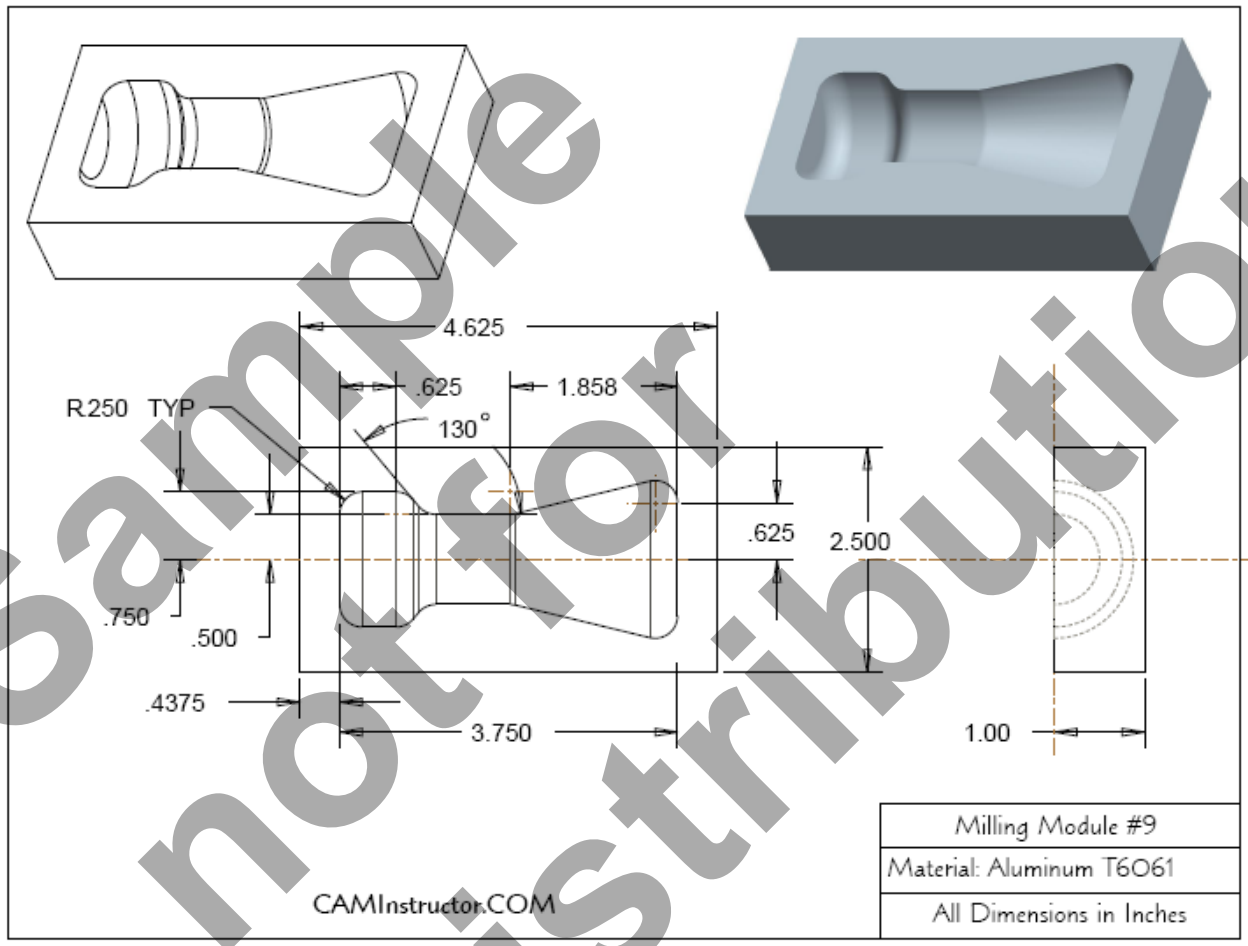
Solidworks and Mastercam X6 Training Guide



Lesson-9-Toolpaths

camInstructor

Objectives

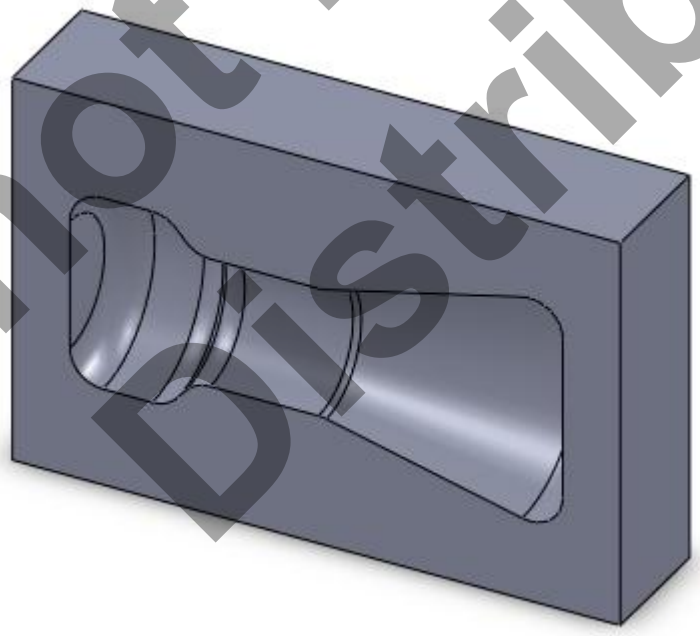
You will import the CAD file for Lesson-9 using **SolidWorks, Inventor, Pro-E or Catia Software**. This Lesson covers the following topics:

➲ **Importing the Solid Model into Mastercam:**
Setting the Environment.
Importing the Solid Model into Mastercam.
Transforming the part to the proper origin.

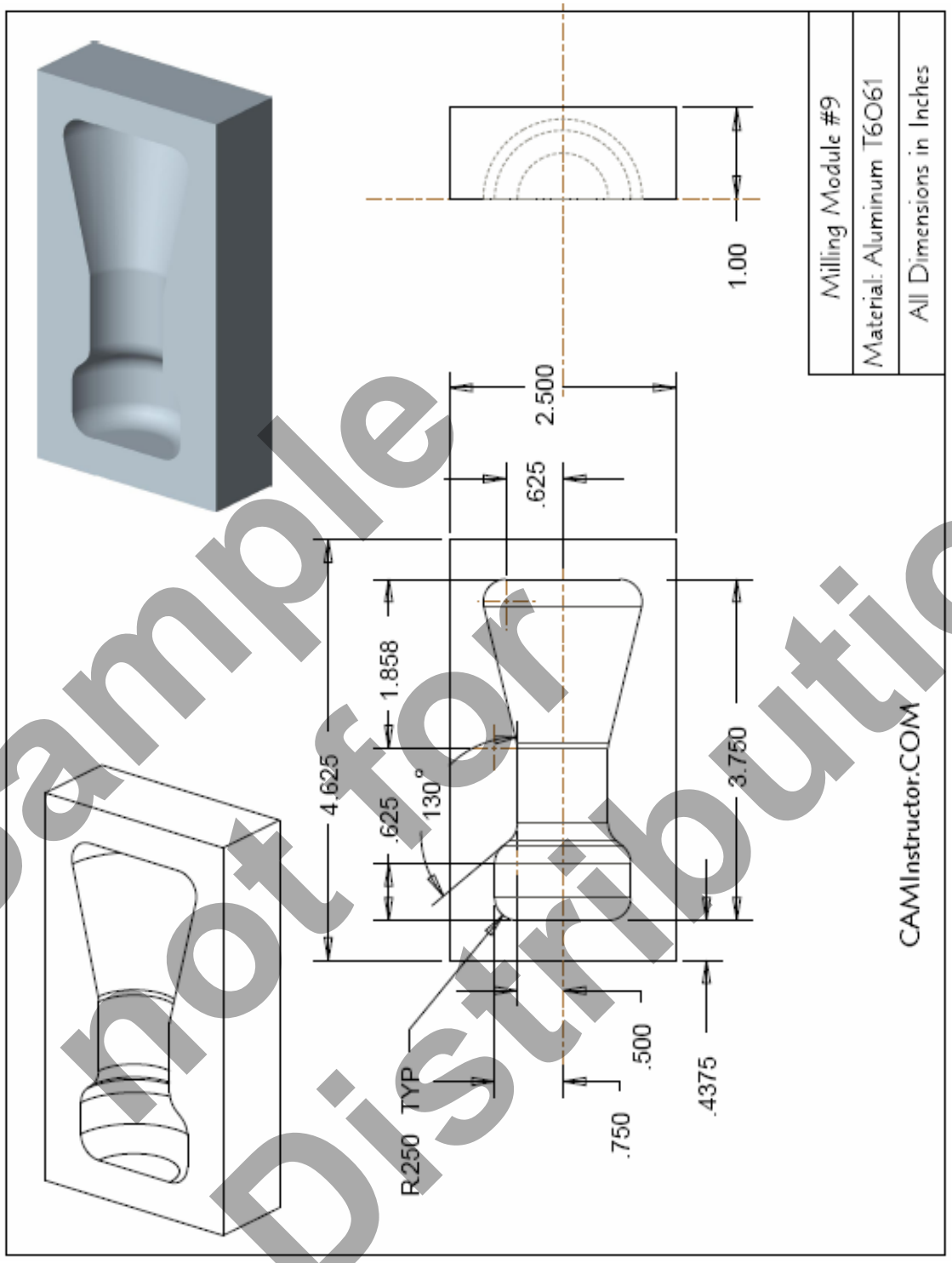
➲ **Establish Stock Setup settings:**
Stock size using Bounding Box.
Material for the part.
Feed calculation.

➲ **Generate a 3-dimensional milling toolpath consisting of:**
Surface Pocket.
Surface High Speed Waterline.

➲ **Inspect the toolpath using Mastercam's Verify and Backplot by:**
Launch the Verify function to machine the part on the screen.
Generate the NC- code.



LESSON-9 DRAWING



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.

Tool List of MILL-LESSON-9.MCX-5

Proj./Part No.: 0			Date	: 05/25/10	
Drawing No. : 1			Customer	: -	
Prog. No. : 9			Programmer	: 1	

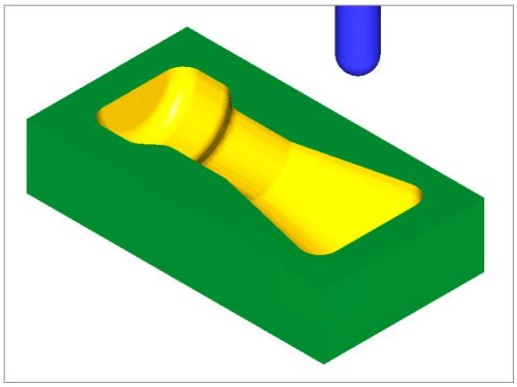
Tool type :	0.5 Endmill3 Bull 1/2 BULL ENDMILL 0.125 RAD				
Manufact.code :					
Chuck :					
Tool Number :	1			Feedrate :	12
Diameter :	0.5	RPM :	2500	Plunge feed r.:	12
Corner radius :	0.125	Tip angle :	0	Diam. offset :	1
Flute length :	1	Material :	ALUMINUM ...	Length offset :	1
Overall length:	3	No flutes :	4		

Tool type :	0.5 Endmill2 Sphere 1/2 BALL ENDMILL				
Manufact.code :					
Chuck :					
Tool Number :	2			Feedrate :	15
Diameter :	0.5	RPM :	2500	Plunge feed r.:	15
Corner radius :	0.25	Tip angle :	0	Diam. offset :	2
Flute length :	1	Material :	ALUMINUM ...	Length offset :	2
Overall length:	3	No flutes :	4		

MILL-LESSON-9 - THE CAM PROCESS

Toolpath Creation

TASK 1: Import and position the CAD file in Mastercam
TASK 2: Define the rough stock using stock setup
TASK 3: Rough the pocket using surface pocket
TASK 4: Finish the pocket using surface high speed waterline
TASK 5: Verify the toolpath
TASK 6: Post and create the CNC code file

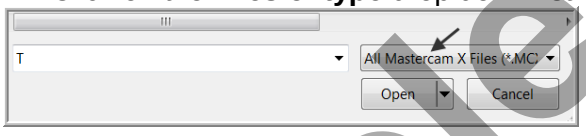


TOOLPATH TASK 1A:
SETTING THE ENVIRONMENT

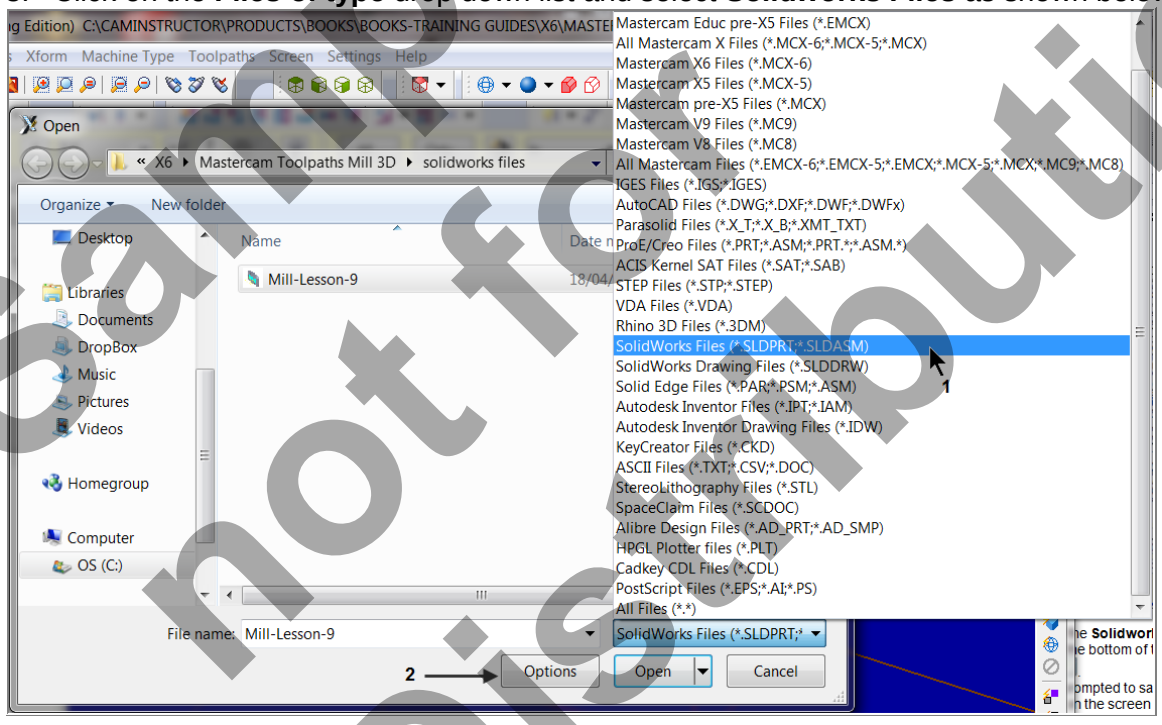
1. Start **Mastercam**
2. Before starting you should set up the grid, toolbars and machine type as outlined in the **Setting the environment** section at the beginning of this text:
    a. Set up the Grid. This will help identify the location of the origin.
    b. Customize the toolbars to machine a 2D part.
    c. Set the machine type to a Haas Vertical Spindle CNC machine.

TOOLPATH TASK 1B:
IMPORT THE CAD FILE IN MASTERCAM

1. Click on **File>Open…**
2. Click on the **Files of type** drop down list.

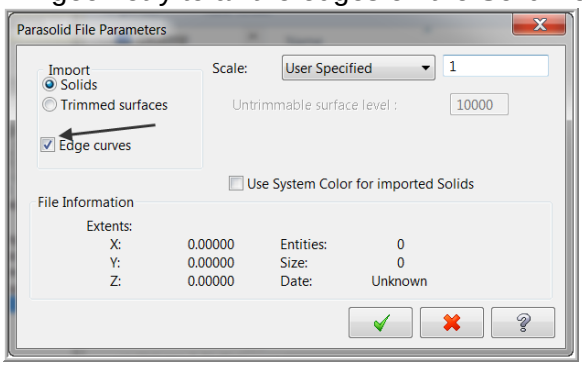


3. Click on the **Files of type** drop down list and select **Solidworks Files** as shown below **(1)**;



4. Find the location of the **Solidworks-Mill-Lesson-9.sldprt** file and click on the file.
5. Click on **Options (2)** at the bottom of the Open screen.

6. Click on the check box beside **Edge curves** as shown below. This will tell Mastercam to add geometry to all the edges on the SolidWorks model once it is imported into Mastercam.

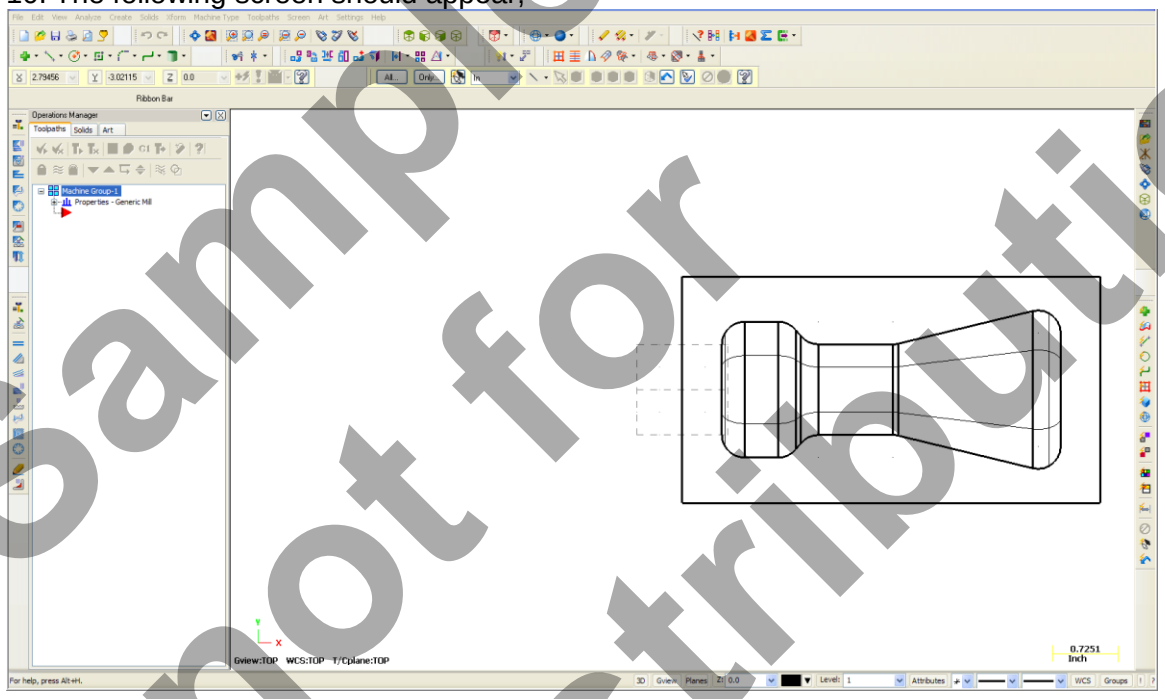


Edge Curves

Turning on Edge Curves tells Mastercam to add Lines and Arcs to all of the Edges on the Solid.

This is necessary so that you can select the geometry on the part to create the Toolpaths.

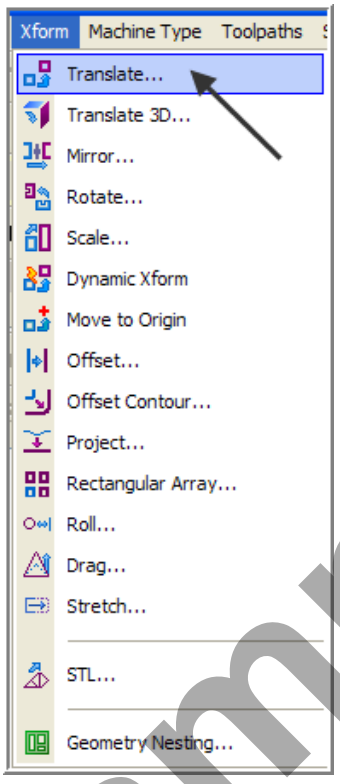
7. Click on **OK** in the **SolidWorks File Parameters** window.
8. Click on **Open.**
9. Note: You may be prompted to save the existing Mastercam File. Click on No, unless there is a Mastercam file on the screen that needs to be saved.
10. The following screen should appear;



11. Click on the **Isometric** icon  and then the **Fit**  icon.

TOOLPATH TASK 1C:
TRANSLATE THE Z0 TO THE TOP OF THE PART

1. Click on **Xform>Translate…**

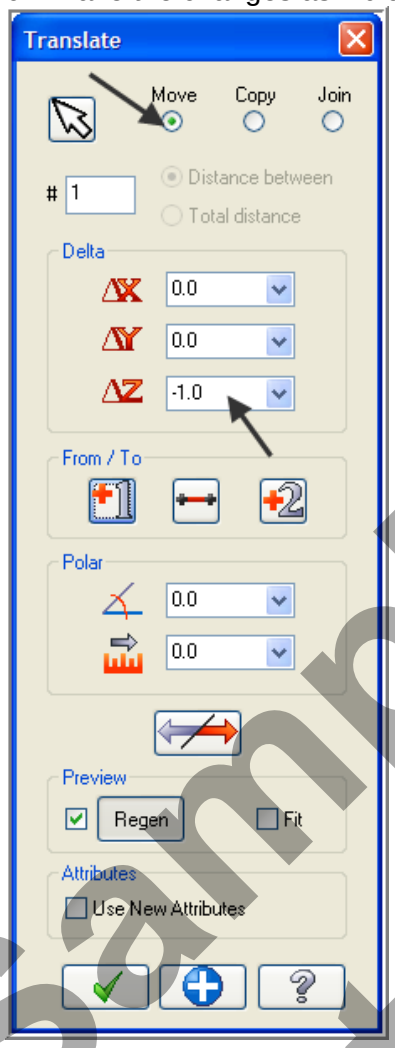


2. You are prompted to **Translate: select entitities to translate**. Click on **All…**as shown below:

3. Click on **OK**  in the **Select All** window.

4. Click on **End Selection**  .

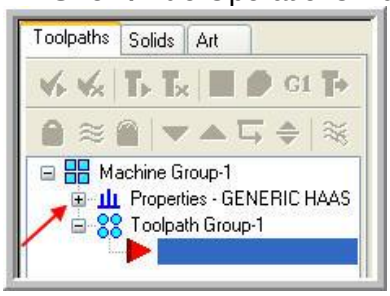
5. Toolpath Task 1b:
6. Make the changes as indicated in the **Translate** parameter screen below:



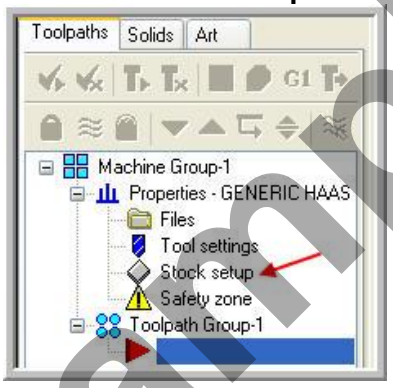
7. Click on **OK** .
8. **Z0** should be at the top of the part.
9. Click on **File>Save As…**
10. Name the file **MILL-LESSON-9**.
11. Click on **OK** .

TOOLPATH TASK 2:
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.

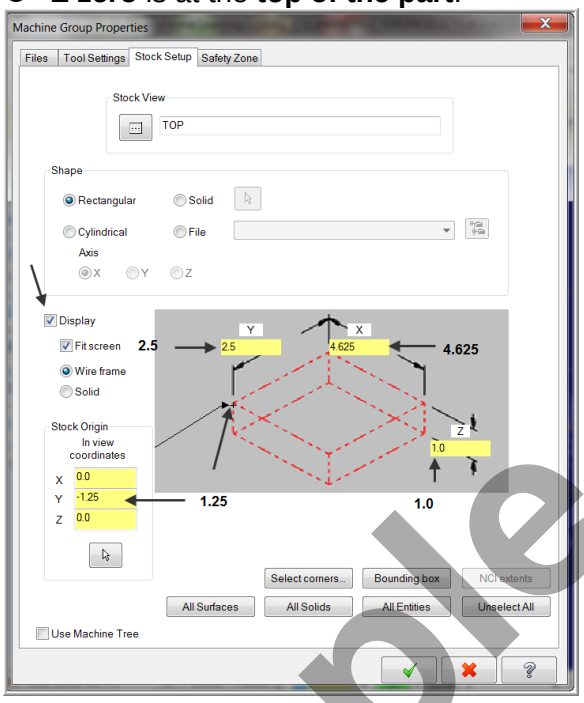


3. Select **Stock setup** in the toolpath manager window.



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:

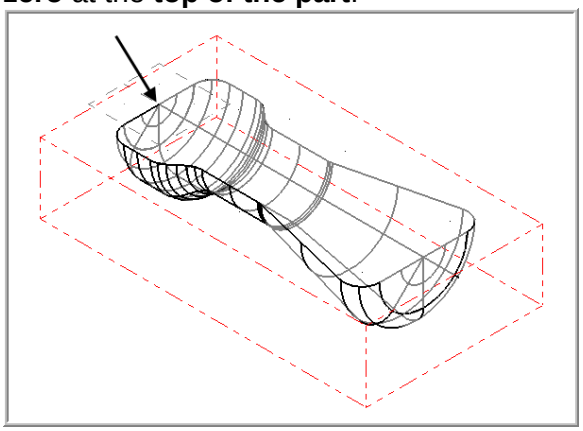
6. To change the Material type to Aluminium 6061 pick the Select button at the bottom of the Tool Settings page.

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.
10. Change the view to Isometric.
11. Fit to Screen.

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**.



Surface roughing toolpaths typically use larger tools, multiple stepovers, and multiple step downs to quickly remove larger volumes of stock and leave an even amount of stock for finishing.

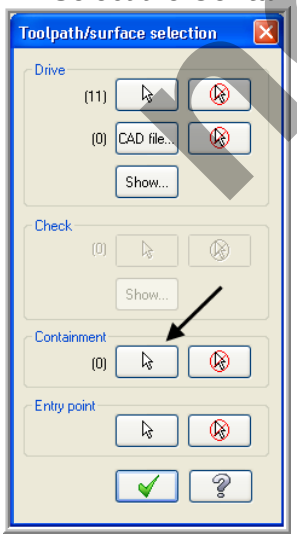
The roughing toolpaths you choose for your part depend on the shape of the part, shape of the stock, and machining situation. Mastercam provides several roughing strategies.

TOOLPATH TASK 3:
ROUGH THE POCKET USING SURFACE POCKET

➲ In this task you will use a 0.5 diameter bull end mill with a 0.125 corner radius to rough out the pocket.

1. From the menu bar select **Toolpaths>Surface Rough>Pocket…**
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:

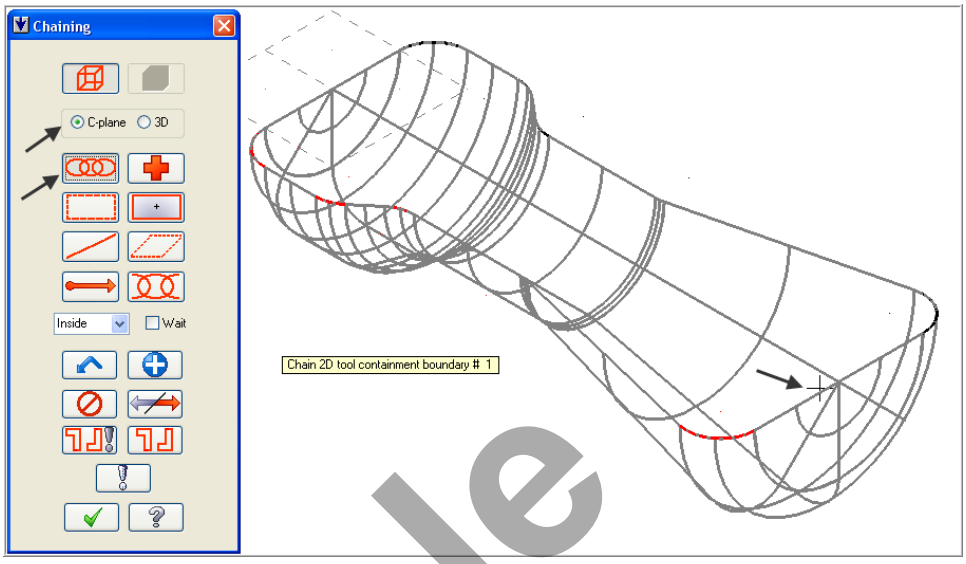
4. The **Select All** dialog box appears on the screen.
5. Click on the OK icon to complete and exit this feature.
6. To move onto the next step you now need to pick the **End Selection** icon .
7. Select the **Containment** button from the **Toolpath/surface selection** dialog box.



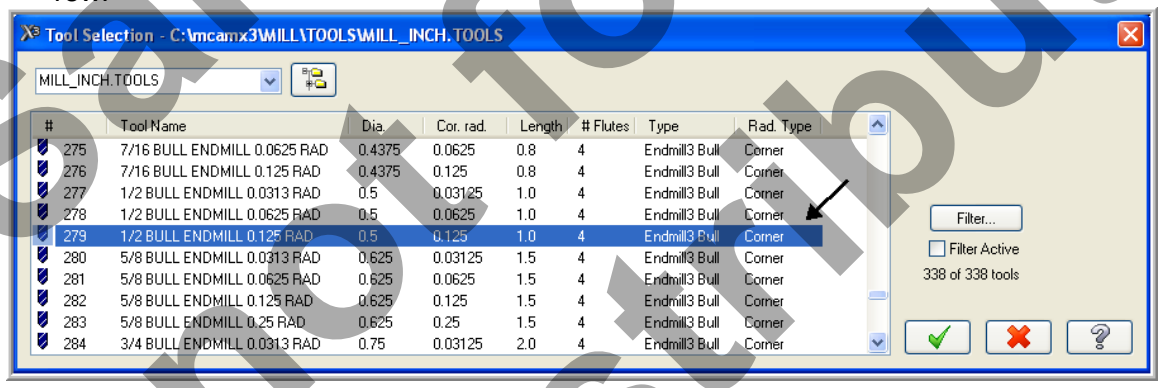
Use this dialog box to select all the geometry needed for a surface toolpath.

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:

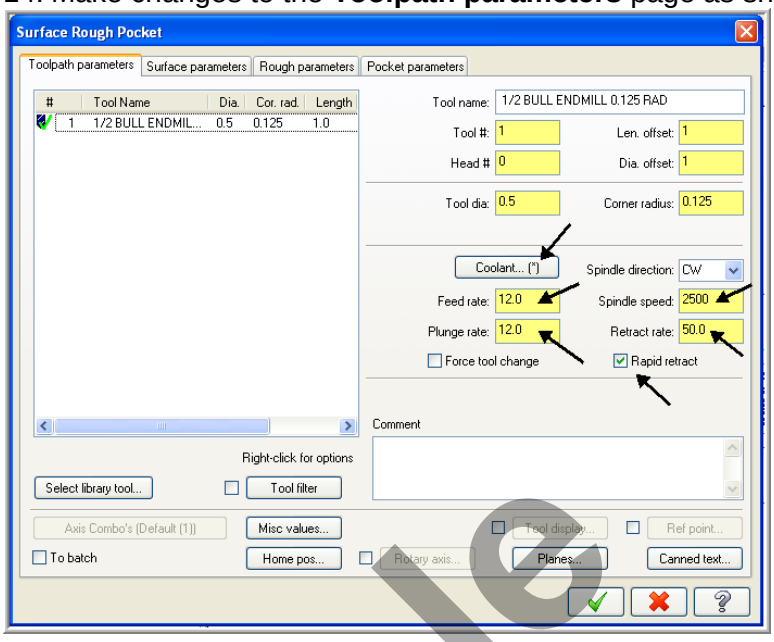


9. After the boundary has been successfully chained select the **OK** button  .
10. Select the OK button  to exit the **Toolpath/surface selection** dialog box.
11. In the lower left corner of the **Toolpath parameters** page select the **Select library tool…** button. **Disable Filter active.**
12. 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.

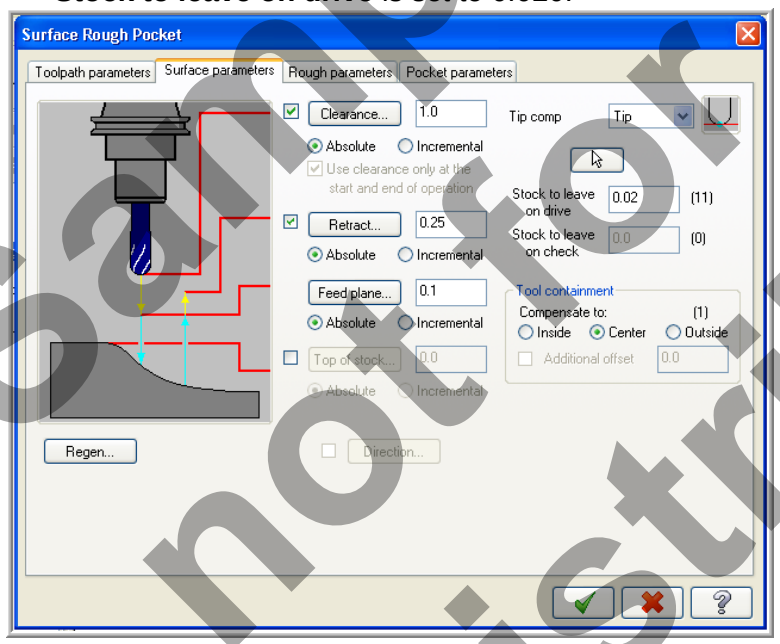


13. Select the OK button  to complete the selection of this tool.

14. Make changes to the **Toolpath parameters** page as shown below. Set coolant on.



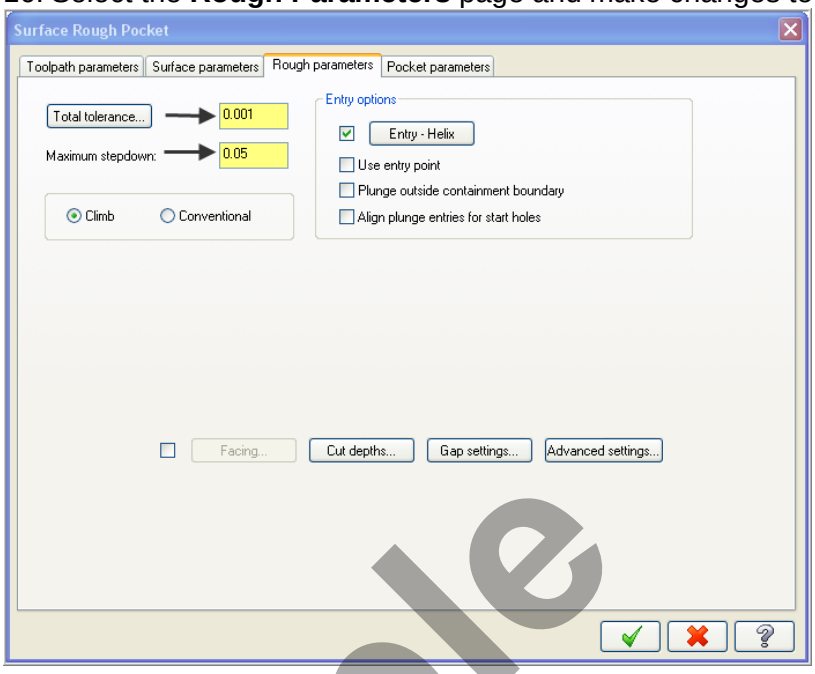
15. Select the **Surface parameters** page and make changes to this page as shown below:
Stock to leave on drive is set to 0.020.



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.

16. Select the **Rough Parameters** page and make changes to this page as shown below:

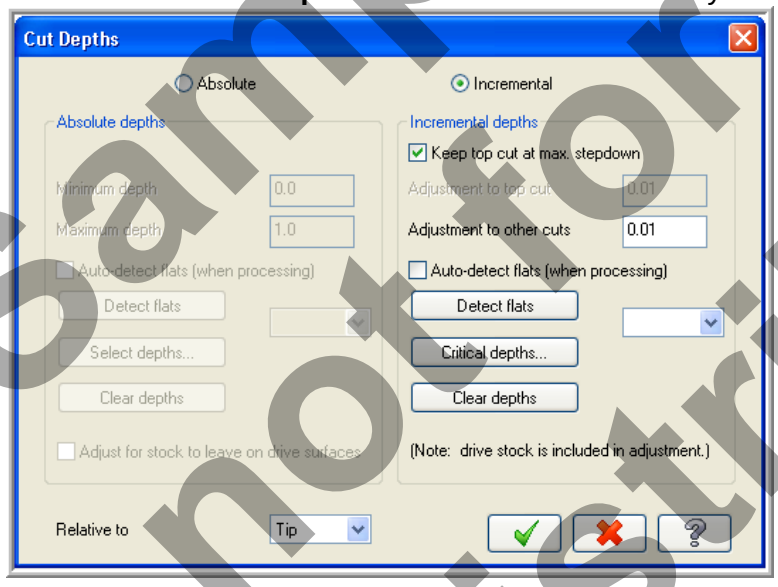


Tolerance selection will vary based on the toolpath type (roughing, semi-finish, or finishing), tolerances of final part, surface finish required, etc.

For the purposes of this tutorial, we will use .001 for both roughing and finishing to find a balance between accuracy and calculation time.

Typical tolerance ranges are from .0001 to .001.

17. Select the **Cuts depths** button make the necessary changes.



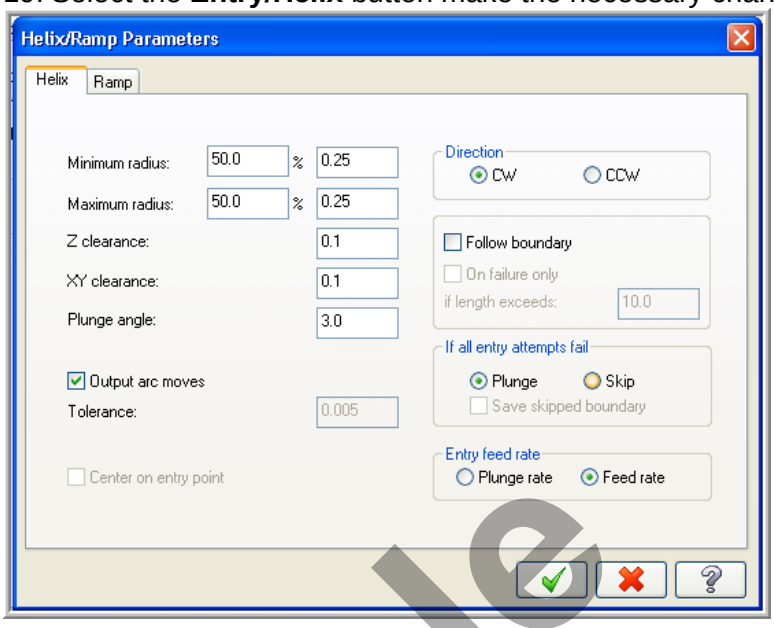
Cut depths specify the placement of Z-axis cuts for all rough surface toolpaths and for finish contour toolpaths.

Incremental cut depths
Incremental cut depths are measured from the top and bottom of the part for most rough surface toolpaths and for finish contour toolpaths.

Keep top cut at max stepdown
Available only for rough pocket toolpaths. Forces the tool to make the first cutting pass at the max cut depth instead of the top of the part.

18. Select the OK button  to complete this feature.

19. Select the **Entry/Helix** button make the necessary changes.

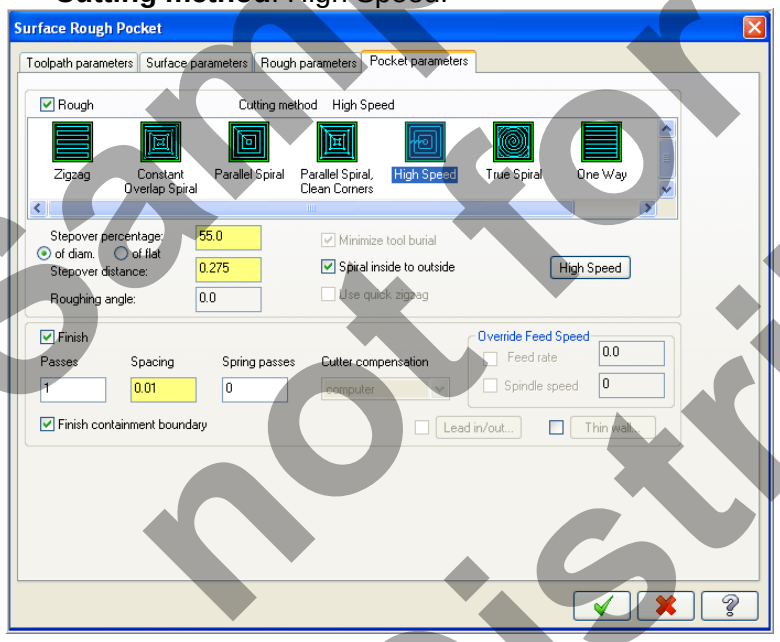


Helix/Ramp
Use this to add a ramp entry move to the pocket roughing operation.

You can add either a helix or a ramp entry, but not both. To switch between ramp or helical entry, just select the other tab and enter the desired entry dimensions.

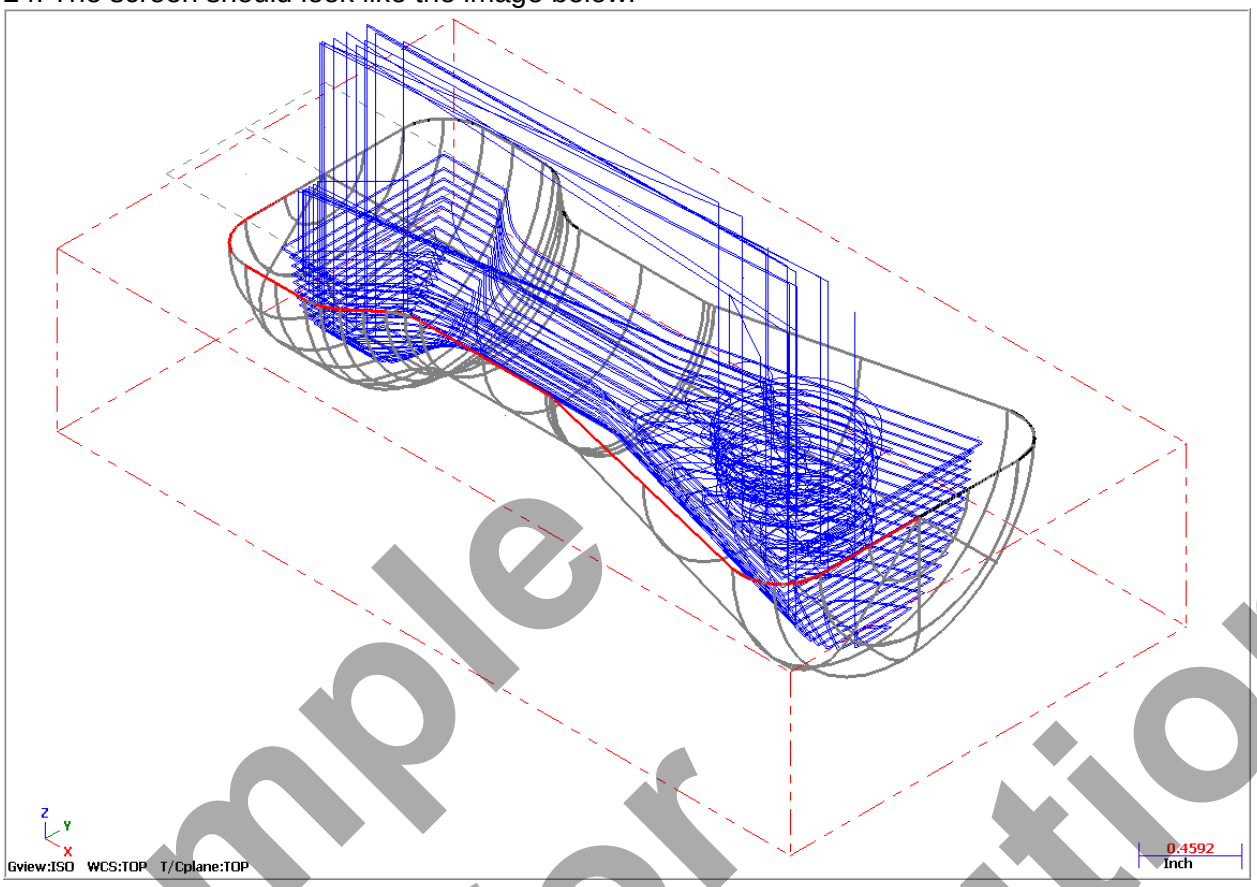
The helix/ramp options are off by default. If no helix/ramp options are set, Mastercam plunges the tool to the pocket depth at the start of the toolpath.

20. Select the OK button  to complete this feature.
21. Select the **Pocket parameters** page and make changes to this page as shown below.
    Cutting method: High Speed.



22. Select the OK button  to exit Pocket parameters.
23. It may take a while for Mastercam to create the toolpath.

24. The screen should look like the image below:



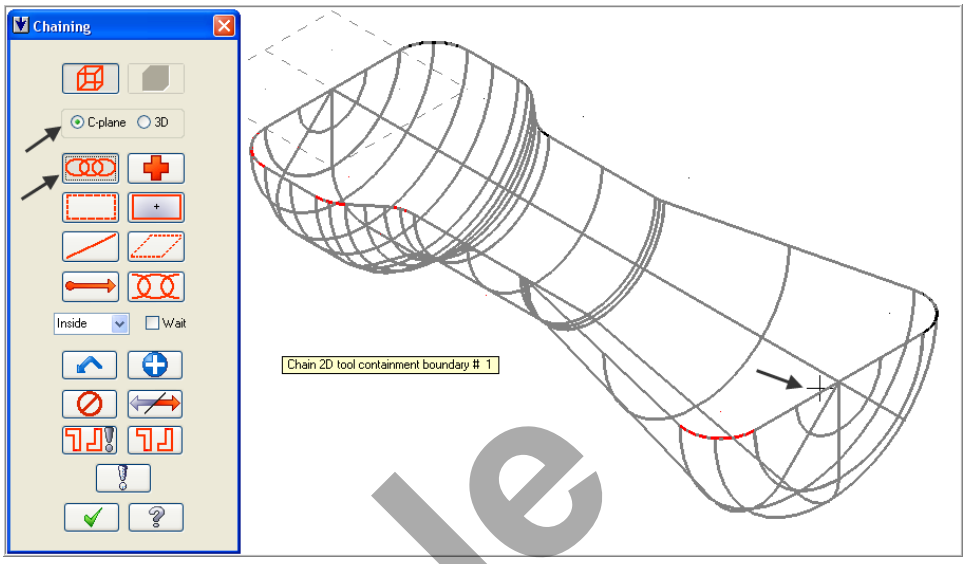
TOOLPATH TASK 4:
FINISH THE POCKET USING SURFACE HIGH SPEED WATERLINE

- In this task you will use a 0.5 diameter Ball end mill to finish the pocket.
- The rough parallel toolpath removes stock quickly using multiple constant Z depth. Surface High Speed Waterline also works at constant Z depth and steps down with cuts directly on the surface.

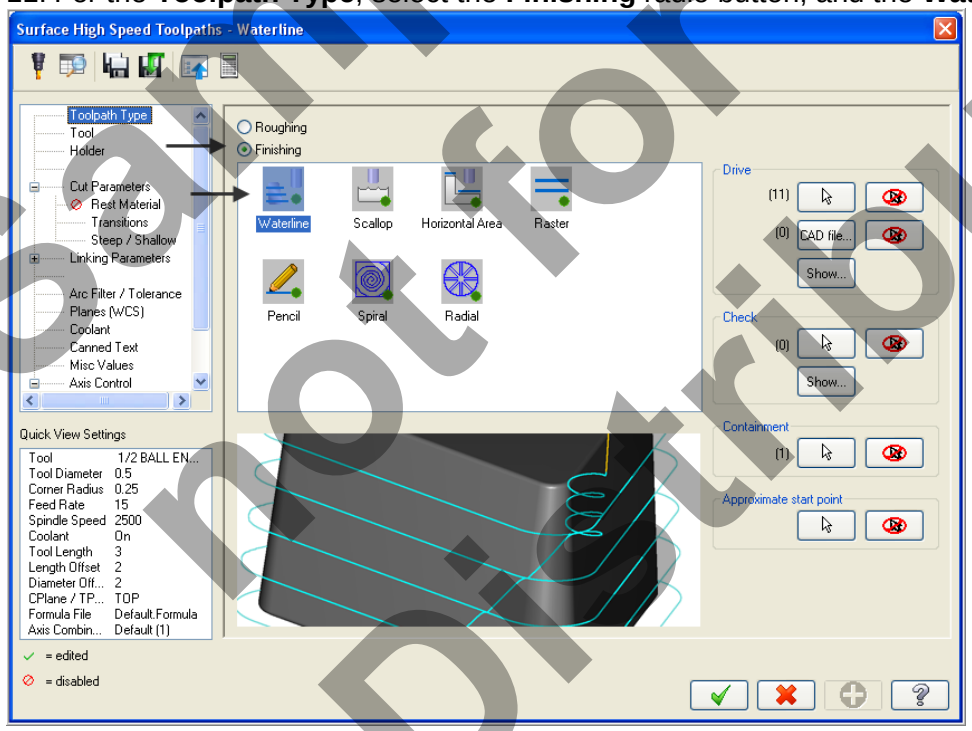
1. From the menu bar select **Toolpaths>Surface High Speed>Waterline...**
2. If you encounter the **"New 3D Advanced Toolpath Refinement Feature!"** dialog box **activate** the radio button for **"I've tried this feature and want to keep it to use. Do not show me this dialog again."** Then Click on the OK icon to complete and exit this feature.
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** button from the **Toolpath/surface selection** dialog box.

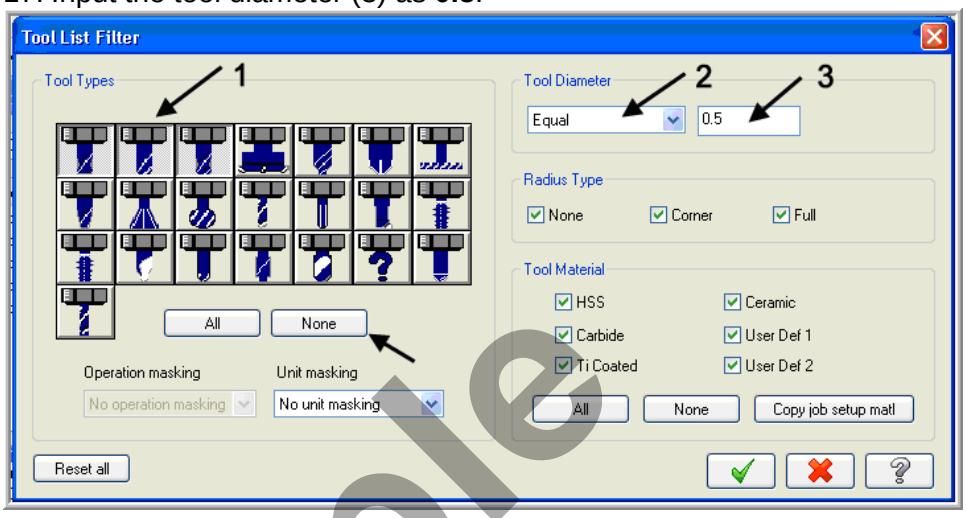
8. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Chain 2D tool containment boundary #1**. Ensure the **C-plane** radio button is activated. Now select the line as shown below:



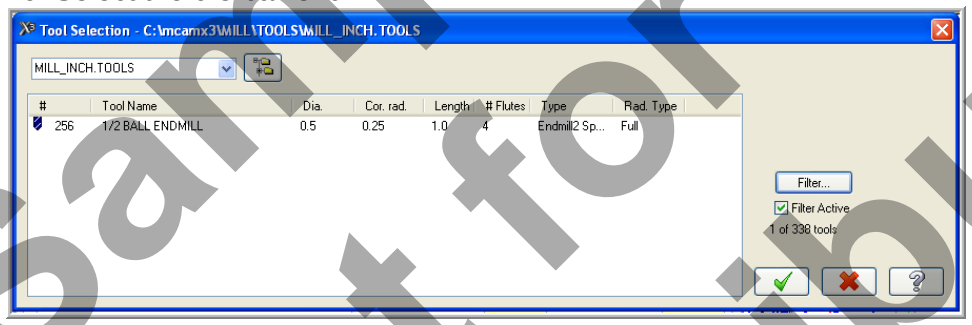
9. After the boundary has been successfully chained select the OK button  .
10. Select the OK button  to exit the **Toolpath/surface selection** dialog box.
11. For the **Toolpath Type**, select the **Finishing** radio button, and the **Waterline** toolpath.



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

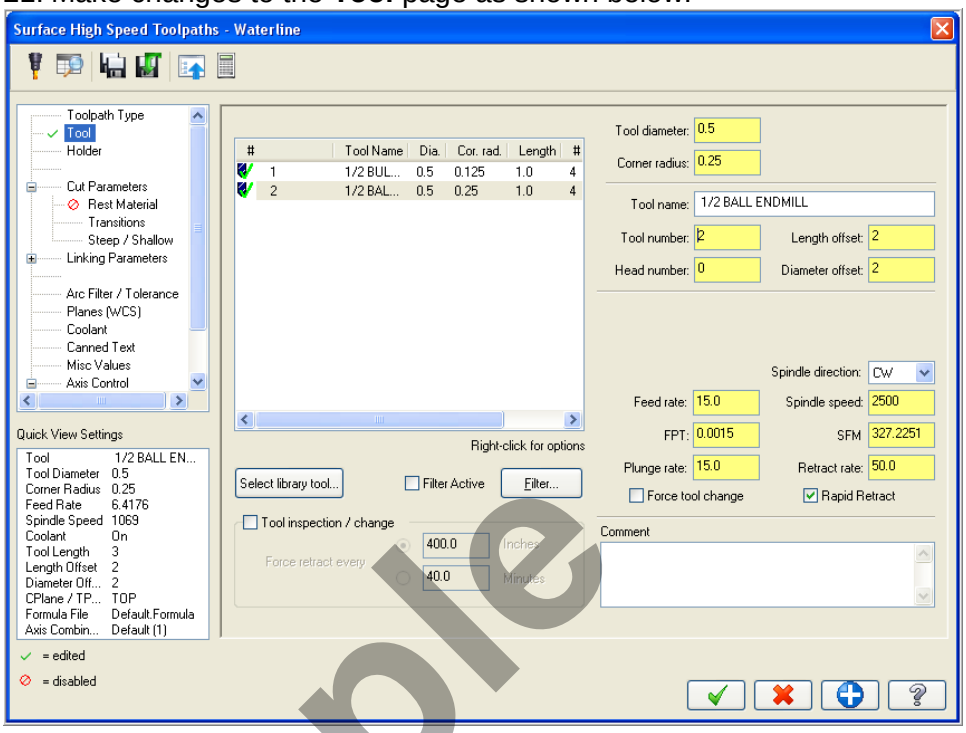


18. Select the OK button to exit.
19. Select the 0.5 ball end mill.

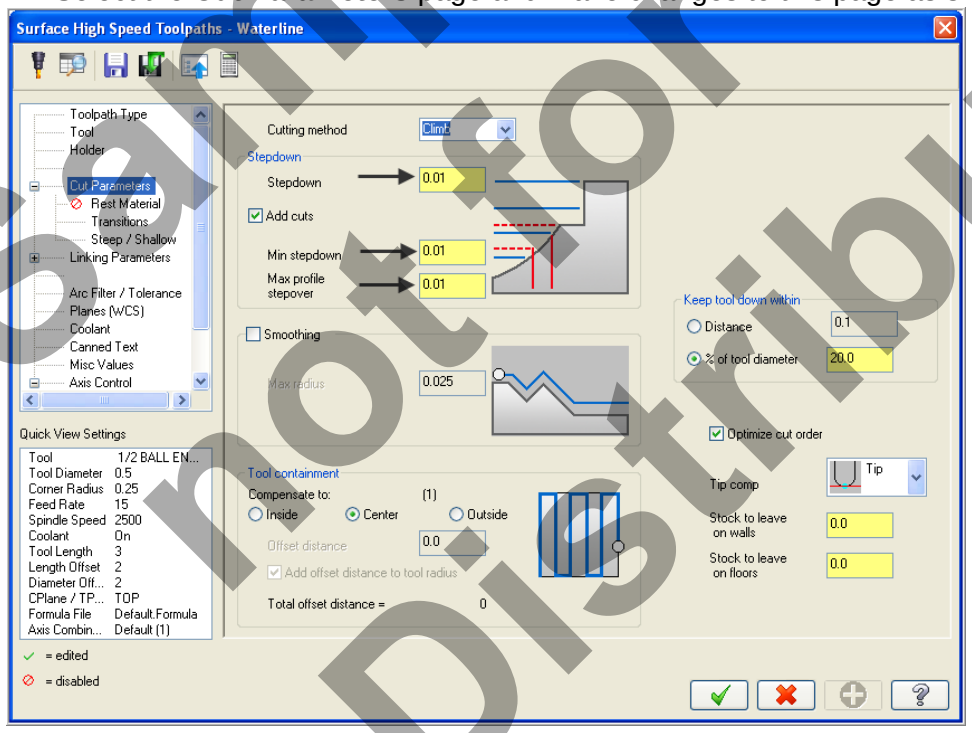


20. Select the OK button to complete the selection of this tool.

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



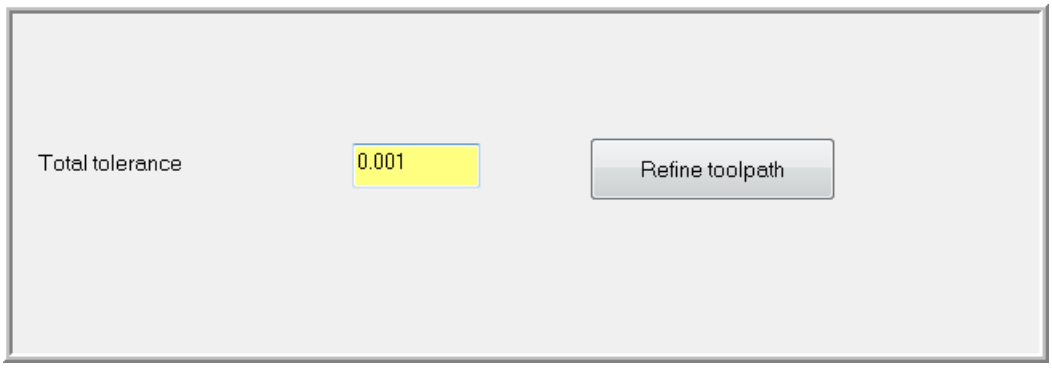
22. Select the **Cut Parameters** page and make changes to this page as shown below:



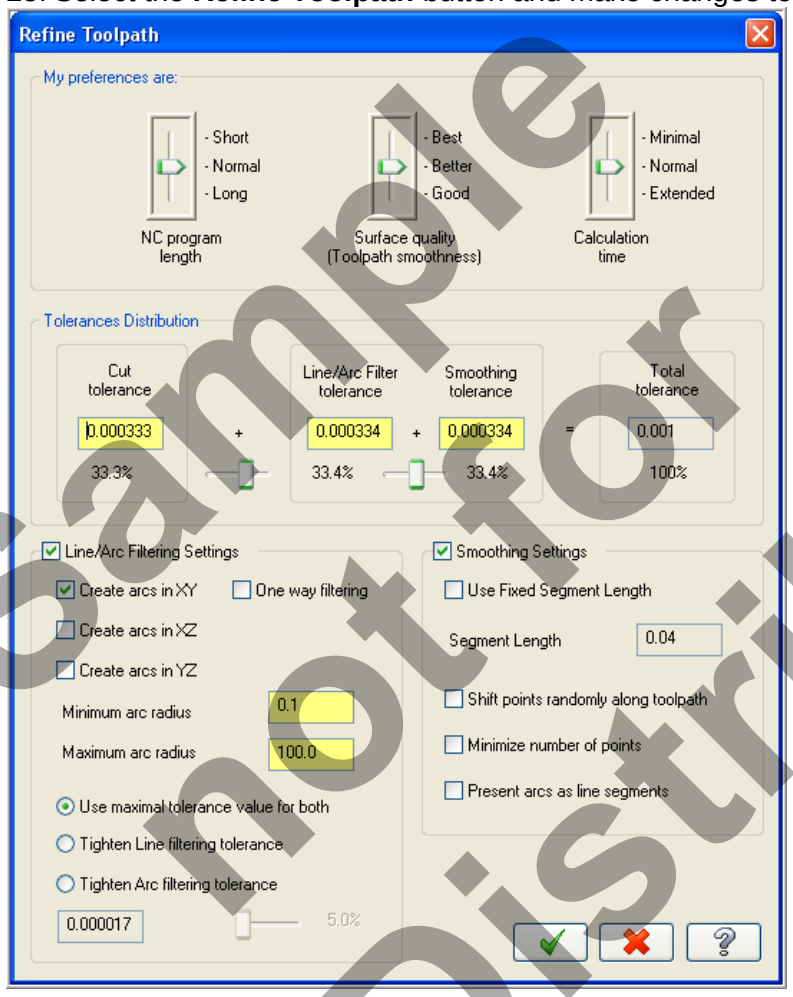
The Add cuts selection allows the user to add cuts in shallow areas to decrease the size of surface scallops.

The programmer should consider the tool corner radius, surface shape, and operation type (rough, semi-finish, finish) when making selections here.

23. Select the **Arc Filter / Tolerance** page,
24. Set the **Total** tolerance.



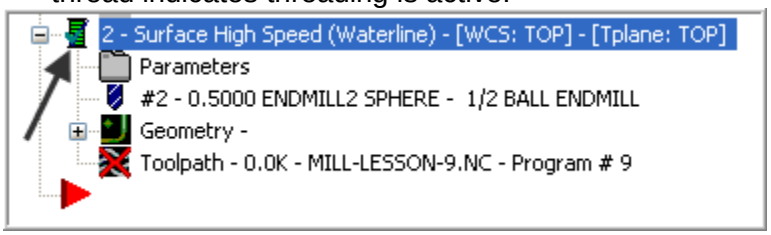
25. Select the **Refine Toolpath** button and make changes to this page as shown below:



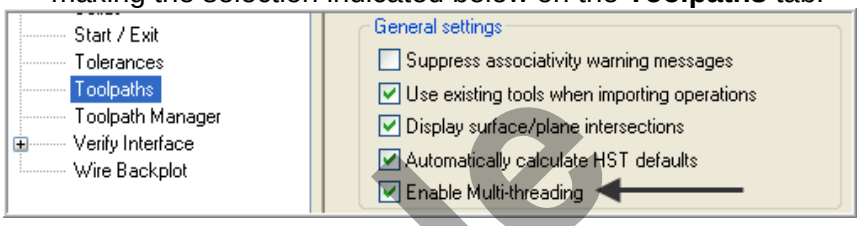
Refine Toolpath
Refine toolpath allows the user to make selections for toolpath calculation tolerance, arc filtering, and smoothness. The beginner programmer can simply adjust the sliders in the preferences section at the top. Advanced users can make selections manually in the windows below.
Tolerances Distribution allows the programmer to manipulate the extent to which the Filtering and Smoothing settings are applied in the toolpath calculation.
Line/Arc Filtering Settings Settings for creating arcs and lines from sections of linear moves.
Smoothing Settings will process the code in varied ways in order to present it to the CNC controller in the preferred way.

26. Select the OK button to exit Refine Toolpath.
27. Select the OK button to exit Surface High Speed Toolpaths.

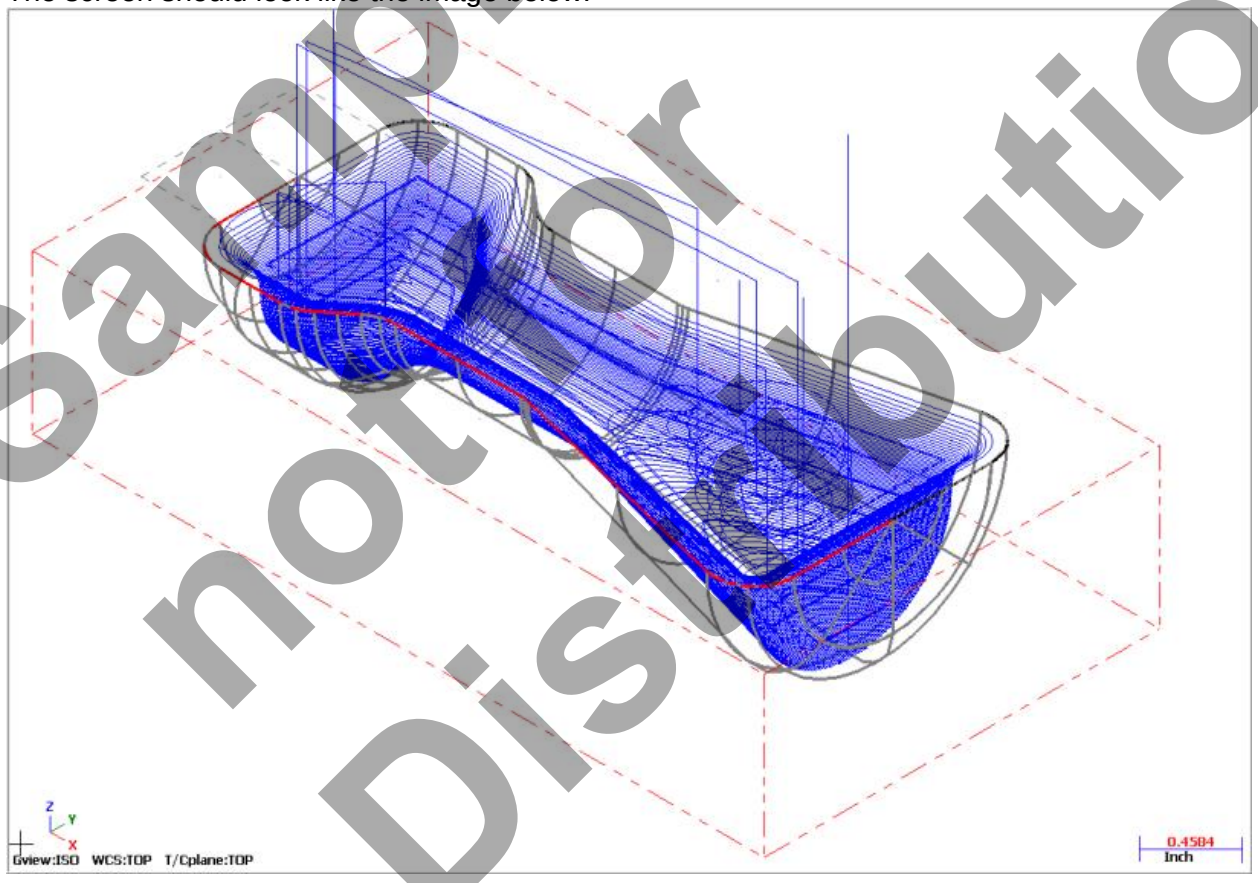
28. It may take a while for Mastercam to create the toolpath. Mastercam's multi-threading functionality will calculate the toolpath while you continue to work. The green spool of thread indicates threading is active.



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

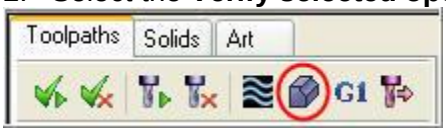


The screen should look like the image below:



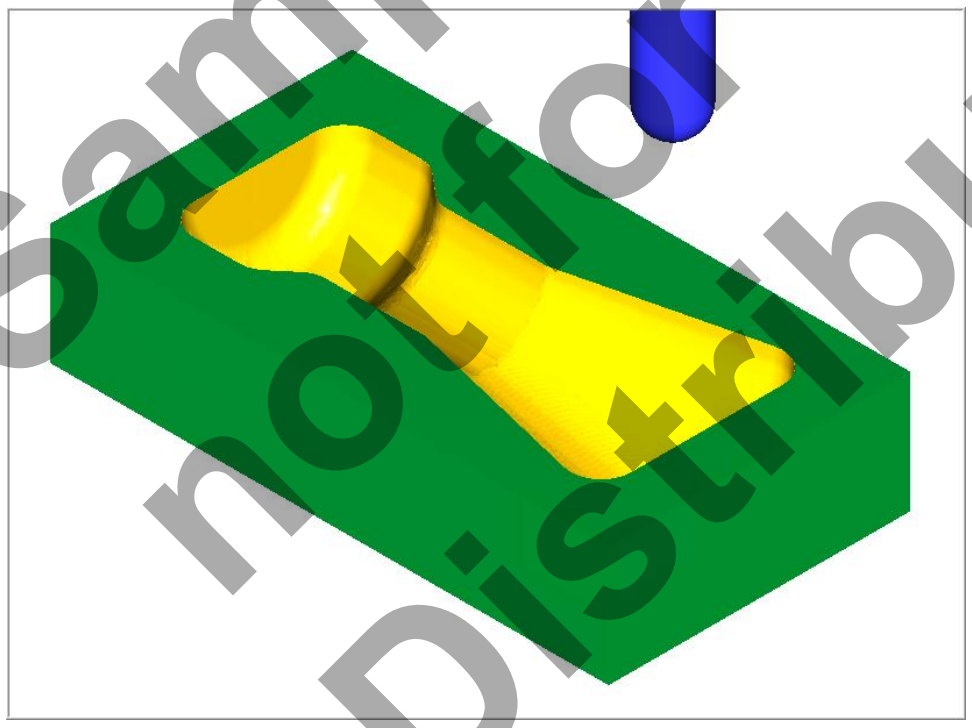
TOOLPATH TASK 5:
VERIFY THE TOOLPATH

1. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon 
2. Select the **Verify selected operations** button circled below:



3. Adjust the Verify speed to fast.

4. Select the **Play** button to verify the toolpath. If it is taking too long to verify the toolpath click on the **Stop** button and go to step 5.
5. Select the **fast forward** button to speed up the verification of the toolpaths.



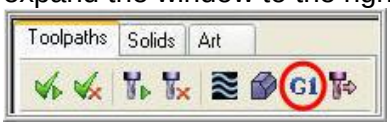
6. Select the OK button  to exit Verify.
7. Save the UPDATED MCX file

TOOLPATH TASK 6:
POST AND CREATE THE CNC CODE FILE

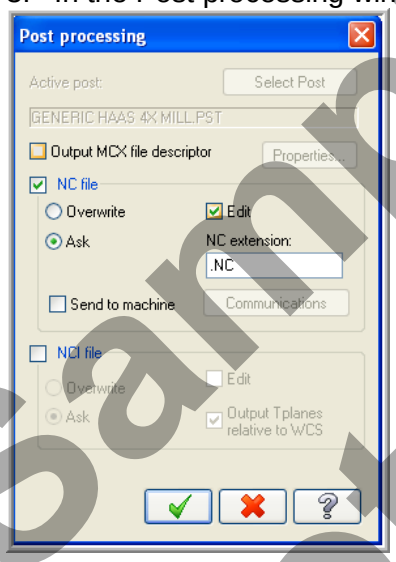
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.

2. Select the **Post selected operations** button from the Toolpath manager.

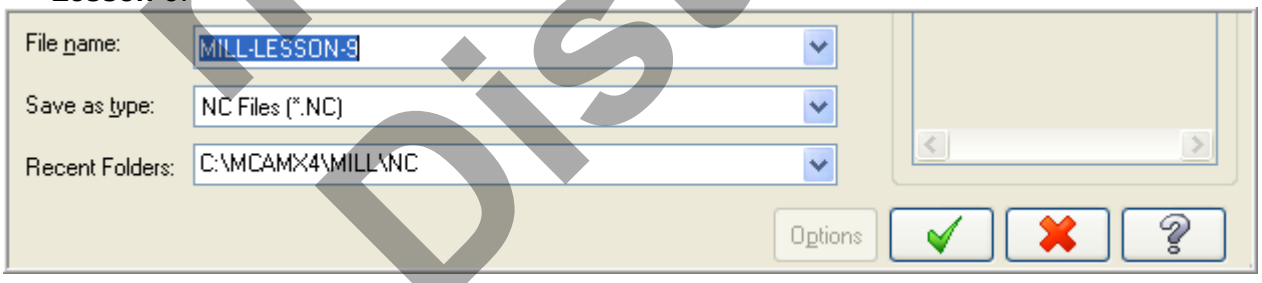
Please Note: If you cannot see **G1** click on the right pane of the Toolpath manger window and expand the window to the right.



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



4. Select the OK button  to continue.
5. Enter the same name as your Mastercam part file name in the NC File name field **Mill-Lesson-9**.



6. Select the Save button.
8. The CNC code file opens up in the default editor:
9. Select the  in the top right corner to exit the CNC editor.

This completes Mill-Lesson-9.

MILL-LESSON-9 EXERCISE