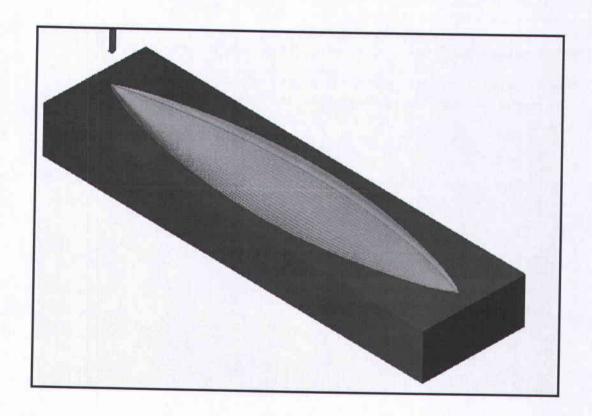
TUTORIAL SERIES FOR

Mastercam.X°

TUTORIAL 10 NET SURFACE, ROUGH POCKET AND FINISH BLEND TOOLPATHS.





Objectives:

The Student will design a 3-dimensional drawing by:

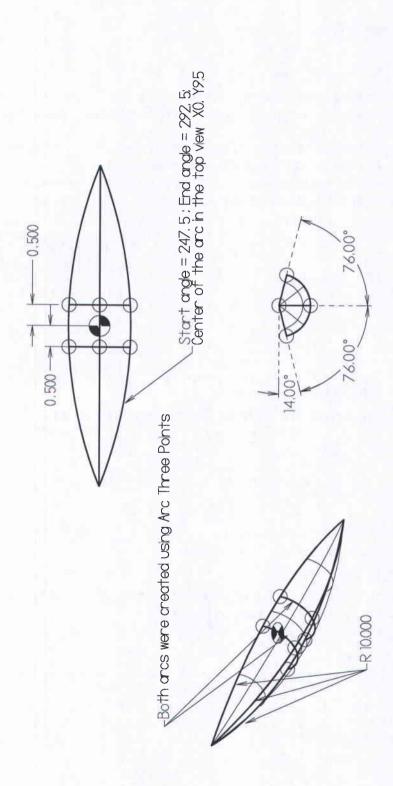
- > Creating an arc knowing center point, angles and the radius in the Top C-plane.
- > Rotating the arc in the Side C-plane.
- > Creating two arcs in the Side C-plane at a given distance, knowing 3 points.
- > Creating a net surface.
- > Changing the view of the part for better visualisation.

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

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

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

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



MIRIA ALUMINUM T6061

DATE: JNE 12, 2005 | eMastercam, com

N -

44

GEOMETRY CREATION

To start a new file from Mastercam:

File

- > New
- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a **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.



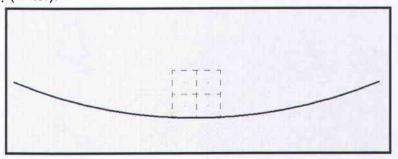
STEP 1:

CREATE THE 10" RADIUS ARCS.

1.1 Create an arc knowing the center point, the initial and final angle, and the radius.

Create

- > Arc
- > Create Arc Polar
- > [Enter the center point]: Select the Fast Point icon.
- Enter into the box to the left of the icon: 0, 9.5 (Enter).
- > Enter the Radius 10.
- To set the other parameters of the arc use the **Tab** key. Note that the diameter value is automatically changed by the system based on the radius.
- > Enter the Start Angle 270-45/2 (Tab).
- > Enter the End Angle 270+45/2 (Enter).
- ➤ Select the **OK** button.
- Select the Fit icon to fit the geometry to the screen.
- > The arc should look as shown to the right.





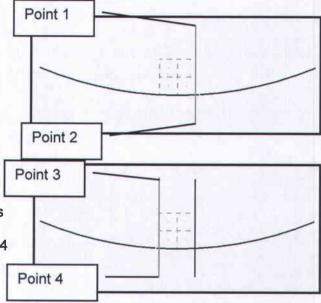
1.2 Creating two vertical construction lines in the Top Construction plane.

Create

- ▶ Line
- > Create Line Endpoints
- > Select the Vertical icon. Vertical
- ➤ [Specify the first endpoint]: Pick Point 1 as shown (sketch the points).

1,00 ---

- > [Specify the second endpoint]: Pick Point 2
- > [Enter the X coordinate]: I 0.5 E
- ➤ Select the **Apply** button.
- ➤ [Specify the first endpoint]: Select Point 3 as shown
- > [Specify the second endpoint]: Select Point 4
- > [Enter the X coordinate]: 10.5
- ➤ Select the **OK** button.

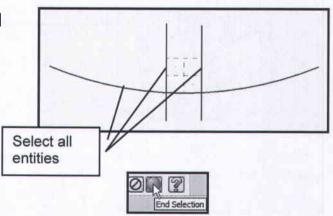


1.3 Break the arc at the intersections with the two vertical construction lines.

Edit

- > Trim/Break
- > Break at Intersection
- ➤ [Select entities to break]: Select the arc and the two lines.

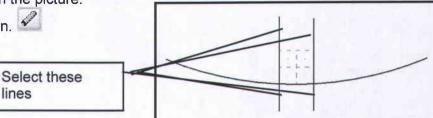
> Select the End Selection button.



1.4 Delete the 4 construction lines.

> Select the lines as shown in the picture.

> Select the **Delete** entity icon.



1.5 Change the G-view to Isometric and the construction plane to right side.

> Select the Isometric View from the view toolbar to see the arc.



Click on the drop-down arrow next to the Set planes icon, and select Right.



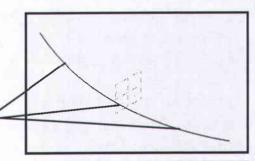
1.6 Rotate-Move the arc.

Xform

- > Xform Rotate
- > [Rotate: select entities to rotate]: Select all three arcs.
- > Click on the End Selection

button.

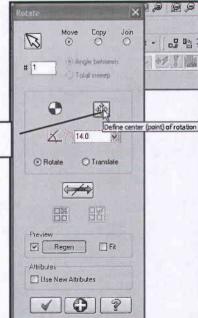
Select all arcs



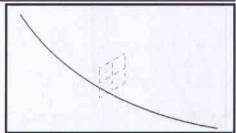
Make sure that you make all the changes as shown in the screenshot.

Select Define center.

- > Enable the **Move** radio button.
- > Change the Angle to 14 degrees.
- > Select Define center (point) of rotation as shown.
- > [Select the point to rotate about]: Select the **Fast Point** icon.
- > Enter into the box to the left of the icon: 0.2612,0 (Enter).



- > If the **Preview** is active you should be able to see the result.
- > Select the **OK** button to exit.

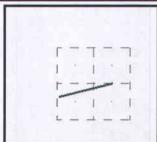


Rotate

> Select the Side View from the view toolbar to see the arc.



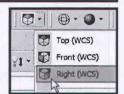
> The part will look as shown to the right.



> Select the **Isometric View** from the view toolbar to see the arc.



➤ Click on the drop-down arrow next to the **Set planes** icon, and select the **Right** plane.



1.7 Rotate-Copy the existing arc to create the other arcs with the same radius.

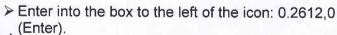
Xform

- > Xform Rotate
- ➤ [Select entities to rotate]: Select all three arcs.
- > Click on the End Selection button.

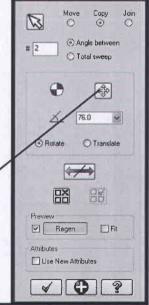


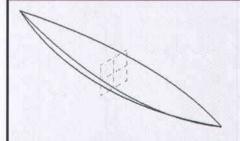
Select Define center

- Make sure that you make all the changes as shown in the screenshot.
- > Enable the Copy radio button.
- > Change the # to 2.
- > Change the Angle to 76 degrees.
- > Select Define center (point) of rotation as shown.
- > [Select the point to rotate about]: Select the Fast Point icon.



- > If the **Preview** is active you should be able to see the result
- > Select the **OK** button to exit.







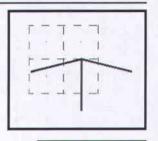
TUTORIAL 10

> See the part from the **Side View** as we did before.

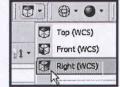


> Change the graphic view back to Isometric.





> Click on the drop-down arrow next to the Set planes icon, and select the Right plane.



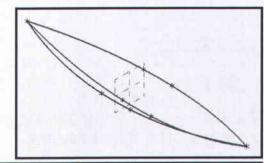
STEP 2

CREATE THE TWO ARCS KNOWING THREE POINTS.

2.1 Create points at the endpoints of the entities.

Create

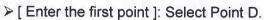
- **➢ Point**
- ➤ Create Point Endpoints
- Note that the points are automatically created.



2.2 Create arcs knowing 3 points.

Create

- > Arc
- > Create Arc 3 Points
- > [Enter the first point]: Select Point A.
- > [Enter the second point]: Select Point B.
- [Enter the third point]: Select Point C.



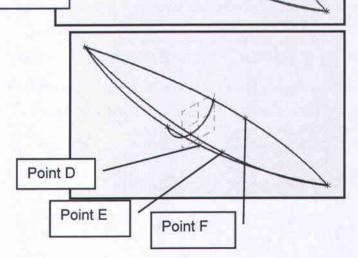
- > [Enter the second point]: Select Point E.
- > [Enter the third point]: Select Point F.





Screen

> Clear colors



Point C

Point A

Point B



STEP 3: CREATE A NET SURFACE.

Net Surface: a surface generated from a network of intersecting curves. Applications: used to create a surface from a grid of curves.

3.1 Join the arcs with 10" radius.

Edit

- > Join entities
- > [Select entities to join]: Select the All button.



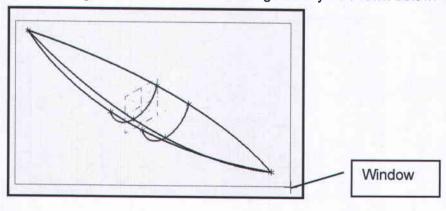


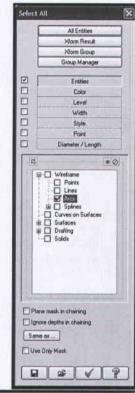
- > Select the **OK** button to exit **Select All**.
- Click on the End Selection button.
- Note that arcs will be automatically joined.

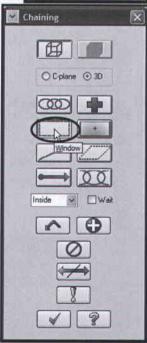
3.2 Create the net surface.

Create

- > Surface
- > Create Net Surface
- > Click on the Chaining dialog box title bar to move it.
- > Select the Apex Point from the Ribbon bar.
- > Select the Window button in the Chaining dialog box.
- ➤ Make a big window around the entire geometry as shown below.





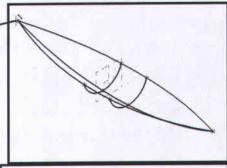


> [Enter search point]: Select Point A

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

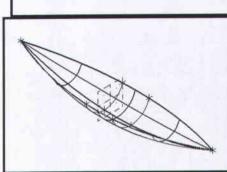


> [Indicate the apex position]: Select the Point A.



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

> The part should look as shown in the screenshot to the right.



> Change the construction plane back to Top.

> To shade the part you can hold down the Alt key and type S.



> Alt + S again will unshade the part.

STEP 4:

TRANSLATE THE GEOMETRY.

Xform

> Xform Translate

> [Translate: select entities to translate]: Select the All button.

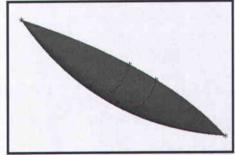


> Select the **OK** button.



> Click on the End Selection button.









- Make sure that you make all the changes as shown in the screenshot.
- > Enable the Move radio button.
- ➤ Enter **Delta Y** 0.2612.
- > Select the **OK** button.

STEP 5: CREATE THE BOX THAT WILL REPRESENT THE STOCK.

Create

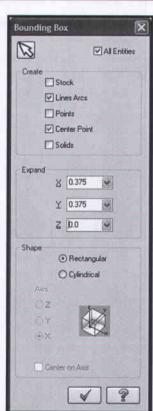
- > Create Bounding Box
- ➤ Enable Create, Lines Arcs and Center Point as shown in the screenshot to the right.
- > Expand the stock with 0.375 along X and Y axis.
- > Select the **OK** button to exit.

STEP 6: SAVE THE FILE.

File

- > Save as
- > File name: "Your Name_10"
- > Select the **OK** button.



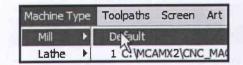


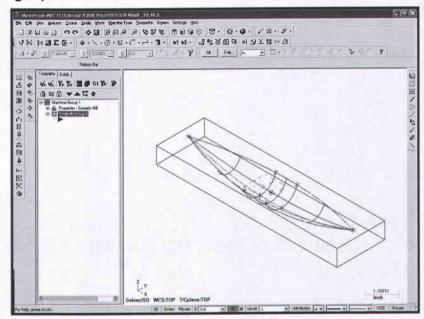


TOOLPATH CREATION STEP 7: SET UP THE STOCK TO BE MACHINED.

Machine type

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





> Use the Fit icon to fit the drawing to the screen.

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

Select the plus

> Select Stock setup.

Select the Stock

Select the S

Toolpaths Solids

Toolpaths | Solids |

VA VA TA TA TO OF TO S

Machine Group 1

Hackine Group 1

Froperties - Generic Mill

Strong 1

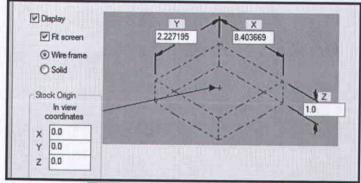


- > Change the parameters to match the screenshot to the right.
- > Select the **Bounding box** button to automatically find the part extents.
- Change allithe expand values to 0
- > Select the **OK** button to exit the **Bounding box** dialog box.



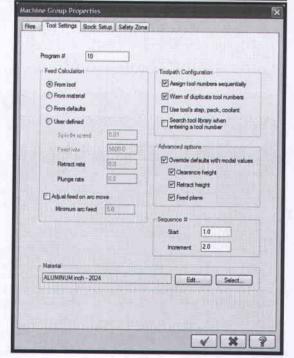


➤ Change the **Z** value of the stock to 1.0 as shown.



- > 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.



Mastercam. **TUTORIAL 10**

STEP 7: ROUGH OUT THE SURFACE USING SURFACE ROUGH POCKET.

Select the

Toolpaths

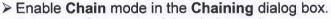
- > Surface Rough
- > Rough Pocket Toolpath
- > Select the **OK** button to accept the NC name.
- > [Select Drive Surfaces]: Select the surface.



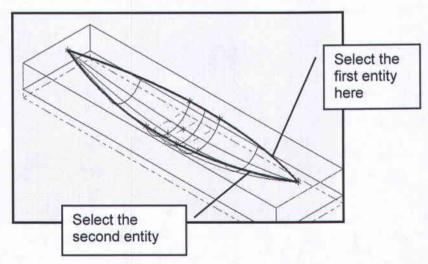
surface

> Click on the End Selection button.

> Select the Containment button as shown.



> Select the first entity of the chain as shown.

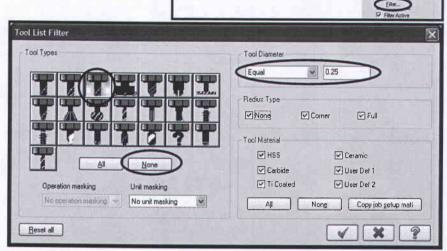


- > [Select next branch]: Select the second entity.
- > Select the End chain button as shown in the dialog box.
- > Select the **OK** button to exit **Chaining**.
- > Select the OK button to exit Toolpath/surface selection.



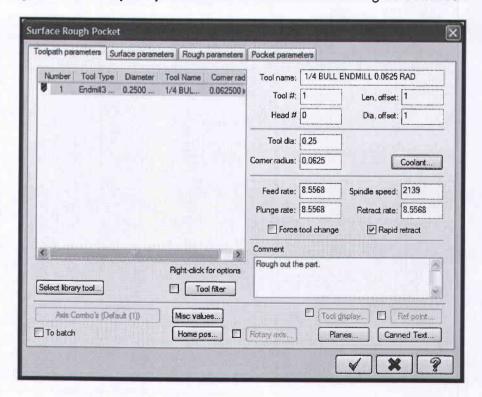


- > Click on Select library tool in the Toolpaths parameters dialog box.
- Select the Filter button in the Tool Selection dialog box.
- Select the None button in the Tool Types area.
- > Select the **Bullnose** tool type button.
- Select the drop-down arrow in the Tool Diameter field, and select Equal.
- > Enter the **Tool Diameter** 0.25.
- Select the **OK** button to exit.



B Radius Type
Comer
Comer
Comer

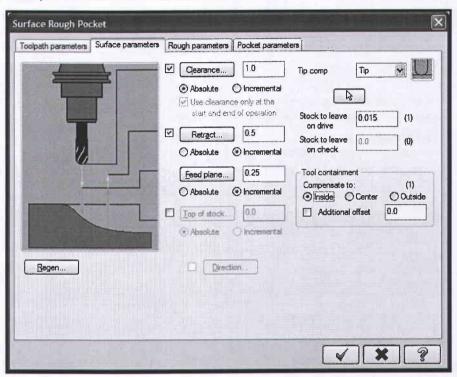
- > Select 0.25" Bull Nose with 0.0625" corner radius in the Tool Selection page.
- > Select the **OK** button to exit.
- > Make the necessary changes in the Toolpath parameters to match the following screenshot.



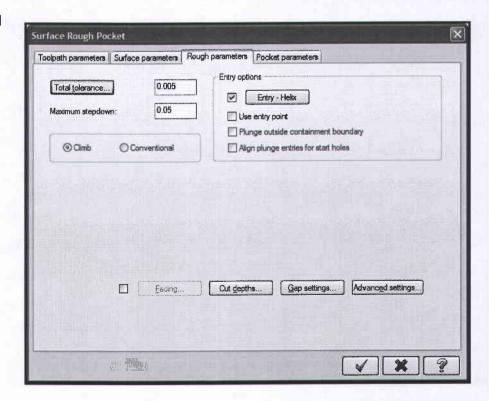
The Feed rate,
Plunge rate, Retract
rate and Spindle
speed are based on
the tool definition.
Change them as
desired.



> Select the **Surface parameters** tab and make the changes as shown. Make sure that you enable **Compensate to Inside**, for the **Tool containment**.

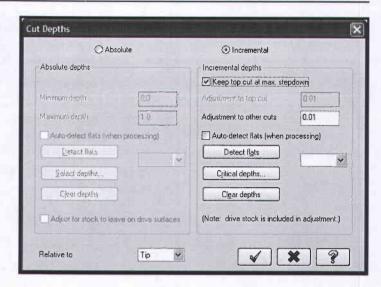


Select the Rough parameters page and make the changes to match the following screenshot.





➤ Select the Cut depths button and enable Keep top cut at max stepdown to force the system to cut 0.05 deep at the first pass.



> Select the OK button to exit Cut

Depths.

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

江

Entry-helix sets the parameters for a helical entry into the part.

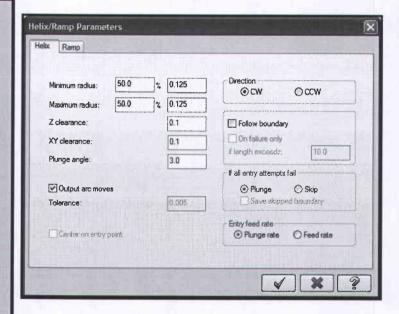
Minimum/Maximum radius value sets the minimum/maximum radius of the helix as a percentage of the tool diameter or as an absolute value.

Z clearance value sets the distance in the Z-axis above the top of the stock where the helix entry begins.

XY clearance sets the minimum distance in the XY axis between the helix and the pocket walls.

Plunge angle sets the helix angle.

Output arc moves instructs the system to output circular movements instead of linear movements.

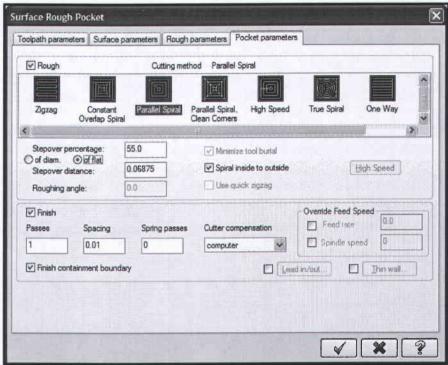


> Select the **OK** button to exit **Helix Parameters**.

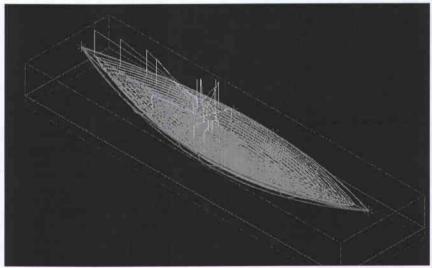




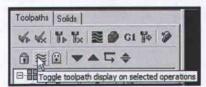
> Select the Pocket parameters and change the Cutting method to Parallel Spiral.



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



- > Select the **Toolpaths** tab to enable the **Toolpaths Manager**.
- > Select Toggle toolpath display on selected operation to remove the toolpath display from the screen.





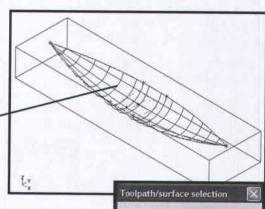
STEP 8:

CLEANING THE AREA WHERE THE PREVIOUS TOOL DID NOT FIT USING SURFACE ROUGH RESTMILL.

Toolpaths

- ➤ Surface Rough
- > Rough Restmill Toolpath
- > [Select Drive Surface]: Select the surface.

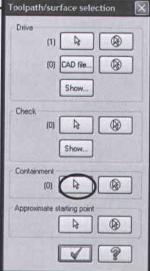
Select the surface



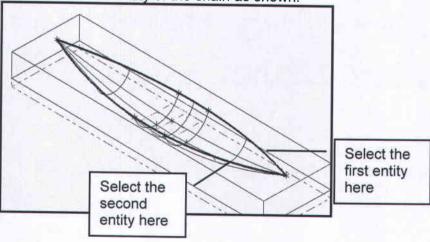
> Click on the End Selection button.



> Select the Containment button as shown.



Select the first entity of the chain as shown.

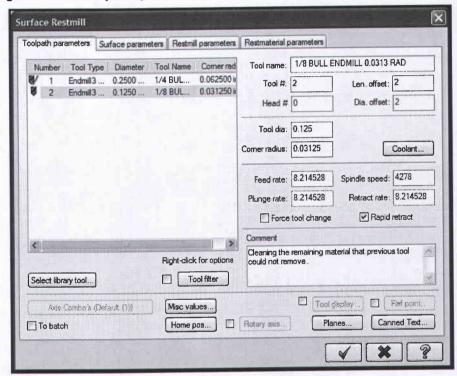


- > [Select next branch]: Select the second entity.
- > Select the End chain button as shown in the dialog box.



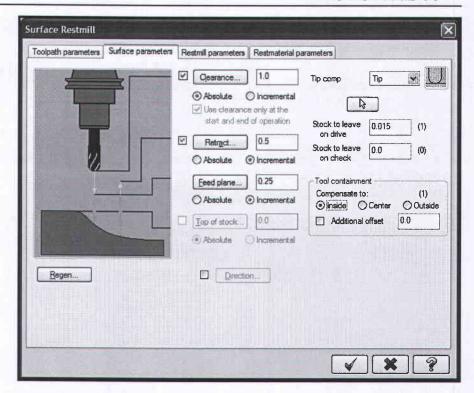


- > Select the **OK** button to exit **Chaining**.
- > Select the OK button to exit Toolpath/surface selection.
- > Click on Select library tool in the Toolpaths parameters dialog box.
- > Select the **Filter** button in the **Tool Selection** dialog box, and following the steps at page 10-14 select the 0.125 Bull nose.
- > Select the **OK** button to exit.
- > Select 0.125" Bull Nose with .03125" corner radius in the Tool Selection page.
- > Select the **OK** button to exit.
- > Make the necessary changes in the Toolpath parameters to match the following screenshot.

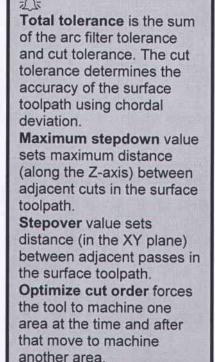


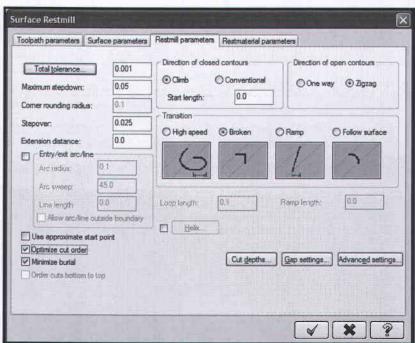


Select the Surface parameters tab and make the changes as shown.



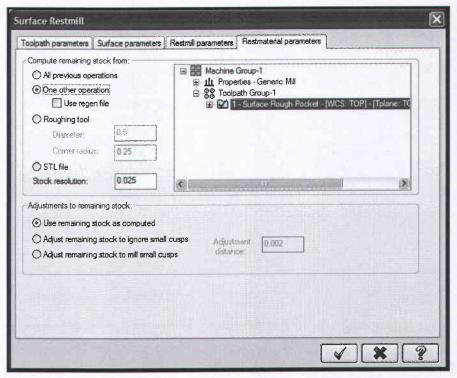
> Select the Restmill parameters page and made the changes as shown.







> Select the **Restmaterial parameters** page to set the options to define how Mastercam calculates remaining stock.



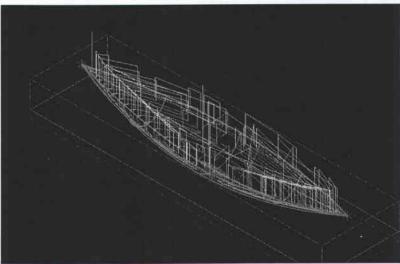
迩

Compute remaining stock from One other operation calculates the remaining stock based on the selected previous operation.

Stock resolution sets the amount the stock model is sampled, entered as a distance. A smaller value tightens up the stock model and creates a more accurate restmill toolpath.

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







STEP 9: FINISH THE SURFACE USING SURFACE FINISH BLEND.

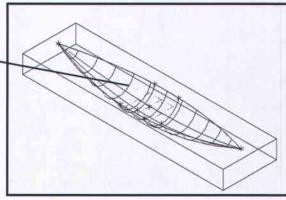
- > Select the Toolpaths tab to enable the Toolpaths Manager.
- > Select Toggle toolpath display on selected operation to remove the toolpath display from the screen.

Toolpaths

- ➤ Surface Finish
- > Finish Blend Toolpath
- ➤ [Select Drive Surface]: Select the surface.

Select the surface





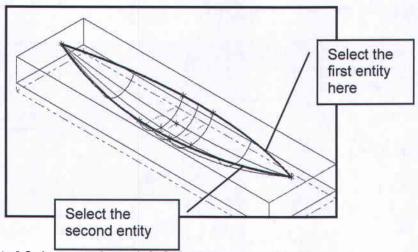
> Select the End Selection button.



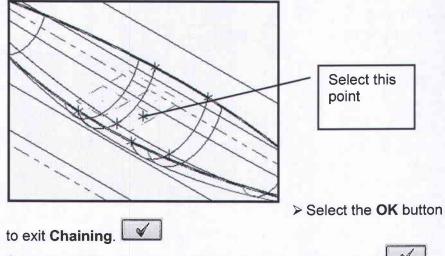
End Selection



> Select the first entity of the chain as shown.



- > [Select next branch]: Select the second entity.
- > Select the End chain button as shown in the dialog box to the right.
- > Change the selection mode to **Point** in the **Chaining** dialog box.
- > Select the Center point as shown in the picture below.



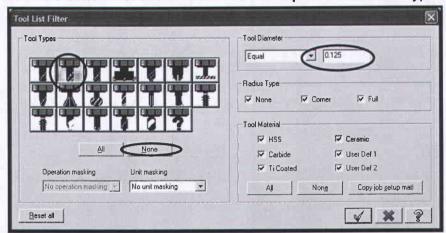
> Select the OK button to exit Toolpath/surface selection.



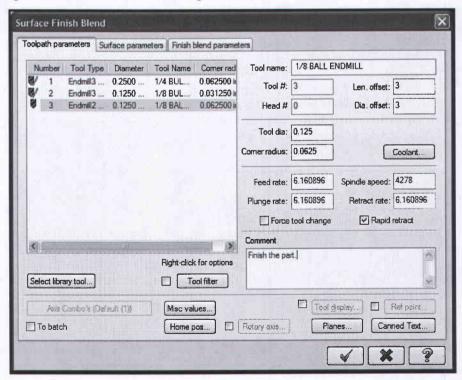




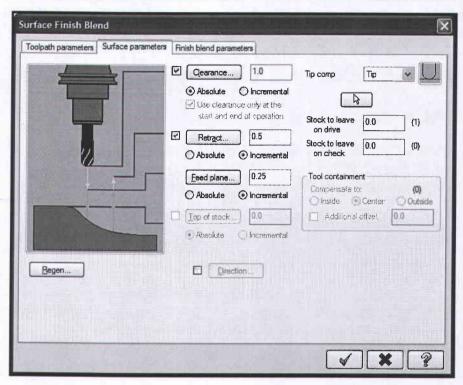
- ➤ Click on Select library tool in the Toolpaths parameters dialog box.
- ➤ Using the Filter option select the 0.125 Ball Endmill.
- > Make sure that you select the None button first and then the Endmill Sphere as the tool type.



- > Select the drop-down arrow in the Tool Diameter field, and select Equal.
- > Enter the Tool Diameter 0.125.
- > Select the **OK** button to exit.
- Make sure that the tool is selected in the Tool Selection window and select the OK button to exit.
- > Make any necessary changes as shown in the following screenshots.



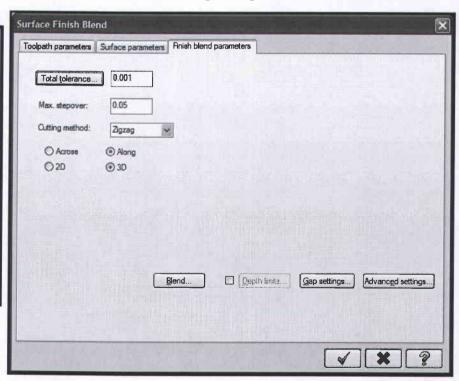




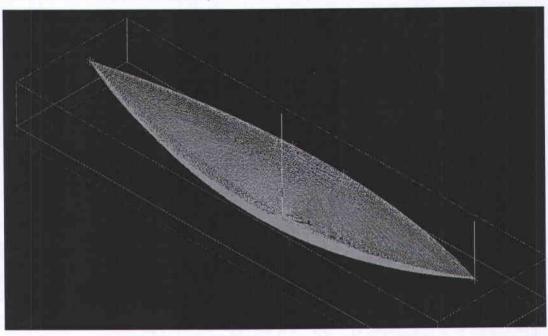
> Select Finish blend parameters and make the following changes.

Blend finish toolpath drives the toolpath along the surfaces using a pattern created from curves that mimic the part features.

Blend along settings dialog box provides control over the temporary mesh created by the along and across tool moves of the Project blend toolpath.



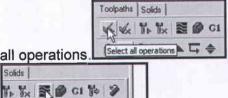
> Select the **OK** button to exit **Finish blend parameters**.



STEP 10: BACKPLOT THE TOOLPATH.

> Click on the Toolpaths tab to enable Toolpaths Manager.

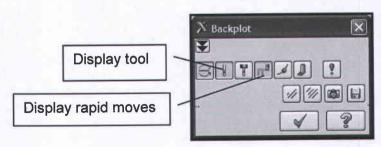
> Choose the Select all operations icon to select all operations.



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

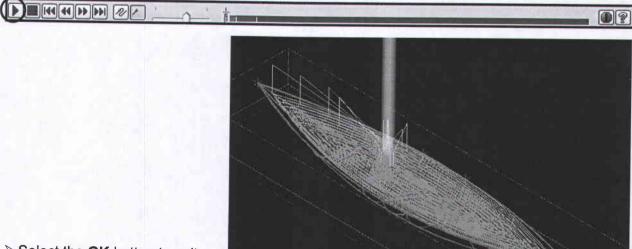
Solids

Display tool Display rapid moves





Select the Play button.



Select the OK button to exit

Backplot.

STEP 11:

VERIFY-TOOLPATH VERIFICATION

> Select the Verify selected operations button.



> Make sure that Turbo button and Stop on collision are enabled.



沇



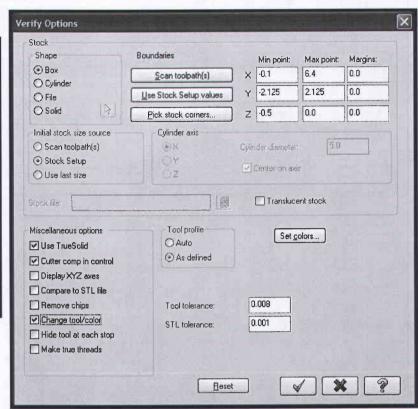
> Select the **Configure** button.



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.

Initial stock size source

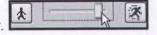
Change tool/color to change the color of the cut stock to indicated tool changes in the toolpath.



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



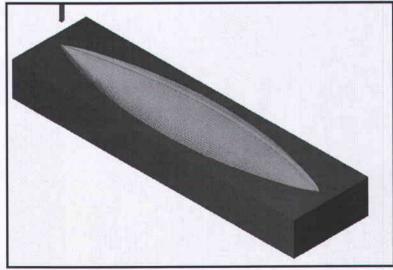
> Set the Verify speed by moving the slider bar in the speed control bar.



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

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

> Select the OK button to exit Verify.





STEP 12: POST PROCESS THE FILE.

> Select the Post selected operations button from Toolpath Manager.



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



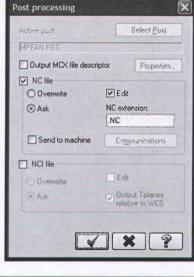
NC file enabled allows you to keep the NC file and to assign the same name as the MCX file.

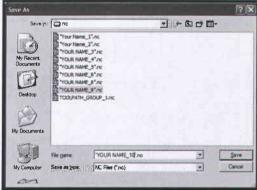
Edit enabled allows you to automatically launch the default editor.

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

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

> Select the Save button.







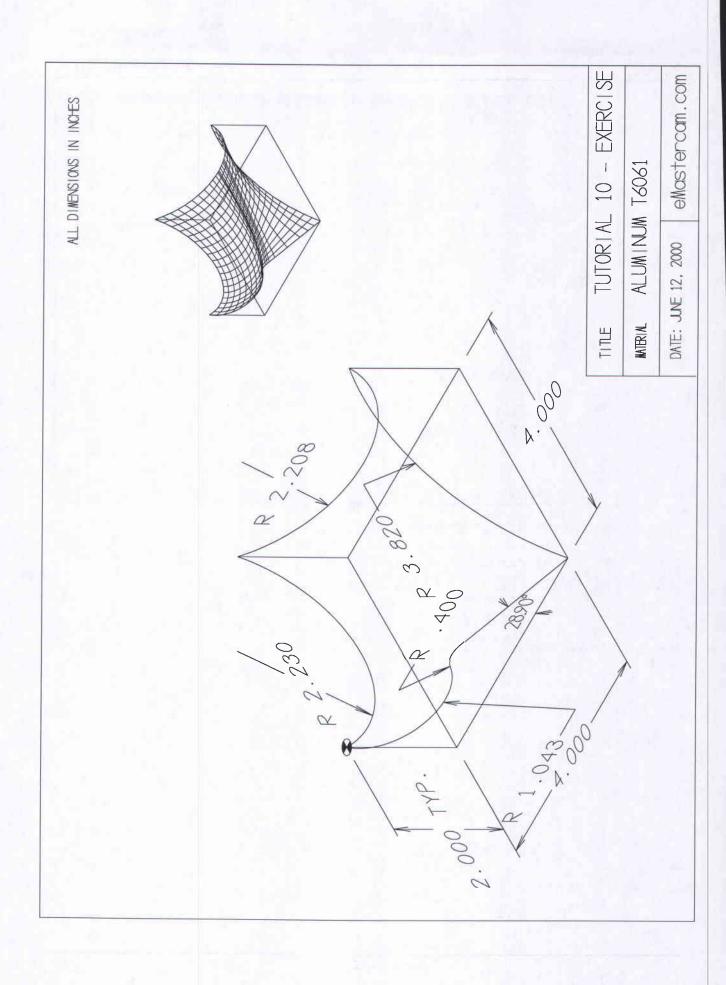
```
### Note For the Contraction (Contraction Contraction Project Contraction Cont
```

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

STEP 13: SAVE THE UPDATED MCX FILE.

> Select the Save icon.





Mastercam.

REVIEW EXERCISES.

Student practise. Create the Toolpath for Exercise-Tutorial 10 as per the instructions below;



1. Establish the **stock** using Bounding box and change the stock size to X =3.75°,

Y =3.75",and z = 2" (Stock Origin X= 2.0, Y= 2.0, Z=0.05)

2. Surface Rough Pocket

Select the net surface

Select the buttom rectangle as Tool Containment boundary

Use 3/4" Bull End Mill rad 1/4

Stock to leave on drive surfaces= 0.03"

Total tolerance = 0.01

Max stepdown =0.1

Disable Entry

Enable Plunge outside tool containment

Cutting method quick zigzag

2. Surface Finish Blend

Select the net surface

Select (in the same direction) the two arcs as the

blend curves as shown

Use 3/8" Ball End Mill

Stock to leave on drive surfaces= 0

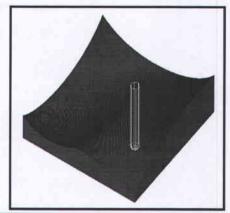
Total tolerance= .001

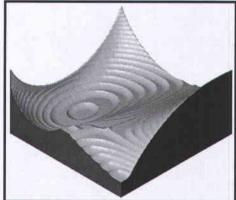
Maximum stepover= 0.05

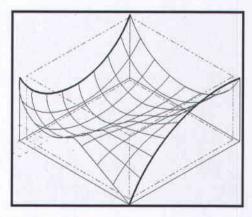
Cutting method zigzag and along

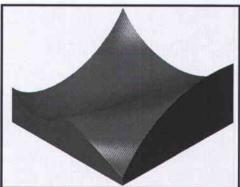
3. Backplot and Verify the toolpaths.

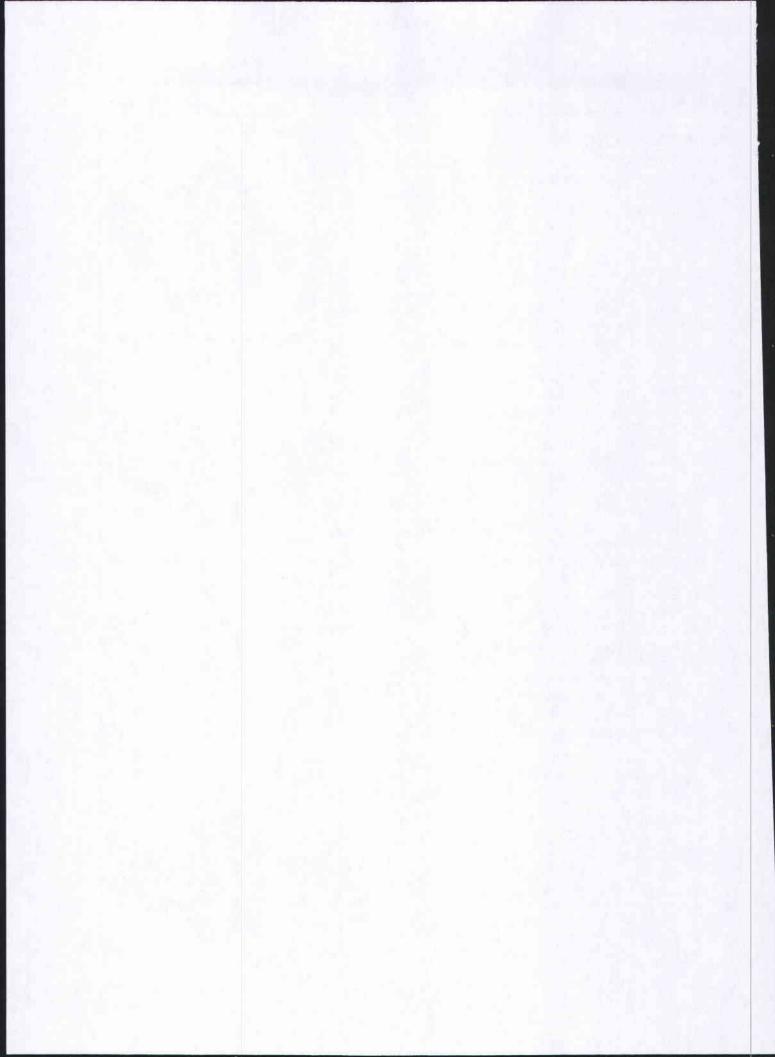
4. Post process the file.







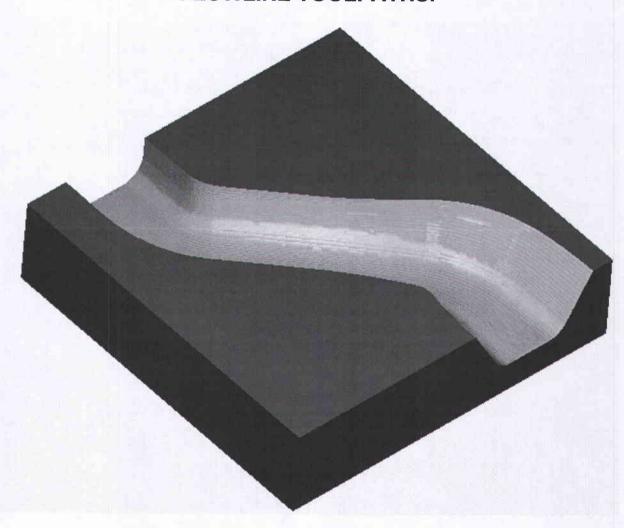




TUTORIAL SERIES FOR



TUTORIAL 11 SWEEP SURFACE, ROUGH FLOWLINE AND FINISH FLOWLINE TOOLPATHS.





Objectives:

The Student will design a 3-dimensional drawing by:

- > Creating rectangles using co-ordinate positioning.
- > Creating parallel lines
- > Creating lines knowing the endpoints.
- > Creating fillet radii.
- > Changing the construction plane to Side and creating half of a rectangular shape and half of a hexagon.
- > Defining a plane from entities.
- > Creating an arc polar knowing the center location.
- > Creating a swept surface.
- > Changing the view of the part for better visualisation.

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

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

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

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



GEOMETRY CREATION

To start a new file from Mastercam:

File

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

STEP 1: CREATE THE BOX WIREFRAME. 1.1 Create a rectangle. Create > Create Rectangle > Enter the Width 4.0 (Tab). > Enter the Height 4.0 (Enter). Anchor to center > Enable Anchor to center. > [Select position of first corner]: Select the **Origin** (center of the grid) as shown. Select the Origin > Select the **OK** button.

♠ During the geometry creation of this tutorial, if you make a mistake, to undo the last step you

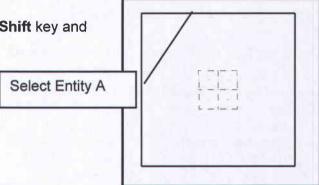
● If you delete or undo a step by mistake, just use the Redo icon. State of the Redo icon.



1.2 Create the 3d box using translate command.

Xform

- > Xform Translate
- > [Select entities to translate]: Hold down the Shift key and select Entity A as shown.
- Note that all four lines of the rectangle are selected.



> Select the End Selection button.



- > Change the parameters in the Translate dialog box as shown to the right.
- ➤ Make sure that the **Z** value is -1.0 and **Join** is enabled.

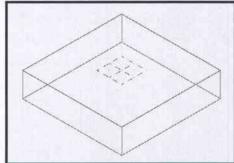
- > Select the **OK** button to exit.
- **6 6 6 6** > Change the graphic view to Isometric.
- ➤ Use the **Fit** icon to fit the drawing to the screen.



Screen

> Clear colors





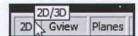


STEP 2: CREATE THE WIREFRAME IN THE TOP CONSTRUCTION PLANE.

> Change the graphic view to back to Top.



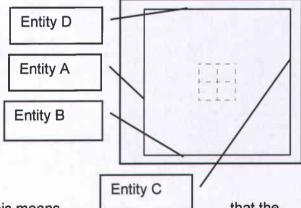
> Select 3D from the status bar. The system will automatically switch to 2D.



2.1 Create parallel lines.

Create

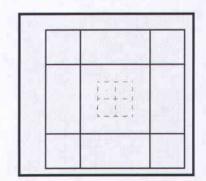
- **>** Line
- ➤ Create Line Parallel
- > [Select a line]: Select Entity A.
- ➤ [Select the point to place a parallel line through]: Pick to the right of the selected line.
- > Type the **Distance** 1.0 (Enter).



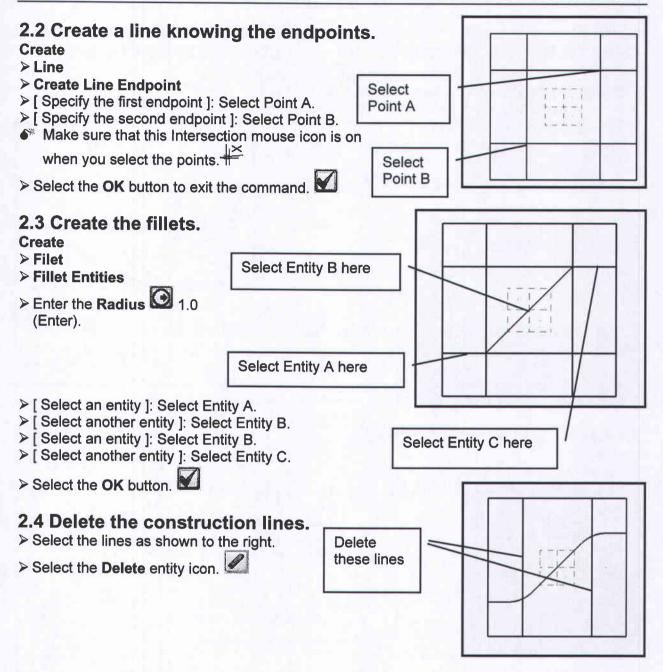
Note that the color of the geometry is cyan. This means entity is "alive" and you can still change the line parameters. To exit the command you can either start a new command or select the **OK** button.

> To create more parallel lines with the same distance click on the distance icon. The distance value will be highlighted in red.

- > [Select a line]: Select Entity B.
- > [Select the point to place a parallel line through]: Pick a point above the selected line.
- > [Select a line]: Select Entity C.
- > [Select the point to place a parallel line through]: Pick a point to the left of the selected line.
- > [Select a line]: Select Entity D.
- > [Select the point to place a parallel line through]: Pick a point below the selected line.
- > Select the **OK** button to exit.
- > The part should look as shown to the right.







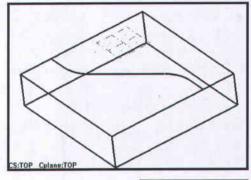


STEP 3: CREATE THE WIREFRAME IN THE SIDE CONSTRUCTION PLANE.

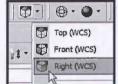
888 > Change the graphic view to **Isometric**.



> The part should look as shown to the right.



> Click on the drop-down arrow next to the Set planes icon, and select the Right plane.



> Select 2D from the Status bar. The system will automatically switch to 3D.

3.1 Create the rectangular shape. Create

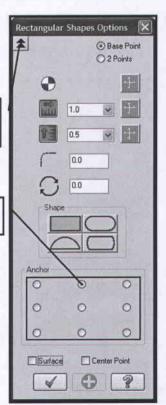
> Create Rectangular Shapes

> Type the Width and the Height as shown in the screenshot to the right.

Select the double arrow to expand

> Select the radio button as shown.

Select this radio button

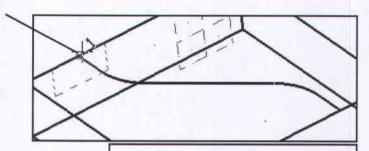




> [Select position of base point]: Select the endpoint of the line as shown.

Select this Endpoint

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



3.2 Delete the extra line.

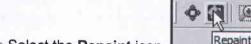
> Select the line as shown to the right.

Delete this line

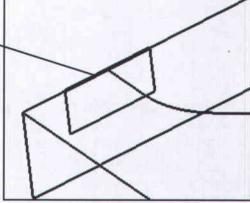
> Select the **Delete** entity icon.



Make sure that you only delete the line from the rectangle.







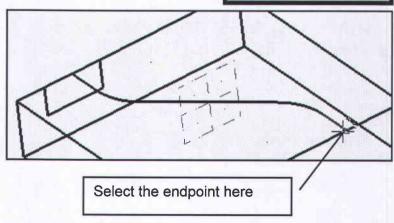
3.3 Create the hexagon.

Create

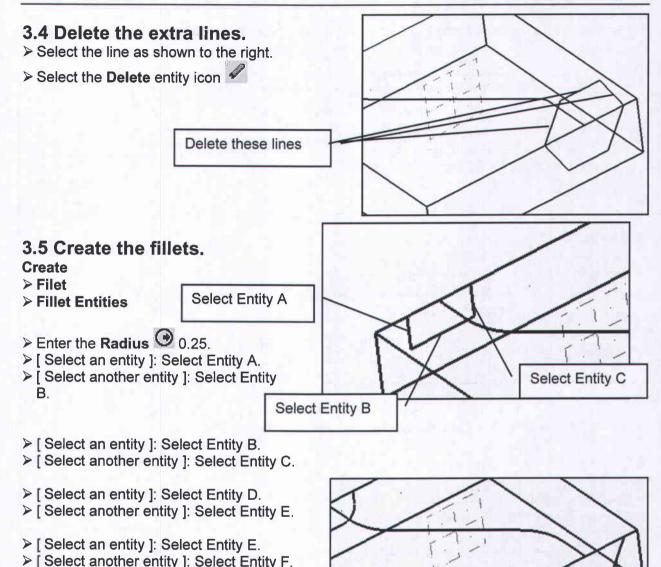
- > Create Polygon
- > Change the number # of sides to 6.
- > Enter the Radius value.
- > Enable Corner for the system to measure the radius to the corner of the polygon.
- > [Select position of base point]: Select the endpoint of the line as shown in the following screenshot.



> Select the OK button to exit the Polygon dialog box.



TUTORIAL 11



STEP 4:

CREATE THE WIREFRAME IN A



Select Entity D

Create

- ➤ Line
- > Create Line Endpoint

➤ Select the **OK** button.

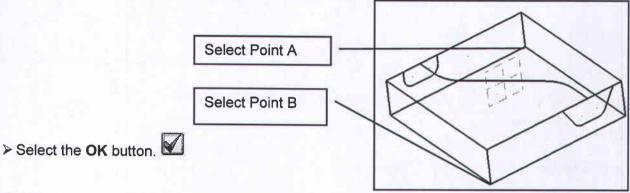
3- 0-0- /

Top (WCS)

Front (WCS)
Right (WCS)

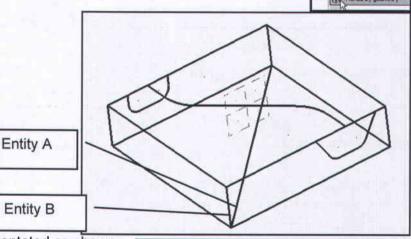
Planes by solid face

- > [Specify the first endpoint]: Select Point A.
- > [Specify the second endpoint]: Select Point B.
- Make sure that this **Endpoint** icon is on when you select the points.



4.2 Define a new construction plane using 2 lines.

- ➤ Select the drop-down arrow next to the **Set plane** icon and select from the list **Planes by geometry**.
- > [Select a flat entity, 2 lines, or 3 points]:

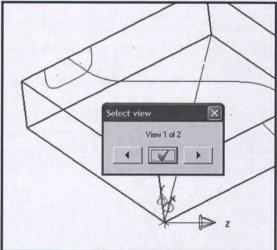




➤ Select Entity B.

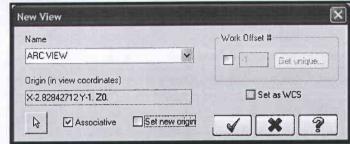
Make sure that the axes are orientated as shown. Otherwise select the right-hand arrow.

➤ Select the **OK** button to accept it.





- > In the Name field enter Arc view.
- ➤ Disable Set new origin
- > Select the **OK** button to exit.



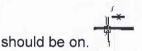
4.3 Create an arc polar knowing the center location, initial and final angles.

Create

- > Arc
- > Create Arc Polar
- > Enter the Radius 0.5.
- To set the other parameters of the arc use the **Tab** key. Note that the diameter value is automatically changed by the system based on the radius.

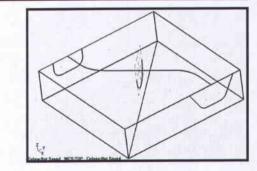
Select the midpoint here

- > Enter the Start Angle 180 (Tab).
- > Enter the End Angle . 0 (Enter).
- ➤ [Enter the center point]: Select the center point at the midpoint of the line as shown in the picture to the right.
- The mouse icon for midpoint



- > Select the **OK** button to exit the command.
- > The wireframe should look as shown.

STEP 5: CREATE THE SWEPT SURFACE



(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).

Create

- ➤ Surface
- > Create Swept Surfaces
- > Make sure that Chain is enabled.
- ➤ [Define the across contour(s) 1]: Select Chain 1 as shown.
- ➤ [Define the across contour(s) 2]: Select Chain 2.
- > [Define the across contour(s) 3]: Select Chain 3.
- Select all the chains in the same directions. Use the Reverse

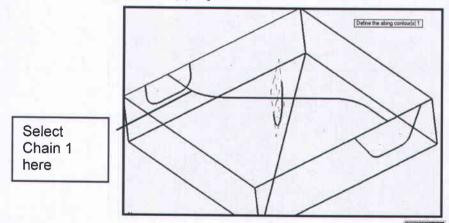
Select
Chain 1
here

Select
Chain 2
here
Select
Chain 3



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

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



here

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

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





> Change the Construction plane to Top.



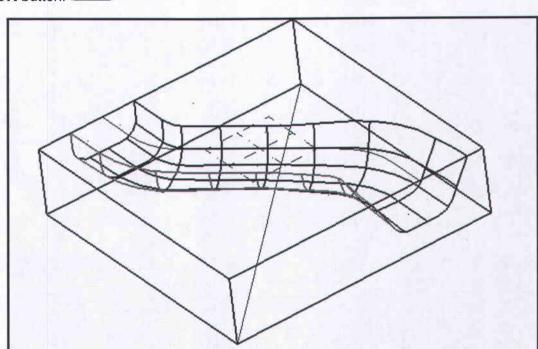
STEP 6: SAVE THE FILE.

File

➤ Save as

> File name: "Your Name_11"

> Select the **OK** button.

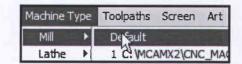


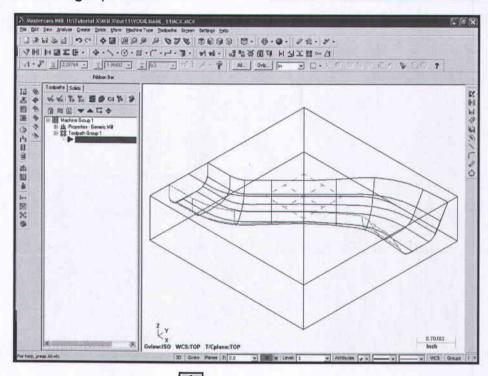


TOOLPATH CREATION STEP 7: SET UP THE STOCK TO BE MACHINED.

Machine type

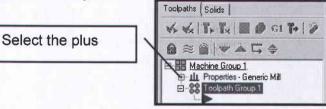
- > Mill
- > Select Default.
- > To display the Toolpaths Manager press Alt + O.



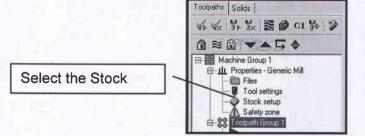


> Use the **Fit** icon to fit the drawing to the screen.

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



> Select Stock setup.

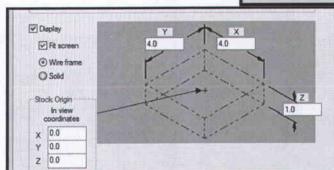


- > Change the parameters to match the following screenshot.
- > Select **Bounding box** button to automatically find the part extents.

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

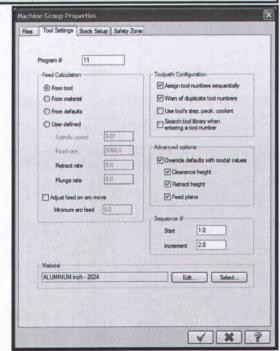






> Select the **Tool Settings** tab to set the tool parameters.

> Select the OK button to exit Toolpath Group Properties.





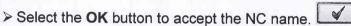
STEP 7: ROUGH OUT THE SURFACE USING SURFACE ROUGH FLOWLINE.



Flowline rough and finish toolpaths allow precise control of the scallops left on the part, creating an exact, smooth finish. You can only use flowline toolpaths on one surface or a row of adjacent drive surfaces.

Toolpaths

- > Surface Rough
- > Rough Flowline Toolpath
- > Select Cavity.
- > Select the **OK** button to exit.

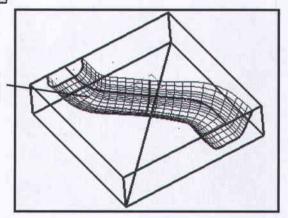


> [Select Drive Surface]: Select the surface.

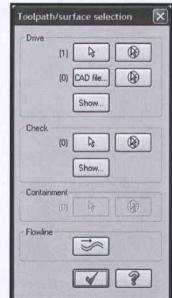


Select the surface

➤ Select the **End Selection** button.



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



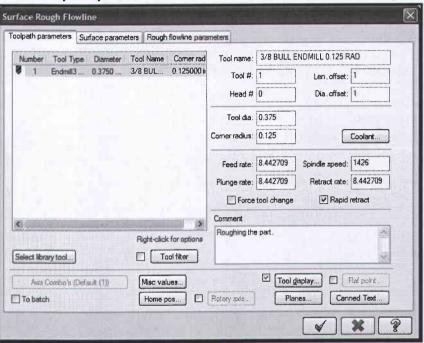


> Click on the Select library tool button.

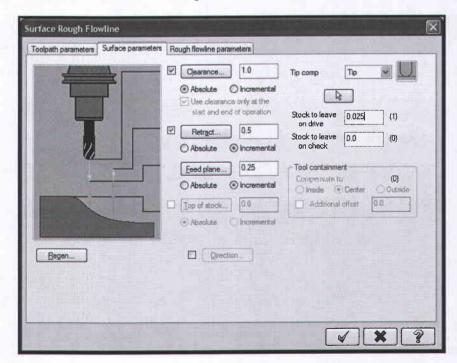
> Select the **Filter** button in the **Tool Selection** dialog box and select 3/8 Bull Nose with 0.125 corner radius. (See tutorial 9 page 8-15)

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

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



> Select the Surface parameters tab and make the changes as shown.





> Select the Rough flowline parameters page and make the changes to match the following

screenshot. Surface Rough Flowline Toolpath parameters | Surface parameters | Rough flowline parameters Cut control Stepover control Distance (Distance 0.1875 Cutting method Zigzeg Total tolerance... 0.005 O Scallop height 0.05 Single row only Check flowline motion for gouge 0.1 Max stepdown: Plunge control Allow multiple plunges along cut Allow negative Z motion along surface O Cut from one side Allow positive Z motion along surface O Cut from both sides Rib cut Cut depths... Gap settings... Advanced settings... 50,0 Resolution (2) of tool Total tolerance settings × > Select the Total Filter ratio 2:1 沁 Total tolerance is the sum of the arc filter tolerance and cut tolerance 0.003333 Filter tolerance determines the accuracy of the surface toolpath using chordal devia Cut tolerance 0.001667 Maximum stepdown value sets maximum distance (along the Z-ax Total tolerance 0.005 n the surface toolpath. One way filtering Pirner snows yeleterminan the tune of axis movement to surface ਾਰਜ਼ਤਰੀਜ਼ ਜਾਣ ਖ਼ਲਾਜ਼ੀ ਜਿਵੇਂਦਦੀ ਜਿਸਤ ਗੁੰਦਤਾ ਦਿਸਤ ਜਾਣਦਾਰੀ a previously clea Allow negative/positive Z metion along surface lots the tool cut a Create arcs in XY Create arcs in XZ Dreate arcs in YZ plunging/retracting. 0.005 Minimum arc radius Maximum arc radius 100.0 tolerance button and set the filter ratio to 2:1. Change the total tolerance value to .005.

➤ Select the **OK** button to exit.

> Select the Gap settings button and change the Gap size as

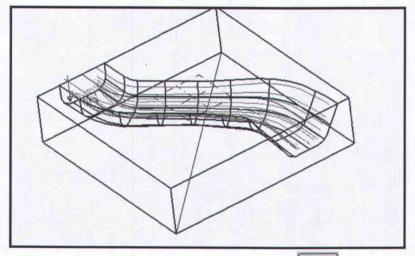


Gap settings sets the way in which the tool moves between gaps or spaces in a surface toolpath.

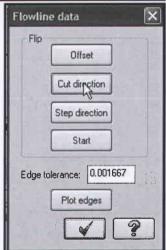
> Select the **OK** button twice to exit the parameter pages.



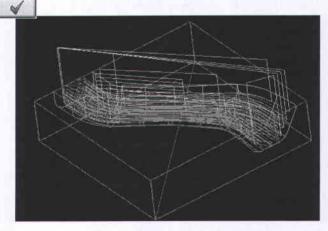
> Select the Cut Direction button to toggle between along and across direction of cutting. The flowlines should look as shown in the following picture (along).







> Select the **OK** button to exit **Flowline data**.





STEP 8: FINISH THE SURFACE USING SURFACE FINISH FLOWLINE.

- > Select the Toolpaths Manager tab to enable it.
- > Select the Select all visible operations button.
- Select Toggle toolpath display on selected operations to remove the toolpaths display from the screen.

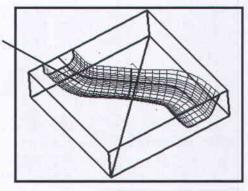




Toolpaths

- > Surface Finish
- > Finish Flowline Toolpath
- ➤ [Select Drive Surface]: Select the surface.

Select the surface



> Select the End Selection button.



> Select the OK button to exit Toolpath/surface selection.

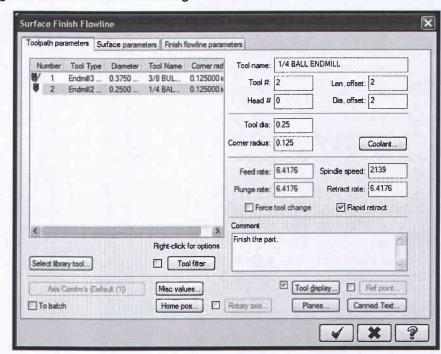


- > Click on the **Select library tool** button.
- ➤ Using the **Filter** option select the 0.25 Ball Endmill.

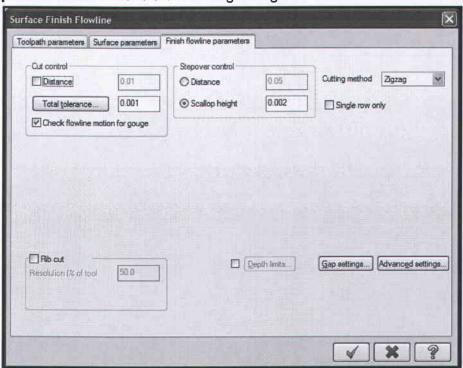




> Make any necessary changes as shown in the following screenshots.

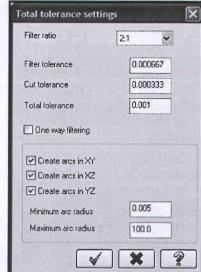


> Select Finish flowline parameters and make the following changes.





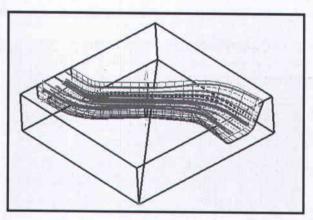
➤ Select the **Total tolerance** button and set the **Filter ratio** to 2:1, maintaining the **Total tolerance** at 0.001.



> Select the OK button to exit Total tolerance settings.

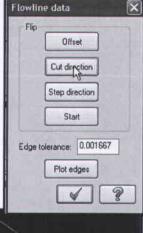


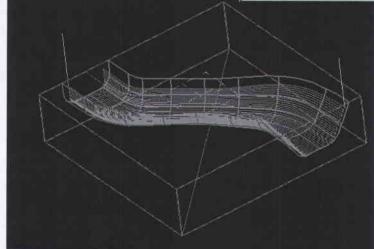
Select the Cut Direction button to toggle between along and across direction of cutting. The flowlines should look as shown in the following picture (along).



➤ Select the **OK** button to exit

Flowline data







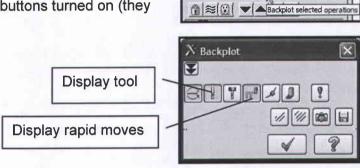
V V V V V S € G1 V 9

KK K K K F O C1 10 3

Select all visible operations

STEP 9: BACKPLOT THE TOOLPATH.

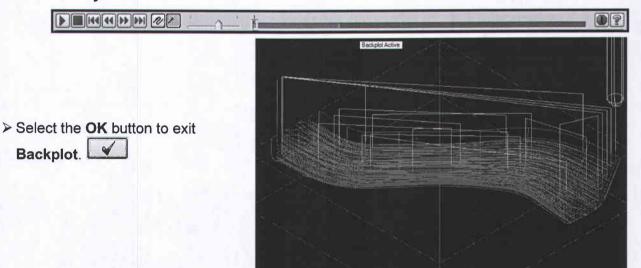
- > Enable Toolpath Manager.
- > Click on the Select all visible operations icon in the Toolpaths Manager to select all operations.
- > Select the **Backplot** selected operations button.
- ➤ Make sure that you have the following buttons turned on (they will appear pushed down).
- ➤ Display tool
- ➤ Display rapid moves



Toolpaths | Solids

Toolpaths | Solids |

Select the Play button





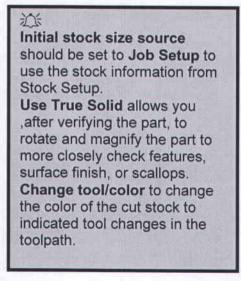
STEP 10: VERIFY.

- > Select the Verify selected operations button.
- > Select Turbo mode as shown.

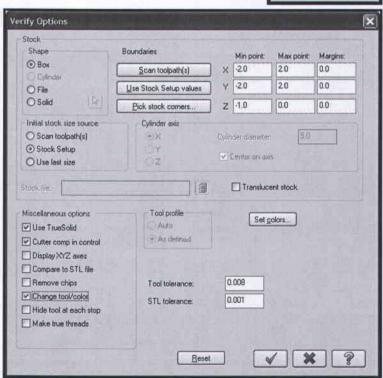




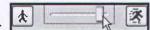
> Select the Configure button.



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

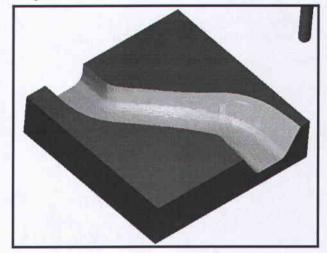


> Set the Verify speed by moving the slider bar in the speed control bar.



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

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

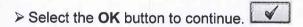


> Select the **OK** button to exit **Verify**.

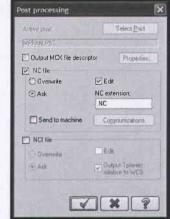
STEP 11: POST PROCESS THE FILE.

Select the Post selected operations button from Toolpath Manager.

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

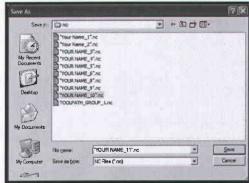




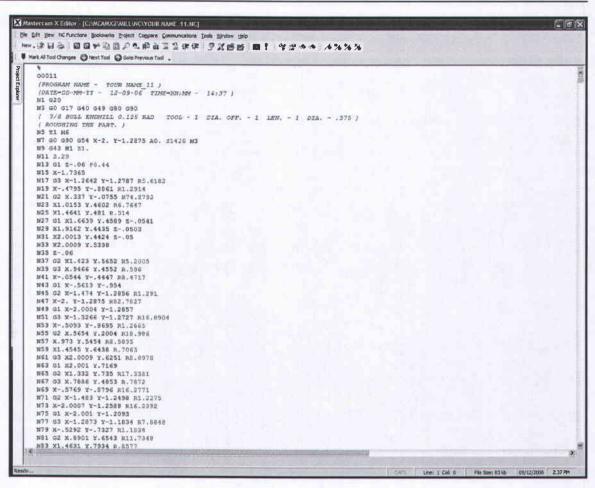


Enter the same name as the geometry name in the NC File name field.

> Select the Save button.







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

STEP 12: SAVE THE UPDATED MCX FILE.



> Select the Save icon.

ALL DIMENSIONS IN INCHES 6.0 18.00 - 6.000 -300

TUTORIAL 11 - EXERCISE TITE

ALUMINUM T6061 MATERIAL

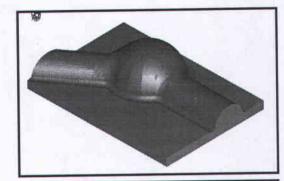
DATE: JUNE 12, 2000

eMastercam.com

Mastercam. TUTORIAL 11

REVIEW EXERCISE.

Student practise. Create the Toolpath for Exercise-Tutorial 11 as per the instructions below;



Tips:

- **1.** Establish the **Stock size** Y = 7, X = 10, Z = 2.5
- **Stock origin** X = -5.25, Y = -1, Z = 2
- 2. Create a rectangle with surface option at Z0, the same size as the stock.

3. Surface Rough Pocket

Select all surfaces

Chain the bottom rectangle for tool containment Use 1" Bull End Mill rad 1/4 Stock to leave on drive surfaces= 0.05"

Total tolerance = 0.01
Max stepdown =0.1
Disable Entry
Plunge outside tool conta

Plunge outside tool containment Cutting method quick zigzag

Disable finish containment boundary

4. Surface Finish Flowline

Select only the "sweep" surface. Flowline toolpath can only machine surfaces that form a row.

Use 1/2" Ball End Mill

Stock to leave on drive surfaces= 0

Filter ratio 2:1

Total tolerance= .002

Scallop height = .002

Cutting method zigzag

Flip options: Cut direction "along Y-axis" as shown above.

- 5. Backplot and Verify the toolpaths.
- 6. Post process the file.

