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

➢ Press **Enter** to end the selection.

➢ Select the **Move** radio button as shown in the screenshot.

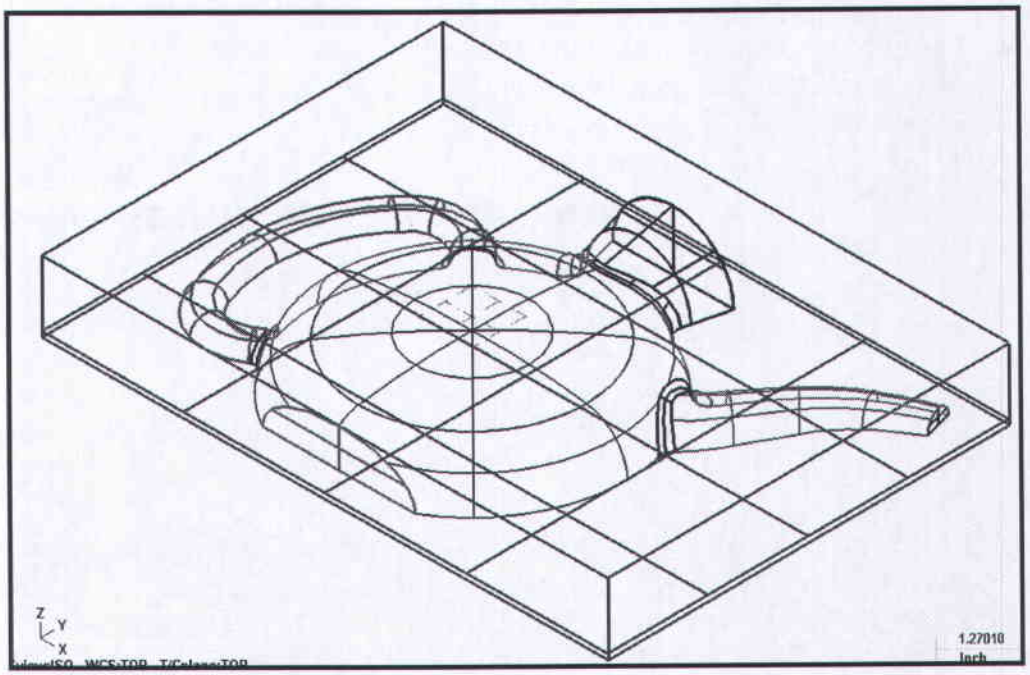
➢ Enter **Delta Z** value -1.225.

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

Screen

➢ **Clear Colors**

➢ The geometry should look as shown below.



STEP 11:
SAVE THE DRAWING.

File

➢ **Save as**

➢ **File name**: "Your Name_15"

➢ Select the **OK** button.

TOOLPATH CREATION
STEP 12:
SET UP THE STOCK TO BE MACHINED.

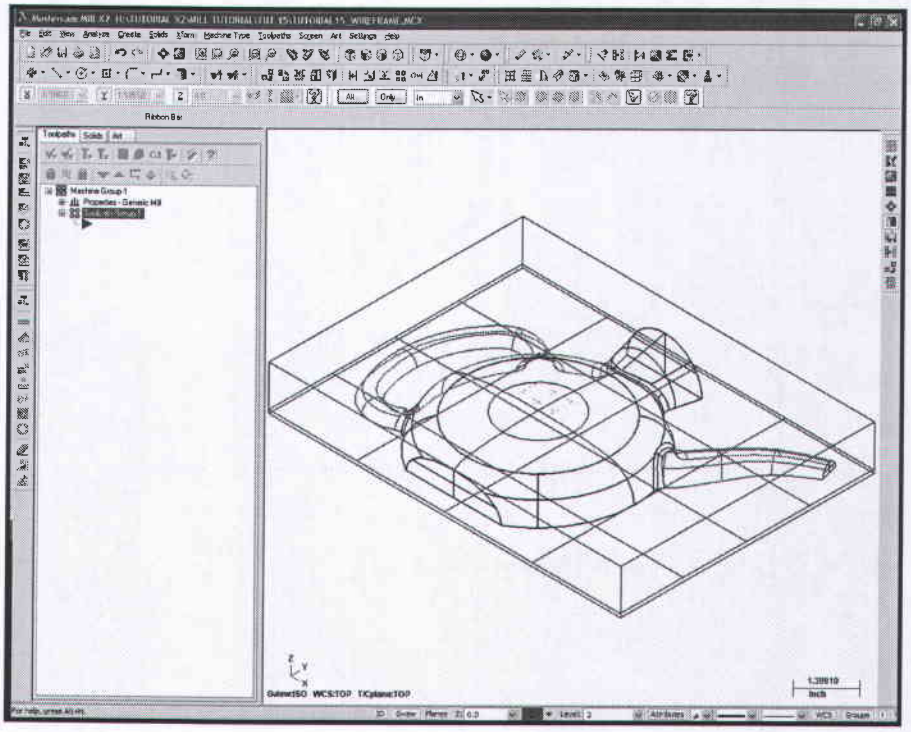
➢ To display the **Toolpaths Manager** press **Alt + O**.

Follow the next step to select the **Mill Default** only if there isn't any machine already selected ; Otherwise, use the existing one.

Machine type

➢ **Mill**

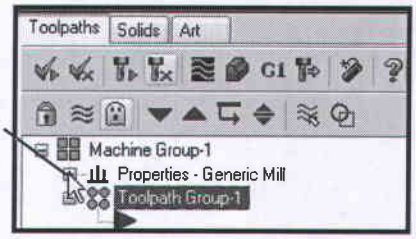
➢ Select **Default**.



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

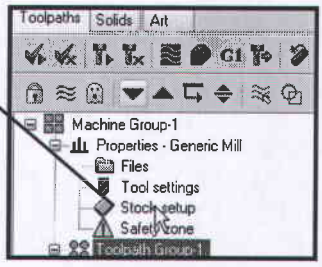
➢ Select the **+** in front of **Properties** to expand the **Toolpaths Group Properties**.

Select the plus

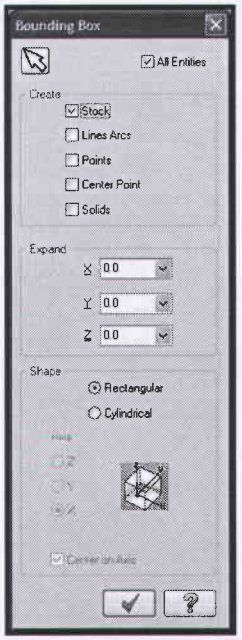


➢ Select **Stock setup**.

Select Stock setup

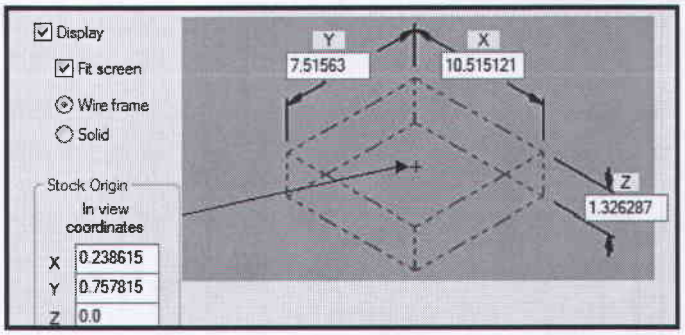


➢ Change the parameters to match the following screenshot.
➢ Select the **Bounding Box** button to automatically find the part extents.

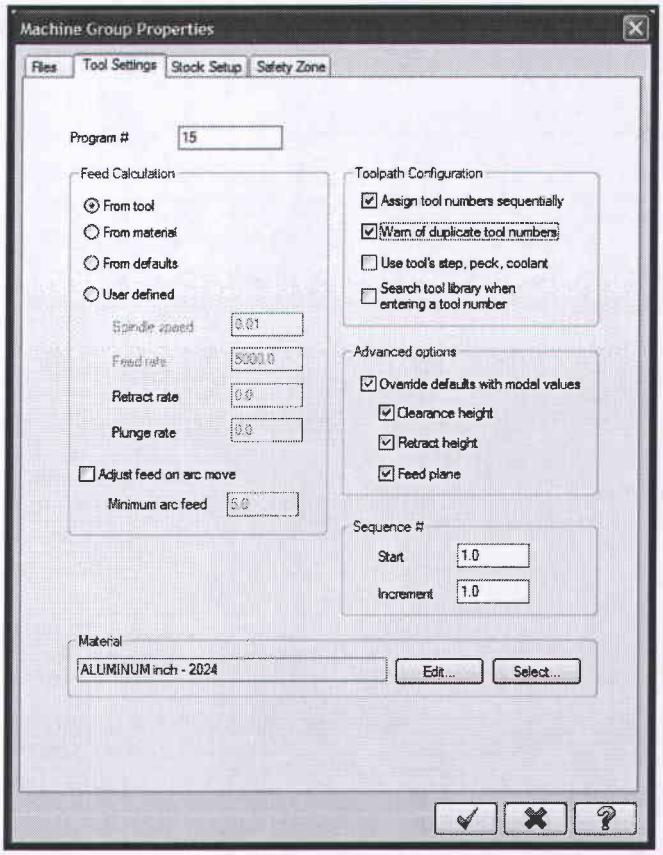


➢ Make sure that the expand values are set to 0.

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



- 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 13:
SURFACE HIGH SPEED - CORE ROUGHING.

Core roughing toolpaths are designed for machining cores which can be approached from the outside. An important feature of core roughing is that Mastercam can change the machining strategy within the same operation if your part has, for example, a mixture of bosses and cavities. In these cases, Mastercam will cut the cavities inside to out (like an area clearance cutting pass), and machine the bosses from the outside.

Toolpaths

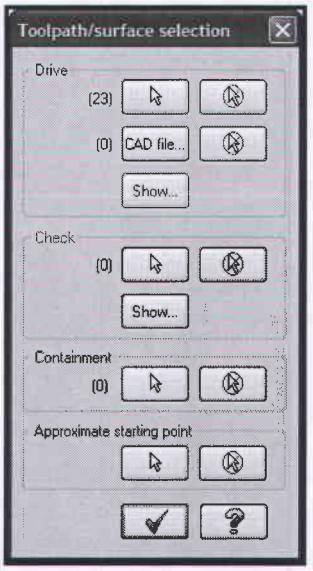
- **Surface High Speed Toolpath**
- Select the **OK** button to accept the **NC name**

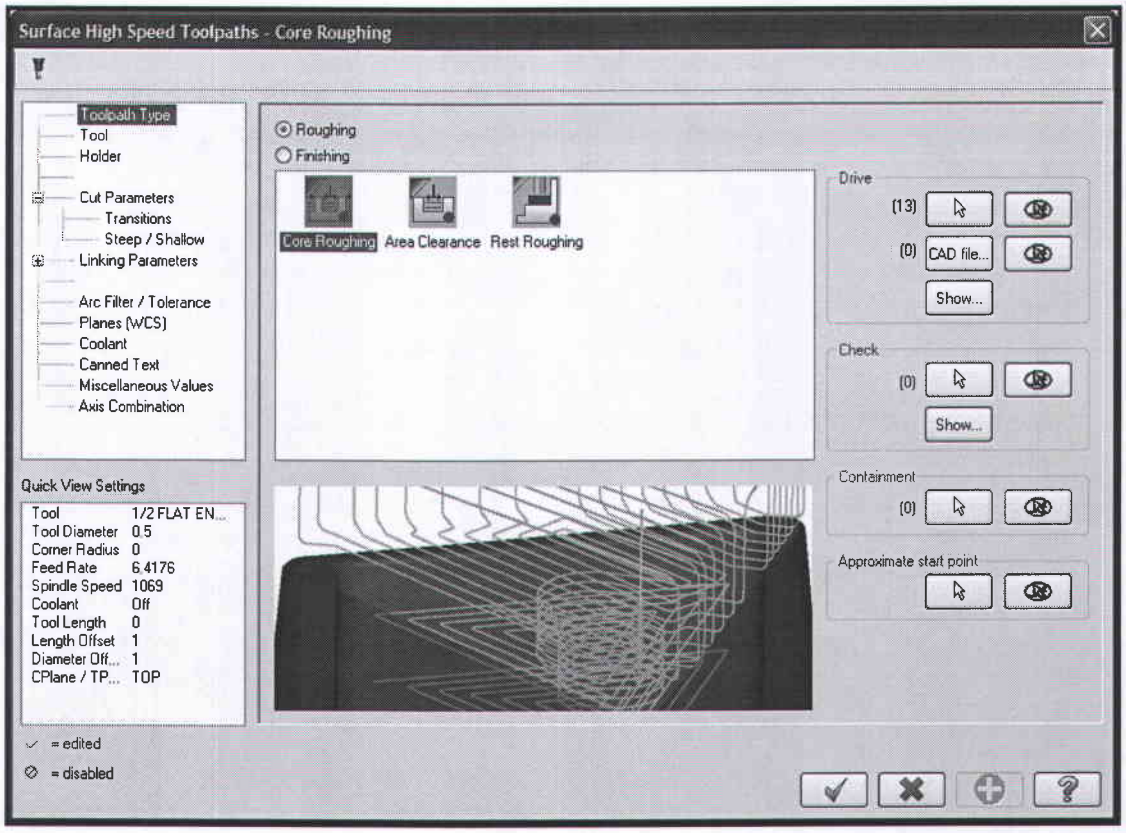
➢ [Select Drive Surface]: Select the **All** button. 

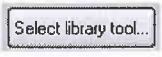
➢ Select the **OK** button to exit. 

➢ Press **Enter** key to end the selection.

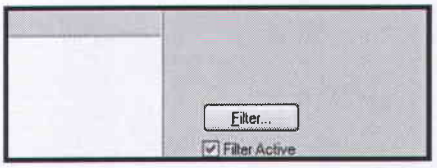
➢ Select the **OK** button to exit **Toolpath/surface selection.** 



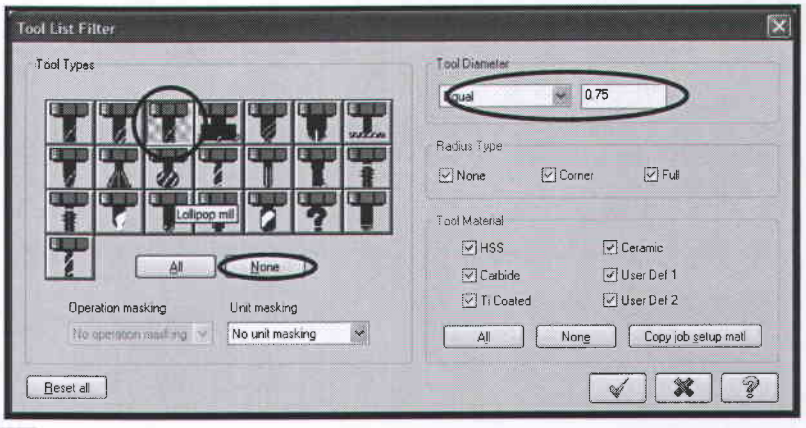


- Enable **Roughing** and select **Core Roughing**.
- Select **Tool** page.
- Click on the **Select library tool** button. 

- Select the **Filter** button in the **Tool Selection** dialog box.
- Select the **None** button in the **Tool Types** area.

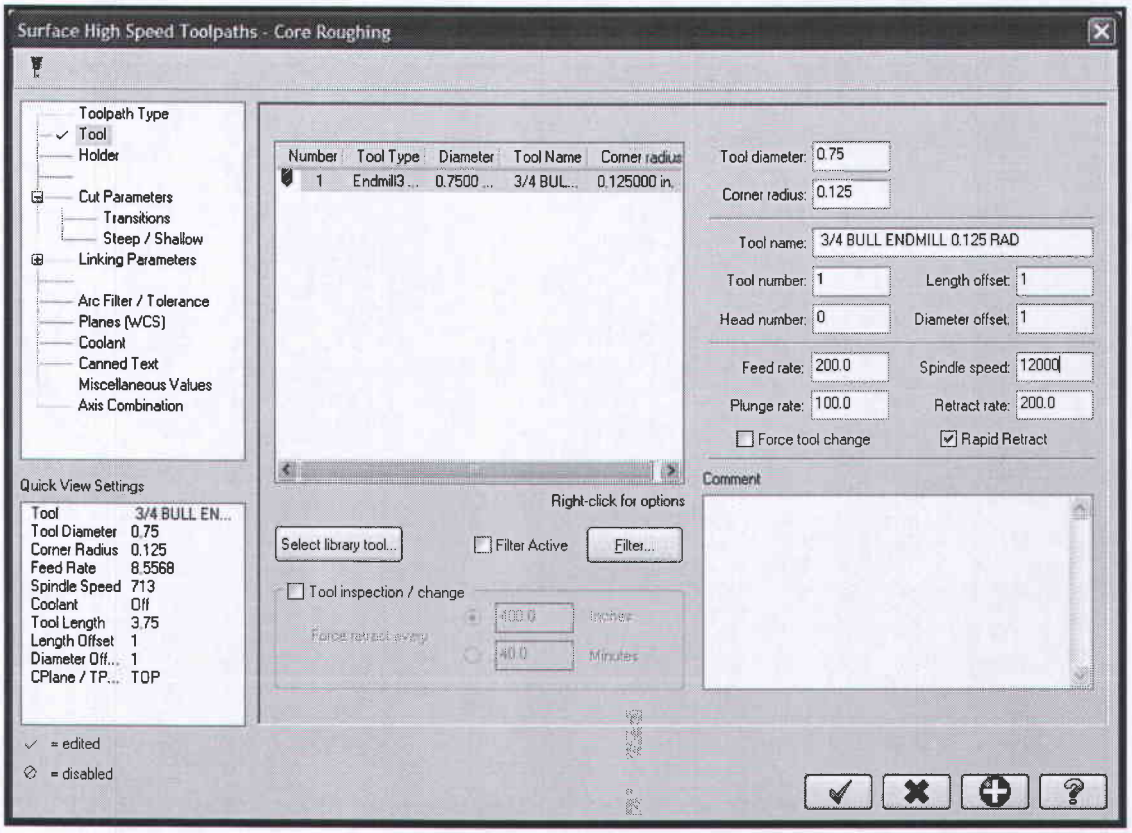


- Click on the **Bullnose** icon to select the tool type.
- Select the **drop-down arrow** in the **Tool Diameter** field, and select **Equal**.
- Enter the diameter 0.75.



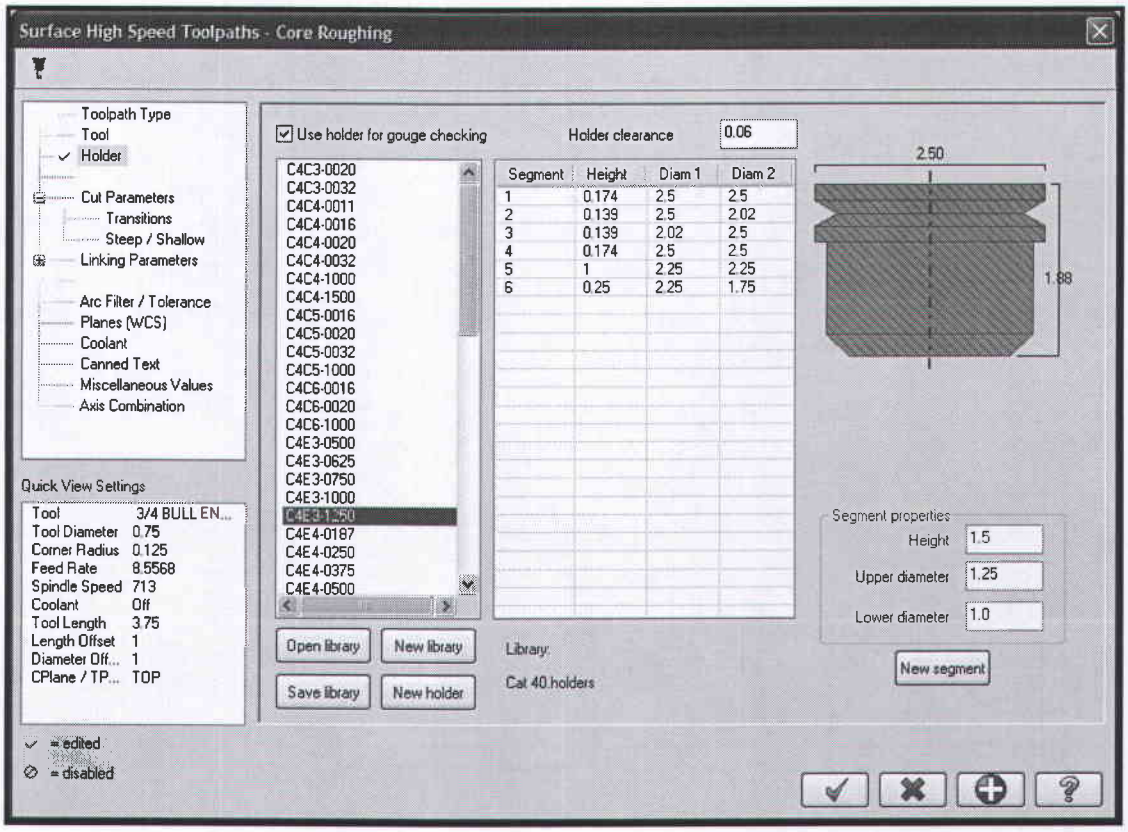
- Select the **OK** button to exit. 

- Select **0.75" Bull Nose** with a **.125" corner radius**.
- Select the **OK** button to exit **Tool Selection**.
- Make the necessary changes in the **Toolpath parameters** to match the following screenshot.



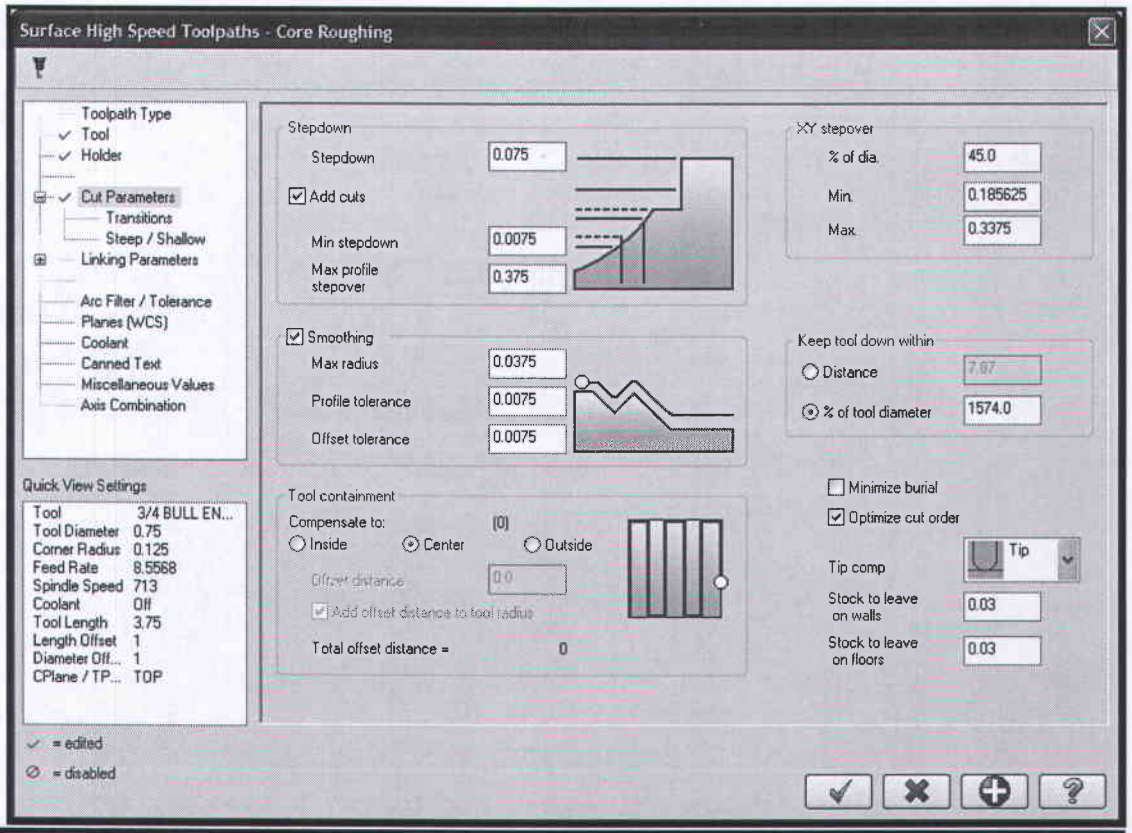
- Select the **Holder** page.

- Select **Open library button.**
- Select the **Cat 40.holders** and **Open** 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 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 to establish the minimum separation between the holder and your surface model. Set the clearance bigger than the stock to leave on the walls.

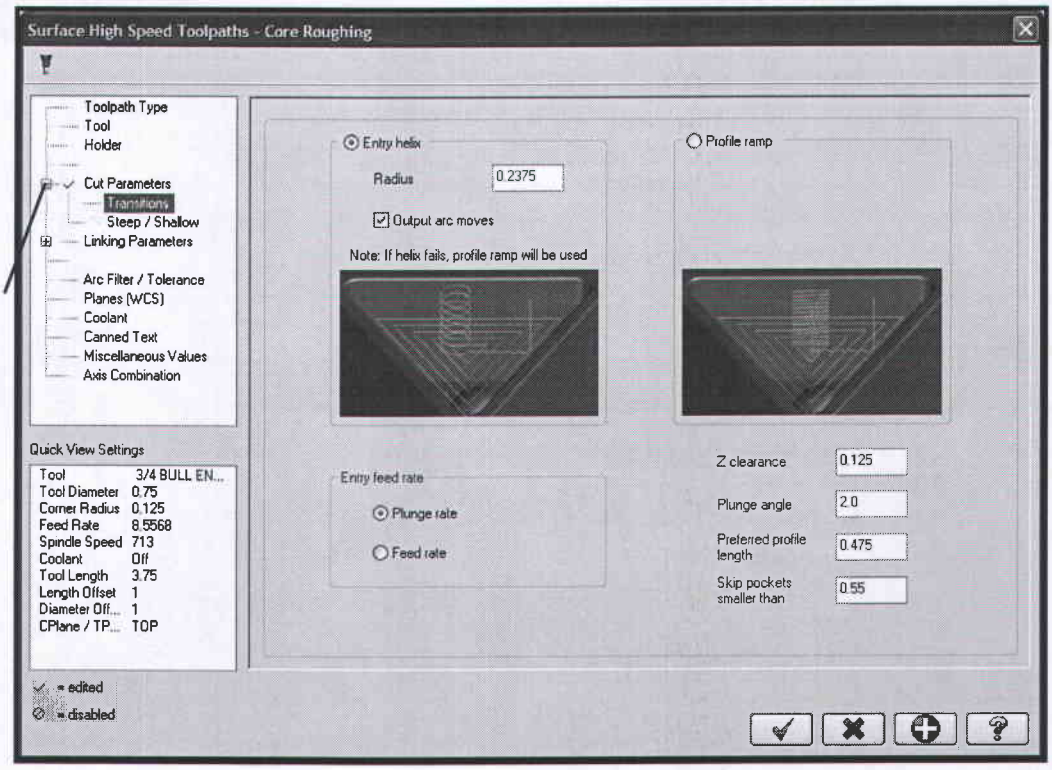
➢ Select the **Cut Parameters** page and make the changes to match the following screenshot.



Stepdown options allows you to configure how Mastercam spaces the cuts in Z.
Stepdown 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 stepdown.
Smoothing settings allows you to insert arcs at corners to create smoother tool motion. The arc radii will depend on the maximum radius that you enter, and the profile/offset tolerances.
Profile tolerance determines the maximum deviation between the smoothed and unsmoothed toolpaths and is only applied on the outermost cutting pass.
Offset tolerance is defined in the same way as the **Profile tolerance**, but it is applied to all the inner passes.
XY stepover settings allows 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.

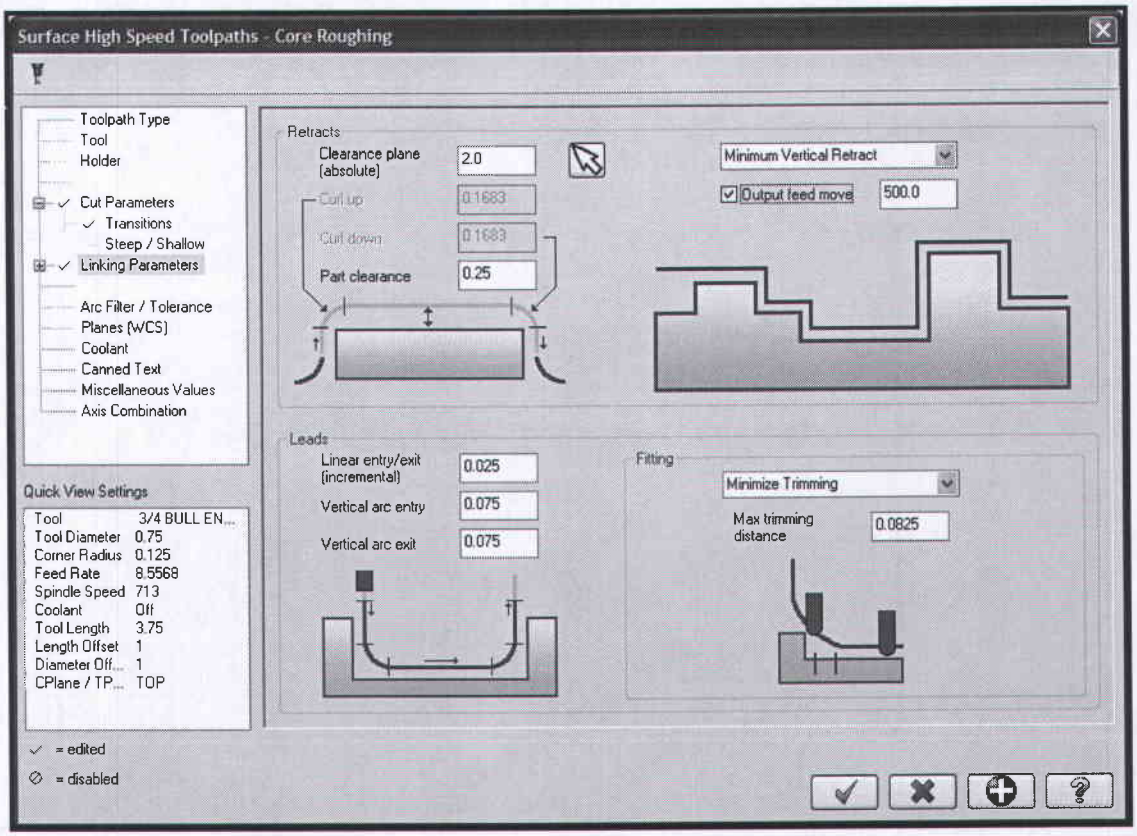
➢ Expand **Cut Parameters** and select **Transitions** to set the **Entry helix**.

Select the plus to expand



Transition allows 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 allows 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.

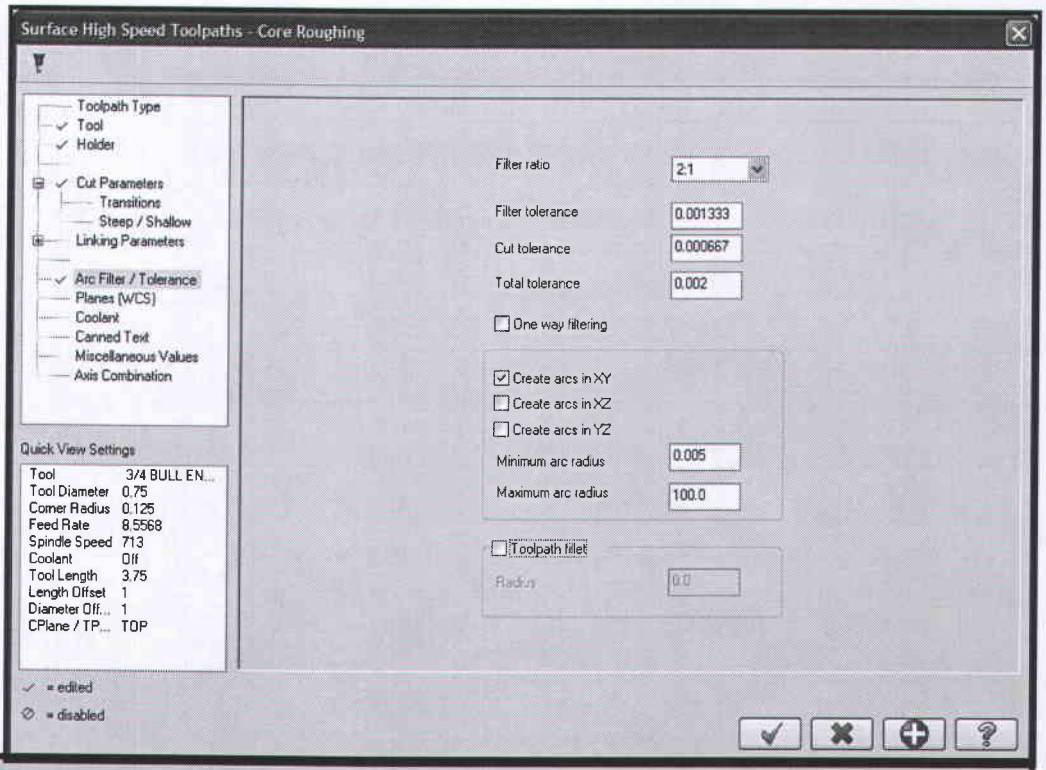
Leads fields set the tool moves onto and off of the part at the start and end of each cutting pass. These moves are applied to each pass no matter which cutting pass is selected.

Fitting settings allows you to choose how the entry and exit arcs fit to the ends of the cutting passes.

Minimize trimming sets the path of the retract to be as close to the surface as possible, maintaining a minimum distance from the surface to fit the arc.

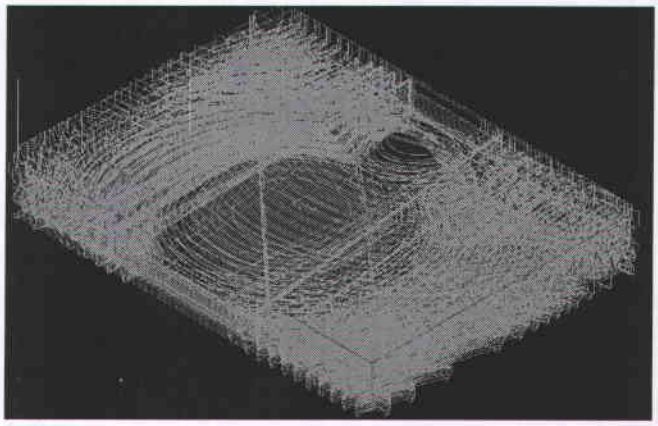
Max trimming distance parameter limits the amount of trimming applied to non-horizontal passes.

➢ Select **Arc Filter/ Tolerance**



Filter ratio is the ratio between the filter tolerance and the cut tolerance and is typically set to 2:1. When you filter a toolpath, Mastercam replaces multiple small toolpath moves that lie within a specified tolerance with a single tool move to simplify the toolpath. You can also optionally replace multiple linear tool moves with an arc move of a specified minimum and maximum radius.
Cut tolerance determines the accuracy of the surface toolpaths using chordal deviation to linearize the toolpath.
Total tolerance is calculated as the sum of the cut tolerance and filter tolerance.

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

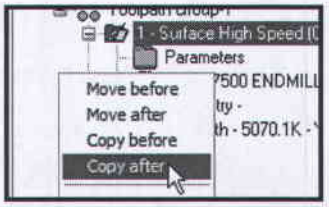


➢ Select **Toggle toolpath display on** selected operations to remove the toolpath display.

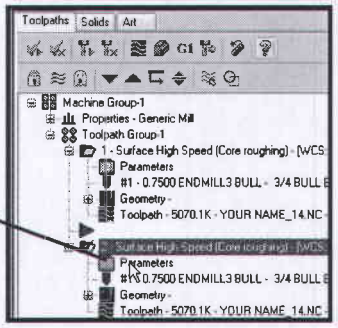
STEP 14:
HIGH SPEED – FINISH HORIZONTAL AREA.

Horizontal Area toolpaths are designed for machining the flat areas of your surface model. Mastercam will create cutting passes at the Z height of each area. Even if the drive surface as a whole is not flat, Mastercam will identify and only machine the flat areas. Mastercam will automatically calculate the toolpath in such a way that the tool does not exit on a sidewall.

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



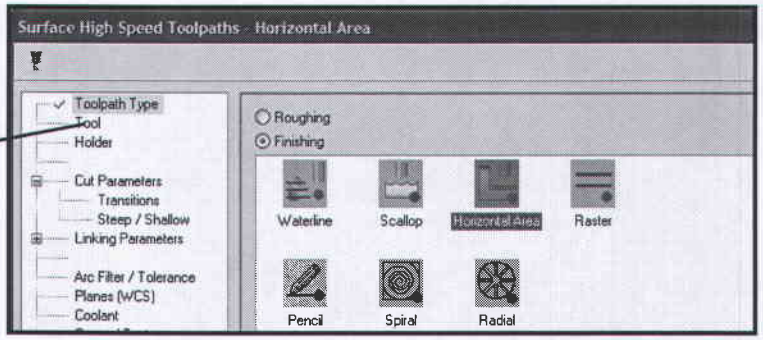
➢ You should now have two operations.
➢ Left-click on the second operation **Parameters**.



Select Parameters

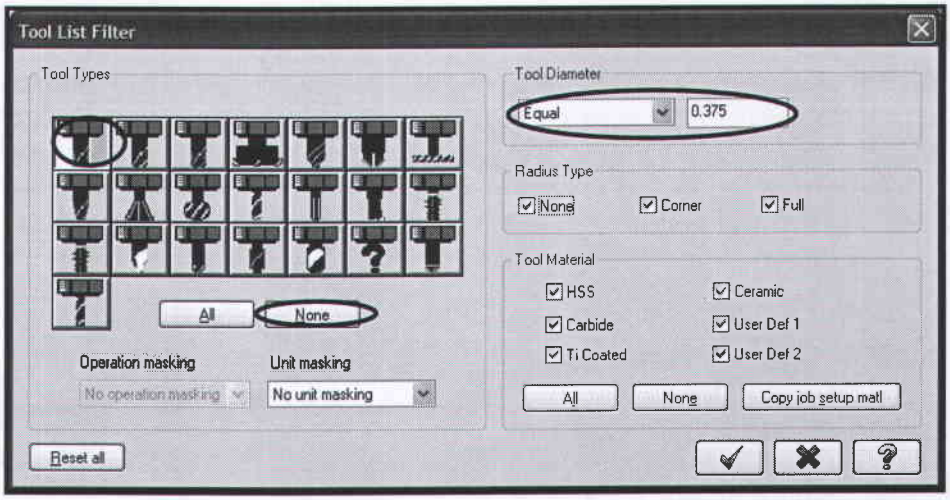
➢ Select **Toolpath Type**
➢ Enable **Finishing and choose Horizontal Area**

Select Tool page



➢ Select **Tool** page and click on the **Select library tool** button. Select library tool...
➢ Select the **Filter** button.

➢ Make sure that you select the **None** button first and then the **Endmill Flat** as the tool type.

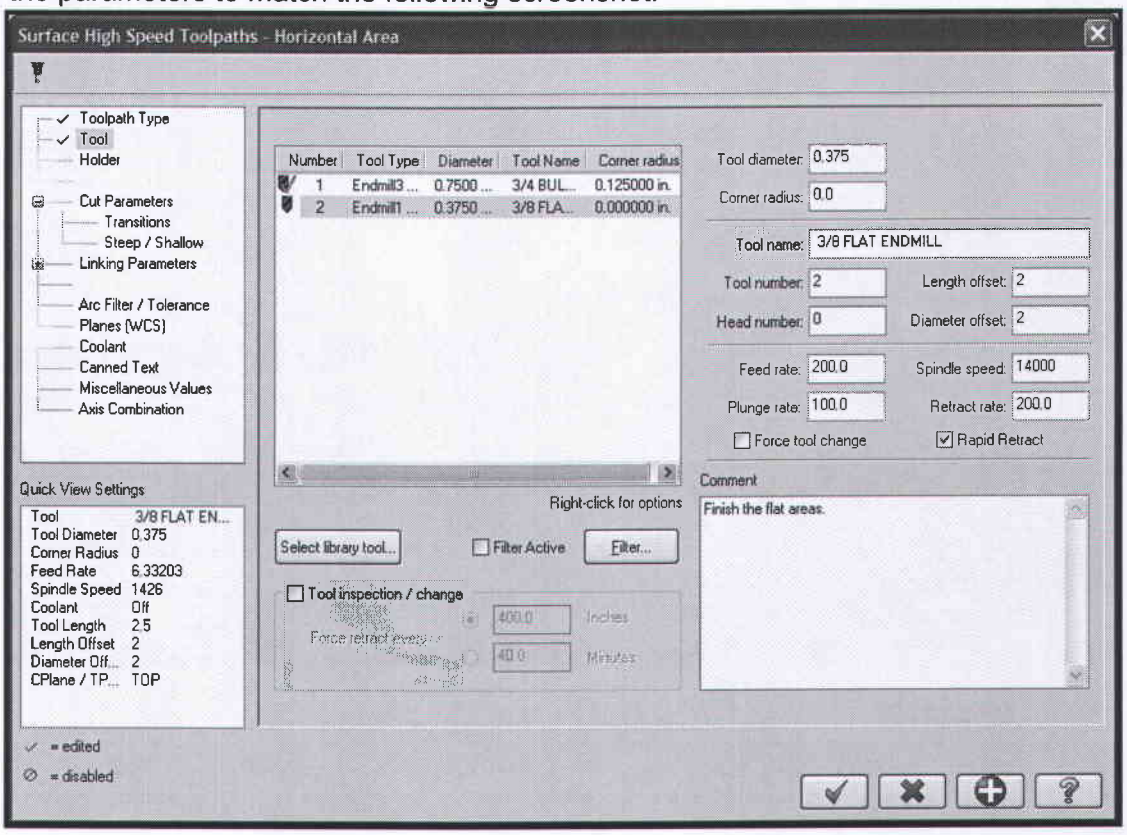


➢ Change the **Tool Diameter** to **Equal** and 0.375".

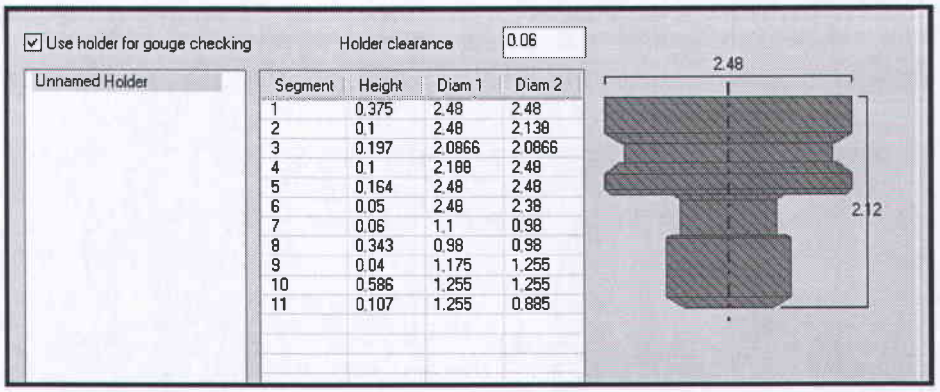
➢ Select the **OK** button to exit **Filter**.

➢ Make sure that the tool is selected and select the **OK** button to exit **Tool Selection**.

➢ Change the parameters to match the following screenshot.

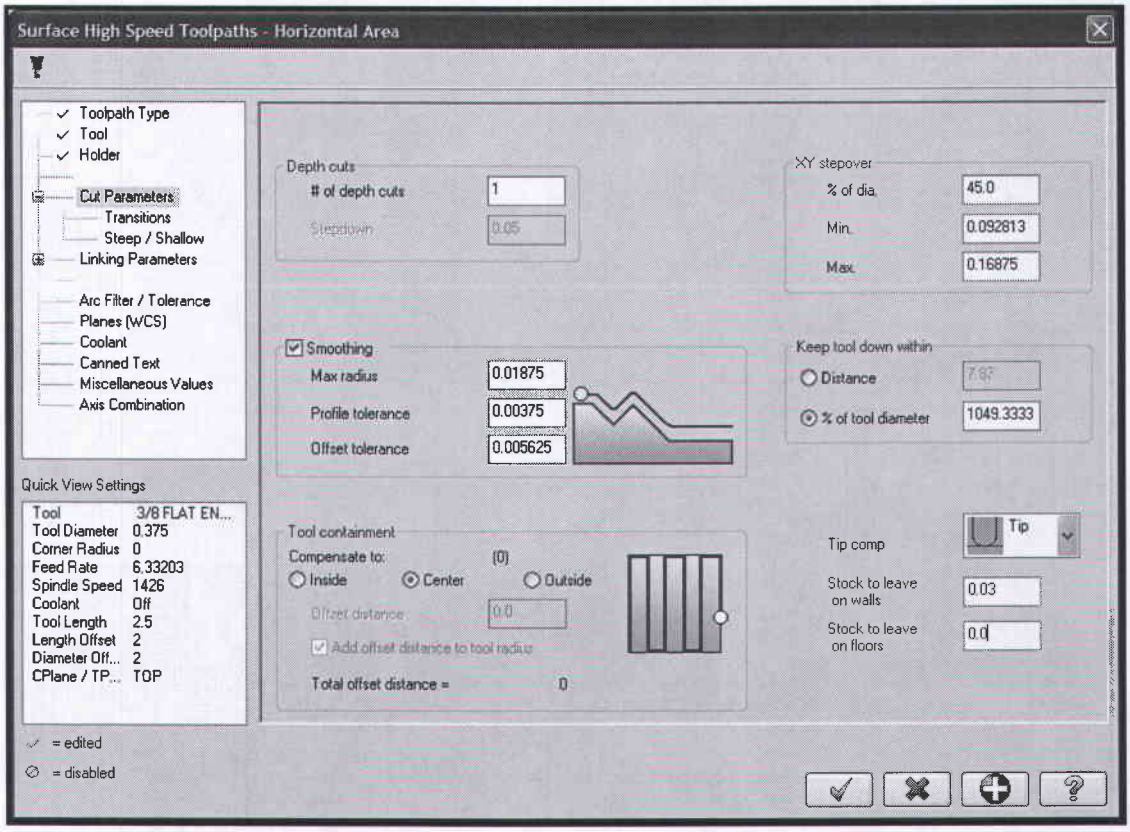


➢ Select the **Holder** page

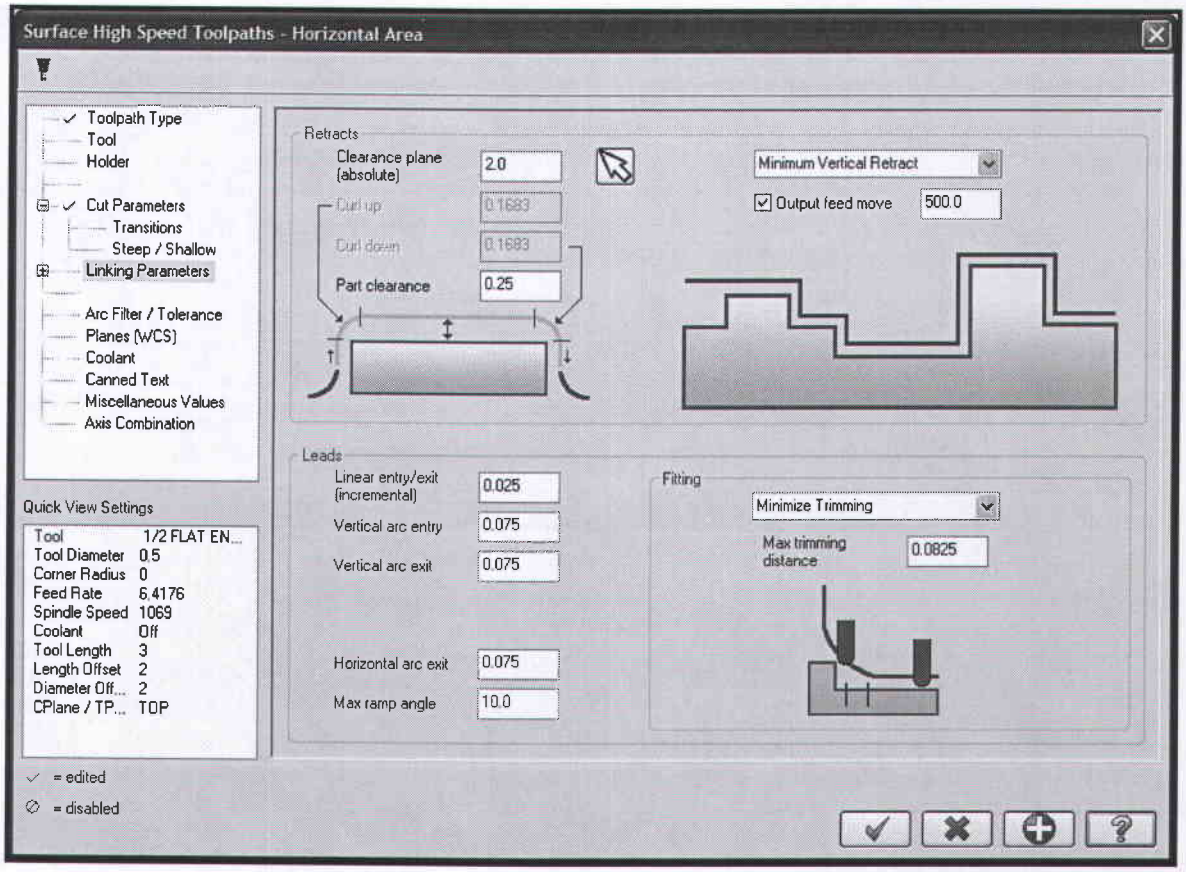


Note the holder is already selected from the previous operation that we copied.

➢ Select the **Cut Parameters** page and change the parameters as shown.

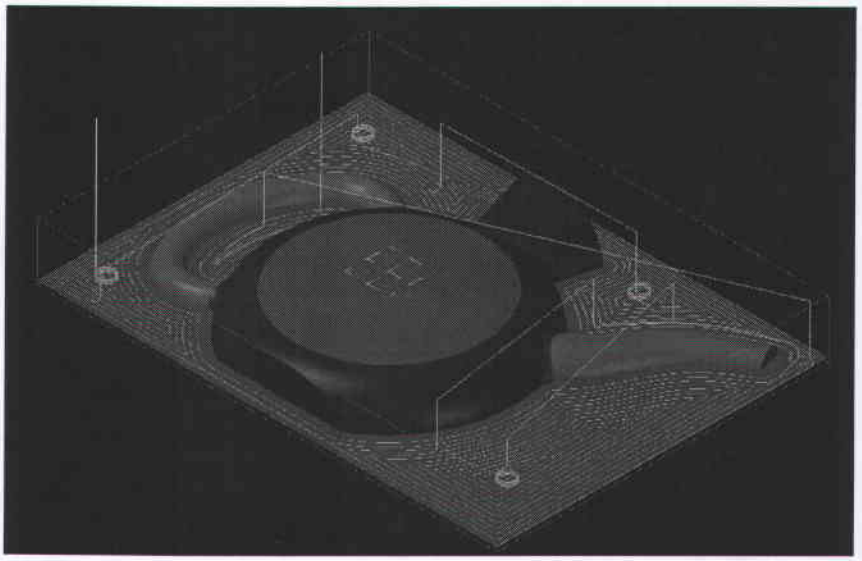


➢ Select **Linking Parameters** page.

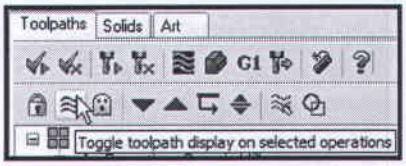


➢ Select the **OK** button to exit parameter pages.

➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.



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

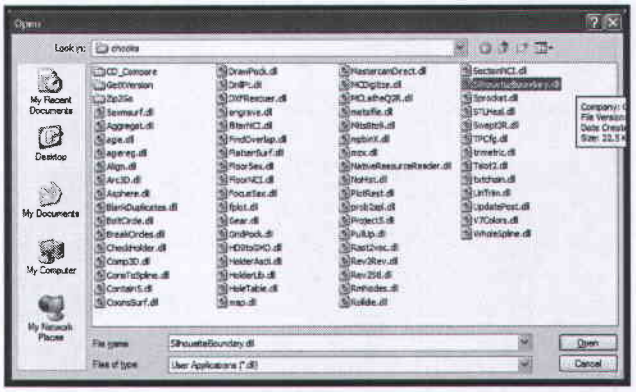


STEP 15:
CREATING A BOUNDARY USING SILHOUETTEBOUNDARY.DLL C-HOOK.

C-Hooks are and **NET-Hooks** are add-in applications or utilities that customize, enhance, or extend Mastercam's functionality. They can be created by individual users, resellers, third-party application developers, or by CNC Software itself.

➢ Change z depth of the construction plane to 2.

➢ To open the **Chooks** directory press **Alt+C**.
➢ Select **SilhouetteBoundary.dll from the list.**



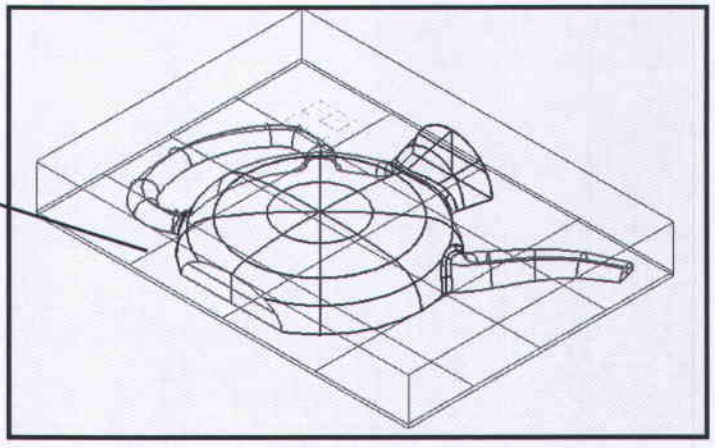
➢ Select the **Open** button.

➢ [Select surfaces, solids or solid faces]: Select the **All** button.

➢ Select the **OK** button.

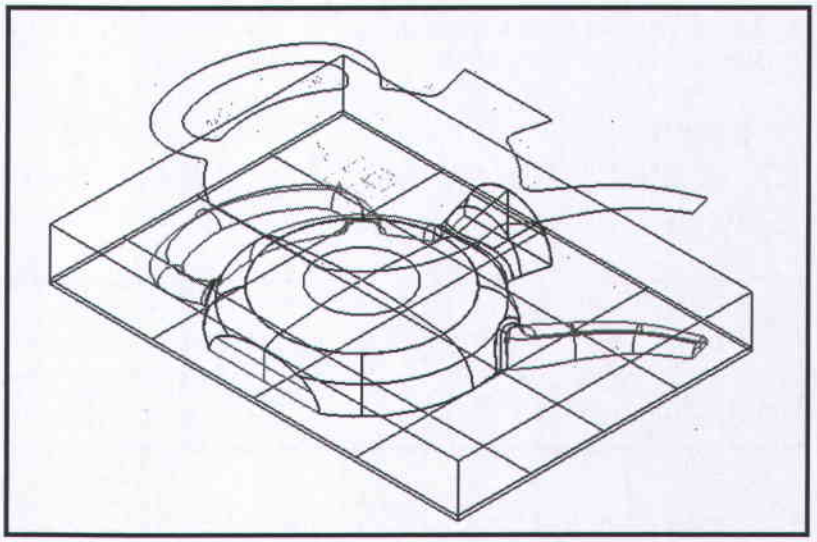
➢ Select the flat bottom surface to unselect it.

Select this flat surface



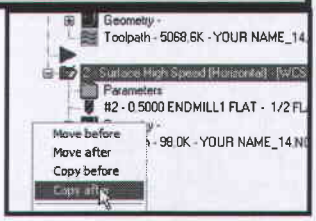
➢ Press **Enter** to finish the selection.

➢ The boundaries will look as shown in the following screenshot.



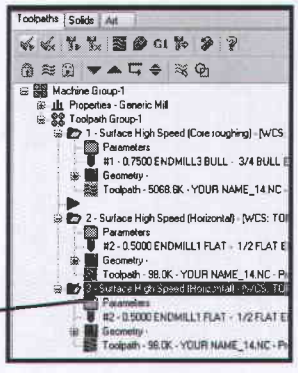
STEP 16:
HIGH SPEED – FINISH SCALLOP.

High speed scallop toolpaths differ from other finish toolpaths in that the stepover distance is a 3D value which is measured along the surface, instead of parallel to the toolplane. This ensures a consistent scallop height across the surface, regardless of the surface direction.



➢ Select **Toolpath Manager.**
➢ **Right-mouse click** (hold it down) on the folder icon in front of the **Surface High Speed (Horizontal)**

➢ Drag the mouse down and release it.
➢ Select **Copy after.**

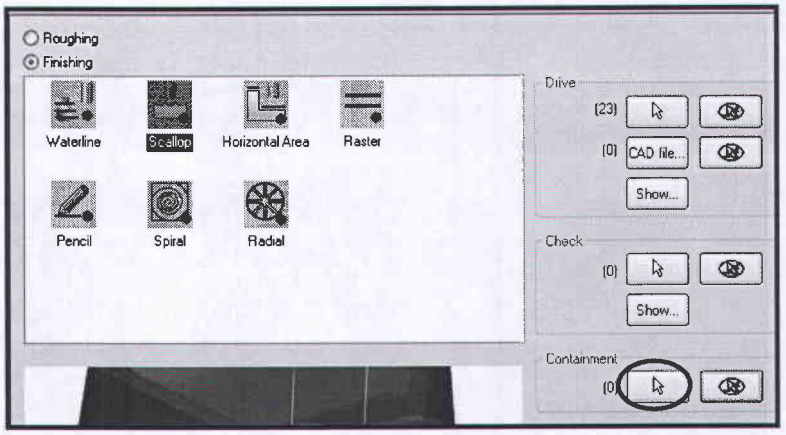


➢ You should now have two **Surface High Speed (Horizontal)** operations.
➢ **Left-click** on the third operation **Parameters**.

➢ Select **Toolpath Type**

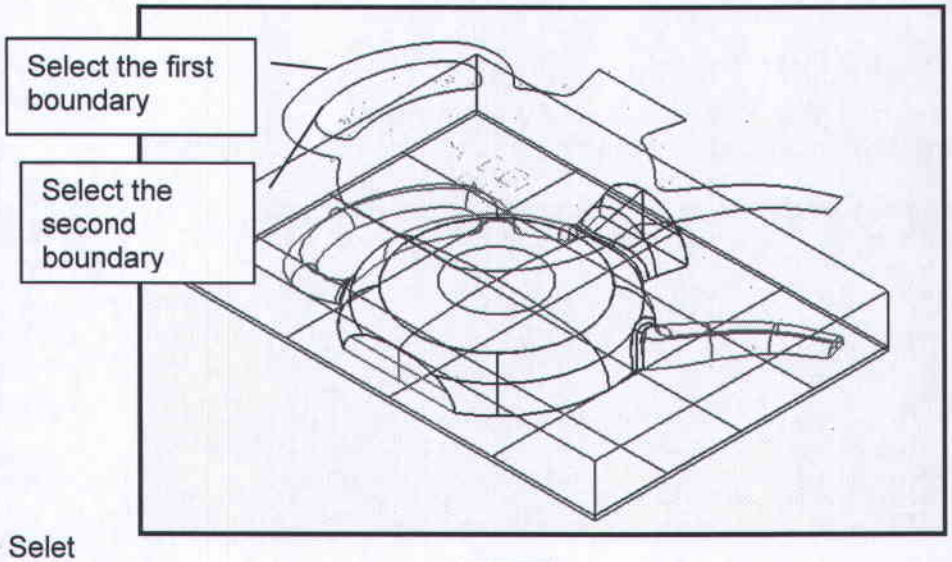
Select Parameters

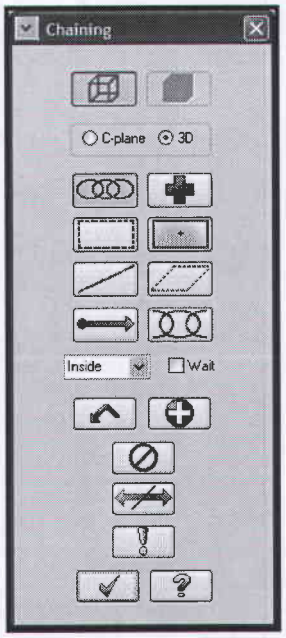
➢ Enable **Finishing** and choose **Scallop**



➢ Click on **Containment** button as shown.

➢ Select the first boundary as shown.





➢ Selet the **End chain** button in the **Chaining** dialog box.

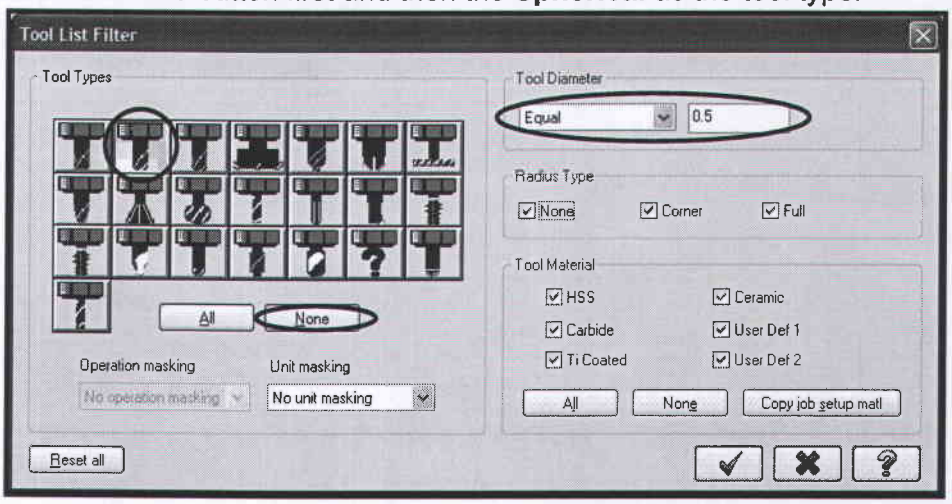
➢ Select the second boundary as shown above.

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

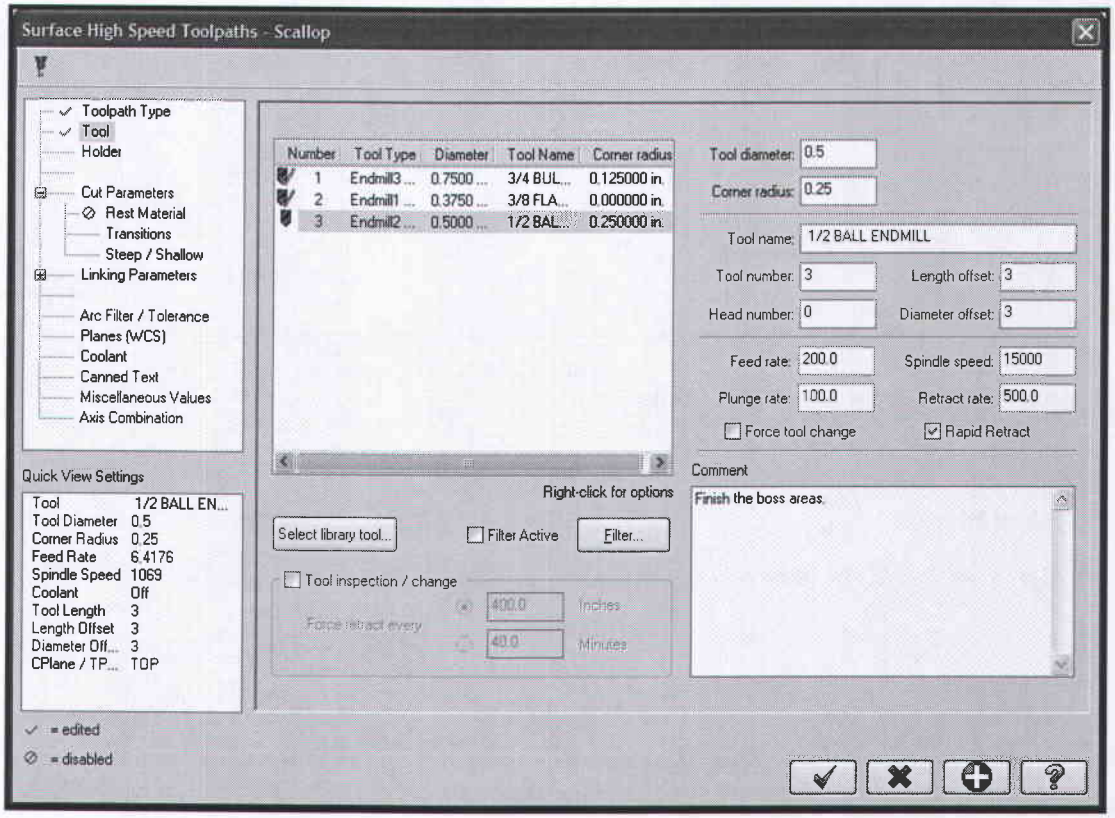
➢ Select the **Tool** page.

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

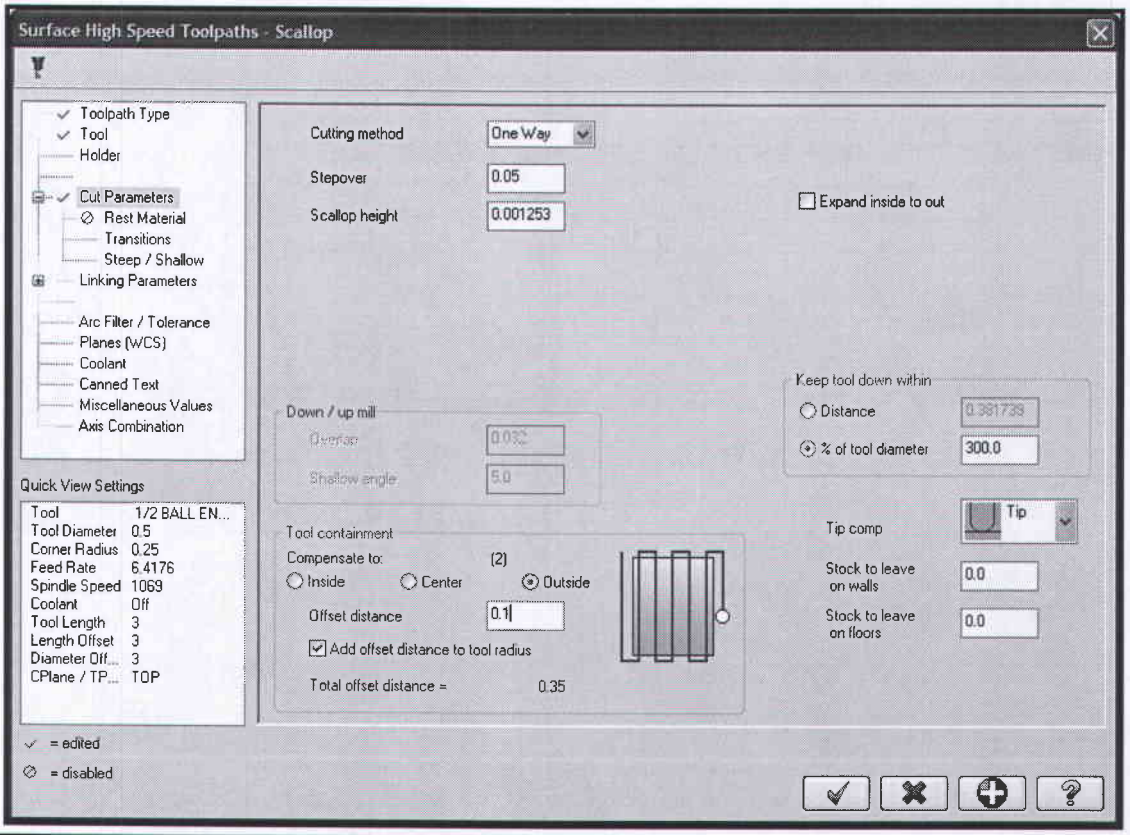
- Use the **Filter** option to select the **0.5 Ball Endmill**.
- Make sure that you select the **None** button first and then the **Spherical** as the tool type.



- Change the **Tool Diameter** to **Equal** and 0.5".
- Select the **OK** button to exit **Filter**.
- Make sure that the tool is selected and select the **OK** button to exit **Tool Selection**.
- Change the parameters as shown in the following screenshots.



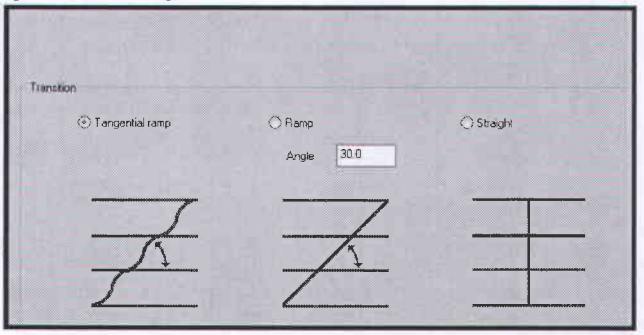
➢ Select the **Cut parameters** page and make the changes as shown.



Stepover is the spacing between cutting passes and is measured as a 3D value along the surface.
Scallop height value is linked to the stepover value. When you enter a value in one of the field the other field value is updated.
Expand inside to out disabled allows the machining to start from the outside towards the center of the part. In this case we avoid cutting with the tool center.

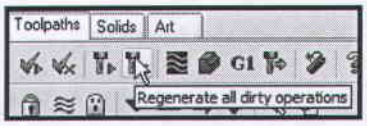
➢ Select the **Transitions page and enable Tangential ramp.**

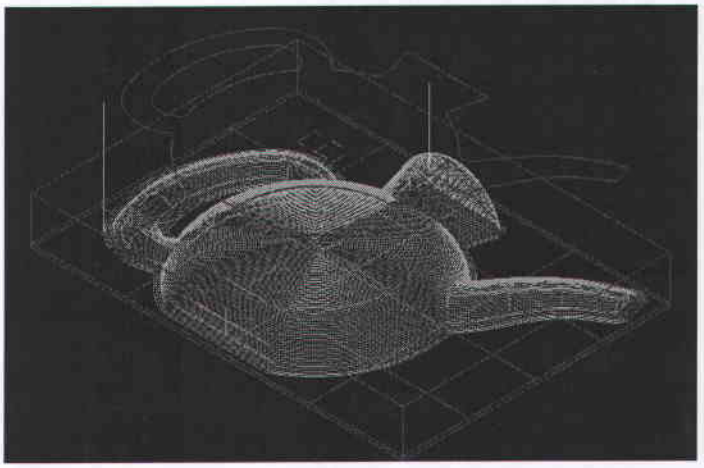
Tangential ramp creates a true high speed transition between the cutting passes. Mastercam inserts arcs at the beginning and end of the ramp for the smoothest tool motion into and out of the move.



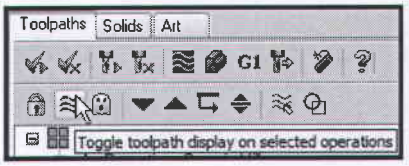
➢ Select the **OK** button to exit parameter pages.

➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.





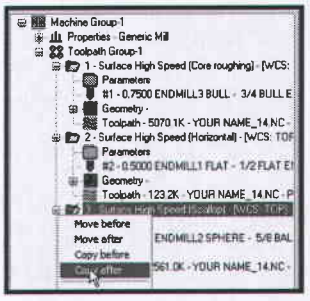
➢ Select **Toggle toolpath display on selected operations** to remove the toolpath display if needed.



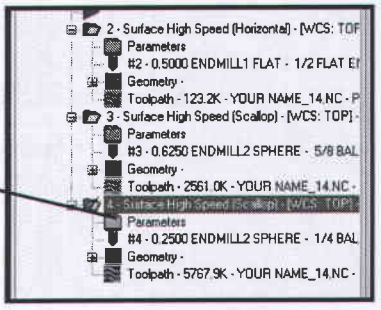
STEP 15:
REMACHINE THE SURFACE (SCALLOP REST PASSES).

Scallop rest passes is a finish toolpath that removes material in areas where the previous tool could not fit.

➢ Select **Toolpath Manager.**
➢ **Right-mouse click** (hold it down) on the folder icon in front of the **Surface High Speed (Scallop)**

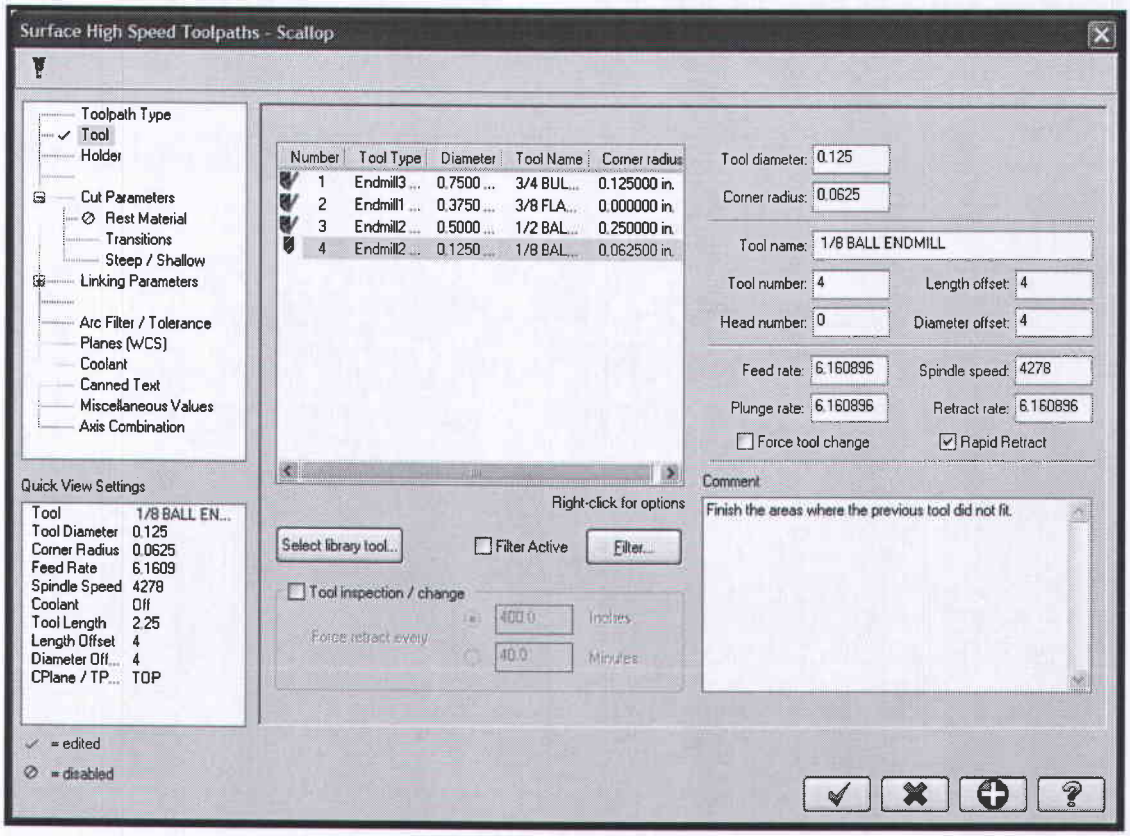


➢ Drag the mouse down and release it.
➢ Select **Copy after.**
➢ Select **Parameters** in the second **Surface High Speed (Scallop)** operation.



Select Parameters

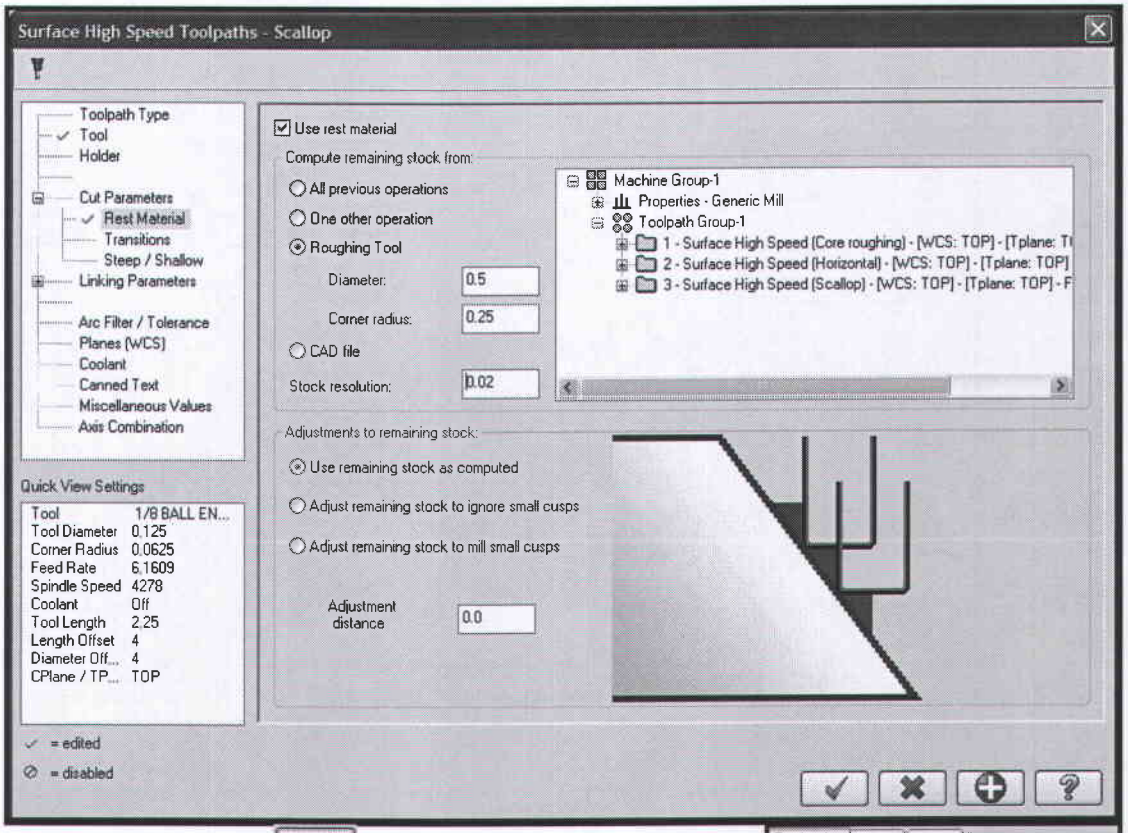
- Select **Tool** page
- Click on **Select library tool** and then on the **Filter to** select the 0.125 diameter End mill ball.
- Change the parameters in the **Tool** page as shown.



- Select the **Rest Material** page.
- Enable **Use rest material.**

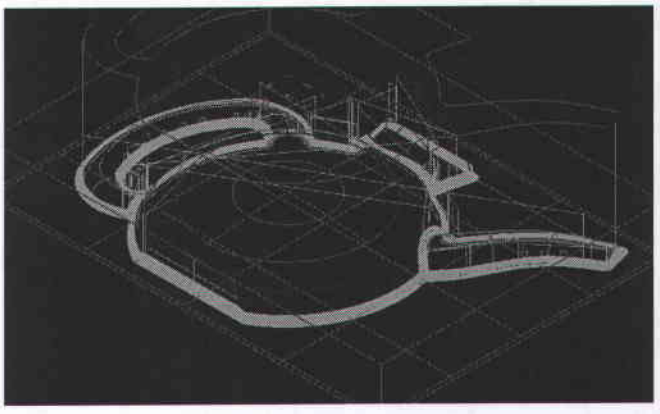
➢ **Enable Compute the remaining stock from the Roughing Tool Diameter** and change the parameters as shown.

Compute remaining stock from Roughing Tool enable Mastercam to calculate the stock based on the areas that could not be cut by a previous tool defined by its diameter and corner radius.

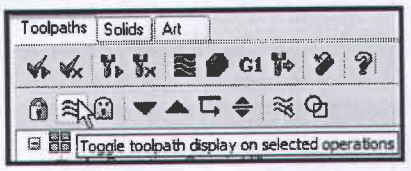


➢ Select the **OK** button to exit.

➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.



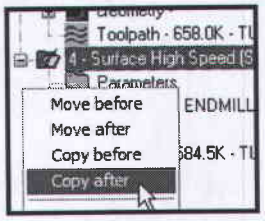
➢ Select **Toggle toolpath display on** selected operations to remove the toolpath display if needed.



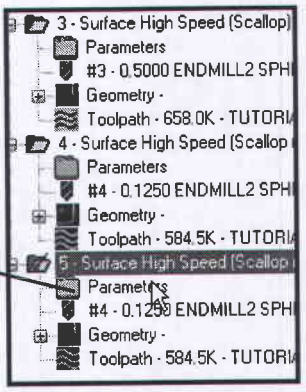
STEP 16:
HIGH SPEED – FINISH PENCIL.

High speed pencil toolpaths can be used as a finish operation to clean leftover materials from corners. The tool follows a contour defined by the intersection of two or more surfaces. You can create pencil toolpaths with either single or multiple passes.

➢ Select **Toolpath Manager.**
➢ **Right-mouse click** (hold it down) on the folder icon in front of the **Surface High Speed (Scallop rest passes)**



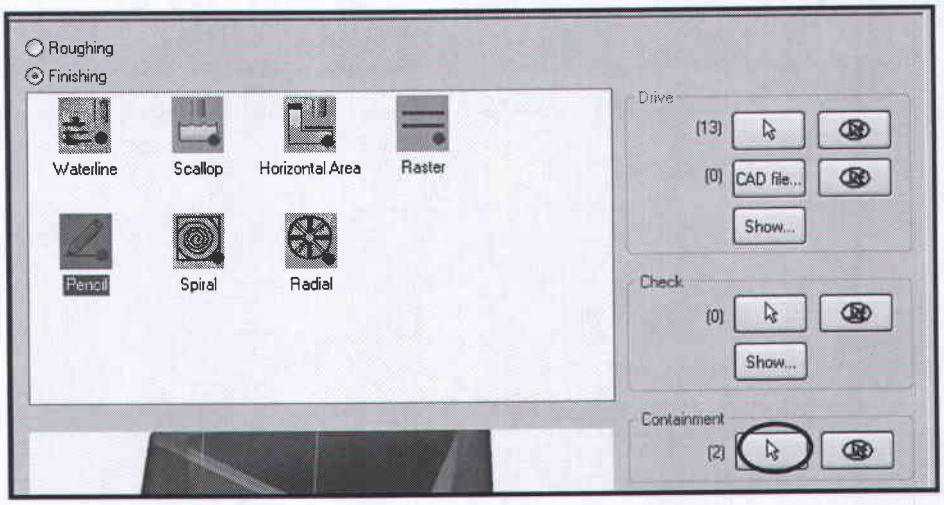
➢ Drag the mouse down and release it.
➢ Select **Copy after.**

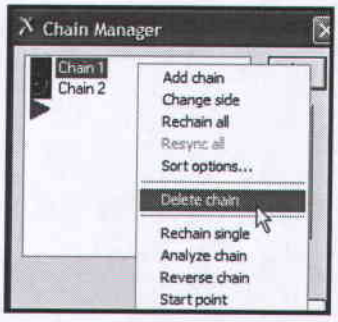


➢ You should now have three **Surface High Speed (Scallop)** operations.
➢ **Left-click** on the fifth operation **Parameters**.

Select Parameters

➢ Select **Toolpath Type**
➢ Enable **Finishing** and choose **Pencil**
➢ Click on **Select** button in the **Containment** field to remove the two boundaries used in scallop.

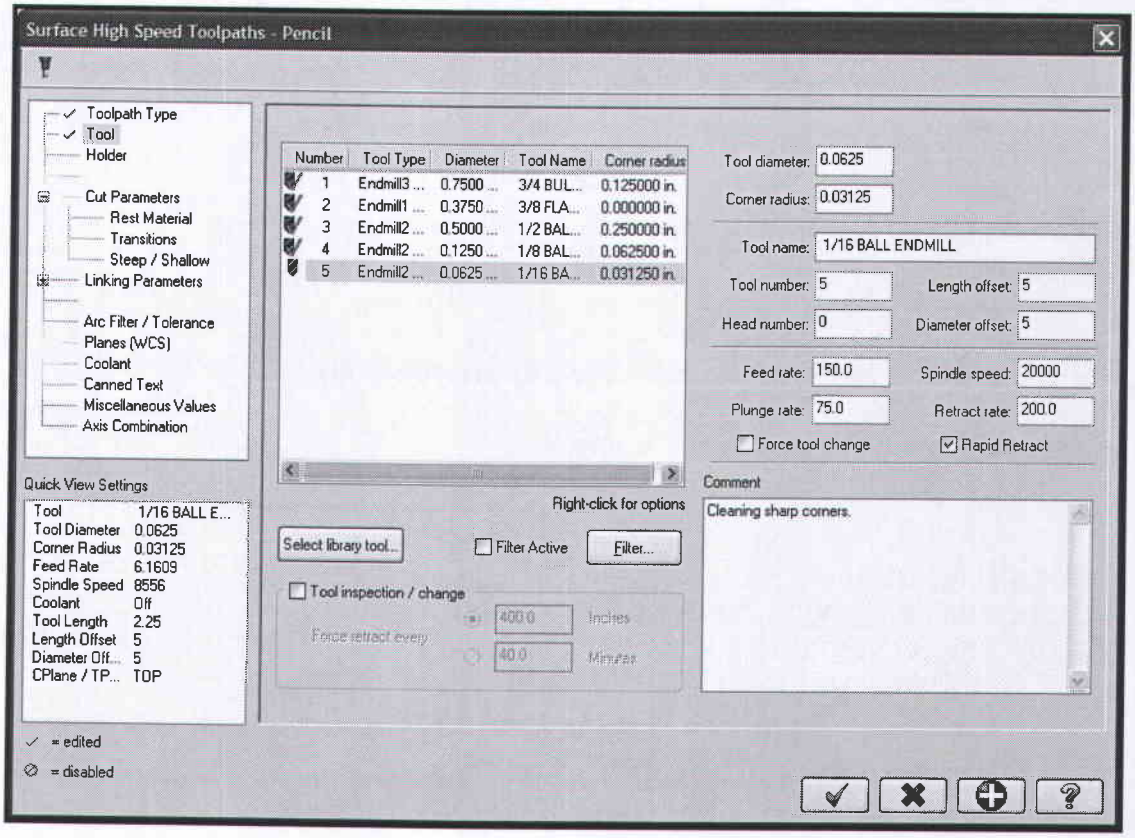




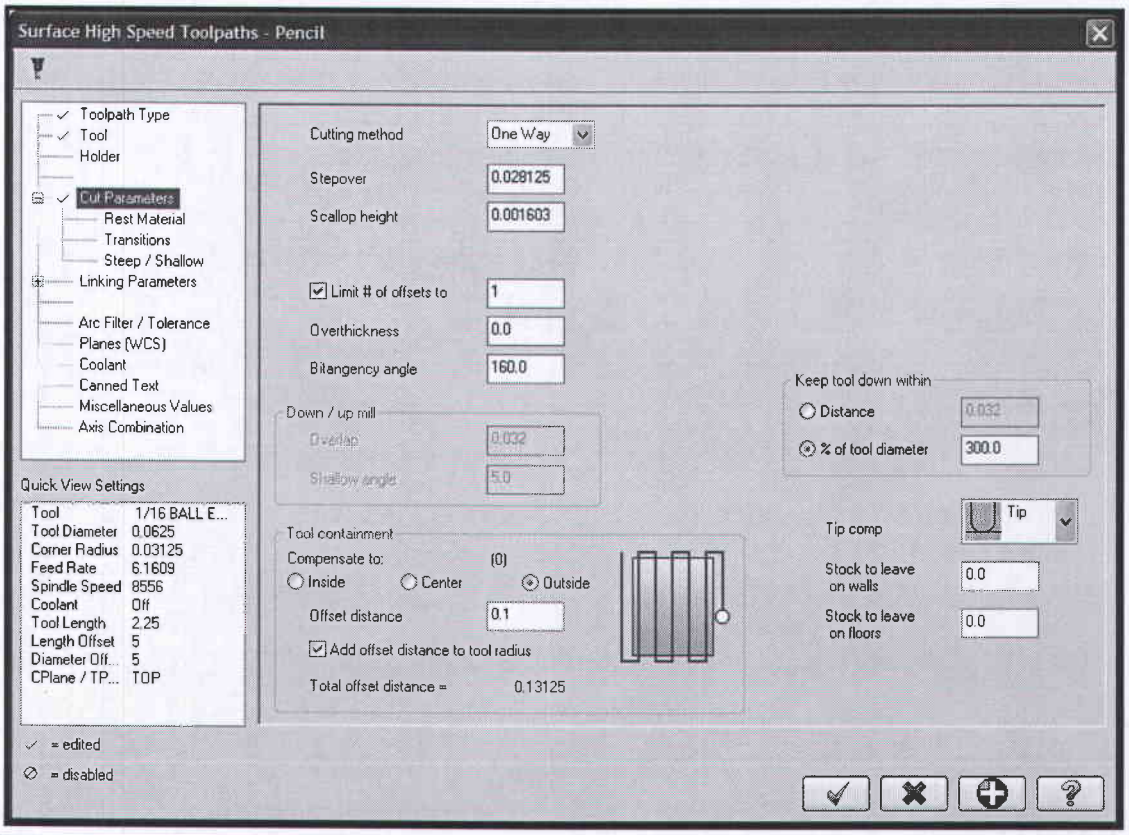
➢ **Right-mouse click** on the **Chain 1** and select **Delete**.
➢ **Right-mouse click** on the **Chain 2** and select **Delete**.
➢ Select the **OK** button to exit **Chain Manager**.

Note that 0 chains are selected in the **Containment** area.

➢ Select **Tool** page.
➢ Click on the **Select library tool** button.
➢ Use the **Filter** option to select the **1/16 Ball Endmill**.
➢ Change the parameters as shown in the following screenshots.

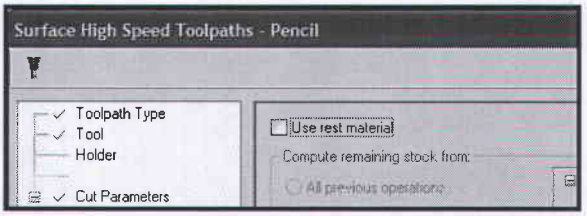


➢ Select the **Cut parameters** page and make the changes as shown.

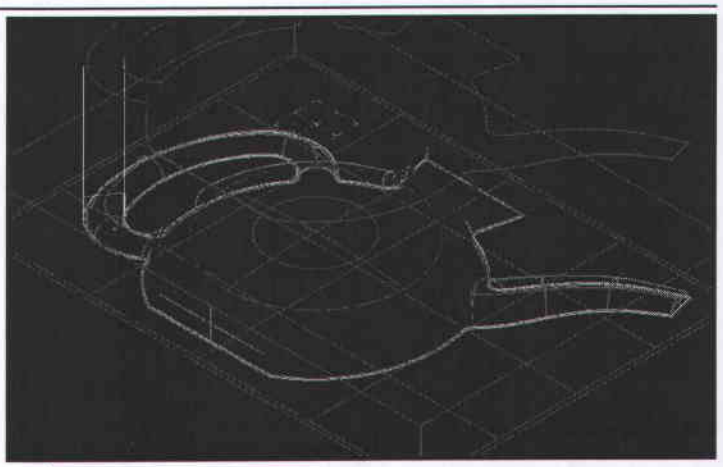


Stepover is the spacing between cutting passes and is measured as a 3D value along the surface.
Scallop height value is linked to the stepover value. When you enter a value in one of the field the other field value is updated.
Limit # of offset to sets the number of machining passes. The total number of passes will be twice the number of offsets, plus one.
Bitangency angle lets you control which intersections are to be machined based on the sharpness of the angle between them.

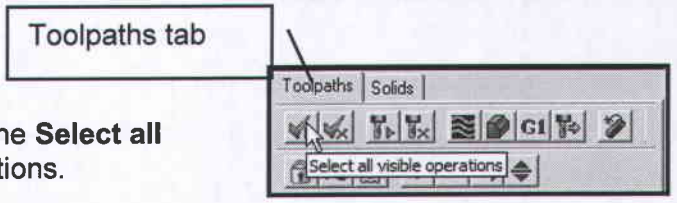
➢ Select the **Rest material** page and disable **Use rest material.**
➢ Select the **OK** button to exit parameter pages.



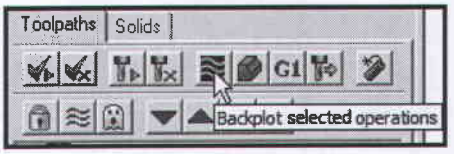
➢ Select the **Regenerate all dirty operation** button to regenerate the toolpath.



STEP 17:
BACKPLOT THE TOOLPATH.

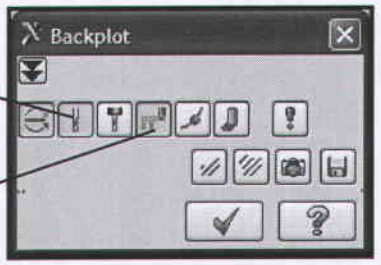


➢ Enable **Toolpaths Manager** and click on the **Select all visible operations** icon to select all operations.

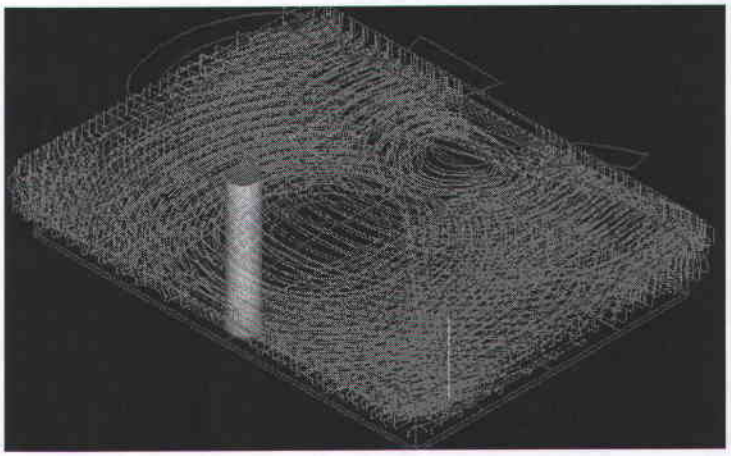


➢ Select the **Backplot selected operations** button.

➢ Make sure that you have the following buttons turned on (they will appear pushed down).
➢ **Display tool**
➢ **Display rapid moves**



➢ Select the **Play** button.



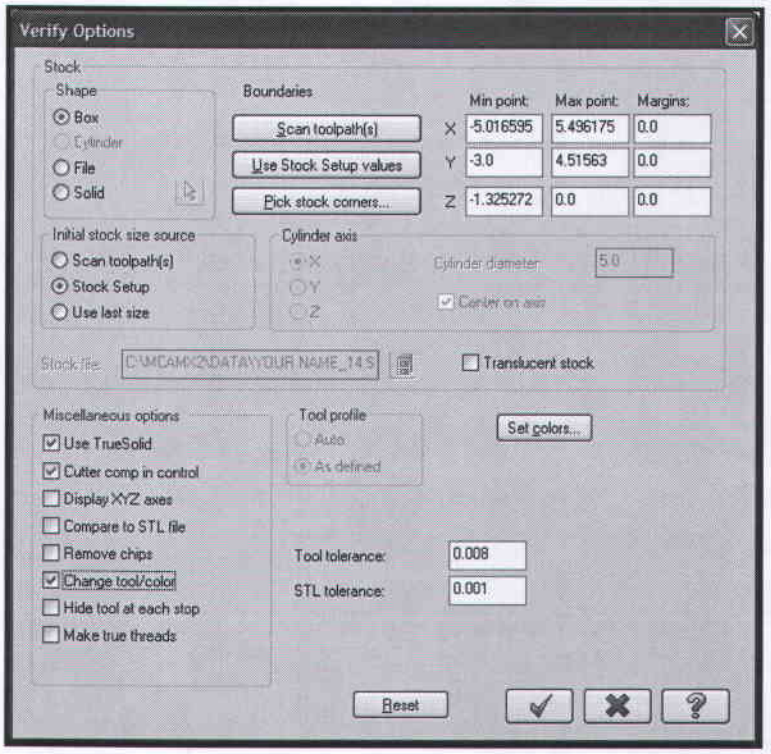
➢ Select the **OK** button to exit **Backplot**.

STEP 18:
VERIFY-TOOLPATH VERIFICATION.

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

Update after each toolpath updates the stock after each operation. **Stop on collision** pauses the verification when the tool touches the part with a rapid move.

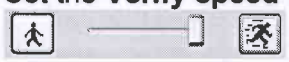
➢ Select the **Configure** button.



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.

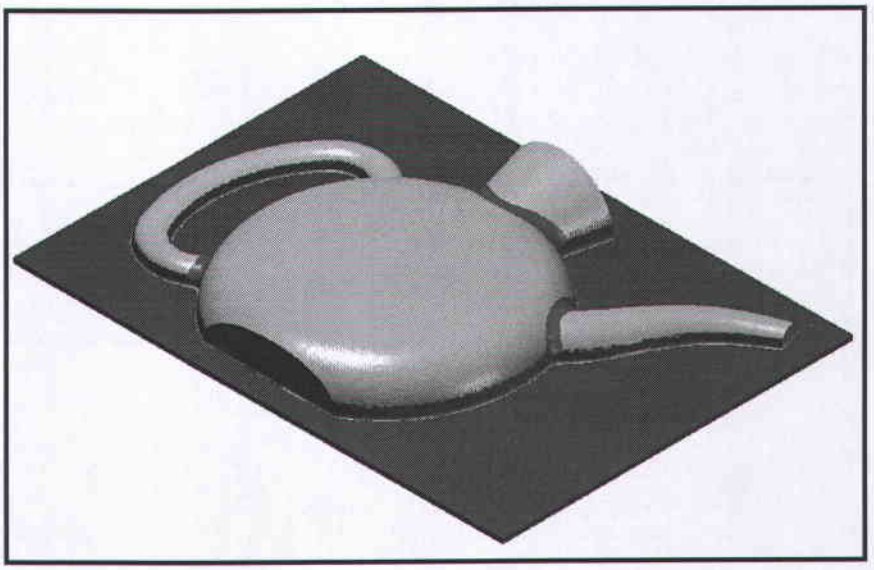
➢ Select the **OK** button to exit **Verify Options**.

➢ Set the **Verify speed** to maximum by moving the slider bar in the speed control bar as shown.



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

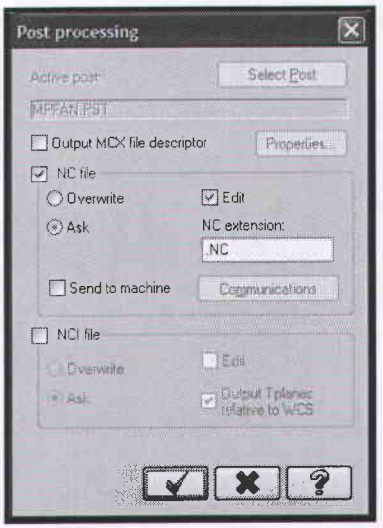
➢ The finished part should appear as shown in the picture to the right.



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

STEP 19:
POST PROCESS THE FILE.

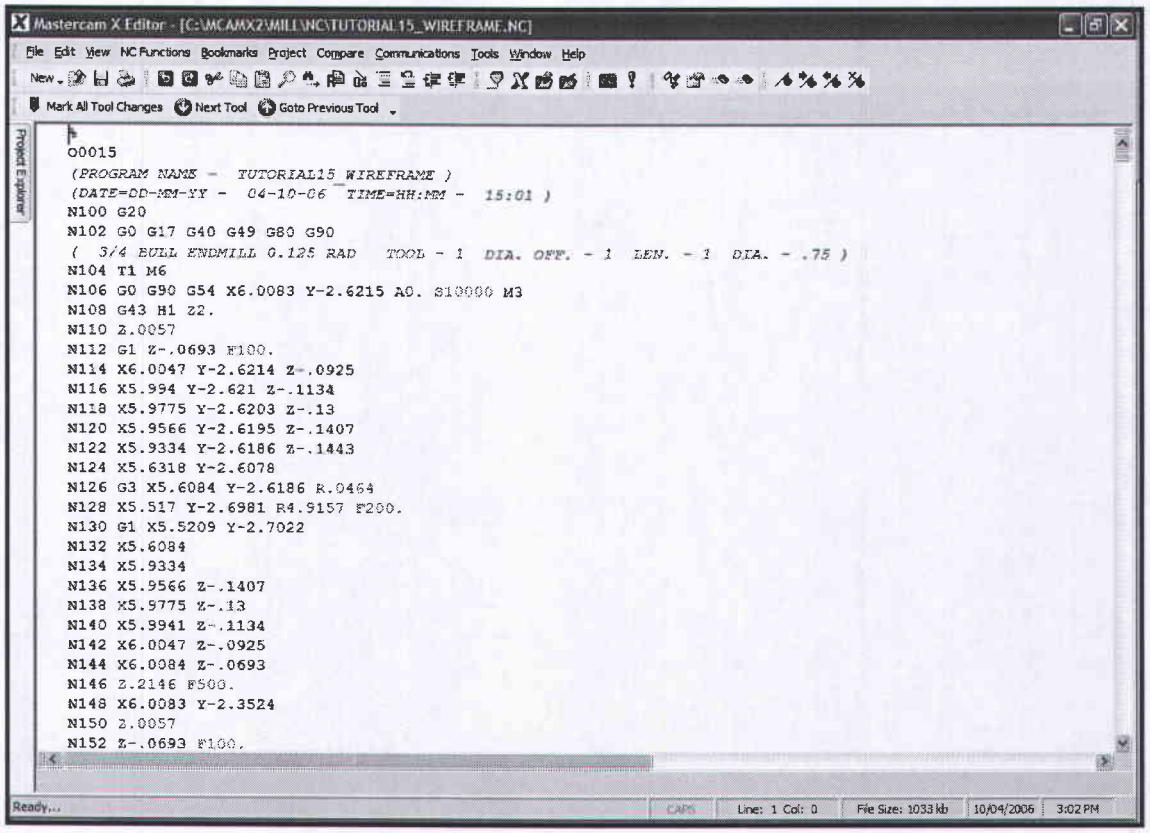
➢ Make sure that all operations are selected, otherwise:
➢ Click on the **Select all visible operations** button.
➢ Select the **Post selected operations** button from **Toolpaths Manager**.



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

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

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



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

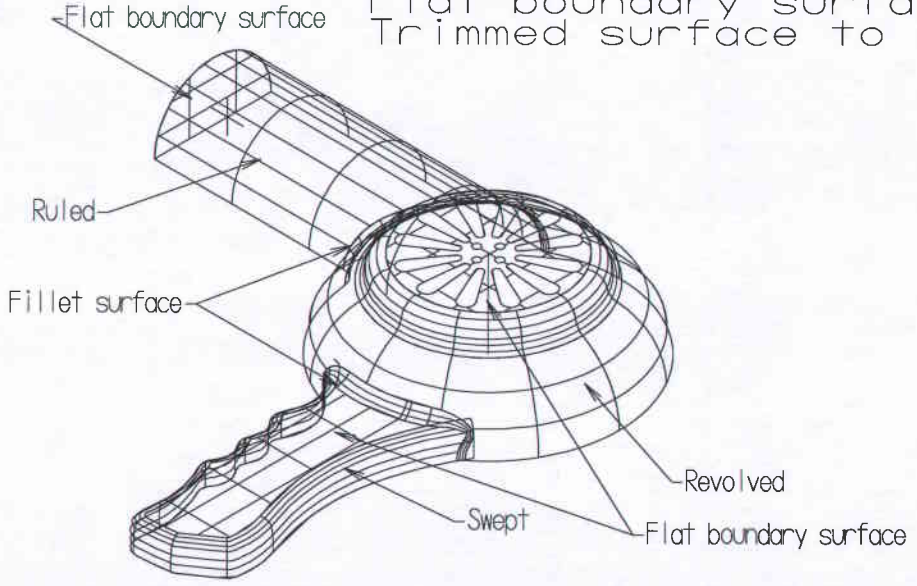
STEP 20:
SAVE THE UPDATED MCX FILE.

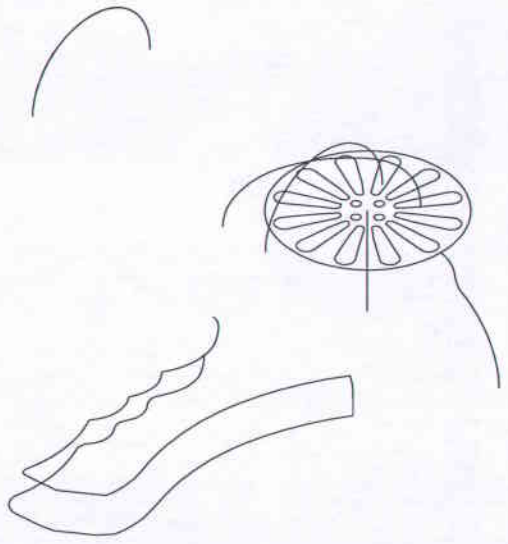
➢ Select the **Save** icon. 

ALL DIMENSIONS IN INCHES

Hairdryer

Swept surface
Flat boundary surface
Revolved surface
Ruled surface
Fillet surface - 1/4 inch
Flat boundary surface
Trimmed surface to curves





Surface modeling

TITLE	TUTORIAL 15-EXERCISE
MATERIAL	ALUMINUM T6061
DATE: MARCH 16, 2005	eMastercam.com

REVIEW EXERCISE

Student practise. Create the Toolpath for **Exercise-Tutorial 15** as per the instructions below;
Download the file from www.emastercam.com/files
Note that the URL address is written with lowercase letters only.

Tips:

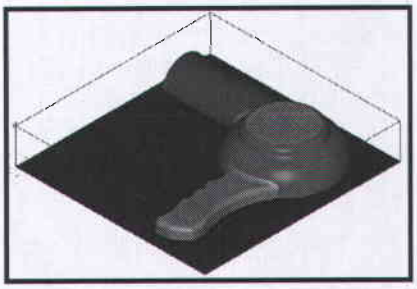
1. Create/ Surface/ Remove Boundary from Trimmed Surfaces
Select the Top surface (green surface)
Slide the arrow to one boundary and accept **Remove all internal boundaries**.

2. Create/ Bounding Box
Select all entities
Enable only Lines Arcs
Expand on X= 0.5; Y= 0.5; Z= 0.1

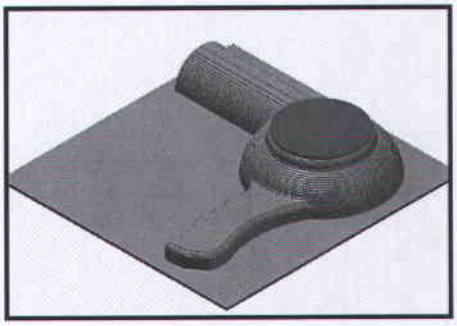
3. Change the z value to -1.59

4.Create/ Rectangle with surface option on

5. Establish the Stock size using bounding box

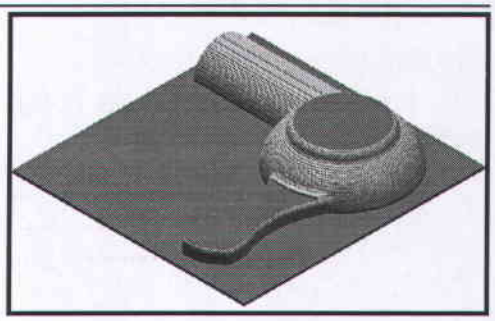


6. High speed Core Roughing
Select All
Use 7/16" Bull End Mill with radius 0.0625
Stepdown =0.05
XY stepover 45 % of diam
Stock to leave on walls and on floors = 0.03
Transitions –Entry helix Radius 0.2
Linking parameters set to Minimum Vertical Retract;
Part clearance =0.25
Keep default Leads values
Minimize Trimming; distance =0.0825



7. High Speed - Finish Horizontal Area

Copy previous operation and change the following parameters;
Use 3/8" Flat End Mill
# of depth cuts =1
Enable Smoothing
Stock to leave on walls = 0.03
Stock to leave on floors = 0
XY stepover = 45% of dia.

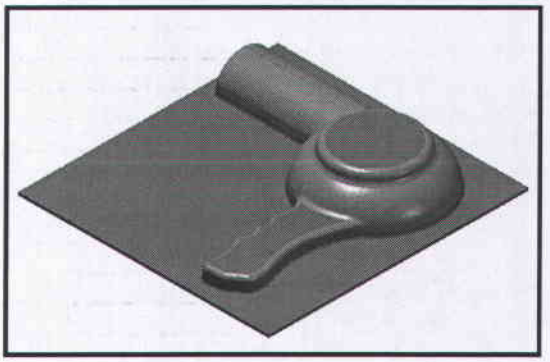


8. High Speed - Finish Waterline

Copy previous operation and change the following parameters;
Use 3/8" Ball End Mill
Stock to leave on walls = 0
Stepdown = 0.0375
Enable Add cuts
Change Max profile stepover to 0.03

9. High Speed Finish Pencil

Copy previous operation and change the following parameters;
Use 1/8" Ball End Mill
Stepover =0.05
Limit # of offsets to 1



10. Backplot and Verify the toolpaths.
11. Post process the file.

INDEX

A

B

C

D

E

F

G

J

K

L

M

N

O

P

R

S

T

V

W

X

Z