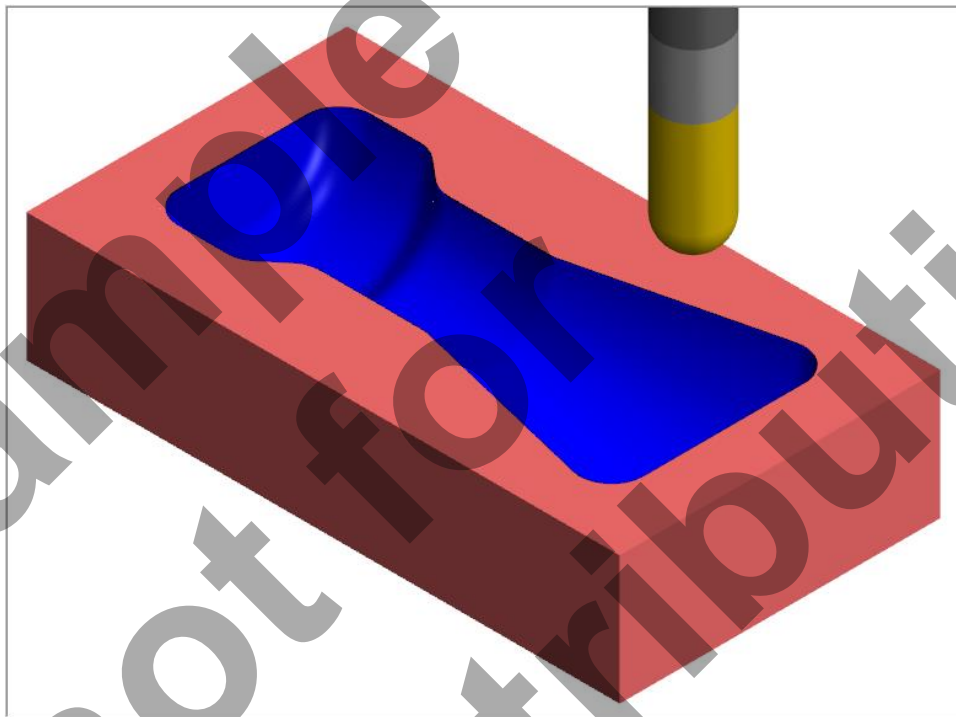


**Mastercam** X<sup>8</sup>

**TRAINING**

**GUIDE**



**MILL-LESSON-9**

**SURFACE HIGH SPEED - OPTIAREA**

**SURFACE HIGH SPEED - HYBRID**

**camInstructor**

### Objectives

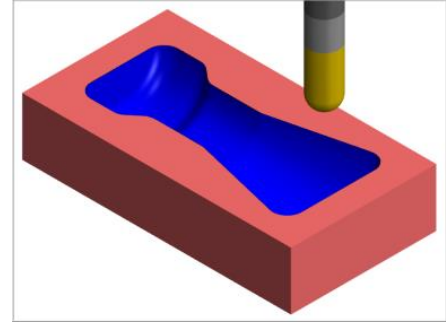
You will create the geometry for Mill-Lesson-9, and then generate the toolpaths to machine the part on a CNC vertical milling machine. This Lesson covers the following topics:

➤ **Create a 3-dimensional drawing by:**

- Creating lines.
- Creating arcs.
- Trimming geometry using Divide / Delete.
- Creating a revolved surface.

➤ **Establish Stock Setup settings:**

- Stock size using Bounding Box.
- Material for the part.
- Feed calculation.



➤ **Generate a 3-dimensional milling toolpath consisting of:**

- Surface High Speed - OptiArea
- Surface High Speed - Hybrid

➤ **Inspect the toolpath using Mastercam's Verify and Backplot by:**

- Launching the Verify function to machine the part on the screen.
- Generating the NC- code.

### High Speed Toolpaths - HST

3D HST Hybrid toolpaths include a new Optimize cut order checkbox which defines the cut order Mastercam applies to different cutting passes in the toolpath. When selected, Mastercam works in tiers, ordering the toolpath's cutting passes based on proximity and safety. Mastercam machines features by area until it reaches a common Z-level, then repeats the process until it reaches the next common ground. The tool stays in a common ground, which reduces unnecessary motion and shortens the length of the toolpath.

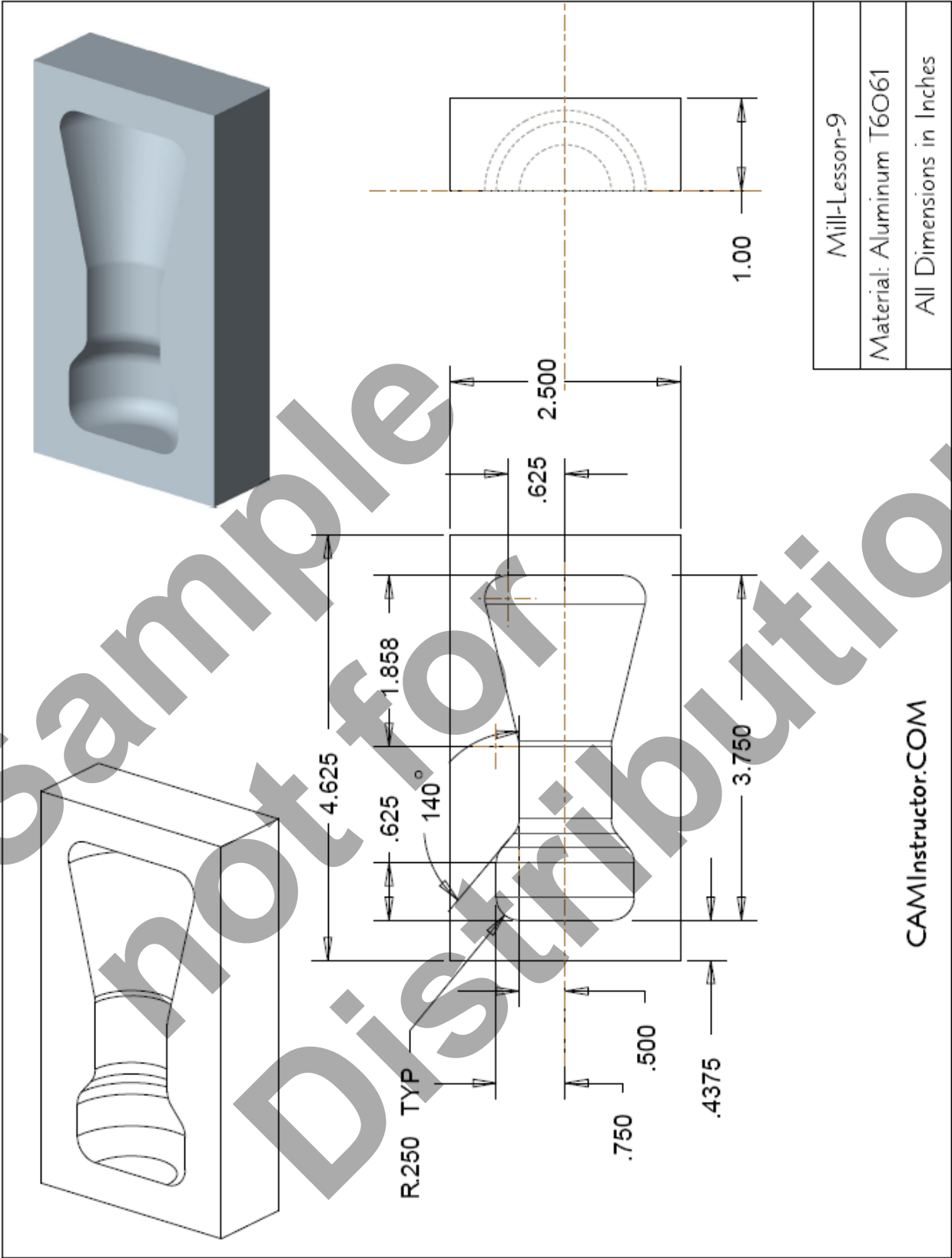
Hybrid toolpaths now offer additional options for handling open and closed contours. Closed contours contain continuous motion without a need for a retract or reversal of direction. You can select climb or conventional machining for these contours. Open contours offer One Way or Zigzag cutting options. Open contours set to One Way will use the cutting method selected for closed contours.

The new Automatic Offset method controls how Mastercam handles the boundaries between steep and shallow areas. If the toolpath engages part surfaces that include core and cavity properties, Mastercam applies upper to lower offset methods to core features and lower to upper offset methods to cavity shapes.

The 3D HST toolpaths now calculate more efficient high speed motion at your part's boundaries. To improve regeneration speed, they now only recalculate the necessary sections of the toolpath instead of processing the entire toolpath when changes are made.

The 3D HST Project toolpath projects either geometry or a toolpath from an earlier operation onto surfaces. Previously, the Project toolpath was only available as a surface roughing toolpath. In Mastercam the full suite of high speed toolpath parameters are available to refine the tool motion.

MILL-LESSON-9 DRAWING



CAMInstructor.COM

## **TOOL LIST**

- 0.500 diameter bull end mill with a 0.125 corner radius to rough machine the pocket.
- 0.500 diameter ball end mill to finish machine the pocket.

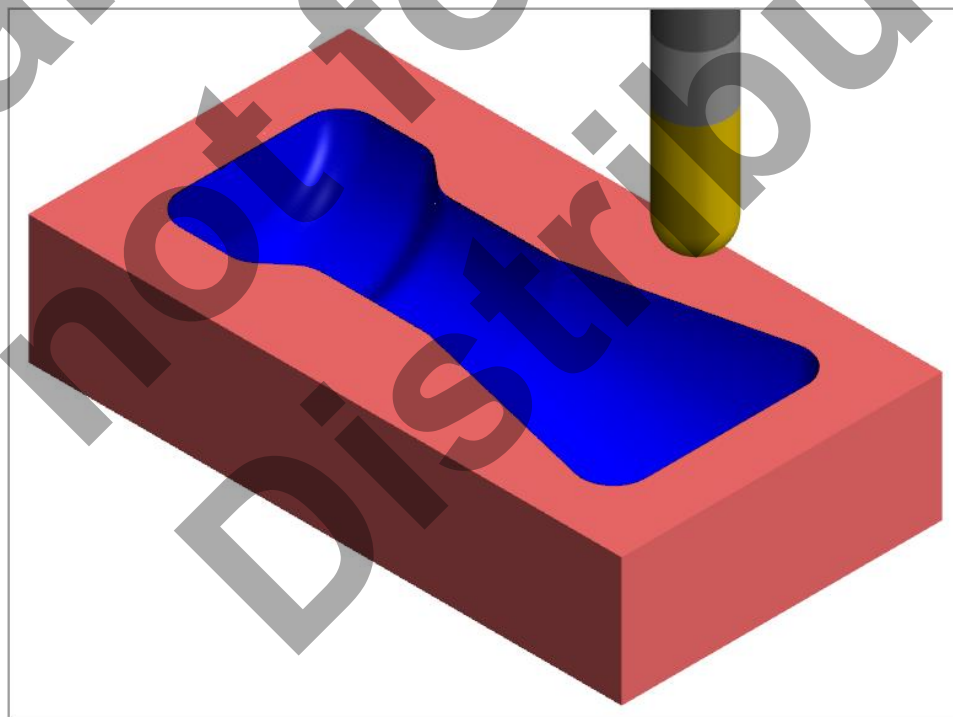
# **MILL-LESSON-9 - THE PROCESS**

## **Geometry Creation**

- TASK 1:** Setting environment
- TASK 2:** Create geometry
- TASK 3:** Trim the geometry
- TASK 4:** Create the revolved surface
- TASK 5:** Create boundary curves
- TASK 6:** Save the drawing

## **Toolpath Creation**

- TASK 7:** Define the rough stock using stock setup
- TASK 8:** Rough the pocket using Surface High Speed OptiArea
- TASK 9:** Finish the pocket using Surface High Speed Hybrid
- TASK 10:** Verify the toolpath
- TASK 11:** Save the updated Mastercam file
- TASK 12:** Post and create the CNC code file



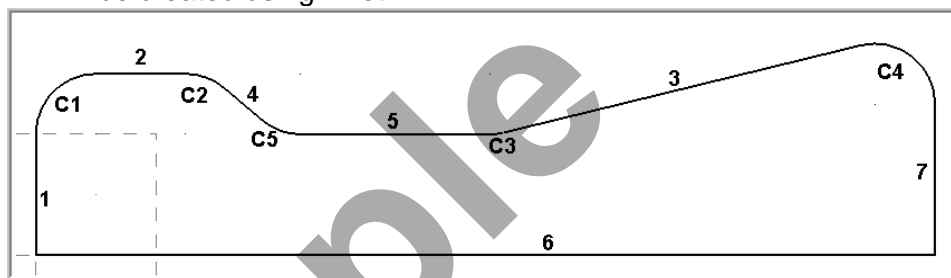
## TASK 1: SETTING THE ENVIRONMENT

Before starting the geometry creation you should set up the grid, toolbars and machine type as outlined in the **Setting the environment** section at the beginning of this text:

1. Set up the Grid. This will help identify the location of the origin.
2. Load the Workspace – **SETTINGS>Load Workspace>3D Toolpaths** to machine a 3D part.
3. Set the machine type to a Haas Vertical Spindle CNC machine.

## TASK 2: CREATE GEOMETRY - X0 Y0 LOWER LEFT CORNER

- In this task you will create the geometry that will be used to create the revolved surface. First you will create the circles, C1 through C4 and then create the lines 1 through 7. The arc C5 will be created using Fillet.



### ➤ Create Circle #1

1. Select **CREATE>Arc>Circle Center point...**
2. The **Circle Center Point** ribbon bar appears and you are prompted to **Enter the center point**.



3. Before moving on, click in the space for **radius** and enter a value of **0.25**. Next **click on the icon for radius** on the ribbon bar as shown below. This **locks** in the radius for the circles. To **unlock** click on the icon for diameter.




4. To satisfy this first prompt click on the **FastPoint** Icon on the Auto Cursor ribbon bar
5. In the space input the **X (0.25)** and **Y (0.5)** values for the center of the circle and **hit enter**. **Note** that you can use a variety of formats to enter coordinates here. This is very helpful when cutting and pasting coordinate values from another document. You do not need to input the Z value for this example.

6. On the ribbon bar click on **Apply** to fix the entity.



### ➤ Create Circle #2

7. To satisfy the prompt **Enter the center point** click on the **FastPoint** Icon on the Auto Cursor ribbon bar.
8. In the space input the **X (0.625)** and **Y (0.5)** values for the center of the circle and then hit the **enter** key.
9. On the ribbon bar click on **Apply** to fix the entity.

### ➤ Create Circle #3

10. To satisfy the prompt **Enter the center point** click on the **FastPoint** Icon on the Auto Cursor ribbon bar.
11. In the space input the **X (3.750-1.858)** and **Y (0.5+0.25)** values for the center of the circle and then hit the **enter** key.
12. On the ribbon bar click on **Apply**  to fix the entity.

### ➤ Create Circle #4


13. To satisfy the prompt **Enter the center point** click on the **FastPoint** Icon on the Auto Cursor ribbon bar.
14. In the space input the **X (3.750-0.25)** and **Y (0.625)** values for the center of the circle and then hit the **enter** key.
15. Click on the OK icon  to complete this feature.
16. Now select the **Fit to screen** icon .

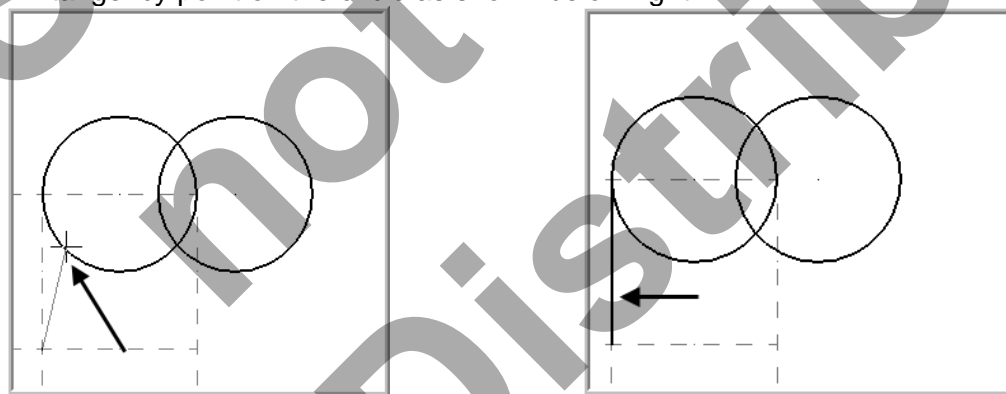
### ➤ Create Line #1

17. Select from the pull down menu: **CREATE>Line>Line Endpoint...**

18. On the graphics screen you are prompted: **Specify the first endpoint** and the Line ribbon bar appears. **Note** that the **Tangent** function is activated.



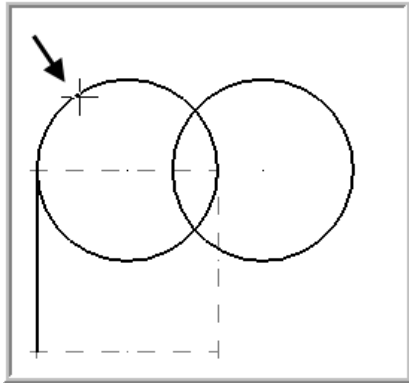
19. To satisfy the prompt **Specify the first endpoint** move the cursor over to the **center of the grid** and as you get close to the **Origin (X0Y0)** a visual cue appears.  This is the cue that will allow you to snap to the origin, with this visual cue highlighted **pick the origin**.
20. To satisfy the prompt **Specify the second endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual snap cue** being displayed and click on this point. As **Tangent** is activated on the line ribbon bar the line will snap to the closest tangency point on the circle as shown below right:



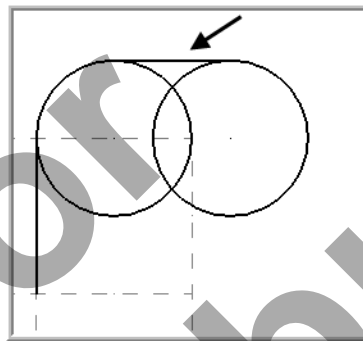
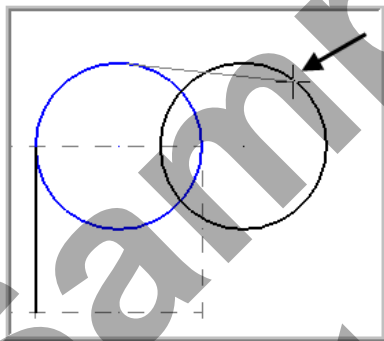
21. On the ribbon bar click on **Apply**  to fix the entity.

➤ **Create Line #2**

22. To satisfy the prompt **Specify the first endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual cue** being displayed and click on this point. As **Tangent** is activated on the line ribbon bar the line will snap to the closest tangency point on the circle.



23. To satisfy the prompt **Specify the second endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual cue** being displayed and click on this point. As **Tangent** is activated on the line ribbon bar the line will snap to the closest tangency point on the circle as shown below right.



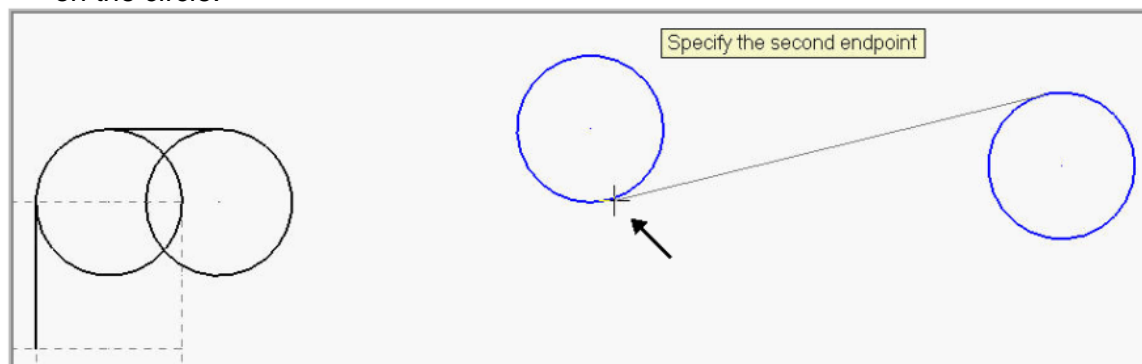
24. On the ribbon bar click on **Apply**  to fix the entity.

➤ **Create Line #3**

25. To satisfy the prompt **Specify the first endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual cue** being displayed and click on this point.



26. To satisfy the prompt **Specify the second endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual cue** being displayed and click on this point. As **Tangent** is activated on the line ribbon bar the line will snap to the closest tangency point on the circle.



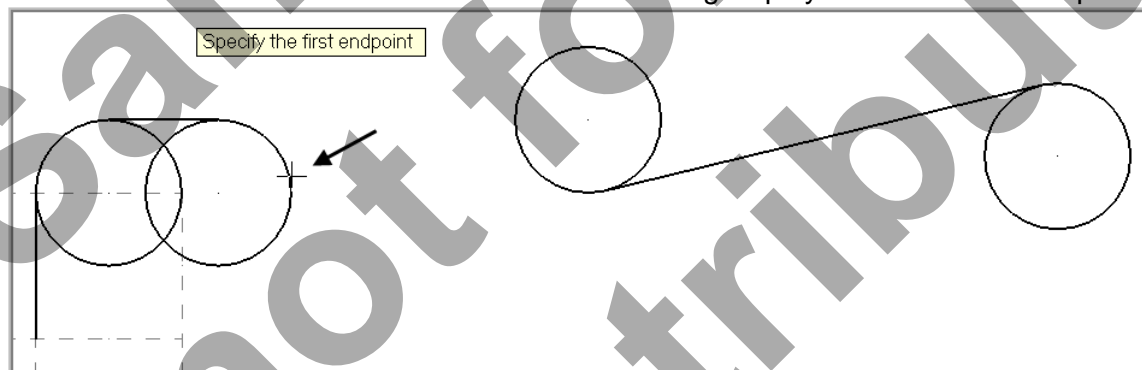
27. On the ribbon bar click on **Apply**  to fix the entity.

### ➤ Create Line #4

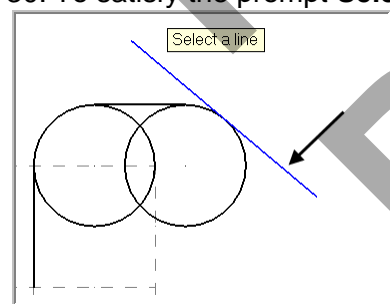
28. Click in the space for **Length** and enter a value of **0.5** and then hit the tab key. In the space for **Angle** and enter a value of **-40.0** and hit enter.



29. To satisfy the prompt **Specify the first endpoint** move the cursor over the area of the circle shown below. Ensure there is **no visual cue** being displayed and click on this point.



30. To satisfy the prompt **Select a line** pick the lower line as shown below:

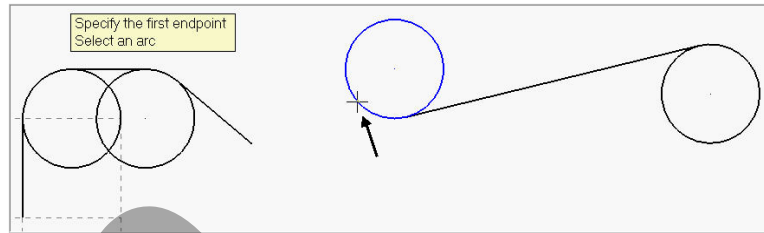
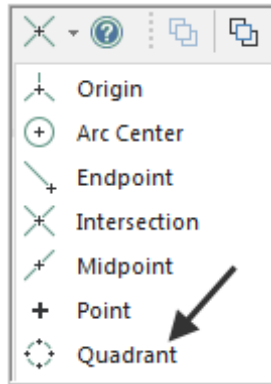



31. On the ribbon bar click on **Apply**  to fix the entity.




➤ **Create Line #5**

32. To satisfy the prompt **Specify the first endpoint** use the **AutoCursor ribbon bar Override drop down list – located at top of screen on left**. Open the drop down and select **Quadrant**. Move the cursor over the area of the circle shown below right and click on the circle:



33. To satisfy the prompt **Specify the second endpoint** move the cursor over to the left of the circle shown below. Ensure the visual cue for **Horizontal/Vertical**  is displayed and click on this point.



34. On the ribbon bar click on **Apply**  to fix the entity.

➤ **Create Line #6**

35. Click on the **Multi-line** button to activate it.




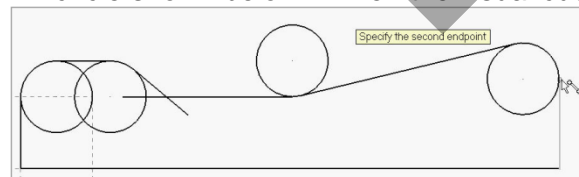
36. To satisfy the prompt **Specify the first endpoint** move the cursor over to the **center of the grid** a visual cue appears.  With this visual cue highlighted **pick the origin**.

37. Click in the space for **Length** and enter a value of **3.75** and then hit the tab key. In the space for **Angle** enter a value of **0** and hit enter.



➤ **Create Line #7**

38. To satisfy the prompt **Specify the second endpoint** move the cursor over to the right of the circle shown below. When the visual cue for **Endpoint**  is displayed click on this point.

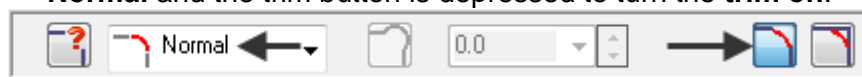


39. Click on the OK icon  to complete this feature.

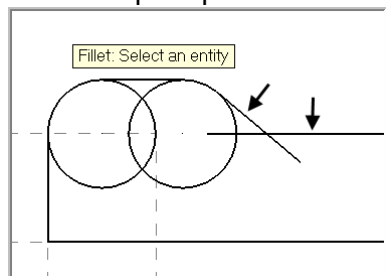
### ➤ Create Fillet #5

40. Select **CREATE>Fillet>Fillet Entities...**

41. On the Fillet ribbon bar enter **.25 for the radius**. Ensure the **Style** of radius is set to **Normal** and the trim button is depressed to turn the **trim on**.



42. When prompted to **Fillet: Select an entity**, select the two lines shown below:



43. Click on the OK icon  to complete this feature.

### TASK 3:

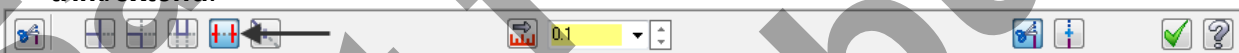
#### TRIM GEOMETRY


➤ In this task you will trim the geometry using **Divide / Delete**. There are many different ways to accomplish this trimming operation, this is just one method. If you make a mistake using **Divide / Delete** use the **Undo** button to start over.

#### ➤ Trim the circles

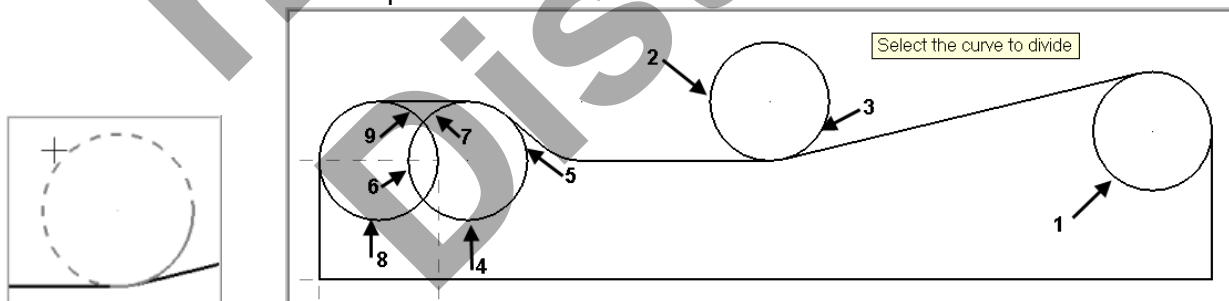
1. Select **EDIT>Trim/Break>Trim/Break/Extend**

2. The Trim / Extend / Break ribbon bar appears and you are prompted to **Select the entity to trim/extend**.




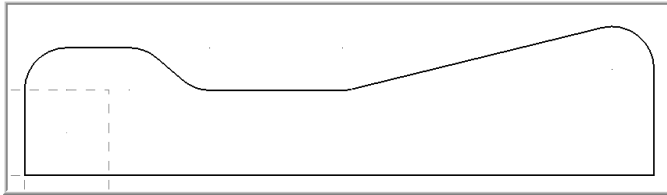
3. Click on the **Divide / Delete**  icon to activate it as shown above. Ensure the icon is pressed down to signal that it is activated.

4. The prompt changes to **Select the curve to divide / delete**. Move the cursor over the various circles and select in order and position as shown below. The arc is trimmed back to the two closest intersections/end points as shown below. Placing the cursor over the entity to divide / delete shows a preview as shown below left.



**Note:** On some of the circles you will need to select them more than once to completely trim them.

5. Click on the OK icon  to complete this feature.  
The completed geometry is shown below:



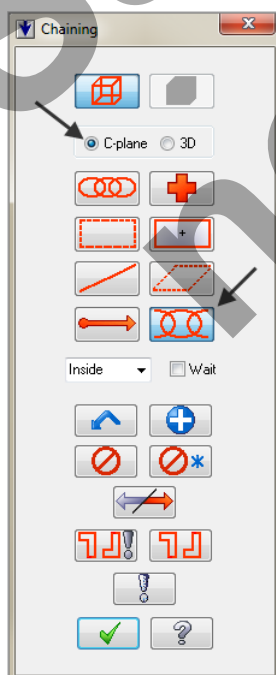
## TASK 4: CREATE THE REVOLVED SURFACE

- In this task you will create a revolved surface using the geometry you have just created and revolve the geometry 180 degrees about the X axis.

**Revolved surfaces** are circular in one direction. You will be prompted to chain an entity that the revolved surface will use as a profile. You may choose multiple entities to create many revolved surfaces at a time.  
You are then prompted to choose a line to use as the axis of rotation and the revolved surface is created.  
If you have chosen multiple entities, a surface for each entity will be created, each having a different profile and position and all using the same axis of rotation. Click Apply or Enter when done.

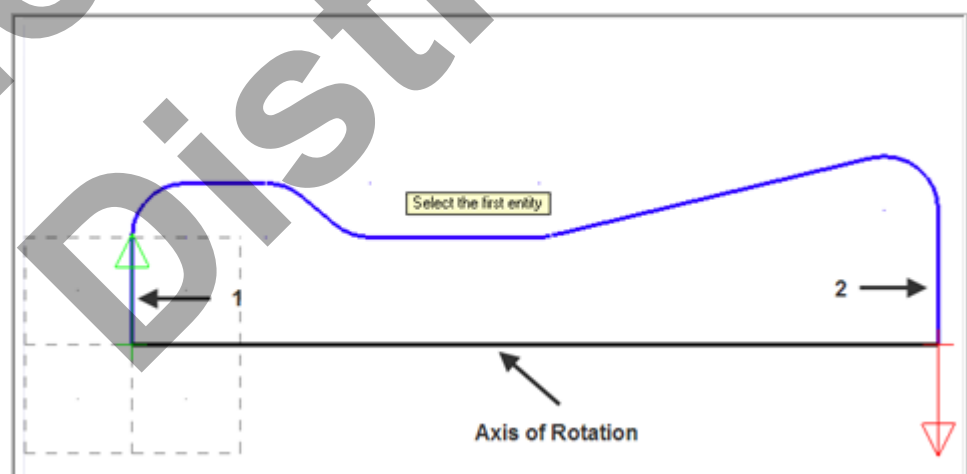
### ➤ Create Revolved Surface


1. Select from the pull down menu: **CREATE>Surface>Revolved...**
2. On the graphics screen you are prompted: **Select profile curve(s) 1.** In the **Chaining** dialog window select **Partial Chain** as shown below:
3. Select **line 1** and ensure the arrow is pointing upwards. If not click on the reverse icon.
4. Then select **line 2** as shown below:





### Cplane

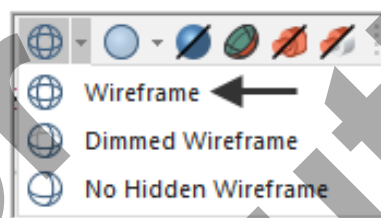
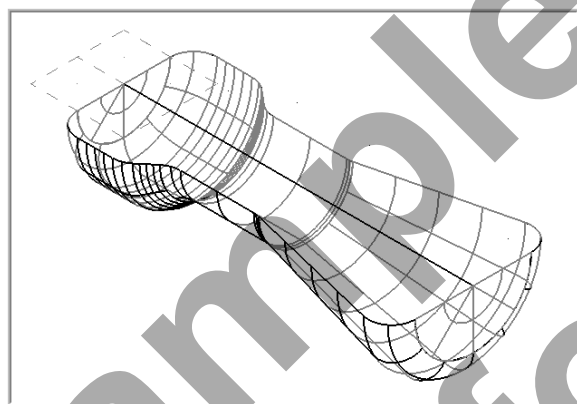
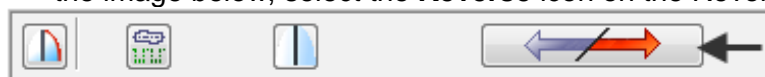
Chains only entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain.




- Click on the OK icon  in the Chaining dialog window.
- Change the **End angle** to **180** as shown below and hit enter:

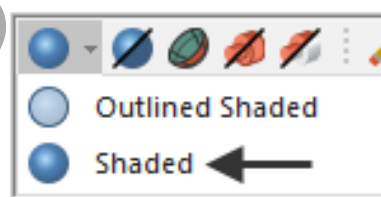
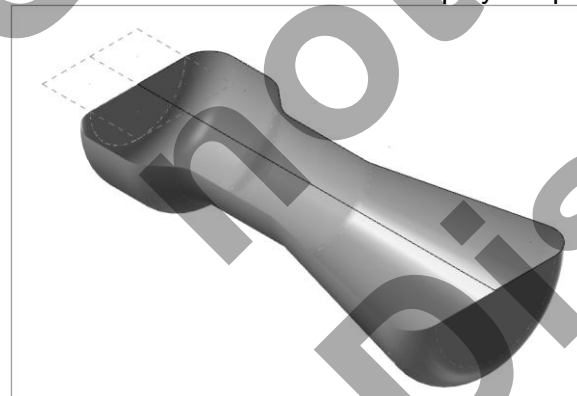




- Change the graphics view to **Isometric** .
- For the prompt on the screen **Select axis of rotation**, select the line shown in the diagram above in step 4.
- Select the **Fit to screen** icon .
- The arrow should be pointing **downwards** and the newly created surface should look like the diagram below: If the arrow is not pointing downwards or the surface does not look like the image below, select the **Reverse** icon on the Revolved surface ribbon bar.

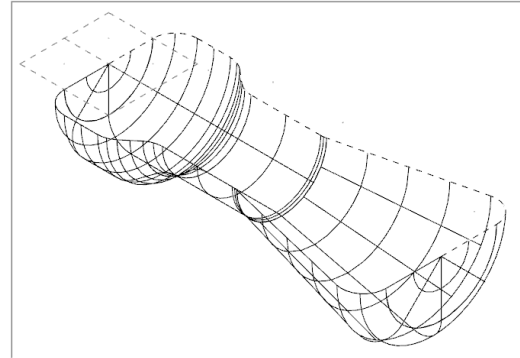
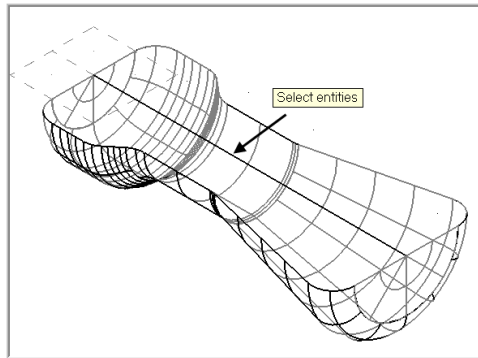


- Click on the OK icon  to complete this feature. The picture above is set to Wireframe display.

- Select the **Shaded** icon to display the part in shaded mode.



13. Select the **Wireframe icon**  to return to wireframe display.
14. **Delete the axis of rotation** as this will not be required anymore. Select the **Delete entities** icon  and when prompted to **Select entities** pick the line shown below and then hit enter.

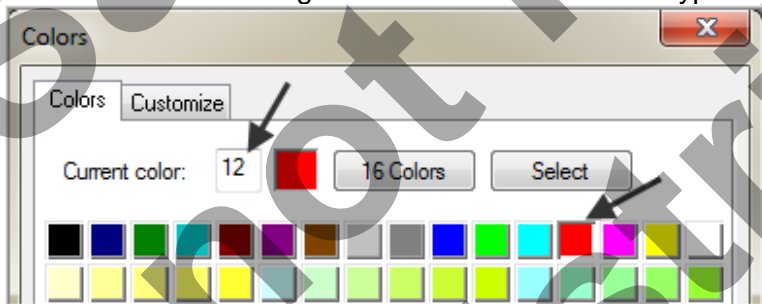


## TASK 5: CREATE BOUNDARY CURVES

- When you machine the Revolved surface you will be prompted to chain an entity that the revolved surface will use as a profile, this task will create that profile.
- **Change the current color to enable better visibility of the entities**
  1. Select **Wireframe color** from the Status bar at the bottom of the screen.



2. In the **Colors** dialog box select the **red** color or type in **12**.



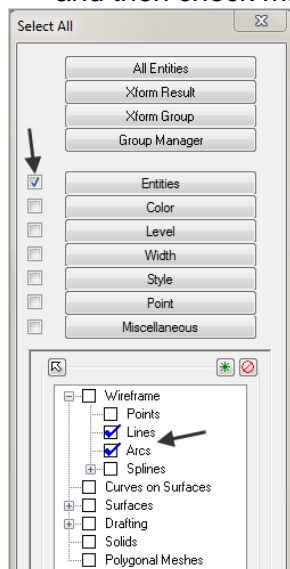
3. Click on the OK icon  to complete this feature.



### ➤ Mirror the initial contour geometry

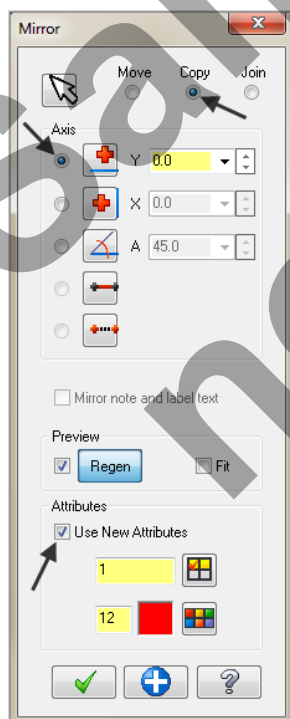
4. Select **Xform>Mirror...**
5. You are first prompted to **Mirror: select entities to mirror**. First select the **All** button on the **General Selection** ribbon bar as shown below:



6. The **Select All** dialog box appears on the left of the screen. Check mark the **Entities** box and then check mark **Lines** hold the **Ctrl** key down and then select **Arcs**.



7. Click on the OK icon  to complete this feature.
8. To move onto the next step you now need to pick the **End Selection** icon .
9. After selecting End Selection the **Mirror window** appears. Set the following values as shown below:





### Use new attributes

Determines the level and color attributes of the transformed entities.

Select this checkbox to display the Level and Color fields where you specify these attributes for the new geometry.

Creating the transformed entities on a different level and in a unique color can make them easier to identify.

To create transformed entities using the attributes of the original entities, deselect this option.

10. Click on the OK icon  to complete this feature.
11. Click on the **Clear Colors**  at the top right hand corner of the screen.

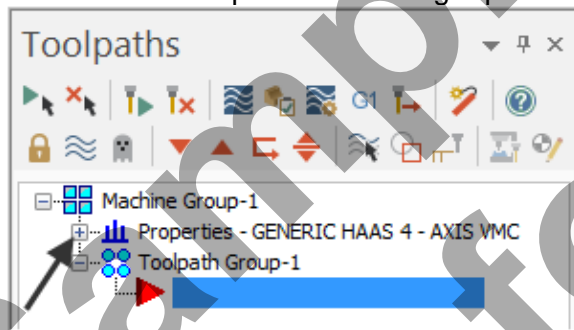
## TASK 6: SAVE THE DRAWING

1. Select **File**.
2. Select **Save as**.
3. In the File name box, type **Mill-Lesson-9**.
4. Save to an appropriate location.
5. Select the **Save** button to save the file and complete this function.

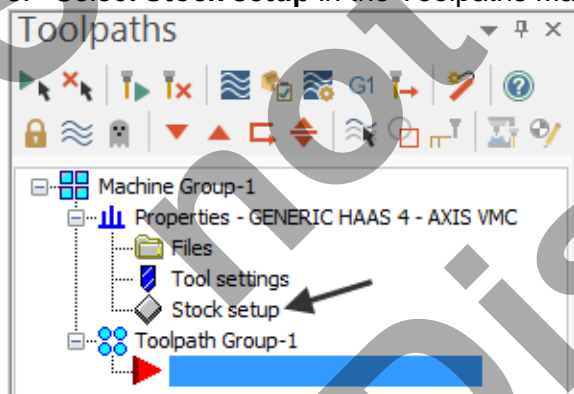
## Toolpath Creation

### TASK 7: DEFINE THE ROUGH STOCK USING STOCK SETUP

1. **Fit to screen** .
2. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will **Show/hide** Operations Manager pane.

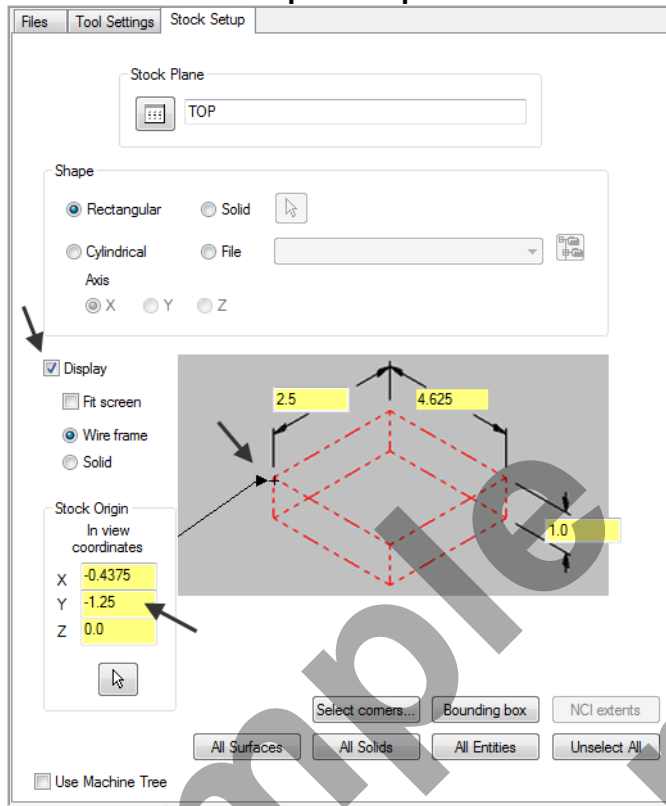


3. Select **Stock setup** in the Toolpaths Manager window.

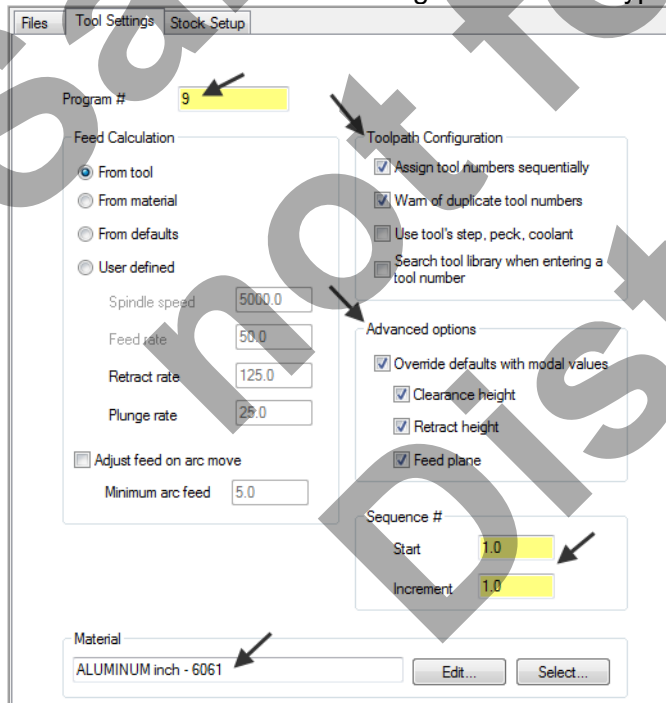


4. Change the parameters to match the **Stock Setup** screenshot below:

➔ **Z zero is at the top of the part.**




5. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. To change the Material type follow the instructions below:




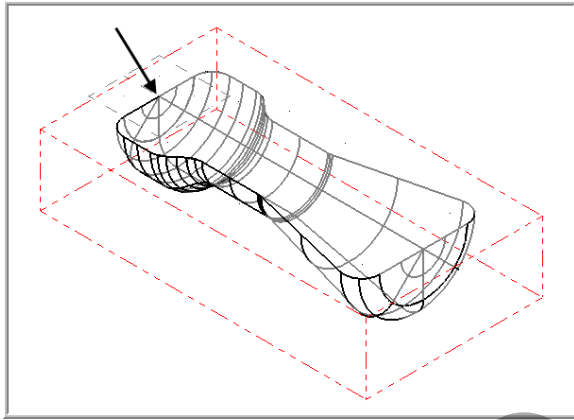
6. To change the Material type to Aluminium 6061 pick the Select button at the bottom of the Tool Settings page.

7. At the **Material List** dialog box open the **Source** drop down list and select **Mill – library**.

8. From the Default Materials list select **ALUMINIUM inch -6061** and then select .



9. Select the OK button  again to complete this Stock Setup function. Your part should look similar to the screen shot below. With **X0 Y0** at the middle left side and **Z zero** at the **top of the part**.




## TASK 8: SURFACE HIGH SPEED OPTI-AREA

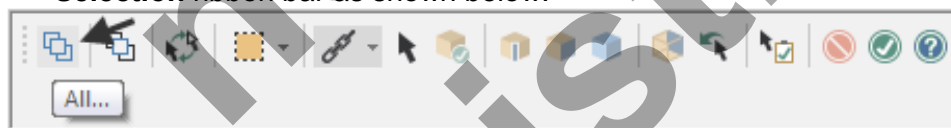
- In this task you will use a 0.5 diameter bull end mill with a 0.125 corner radius to rough out the pocket using **Surface High Speed>OptiArea**.



### Optimized toolpaths

The 3D surface high speed OptiCore, OptiArea, and OptiRest toolpaths support cutters capable of machining very large depths of cut. It uses an aggressive, fast, intelligent roughing algorithm based on Mastercam's 2D high speed dynamic milling motion.

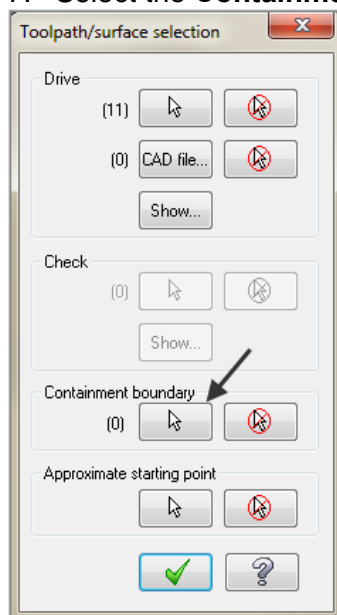
A single Optimized toolpath can cut material in two directions, on stepdowns (-Z) and step ups (+Z). This highly efficient bi-directional cutting strategy removes the maximum amount of material with the minimum of stepdowns, significantly reducing cycle times.

1. From the menu bar select **TOOLPATHS>Surface High Speed>OptiArea...**
2. When prompted to **Enter new NC name** ensure **Mill-LESSON-9** is entered and then select the OK button .
3. You are first prompted to **Select Drive surfaces**, select the **All** button on the **General Selection** ribbon bar as shown below:



4. The **Select All** dialog box appears on the screen.
5. Click on the OK icon  to complete and exit this feature.
6. To move onto the next step you now need to pick the **End Selection** icon .

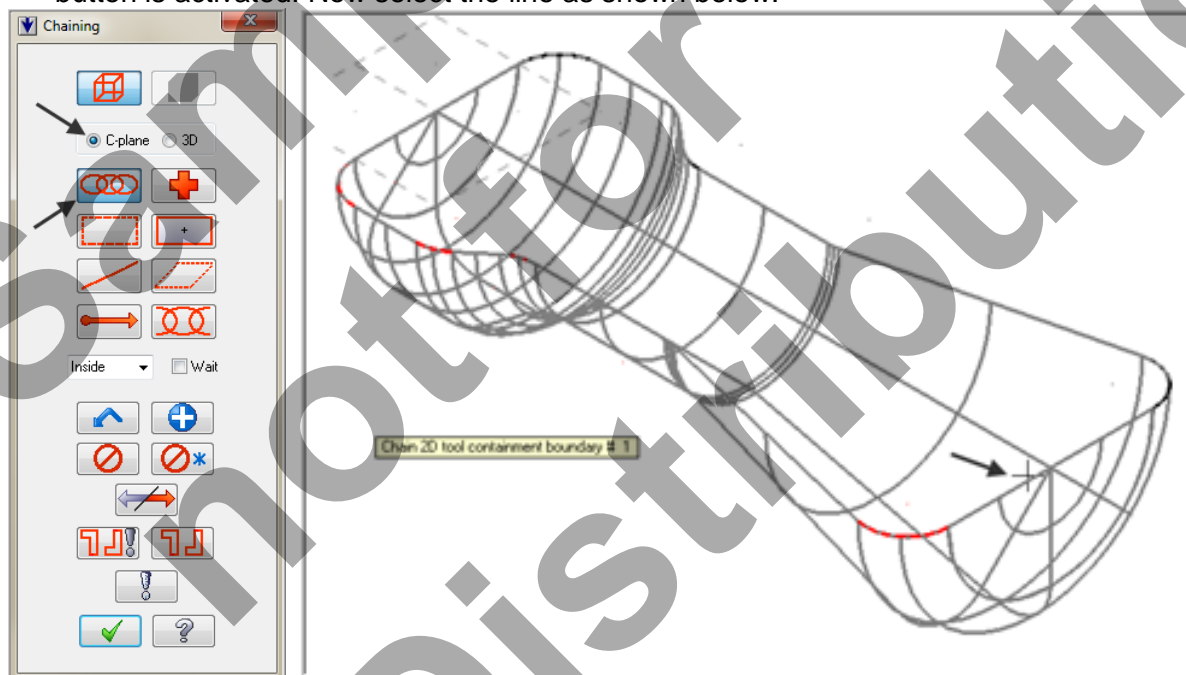
7. Select the **Containment boundary** button from the **Toolpath/surface selection** dialog box.



Use this dialog box to select all the geometry needed for a surface toolpath.

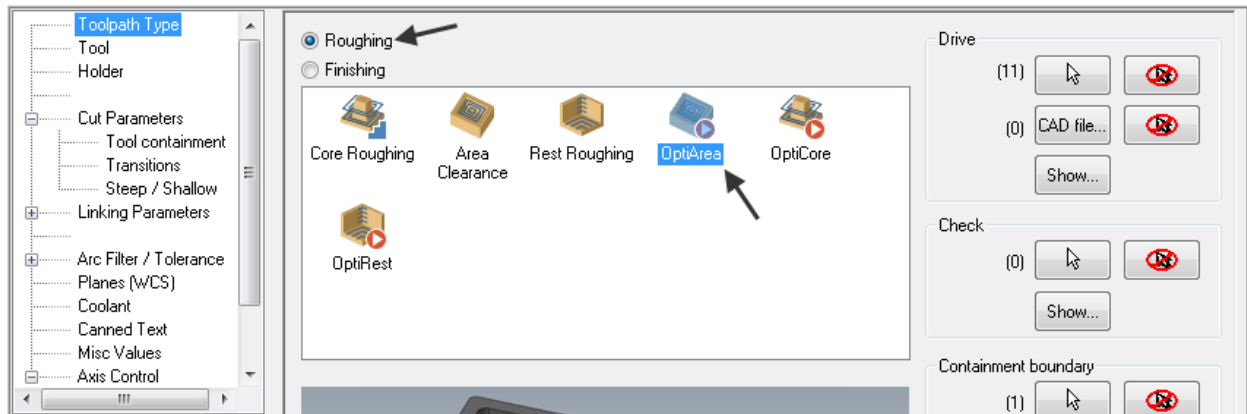
Each surface toolpath requires drive geometry selected from the graphics window or from a CAD file.


8. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Chain 2D tool containment boundary #1**. Ensure the **C-plane** radio button is activated. Now select the line as shown below:

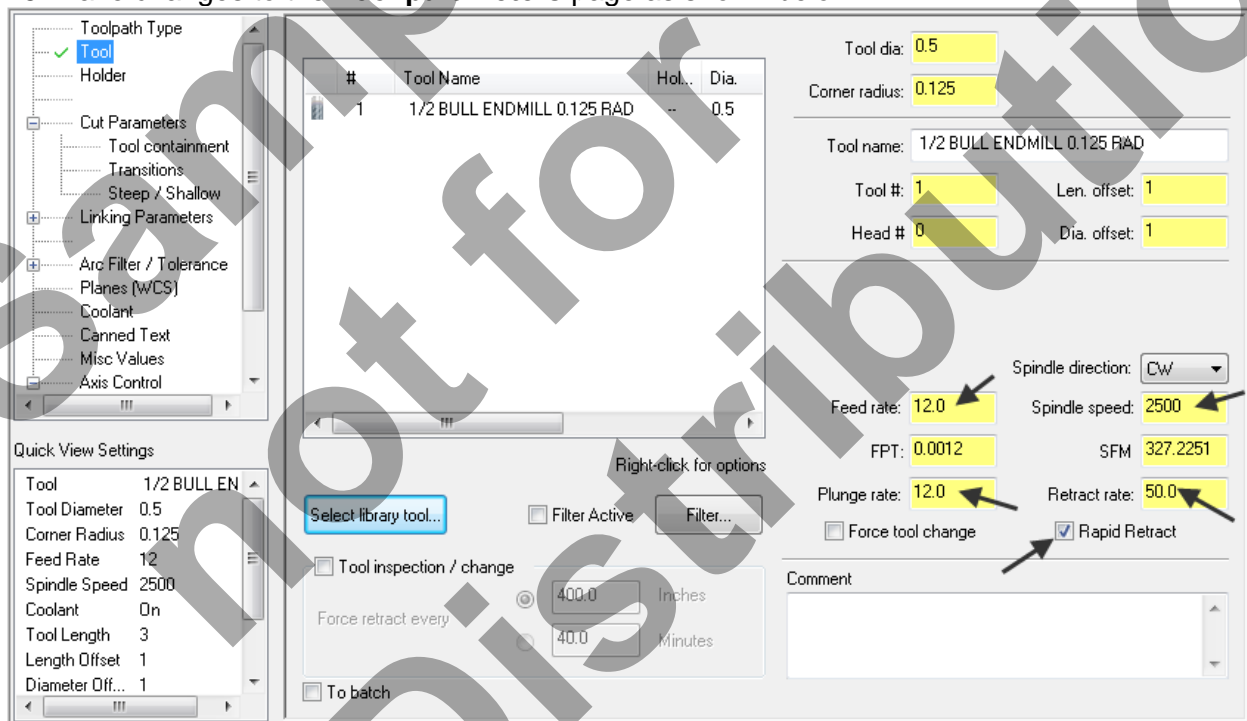


9. After the boundary has been successfully chained select the OK button .
10. Select the OK button  to exit the **Toolpath/surface selection** dialog box.

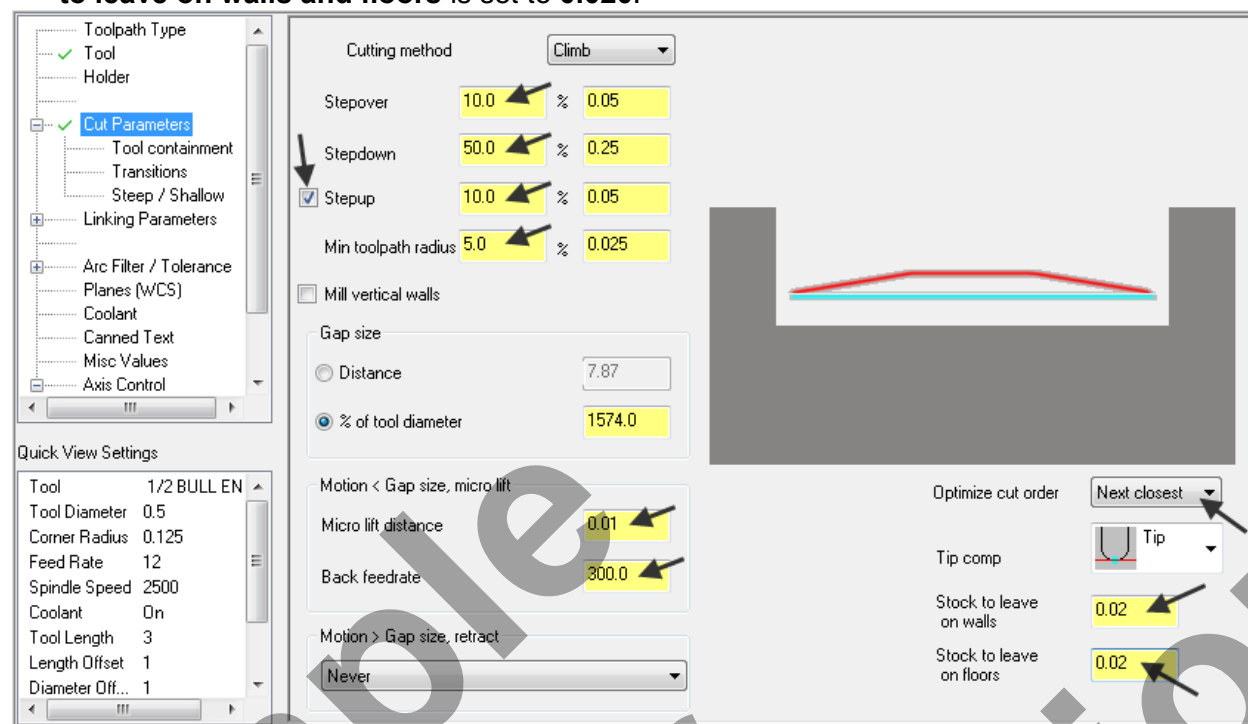
11. On the **Toolpath Type**, ensure the **Roughing** radio button and the **OptiArea** toolpath are selected.



12. Now click on **Tool** and in the lower left corner of the **Toolpath parameters** page select the **Select library tool...** button. **Disable Filter active.**
13. Use the slider bar on the right of this dialog box to scroll down and locate a **0.5 diameter bull end mill with a 0.125 corner radius**. Select the end mill by picking anywhere along its row.
14. Select the OK button  to complete the selection of this tool.
15. Make changes to the **Tool parameters** page as shown below.



16. Select the **Cut parameters** page and make changes to this page as shown below: **Stock to leave on walls and floors** is set to **0.020**.



### Cut Parameters - Optimized Toolpaths

As stepdowns are calculated, the dynamic roughing motion clears the material in the -Z direction, avoiding any islands.

If you activate step up cuts in the toolpath's Cut Parameters, Mastercam calculates additional slices in the +Z direction to remove any material left behind between the step downs—for example, islands or large scallops on angled walls.

The step up cuts use a 2D high speed dynamic rest motion to clear leftover material. Step up cuts are ordered between stepdowns.

#### Optimize cut order

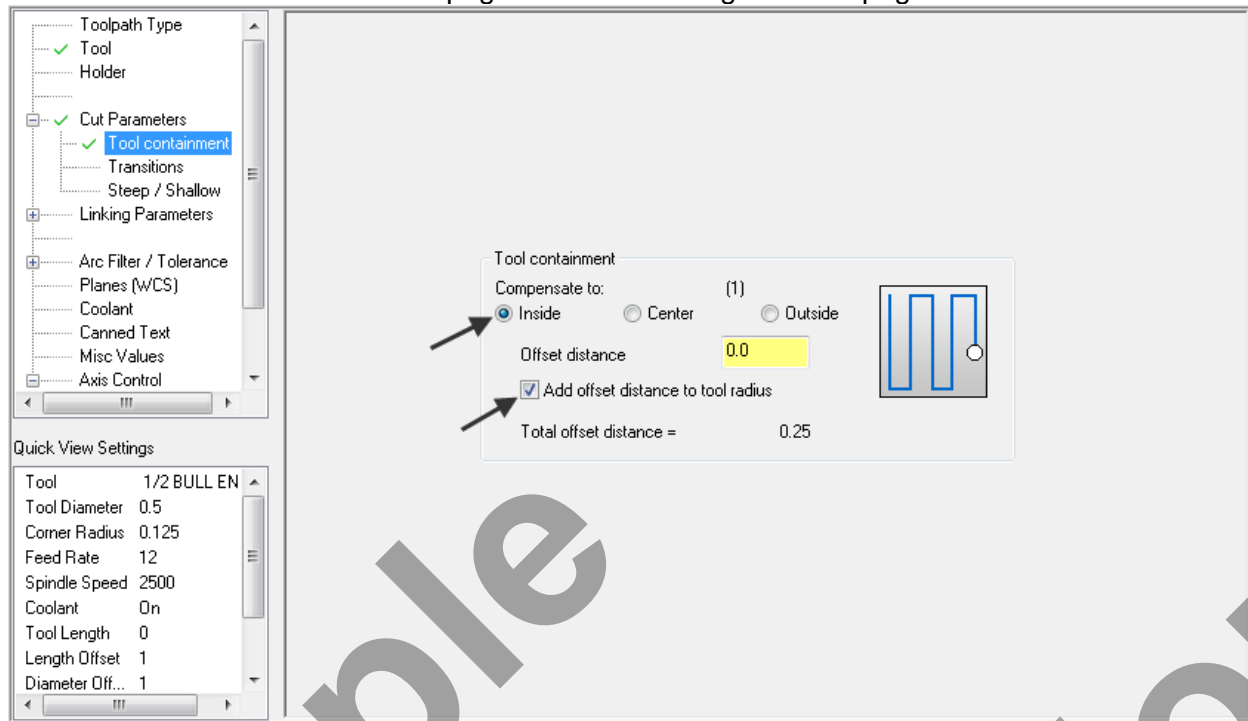
Defines the cut order Mastercam applies to different cutting passes in the toolpath.

**By Depth:** Mastercam machines all cut passes z-level by z-level.

**Next closest:** Mastercam moves to the closest cut from where it finished the previous cut.

**By pocket:** Mastercam first machines all the stepdowns, moving from pocket to pocket. After all stepdowns on a z-level are machined to completion, Mastercam machines the step-ups by next closest, in the safest cut order.

17. Select the **Tool containment** page and make changes to this page as shown below.



### Tool Containment

Specify the behaviour of the containment boundary.

#### Compensate to Inside

Keep the tool inside this boundary.

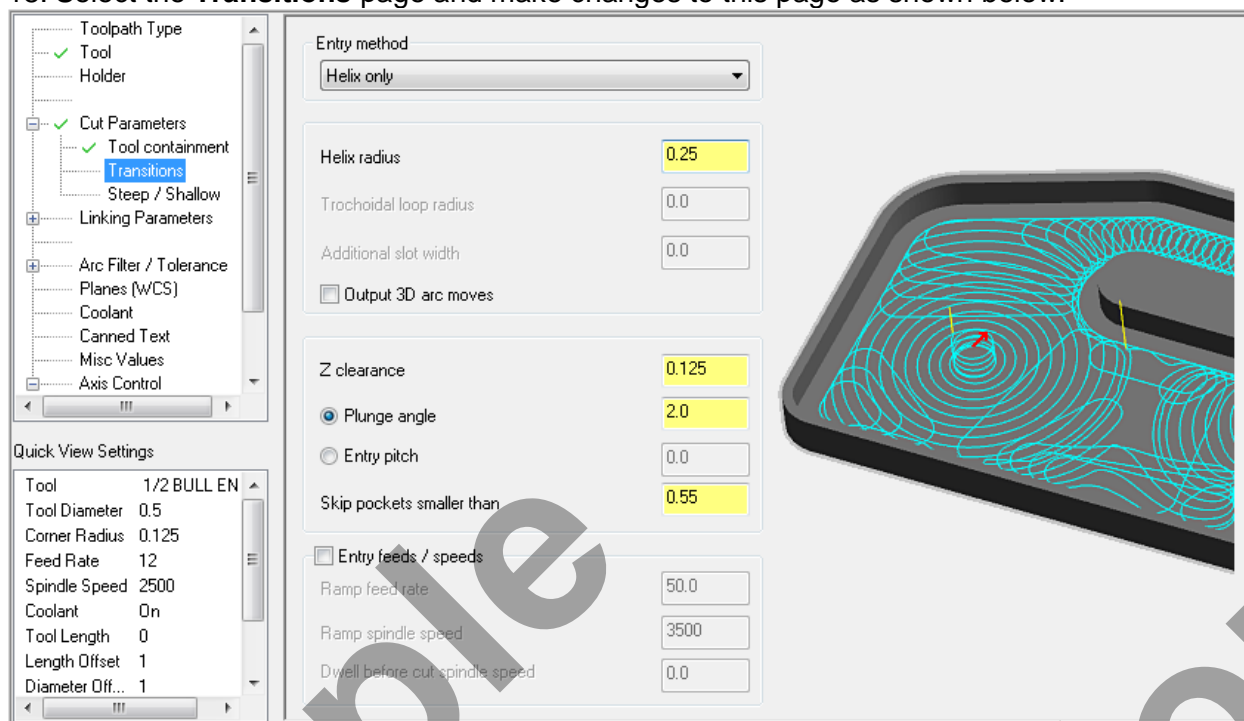
#### Compensate to Center

Keep the tool centerline inside the boundary (allow up to half of the tool to exit the boundary).

#### Compensate to Outside

Allow the entire tool to exit the boundary but keep the tool edge in contact with the boundary.

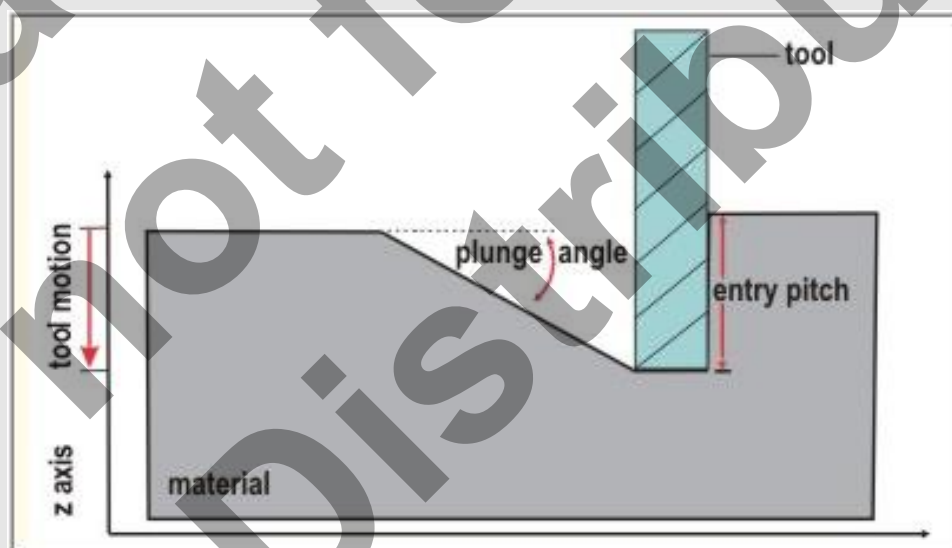
18. Select the **Transitions** page and make changes to this page as shown below.



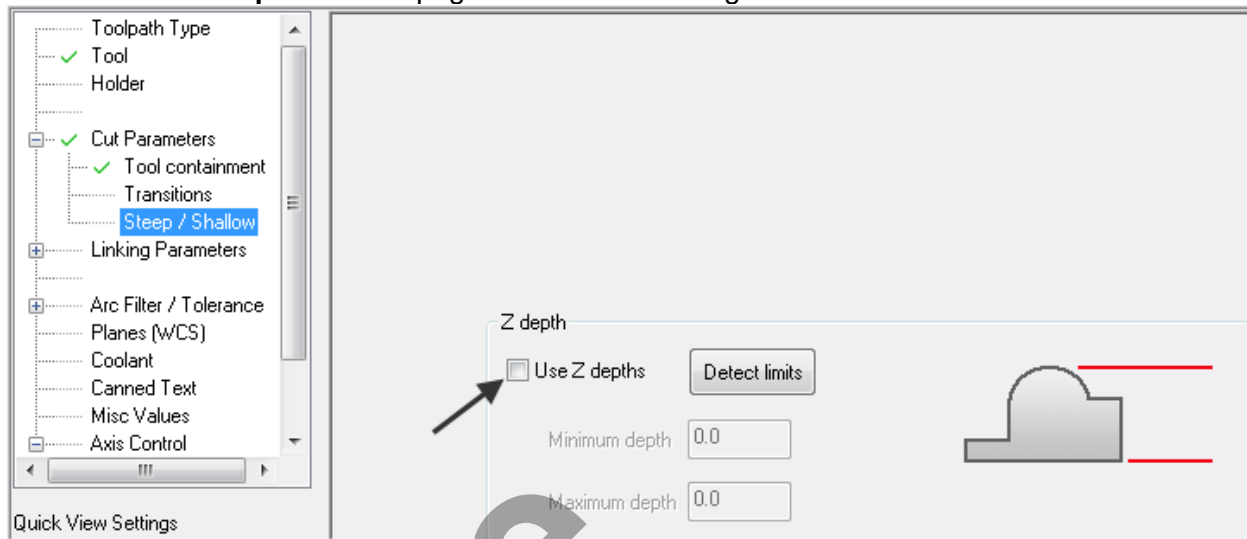
### Transitions

Use this page to configure an entry method for the toolpath. This defines how and where the tool enters the part.

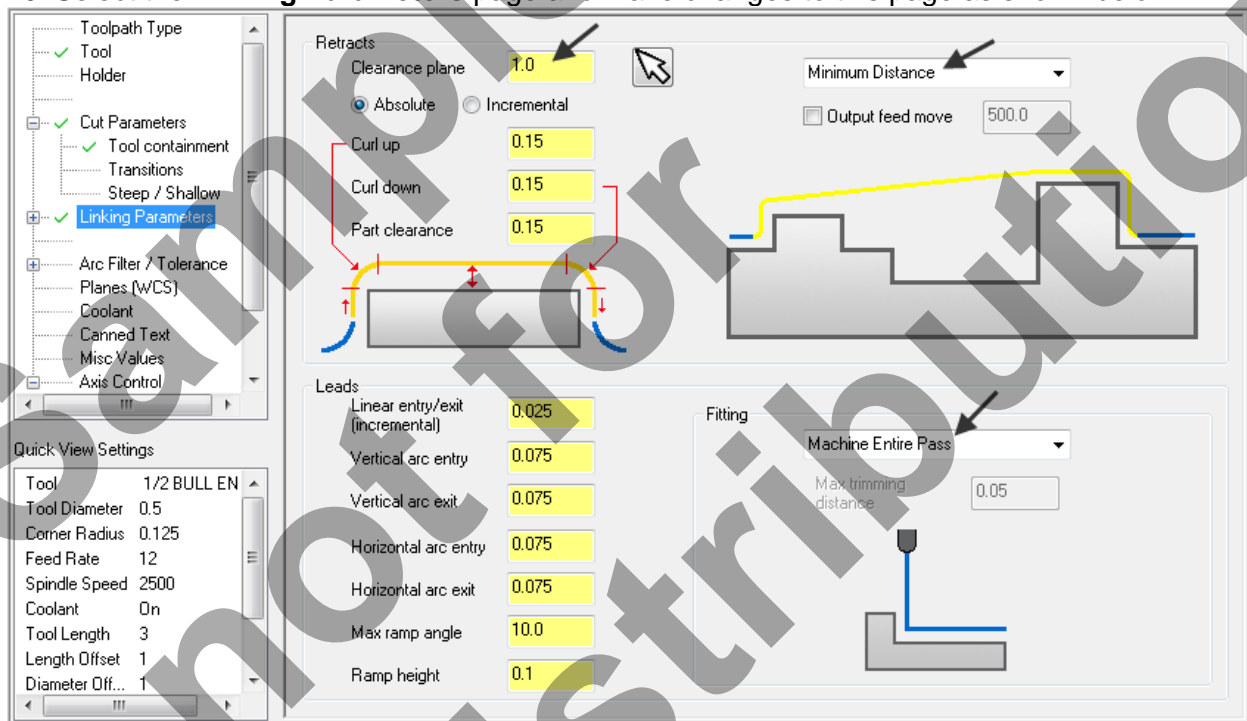
Other parameters let you further refine the entry moves, and set entry feeds and speeds.



19. Select the **Steep / Shallow** page and ensure nothing is activated here.



20. Select the **Linking Parameters** page and make changes to this page as shown below.



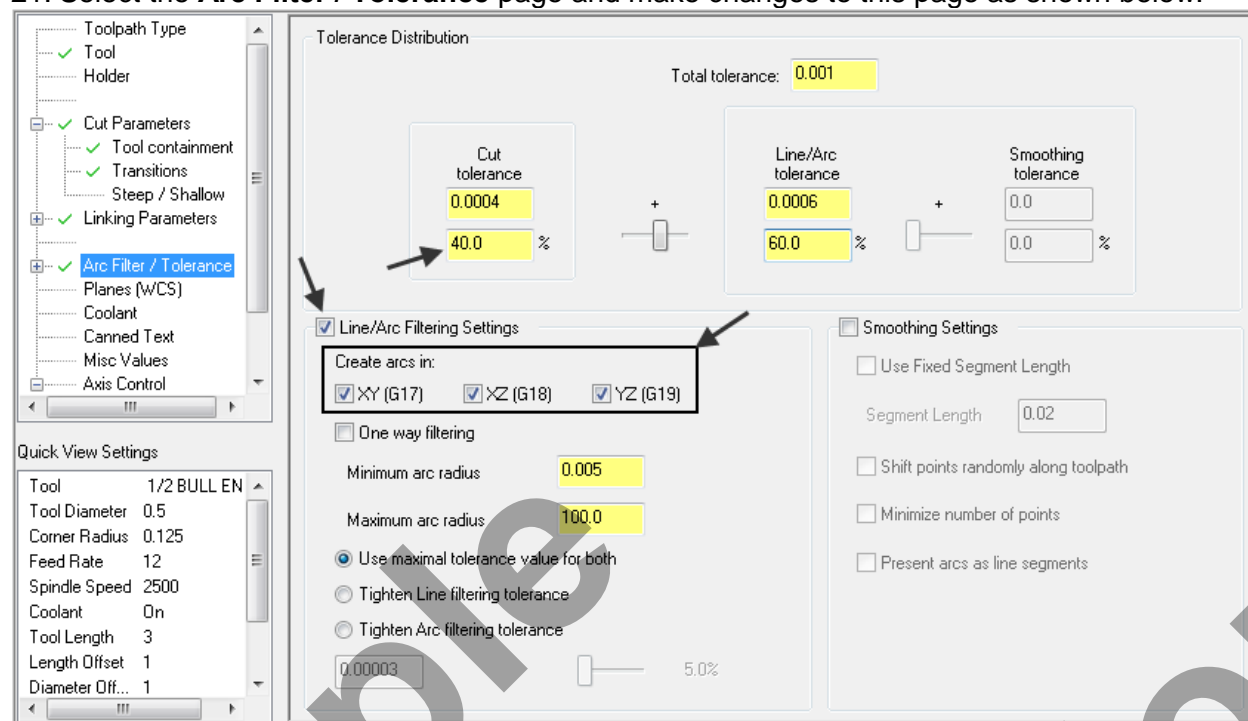
### Linking Parameters

Use this page to create the **links between the cutting passes**. In general, you can think of linking moves as air moves when the tool is not in contact with the part, compared to cutting moves which are configured on the toolpath's Cut parameters page.

**First**, select a retract method. This determines how the tool will move between the end of one pass and the beginning of another. Then, use the **Leads fields** to control how 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.

Finally, select a **Fitting option** to control how the entry and exit arcs will be fit into each pass.

21. Select the **Arc Filter / Tolerance** page and make changes to this page as shown below.



### Arc Filter/Tolerance

Use this page to control toolpath tolerances. Typically, this involves several sets of variables. Mastercam uses the values you enter here to convert the toolpath—originally created using G1, G2, G3 motions—to a refined set of "smoothed" G1 motions wherever possible, and within the tolerances you specify.

### Tolerances Distribution

The Tolerances Distribution fields display the total tolerance and the formula used to derive it. Adjust the total tolerance by entering a value in the Total tolerance field or by changing the values of the Cut, Line/Arc and/or Smoothing tolerances.

### Line/Arc Filtering and Smoothing

Toolpath filtering lets you replace multiple very small linear moves — within the filter tolerance — with single arc moves to simplify the toolpath. Smoothing distributes a toolpath's node points, avoiding the clustering and grouping of points that can cause marks and other imperfections.

Use the different checkboxes to enable or restrict arc creation in specific planes.

Enter minimum and maximum arc radius values to control the size of the arcs Mastercam creates in the filtered toolpath.

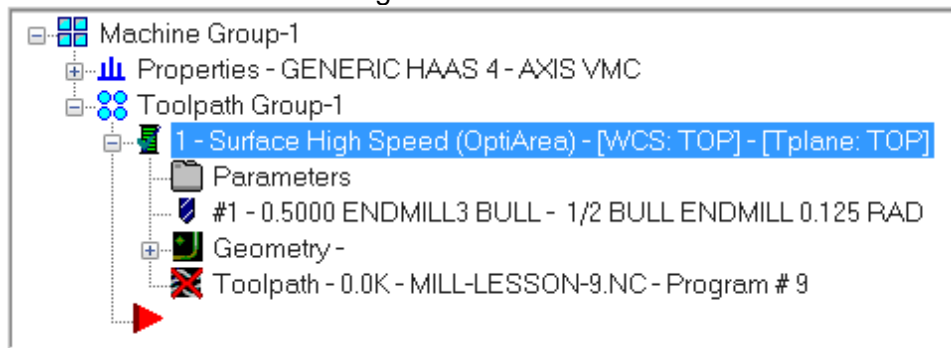
Define the smoothing tolerance by shifting, removing and/or adding node points along your toolpath.

22. Turn the **Coolant Flood** on.

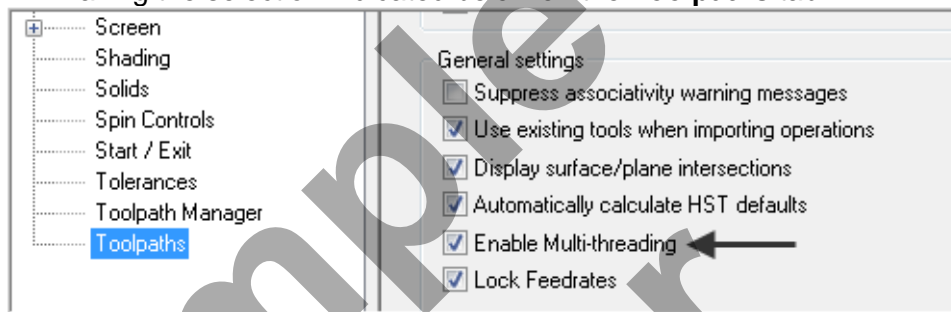
23. Select the OK button  to complete this toolpath.



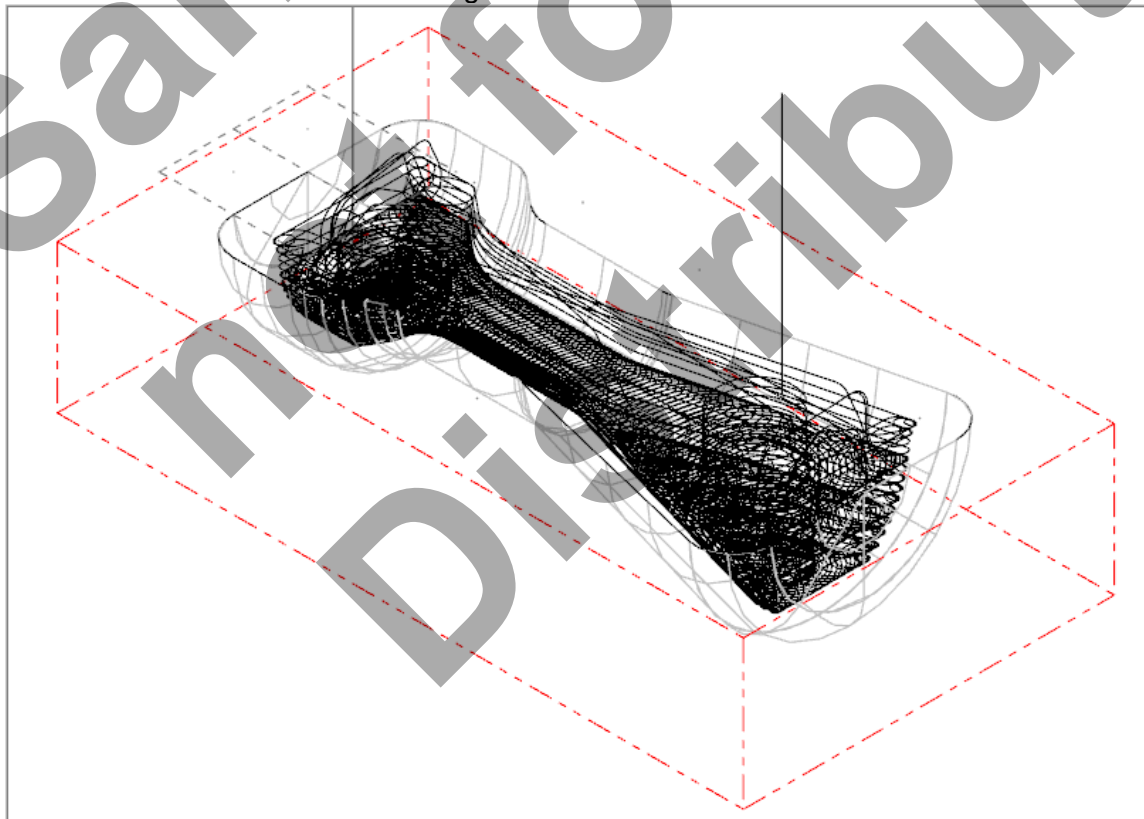
24. It may take a while for Mastercam to create the toolpath. Mastercam's multi-threading functionality will calculate the toolpath while you continue to work. The green spool of thread indicates threading is active.



25. If threading is not active, you can turn it on by selecting **Settings>Configuration...** then making the selection indicated below on the **Toolpaths** tab.



The screen should look like the image below:





### TASK 9:

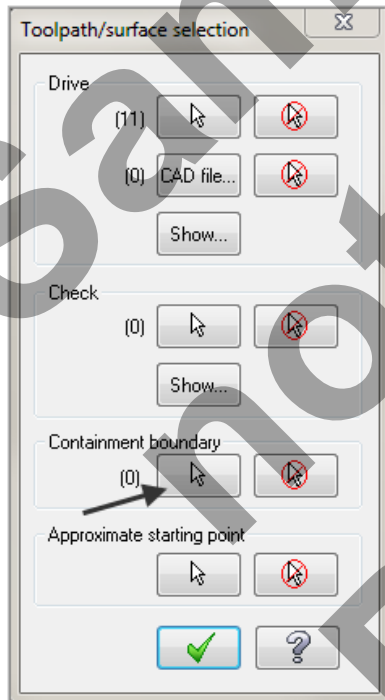
#### FINISH THE POCKET USING SURFACE HIGH SPEED HYBRID

- In this task you will use a 0.5 diameter Ball end mill to finish the pocket.
- Mastercam's Hybrid finishing toolpath addresses steep and shallow areas utilizing both scallop and constant Z approaches in a single toolpath.
- The toolpath switches seamlessly between both methods and cuts in a logical optimized order.
- Most of the parameters you define for the toolpath are similar or the same as other surface high speed finishing toolpaths. However, the Step group of settings on the Cut Parameters page are unique. Use them to control how Mastercam defines steep and shallow areas.

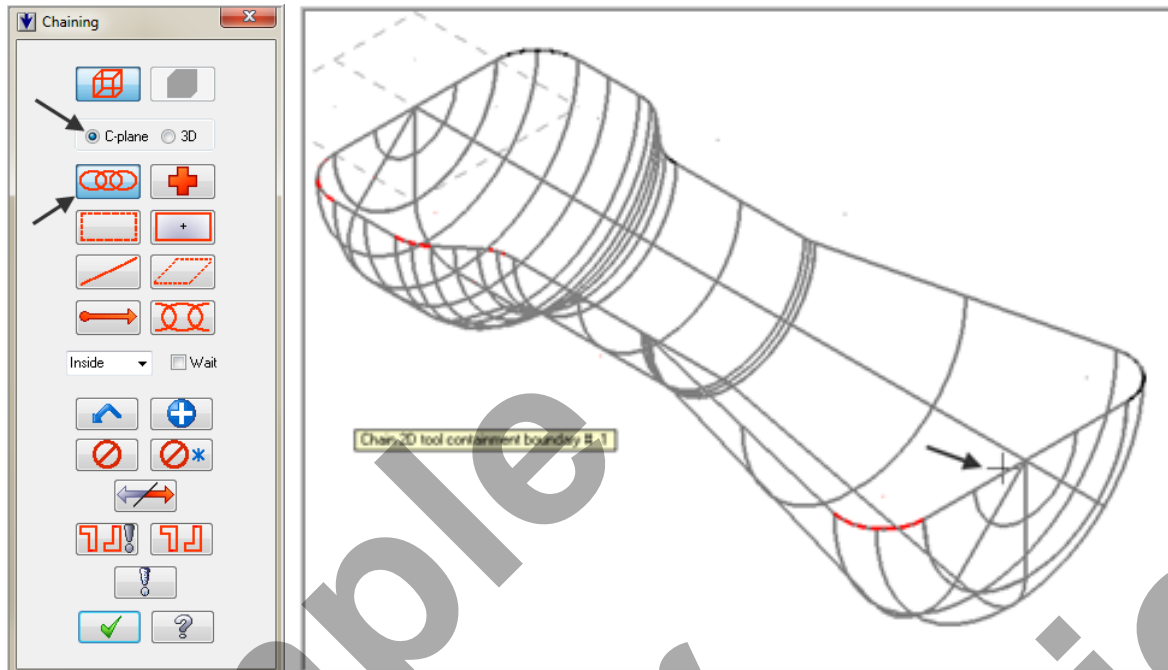
1. From the menu bar select **TOOLPATHS>Surface High Speed>Hybrid...**
2. You are first prompted to **Select Drive surfaces**, select the **All** button on the **General Selection** ribbon bar as shown below:



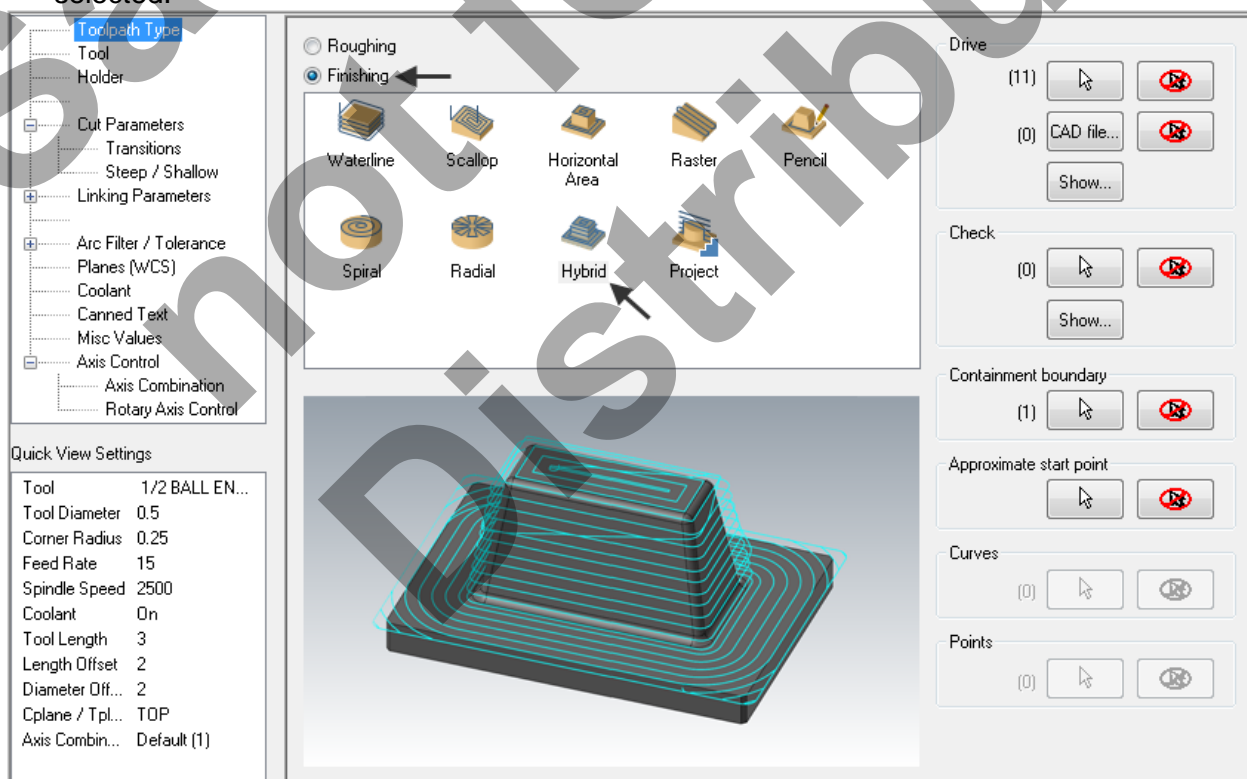
3. The **Select All** dialog box appears on the screen.
4. Click on the OK icon  to complete and exit this feature.
5. To move onto the next step you now need to pick the **End Selection** icon .
6. Select the **Containment boundary** button from the **Toolpath/surface selection** dialog box.



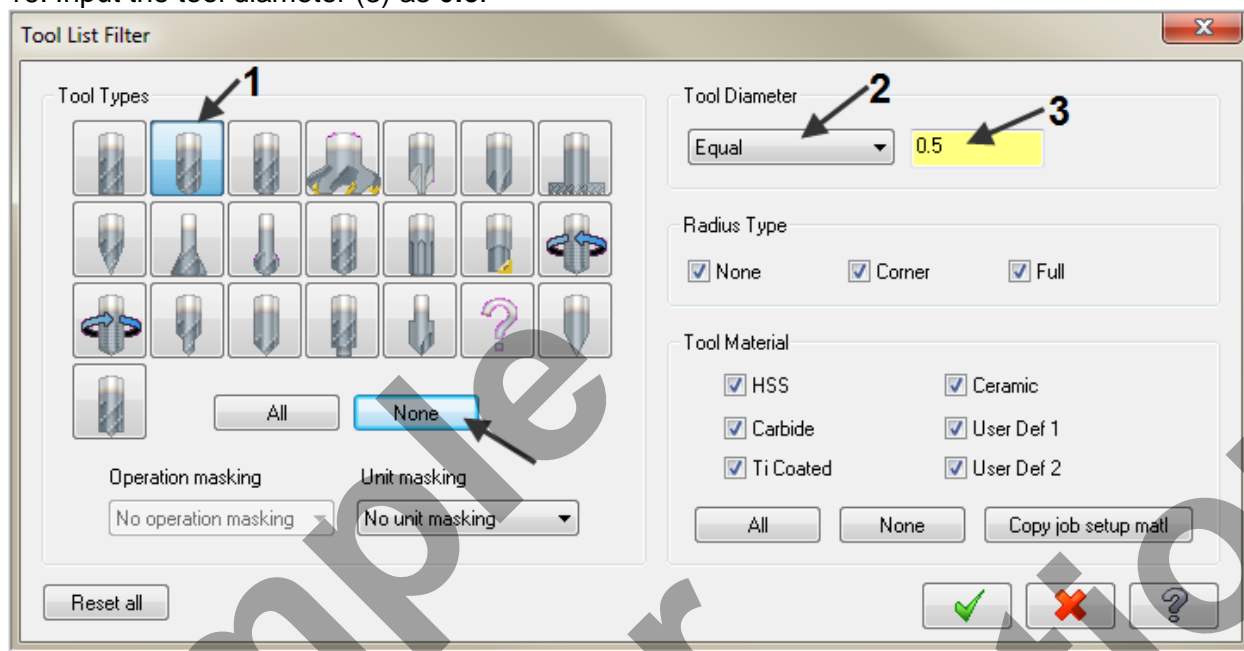
- On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Chain 2D tool containment boundary #1**. Ensure the **C-plane** radio button is activated. Toolpath display has been turned off for clarity **ALT + T**. Now select the line as shown below:




- After the boundary has been successfully chained select the OK button .
- Select the OK button  to exit the **Toolpath/surface selection** dialog box.
- On the **Toolpath Type**, ensure the **Finishing** radio button and the **Hybrid** toolpath are selected.



11. Select the **Tool** selection page, then **Select library tool...** button.
12. Select the **Filter** button on the right side of the **Tool selection** dialog box.
13. Select the **None** button in the Tool Types section.
14. Click on the **Endmill2 Sphere** type icon as shown in the picture (1) below:
15. Select the drop down arrow in the **Tool diameter** (2) field and set it to **Equal**.
16. Input the tool diameter (3) as **0.5**.



17. Select the OK button  to exit.
18. Select the 0.5 ball end mill.



19. Select the OK button  to complete the selection of this tool.

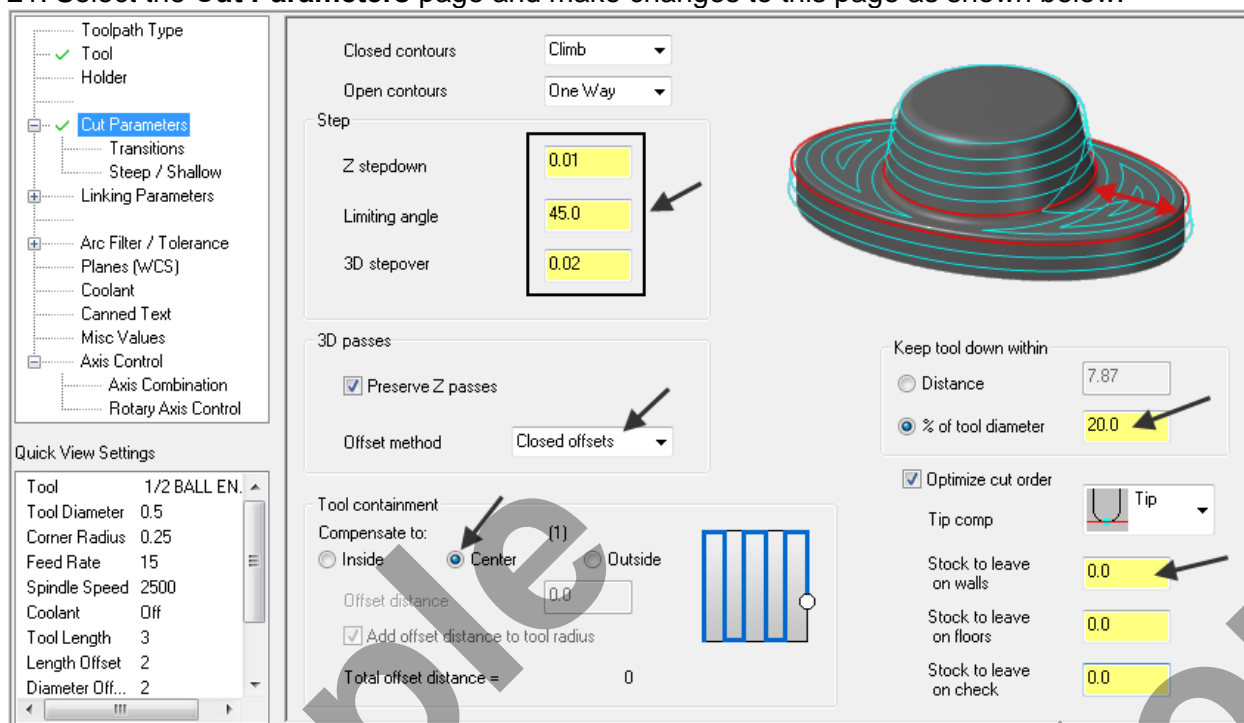
20. Make changes to the **Tool** page as shown below:

The screenshot shows the 'Tool' page in a CAD software interface. The left sidebar contains a tree view with the following items: Toolpath Type, Tool (selected), Holder, Cut Parameters, Transitions, Steep / Shallow, Linking Parameters, Arc Filter / Tolerance, Planes (WCS), Coolant, Canned Text, Misc Values, Axis Control, Axis Combination, and Rotary Axis Control. Below the tree is a 'Quick View Settings' panel showing the following values: Tool: 1/2 BALL EN., Tool Diameter: 0.5, Corner Radius: 0.25, Feed Rate: 15, Spindle Speed: 2500, Coolant: On, Tool Length: 3, Length Offset: 2, and Diameter Off...: 2. The main area contains a table of tools with the following data:

#	Tool Name	Dia.	Cor. rad.
1	1/2 BULL ENDMILL 0.125 RAD	0.5	0.125
2	1/2 BALL ENDMILL	0.5	0.25

Below the table is a 'Select library tool...' button and a 'Filter Active' checkbox. To the right of the table is a 'Right-click for options' button. Below the table is a 'Tool inspection / change' section with a 'Force retract every' dropdown set to 400.0 Inches and a 'To batch' checkbox. The right side of the page contains various tool parameters: Tool dia: 0.5, Corner radius: 0.25, Tool name: 1/2 BALL ENDMILL, Tool #: 2, Len. offset: 2, Head #: 0, Dia. offset: 2, Spindle direction: CW, Feed rate: 15.0, Spindle speed: 2500, FPT: 0.0015, SFM: 327.2251, Plunge rate: 15.0, Retract rate: 50.0, Force tool change checkbox, and Rapid Retract checkbox. A 'Comment' text area is at the bottom right.

21. Select the **Cut Parameters** page and make changes to this page as shown below.



### Cut Parameters

Use this page to configure the cutting passes for a hybrid toolpath. This is a finishing toolpath which generates waterline-style cut passes for steep regions and scallop cut passes for shallow regions. Mastercam switches smoothly between both styles to cut in a logical optimized order.

#### Z stepdown

Defines the constant Z distance between adjacent stepdowns. Mastercam uses these stepdowns in combination with the Limiting Angle and 3D stepover to calculate cut passes for the hybrid toolpath.

Mastercam first slices the entire model into sections defined by the Z Stepdown distance. It then analyzes the drive surface's slope transition between each stepdown along the specified

#### Limiting Angle.

If the drive surface's slope transition within the stepdown distance is less than the applied limiting angle, the hybrid toolpath considers it steep and a single 2D waterline cutting pass is generated. Otherwise, the stepdown is defined as shallow. Mastercam uses the 3D stepover value to create 3D scallop cut passes along the shallow slope.

#### 3D stepover

Defines the spacing between 3D scallop-style cutting passes in shallow stepdowns.

#### Preserve Z passes

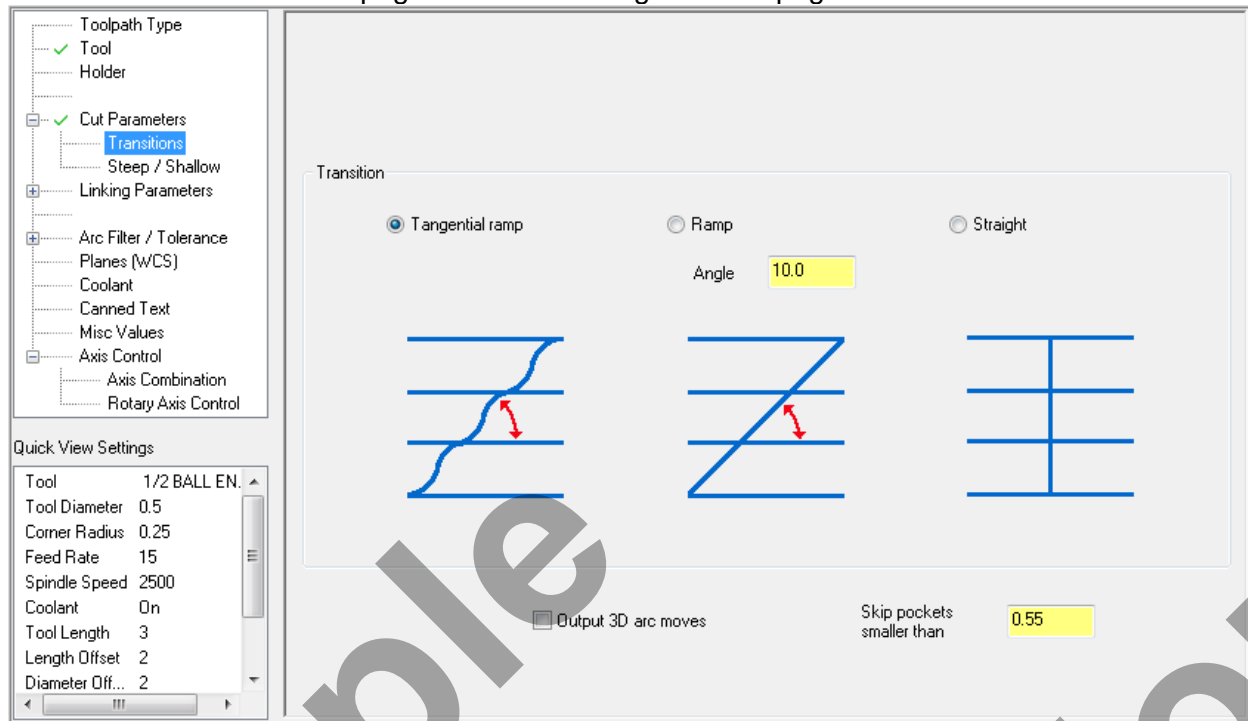
Select to maintain Z passes in steep areas. Passes in shallow areas are calculated based on the Offset method. Deselect to calculate collapse motion across the entire part when a shallow region is encountered.

#### Offset method

Select a method for handling the boundaries between steep and shallow areas from the drop-down.

**Closed offsets:** The boundaries are treated as closed chains when calculating the 3D collapse motion.

22. Select the **Transitions** page and make changes to this page as shown below.



### Transition Page - Hybrid

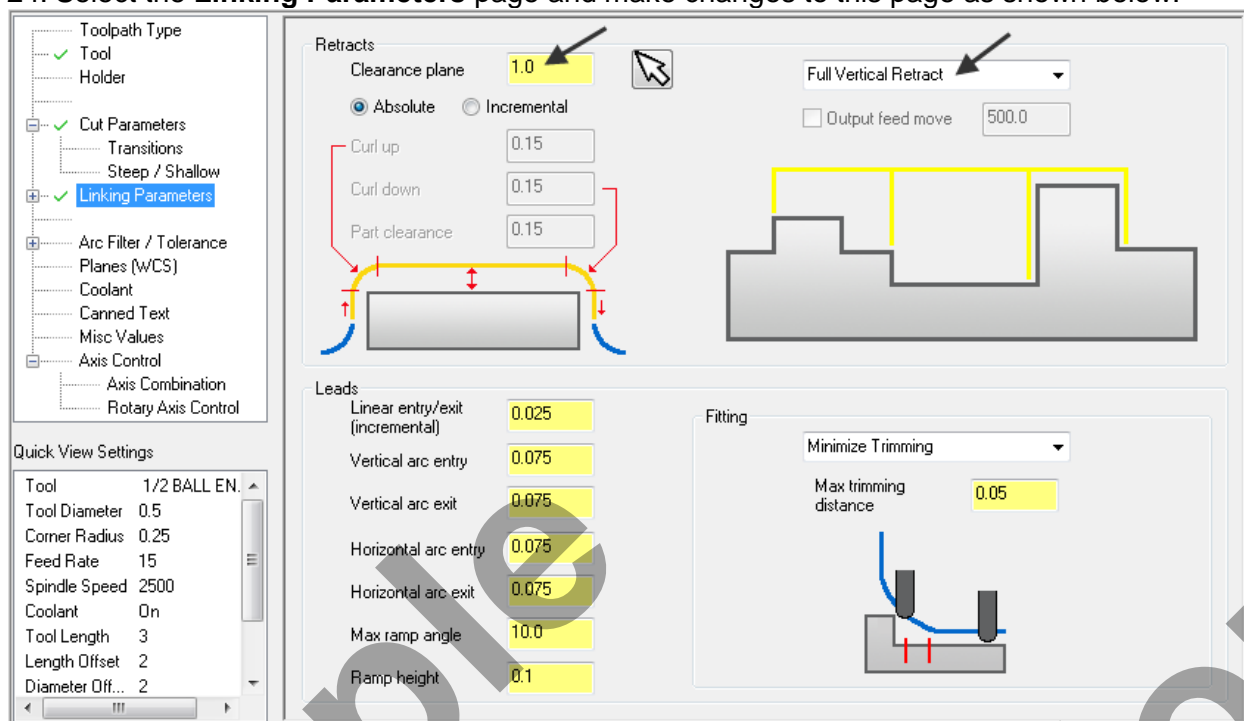
Use this page to configure the entry moves that the tool will make as it transitions to new Z levels. These moves control the transition to a new set of cuts on a different Z level. toolpath.

Select **Tangential ramp** to create 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.

23. Select the **Steep / Shallow** page and ensure nothing is activated here.



24. Select the **Linking Parameters** page and make changes to this page as shown below.

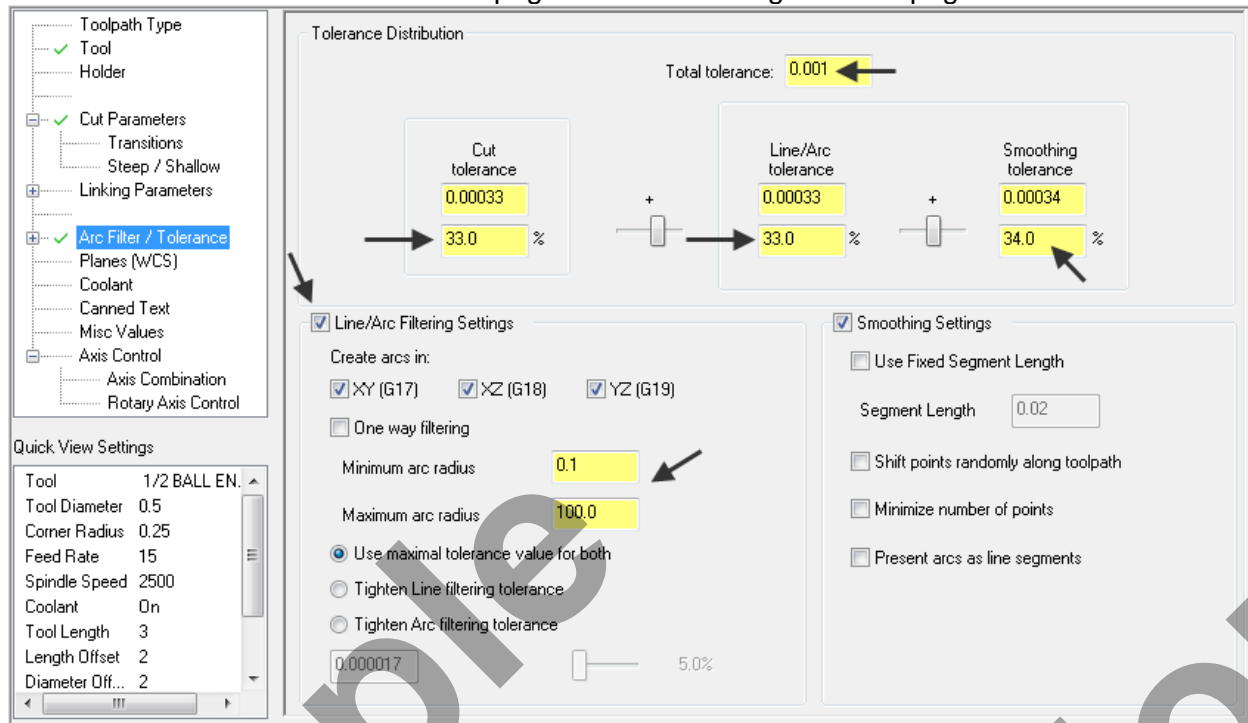


### Linking Parameters

Use this page to create the **links between the cutting passes**. In general, you can think of linking moves as air moves when the tool is not in contact with the part, compared to cutting moves which are configured on the toolpath's Cut parameters page.



25. Select the **Arc Filter / Tolerance** page and make changes to this page as shown below.



### Arc Filter/Tolerance

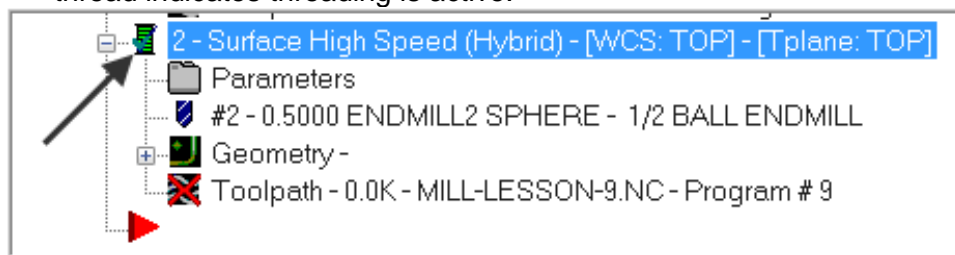
Use this page to control toolpath tolerances. Typically, this involves several sets of variables. Mastercam uses the values you enter here to convert the toolpath—originally created using G1, G2, G3 motions—to a refined set of "smoothed" G1 motions wherever possible, and within the tolerances you specify.

Define the smoothing tolerance by shifting, removing and/or adding node points along your toolpath.

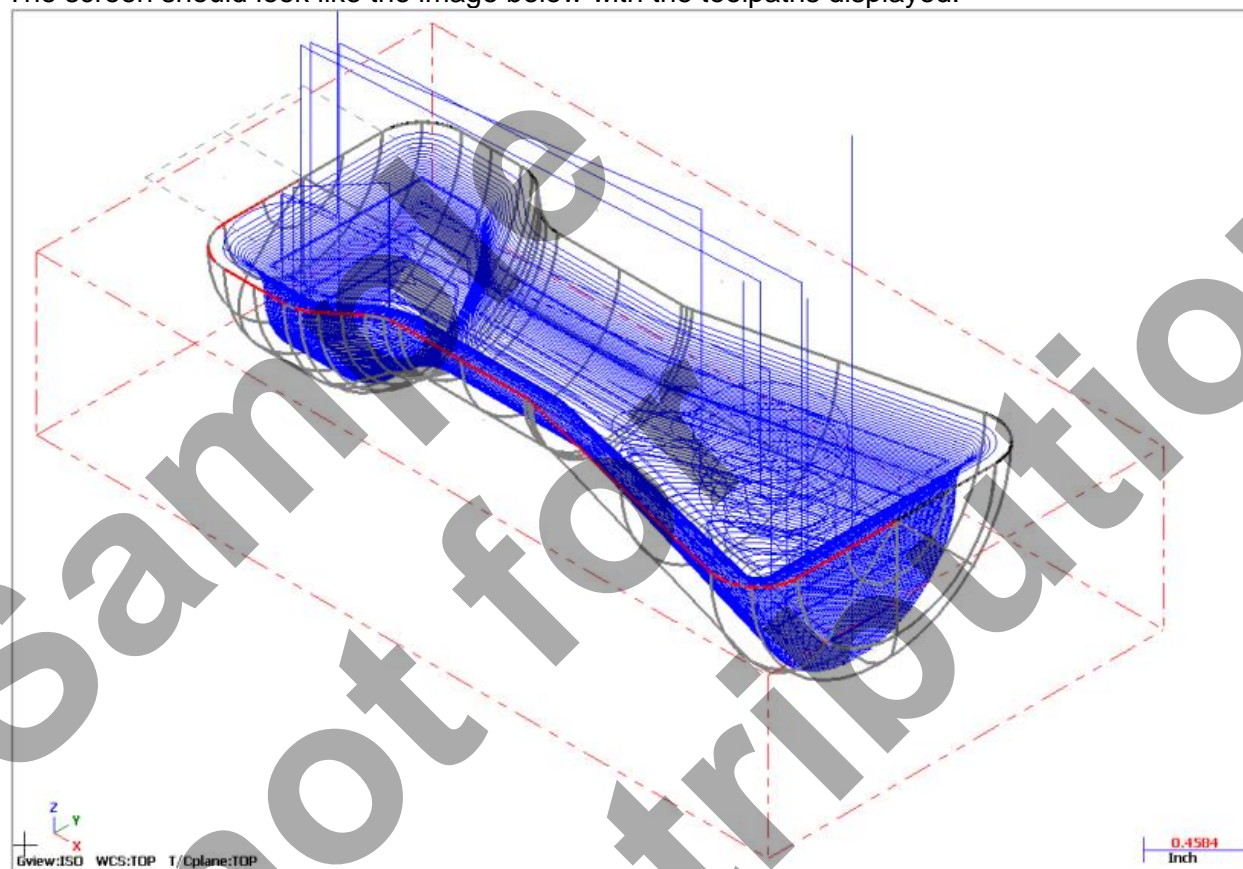
26. Turn the **Coolant Flood** on.

27. Select the OK button  to complete this toolpath.

28. It may take a while for Mastercam to create the toolpath. Mastercam's multi-threading functionality will calculate the toolpath while you continue to work. The green spool of thread indicates threading is active.



The screen should look like the image below with the toolpaths displayed:



## TASK 10: VERIFY THE TOOLPATH

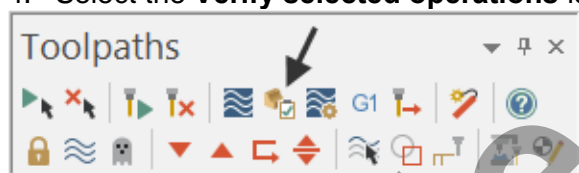
1. Set the graphics view to the Isometric View by using the toolbar icon at the top of the screen.



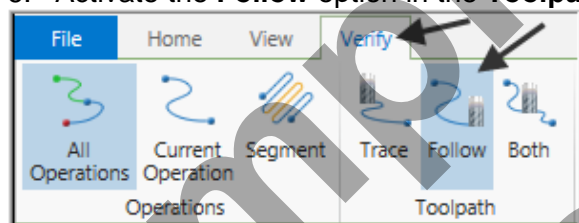
2. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen.
3. In the **Toolpaths Manager** pick all the operations to verify by picking the **Select All** icon



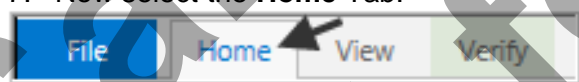
4. Select the **Verify selected operations** icon shown below:



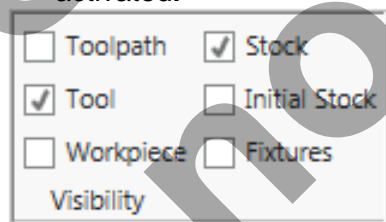
5. **Maximize** the Backplot/Verify window if required.
6. Activate the **Follow** option in the **Toolpath** section of the **Verify** tab.



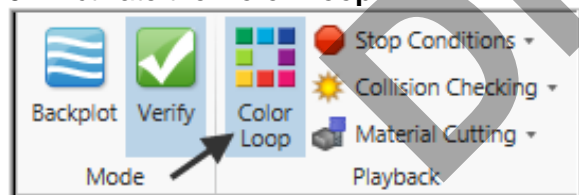
7. Now select the **Home** Tab.



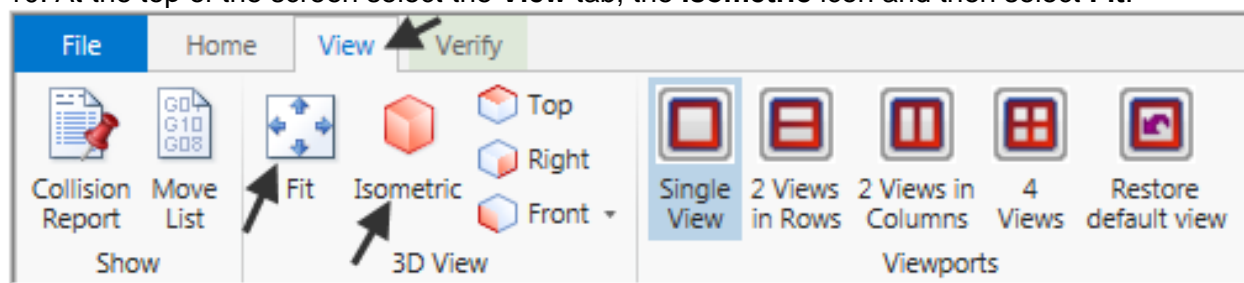
8. Activate the options shown below in the **Visibility** section of the **Home** tab. **Initial Stock** not activated.



9. Activate the **Color Loop**.



10. At the top of the screen select the **View** tab, the **Isometric** icon and then select **Fit**.



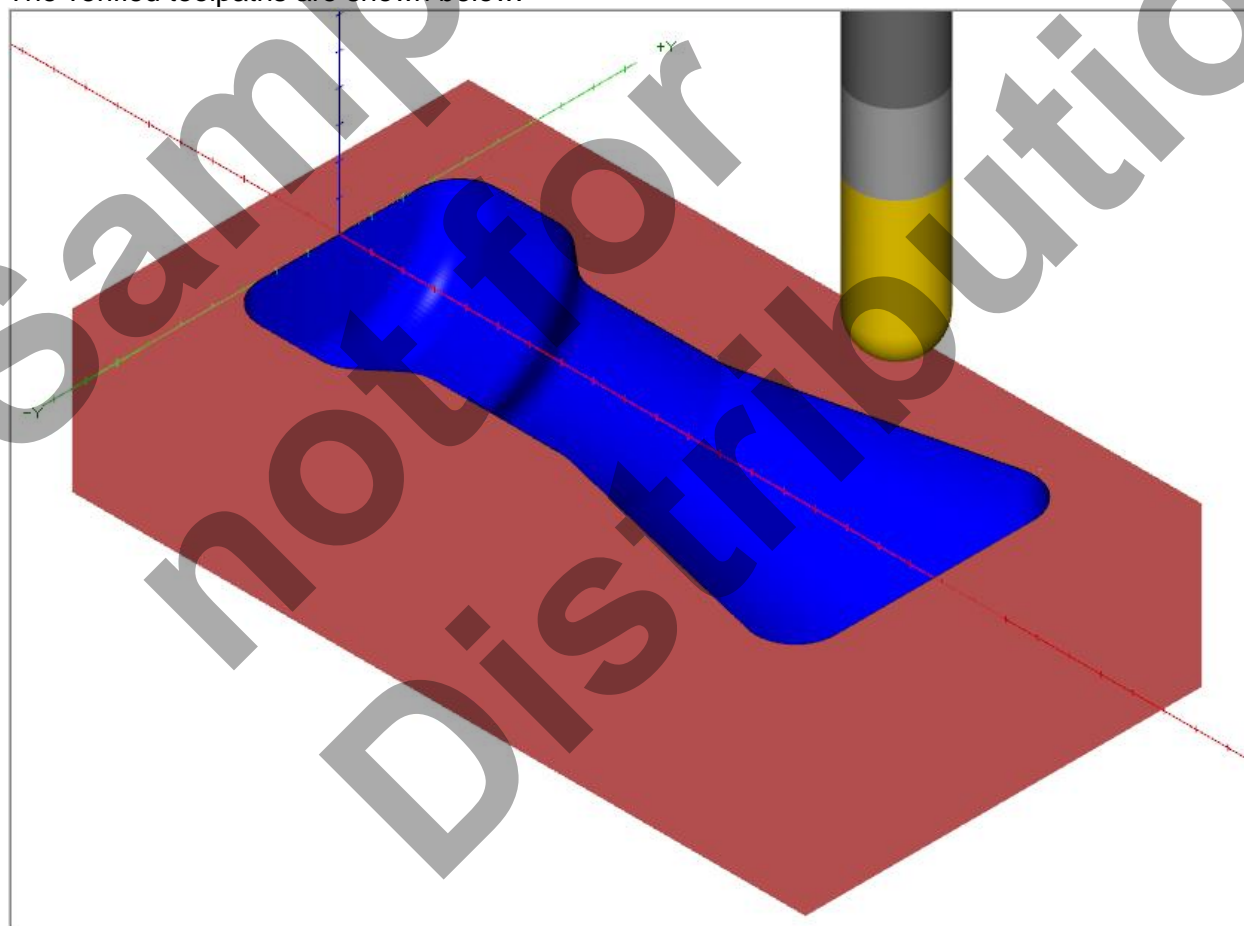
11. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.




12. Now select the **Play Simulation** button to review the toolpaths.



The verified toolpaths are shown below:



13. Select the **Close** button  in the top right hand corner to exit Verify.

## TASK 11: SAVE THE UPDATED MASTERCAM FILE

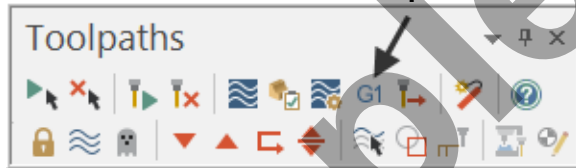
1. Select the save icon from the toolbar.

## TASK 12: POST AND CREATE THE CNC CODE FILE

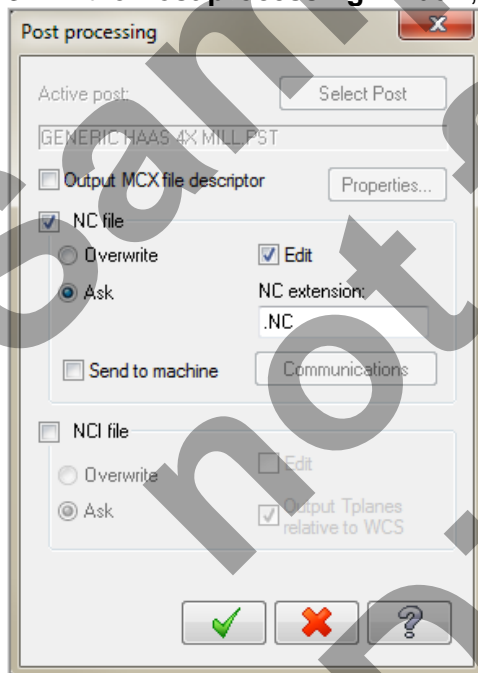
### Please Note:

Users of the Mastercam **Home Learning Edition** (HLE) will not be able to Post and Create the CNC code file.

1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpaths manager.
2. Select the **Post selected operations** icon **G1** from the **Toolpaths manager**.



3. In the **Post processing** window, make the necessary changes as shown below:



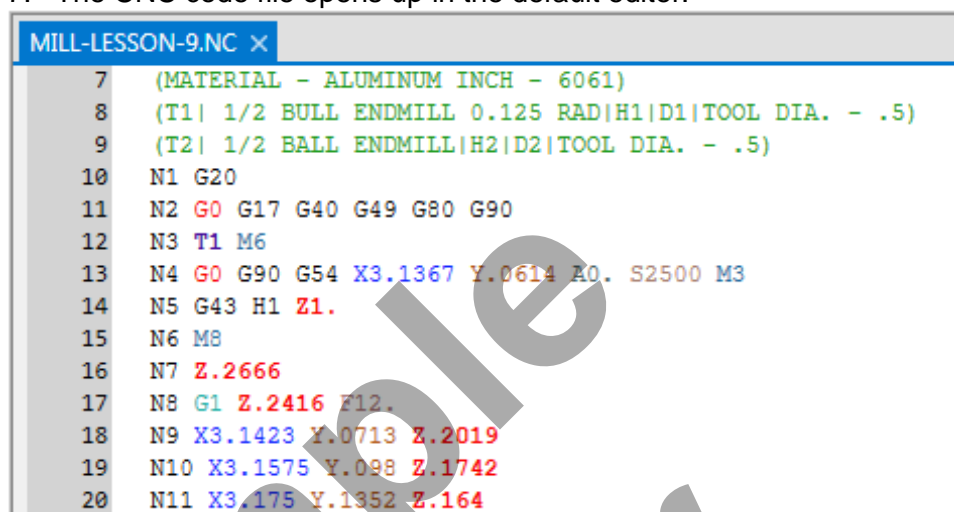
4. Select the **OK** button  to continue.

5. Ensure the same name as your Mastercam part file name is displayed in the File name window as shown below:



File name: MILL-LESSON-9  
Save as type: NC Files (\*.NC)

6. Select the **Save** button.
7. The CNC code file opens up in the default editor:



```
MILL-LESSON-9.NC x
7 (MATERIAL - ALUMINUM INCH - 6061)
8 (T1| 1/2 BULL ENDMILL 0.125 RAD|H1|D1|TOOL DIA. - .5)
9 (T2| 1/2 BALL ENDMILL|H2|D2|TOOL DIA. - .5)
10 N1 G20
11 N2 G0 G17 G40 G49 G80 G90
12 N3 T1 M6
13 N4 G0 G90 G54 X3.1367 Y.0614 Z0. S2500 M3
14 N5 G43 H1 Z1.
15 N6 M8
16 N7 Z.2666
17 N8 G1 Z.2416 F12.
18 N9 X3.1423 Y.0713 Z.2019
19 N10 X3.1575 Y.098 Z.1742
20 N11 X3.175 Y.1352 Z.164
```

8. Select the  in the top right corner to exit the CNC editor.

**This completes Mill-Lesson-9.**

MILL-LESSON-9 EXERCISES

