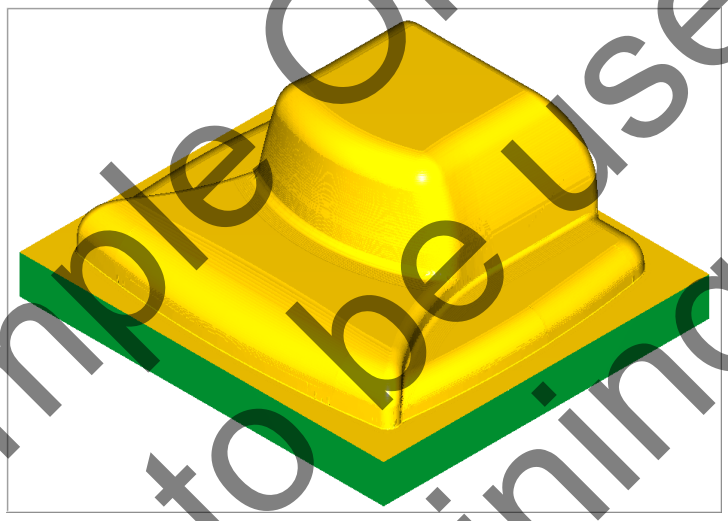
Mastercam X4

Training Guide



Mill-Lesson-15
Core Roughing, Waterline, and Surface Finish Leftover

camInstructor

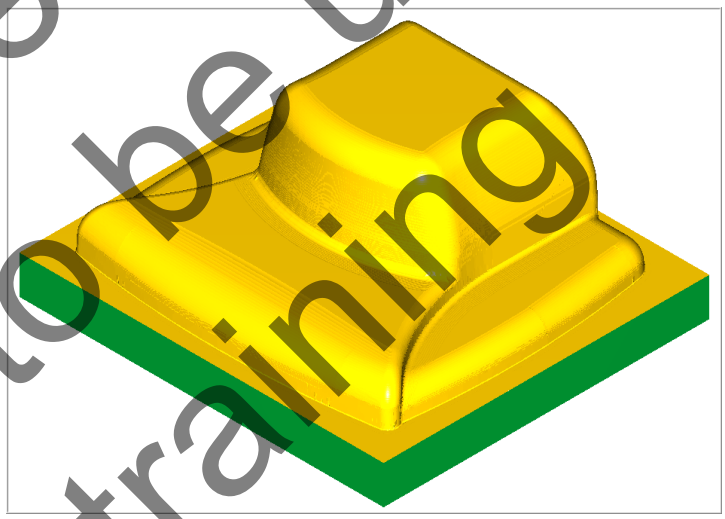
Objectives

You will use a provided model for Mill-Lesson-15, then generate the toolpaths to machine the part on a CNC vertical milling machine. This Lesson covers the following topics:

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

➲ **Generate 3-dimensional milling toolpaths consisting of:**
Core Roughing
Waterline
Surface Finish Leftover

➲ **Inspect the toolpath using Mastercam's Verify by:**
Launching the Verify function to machine the part on the screen.
Comparing a verified part to the original stock stl file.
Generating the NC- code.

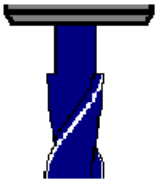


TOOL LIST

- 1.000 diameter flat end mill to rough and finish machine.
- .500 diameter ballnose to finish machine
- .125 diameter ballnose to finish machine
- .0313 diameter ballnose to finish machine

Tool List of MILL-LESSON-15.MCX

Proj./Part No.: 0		Date : 06/22/09	
Drawing No. : 1		Customer : -	
Prog. No. : 15		Programmer : 1	



Tool type	:	1 Endmill1 Flat 1 INCH FLAT ENDMILL					
Manufact.code	:						
Chuck	:						
Tool Number	:	243				Feedrate	: 19.552
Diameter	:	1	RPM	: 1222		Plunge feed r.:	9.776
Corner radius	:	0	Tip angle	: 0		Diam. offset	: 243
Flute length	:	2	Material	: ALUMINUM ...		Length offset	: 243
Overall length:		4	No flutes	: 4			

Tool type	:	0.0313 Endmill2 Sphere 1/32 BALL ENDMILL					
Manufact.code	:						
Chuck	:						
Tool Number	:	246				Feedrate	: 15.0205
Diameter	:	0.0313	RPM	: 39116		Plunge feed r.:	7.5103
Corner radius	:	0.0156	Tip angle	: 0		Diam. offset	: 246
Flute length	:	0.375	Material	: ALUMINUM ...		Length offset	: 246
Overall length:		2.25	No flutes	: 4			

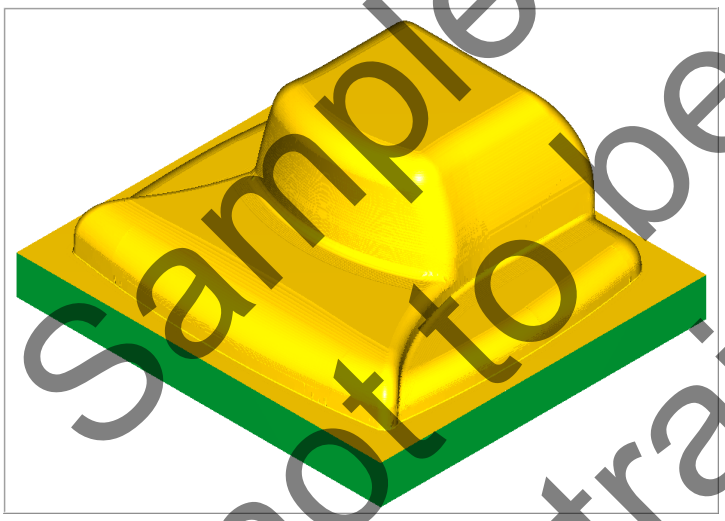
Tool type	:	0.125 Endmill2 Sphere 1/8 BALL ENDMILL					
Manufact.code	:						
Chuck	:						
Tool Number	:	249				Feedrate	: 15.0205
Diameter	:	0.125	RPM	: 9779		Plunge feed r.:	7.5103
Corner radius	:	0.0625	Tip angle	: 0		Diam. offset	: 249
Flute length	:	0.375	Material	: ALUMINUM ...		Length offset	: 249
Overall length:		2.25	No flutes	: 4			

Tool type	:	0.5 Endmill2 Sphere 1/2 BALL ENDMILL					
Manufact.code	:						
Chuck	:						
Tool Number	:	256				Feedrate	: 15.6416
Diameter	:	0.5	RPM	: 2444		Plunge feed r.:	7.8208
Corner radius	:	0.25	Tip angle	: 0		Diam. offset	: 256
Flute length	:	1	Material	: ALUMINUM ...		Length offset	: 256
Overall length:		3	No flutes	: 4			

MILL-LESSON-15 - THE PROCESS

Toolpath Creation

TASK 1: Setting the environment
TASK 2: Open existing file from the multimedia CD
TASK 3: Define the rough stock using stock setup
TASK 4: Rough mold core using Surface High Speed (Core Roughing)
TASK 5: Finish profile surfaces using Surface High Speed (Waterline)
TASK 6: Finish all remaining stock using Surface Finish Leftover
TASK 7: Verify the toolpath and compare to STL file
TASK 8: Save the updated MCX file
TASK 9: Post and create the CNC code file
TASK 10: Create ActiveReport



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:

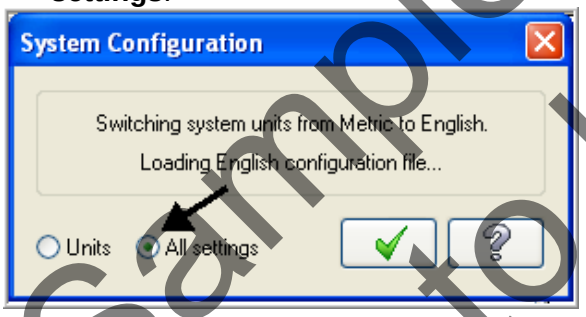
1. Set up the Grid. This will help identify the location of the origin.
2. Customize the toolbars to machine a **3D part**.
3. Set the machine type to a Haas Vertical Spindle CNC machine.

Toolpath Creation

TASK 2:
OPEN EXISTING FILE FROM THE MULTIMEDIA CD

- On the multimedia CD that came with this text is **a folder called Mastercam-Files**. The file is in **inch** units and contains the wireframe and surface (solid) geometry of the part.
- The part is already setup for a: **GENERIC HAAS 4 – AXIS VMC.**

1. Select **File>Open> Mill-Lesson-15.MCX.**
2. If confronted with the System Configuration dialog box activate the radio button for **All settings**.

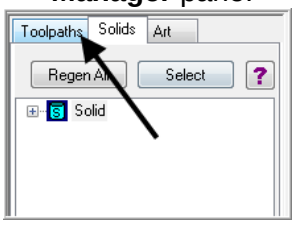


- **System Configuration (switch units) dialog box**
- When you open a part file that uses different units (English or metric) from those currently in use, Mastercam automatically displays this dialog box, which informs you that Mastercam is switching units and loading an alternate default configuration file. In order to complete the operation, select one of the following options:
- **Units:** Tells Mastercam to use only the units from the new configuration file. (default)
- **All settings:** Tells Mastercam to load all settings from the new configuration file.

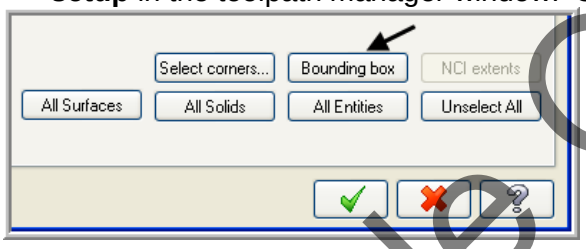
3. Select the OK button to exit the **System Configuration** dialog box.
4. Activate a shaded view by selecting the icon at the top of the screen.

TASK 3:
DEFINE THE ROUGH STOCK USING STOCK SETUP

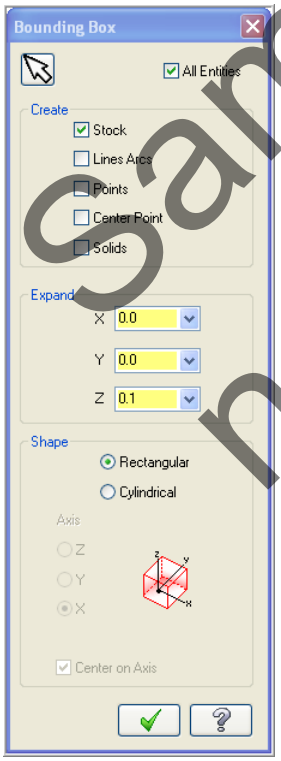
1. Click on the Toolpaths Tab as shown below: Note; **Alt-O** will **Show/hide** the **Operations Manager** pane.

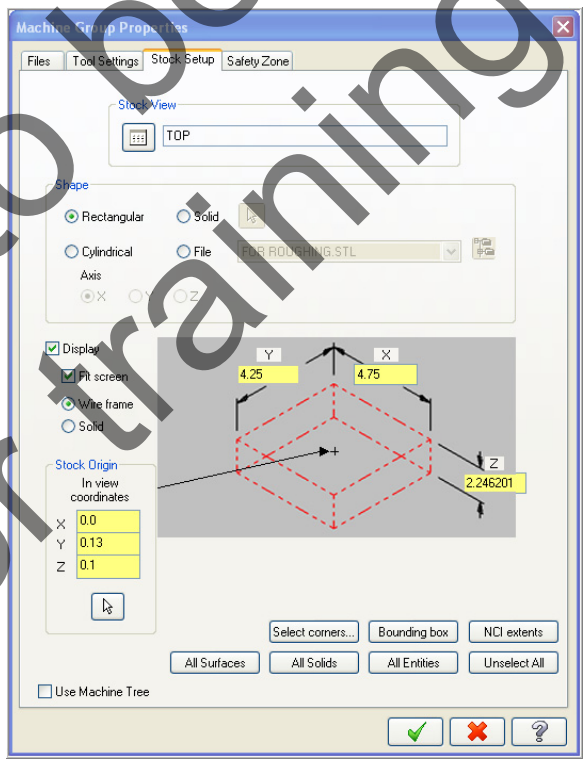


2. Select the **+** in front of **Properties** to expand the Toolpaths Group Properties. Select **Stock setup** in the toolpath manager window. Select the **Bounding box** button:



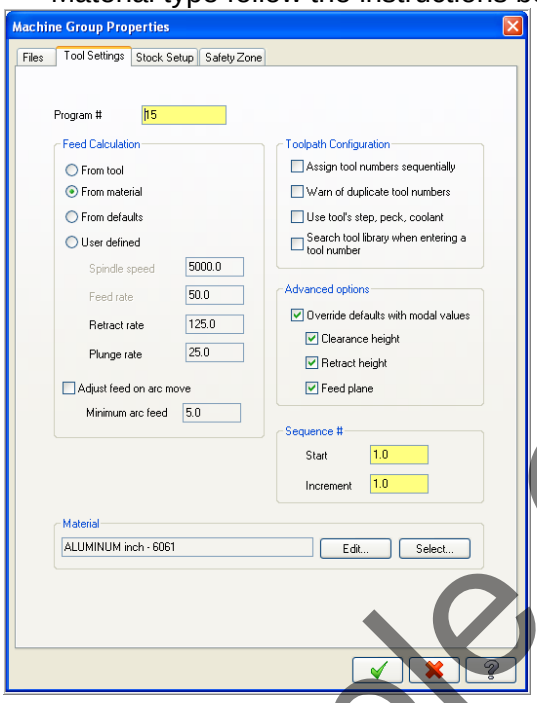
3. Set the parameters to match the **Bounding Box** screenshot in the left image below, then select the **OK button**.





4. Set the parameters to match the **Stock Setup** screenshot shown in the above right image. Since the expand option in the bounding box window increased the size of the stock in both the negative and positive directions, the Z dimension needs to be reduced by 0.1. Simply select the Z dimension and add **– 0.1** and hit **enter**.

5. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. **Note**: The **Feed Calculation** is set to **From materia**l. To change the Material type follow the instructions below:

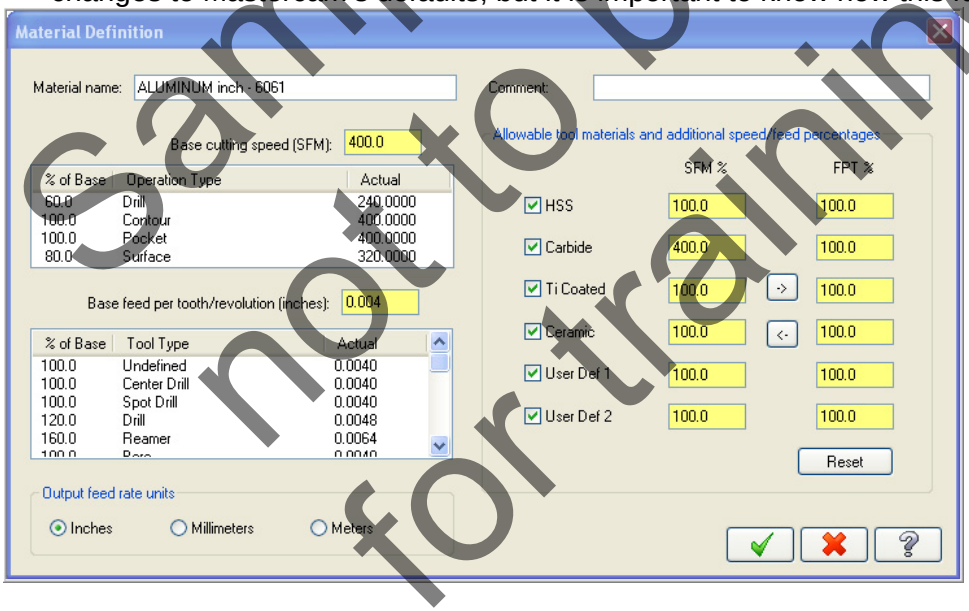


To change the **Material** type to **Aluminium 6061** pick the **Select** button at the bottom of the **Tool Settings** page.

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

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

6. Select the **Edit** button to enter the material definition. At this time we will not make any changes to Mastercam's defaults, but it is important to know how this function works.



Material Definition allows the user to enter the **Base cutting speed** (Surface Feet per Minute) and **Base feed per tooth/revolution** (Chip Load). These base values can be arrived at based on the material used. Mastercam's default values are very conservative so we will use them for safety purposes during this tutorial.

% of Base by Operation Type allows the user to specify a variation in SFM based on the operation type. Eg, face milling will have a much higher SFM than profile milling.

% of Base by Tool Type allows the user to vary feed per tooth by the tool. Eg. endmills will typically have a much higher feed per tooth (FPT) than a ballnose tool will.

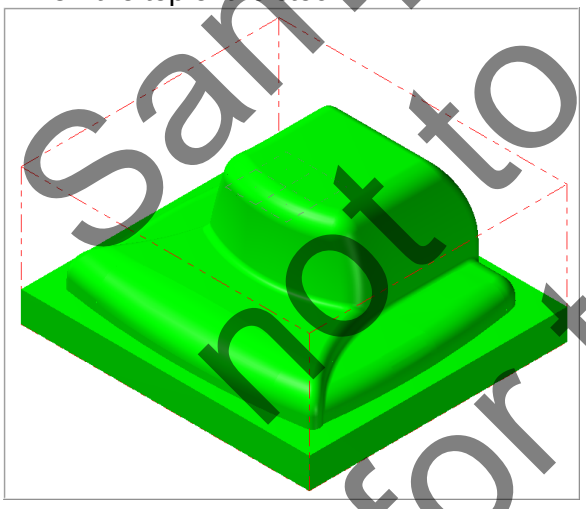
Allowable tool materials and additional speed/feed percentages allows the user to further customize based on the tool type.

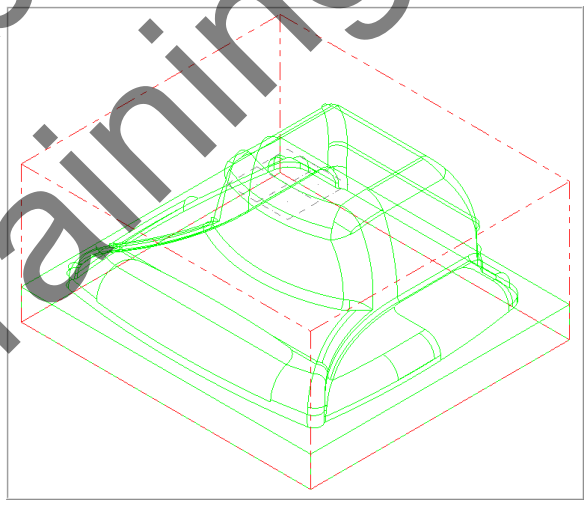
The user can further adjust SFM and chip load percentages in each tool definition. This is demonstrated during the first operation in **TASK 3**.

If cutting on a machine, it is extremely important that you research recommended SFM and FPT for your tools and material and make the appropriate settings in Mastercam.

7. Select the **OK button** again to complete this function.
8. Select the **OK button** again to exit the **Machine Group Properties**.
9. Now select the **Fit to screen** icon then select the **Isometric View** icon.

➲ Your part should look similar to the screen shot below: With **X0 Y0** at the center and **Z zero on** the top of the stock.



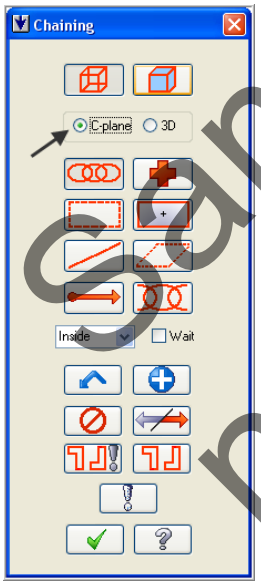


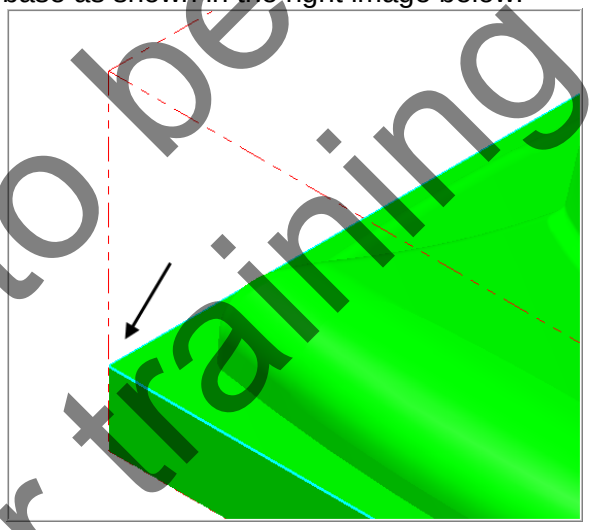
TASK 4:
ROUGH OUT MOLD CORE USING SURFACE HIGH SPEED (CORE ROUGHING) TOOLPATH

➲ In this task you will use a 1.000 diameter end mill to rough the part.

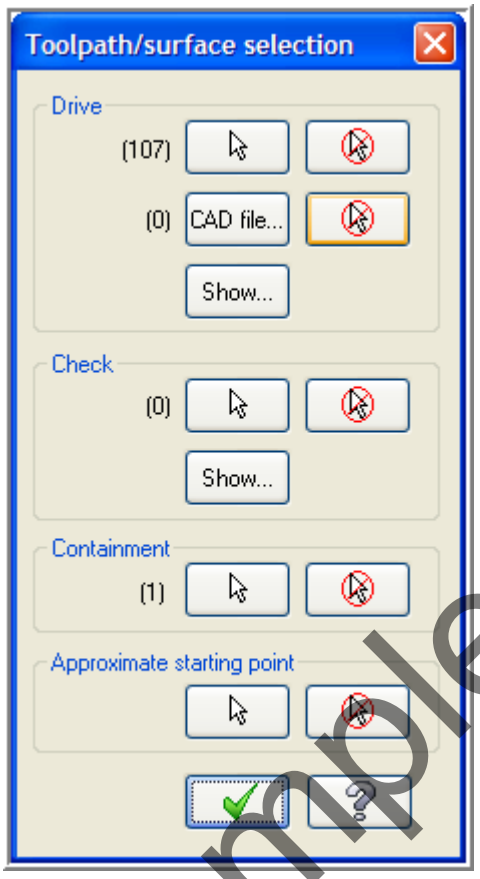
1. From the bottom **Status** menu bar open the **Level Manager** and ensure **level 1 and level 2** are turned on.
2. From the menu bar select **Toolpaths>Surface High Speed…**
3. If prompted with the **New 3D Advanced Toolpath Refinement Feature** window, select **Yes, I want to activate. Show me this dialog again.**
4. When prompted to **Enter new NC name** ensure **Mill-LESSON-15** is visible and then select the **OK button**  .
5. You are first prompted to **Select Drive surfaces,** window select all of the entities on the screen.
6. Click the **End Selection** icon  .
7. Click on the **Containment** Button.

8. Select **C-plane** in the Chaining window as shown in the left image below. Then select the chain around the top of the mold base as shown in the right image below:

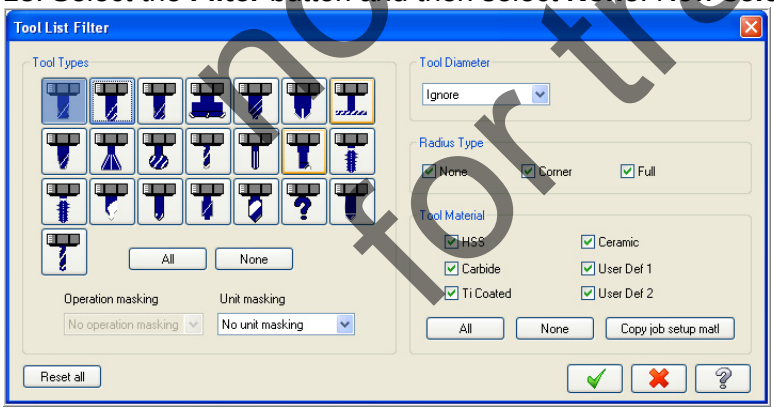




9. Click on **OK**  to exit **Chaining** and then confirm the number of drive surfaces as shown below:

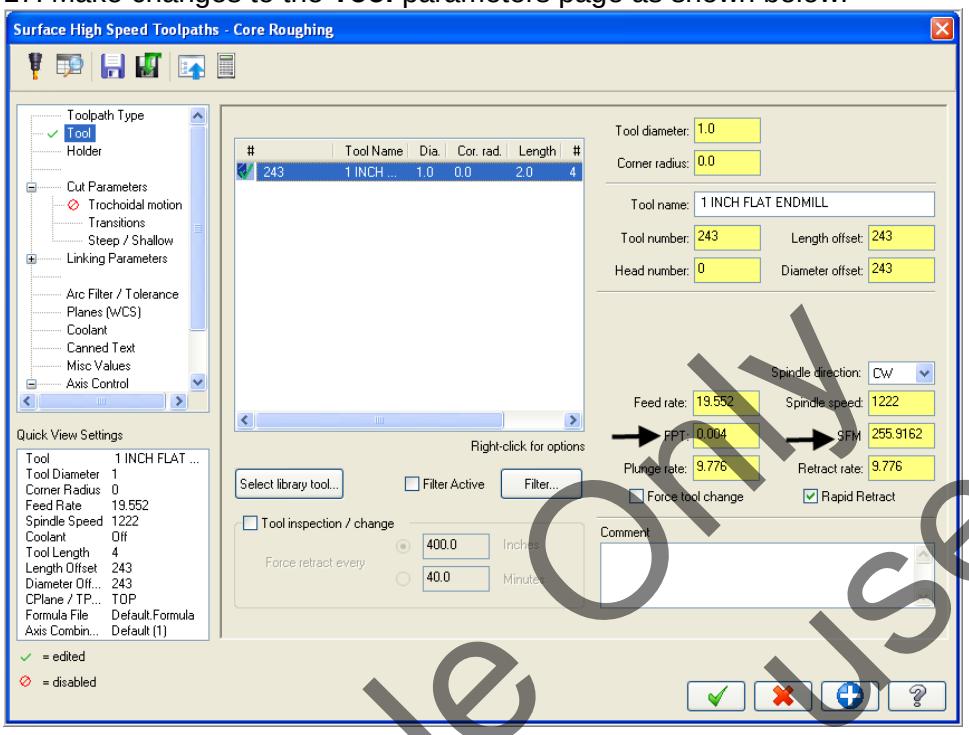


10. Click the **OK button**  to enter the toolpath parameters.
11. Select the **Toolpath Type** tab and activate the **Roughing** radio button, and then select the **Core Roughing** toolpath.
12. Next, select the **Tool** tab. In the lower left corner of the page select the **Select library tool…** button.
13. Select the **Filter** button and then select **None**. Now **select only** the **Endmill1 Flat** icon:



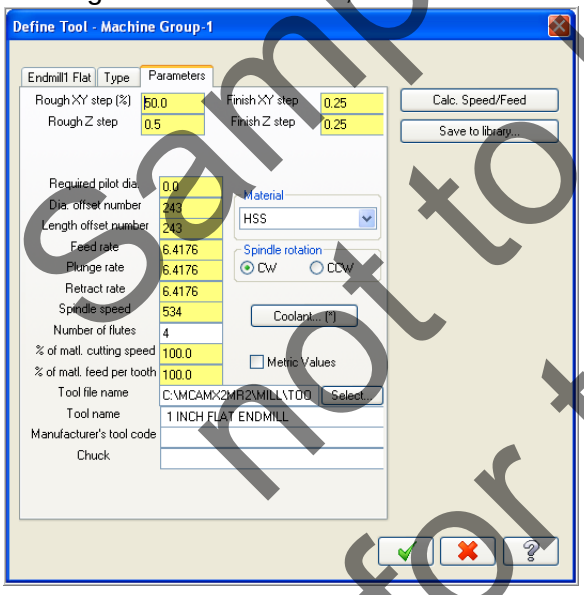
14. Select the **OK button**  to exit the **Tool List Filter**.
15. Use the scroll bar on the right of this dialog box to locate and select the **1 Inch Flat Endmill**.
16. Select the **OK button**  to complete the selection of this tool.

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



By clicking on the tool on this page, you will automatically load the values for **FPT** (feed per tooth) and **SFM** (surface feet per minute). This is because we selected **From Material** on the **Tool Settings** tab of the **Machine Group Properties**.

18. Right Click on the tool, select **Edit Tool**. Then go to the **Parameters** tab.



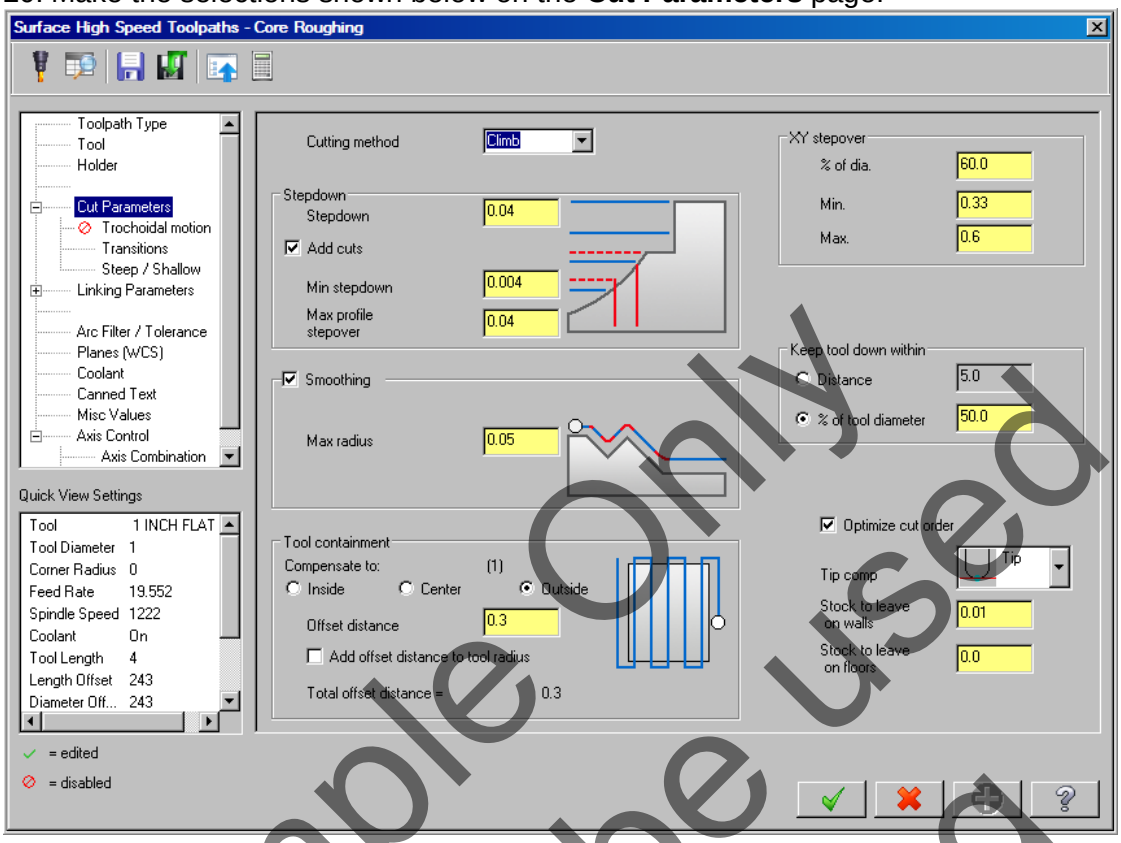
% of matl cutting speed allows the user a further adjustment of **SFM** based on the particular tool.

% of matl feed per tooth allows the user a further adjustment of **FPT** based on the particular tool.

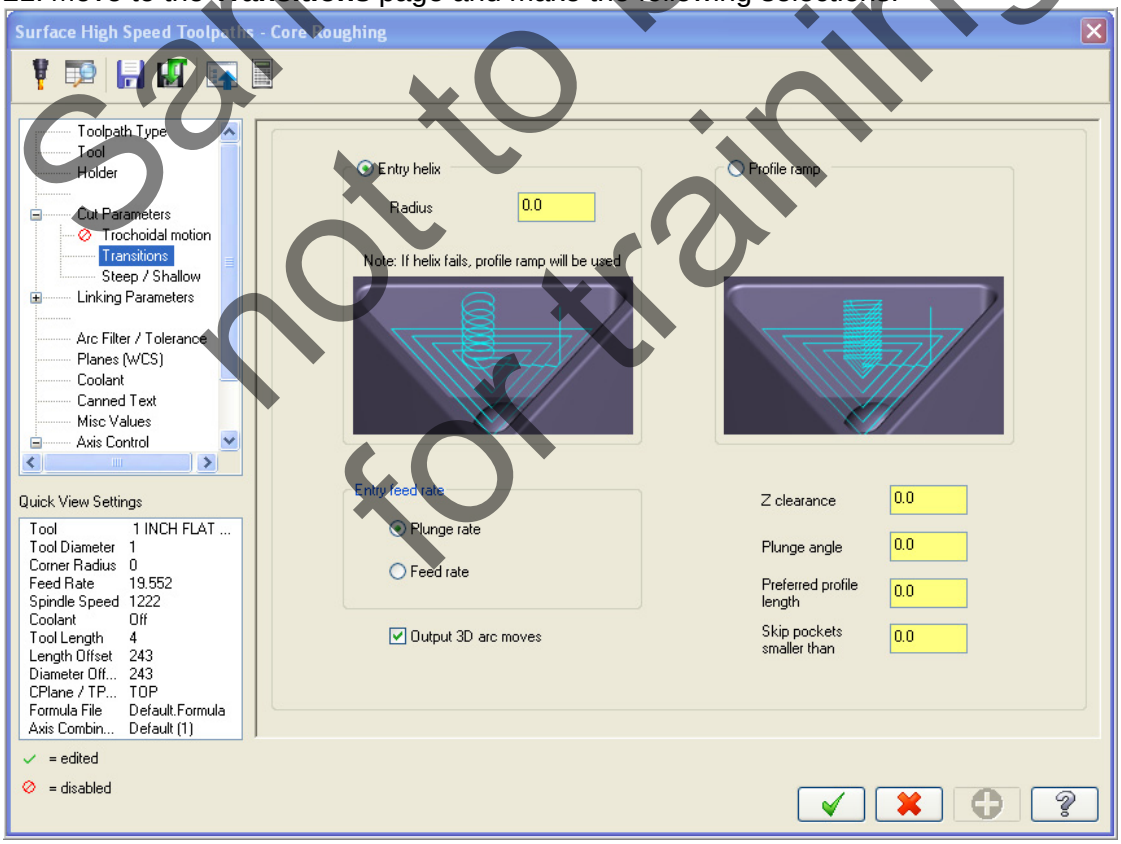
Refer to **TASK 2** for material library information.

19. Click the **OK button**  to return to the **Tool Parameters** page.

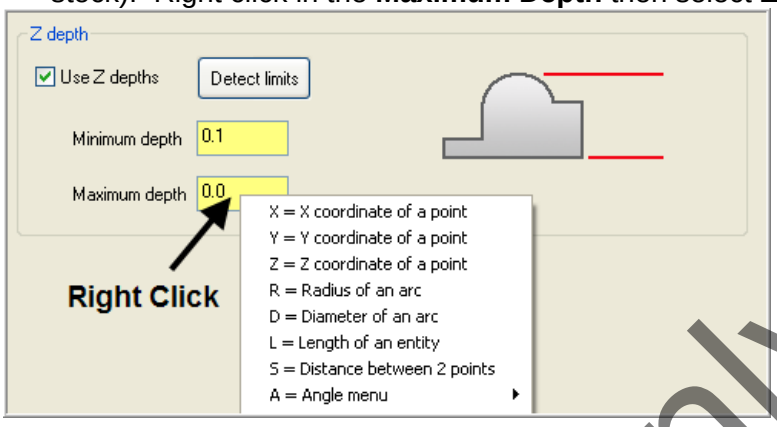
20. Make the selections shown below on the **Cut Parameters** page:



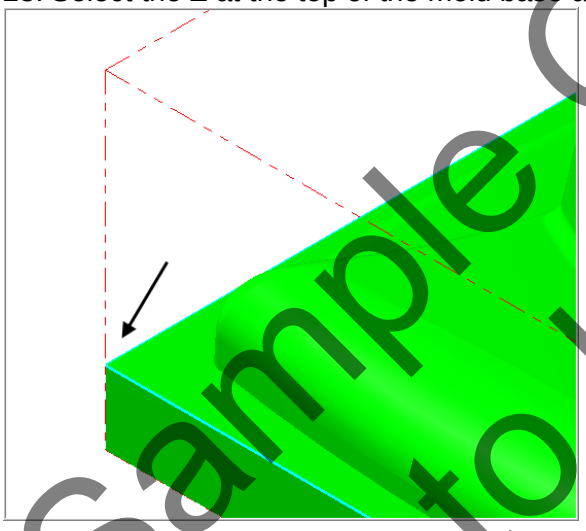
21. Move to the **Transitions** page and make the following selections:



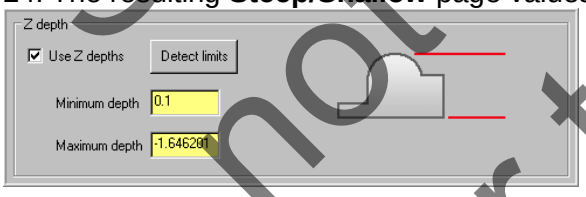
22. On the **Steep/Shallow** page set the **Minimum Depth** of the cut to **0.1** (this is the top of stock). Right click in the **Maximum Depth** then select **Z = Z coordinate of a point**.



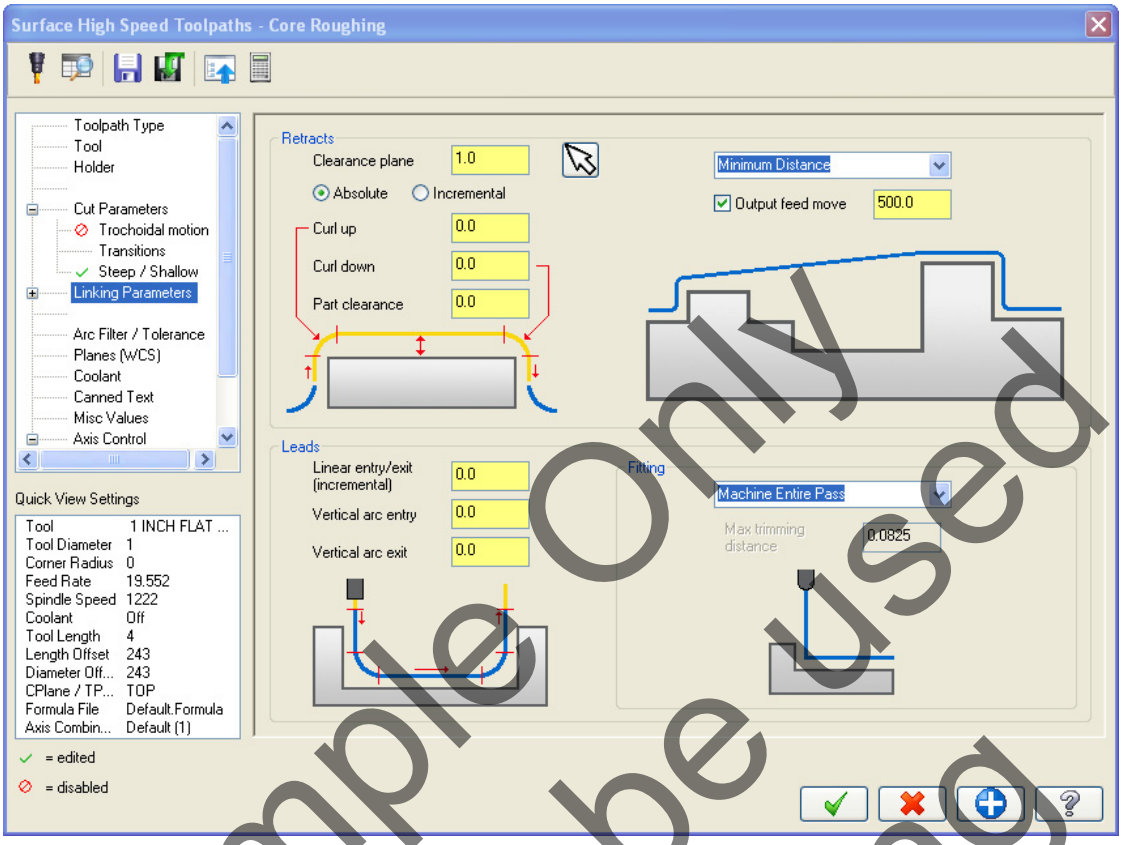
23. Select the Z at the top of the mold base as shown below:



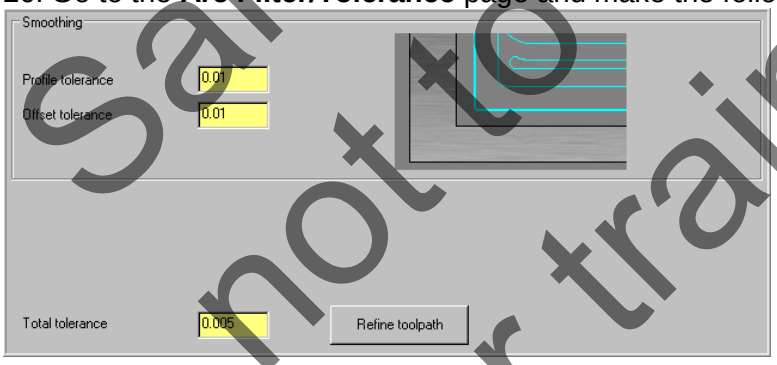
24. The resulting **Steep/Shallow** page values should match the following image:



25. Make the following selections on the **Linking Parameters** page: Open up the top drop down menu first and select **Minimum Distance**.



26. Go to the **Arc Filter/Tolerance** page and make the following selections:

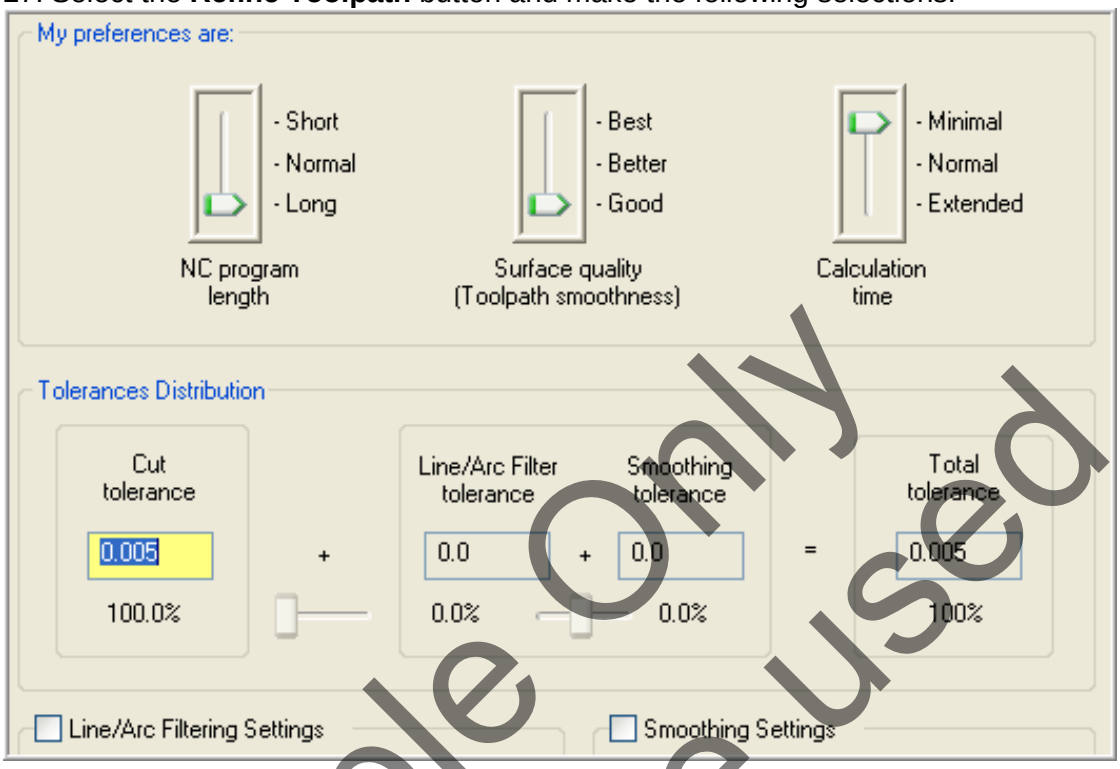


Profile Tolerance must always be set at least as tight as the stock to leave value.

For the purposes of this tutorial we will do all roughing at .005 overall tolerance to speed calculation times.

If cutting on a CNC machine, typical tolerance selections would range from .001 to .0001.

27. Select the **Refine Toolpath** button and make the following selections:



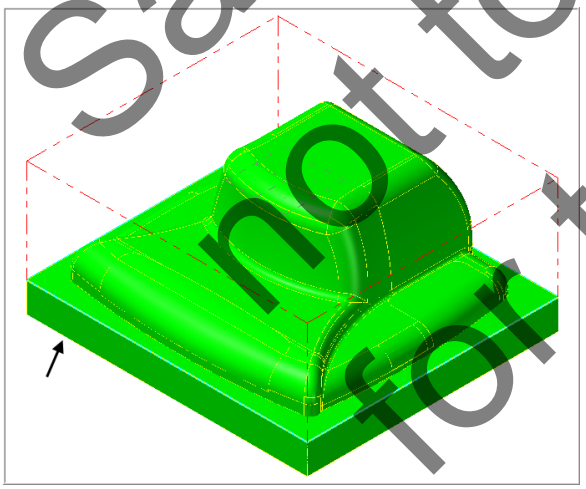
28. Select the **OK button** to exit the **Refine Toolpath** window.
29. Finally, move to the **Coolant** tab and turn **Flood** coolant on.
30. Select the **OK button** to complete the toolpath.
31. **Verify** the new toolpath.

TASK 5:
FINISH PROFILE SURFACES USING SURFACE HIGH SPEED (WATERLINE)

- Next you will finish the flat portions of the part using the Surface High Speed (Waterline) toolpath.
- You may find it useful to toggle the display of toolpaths on and off during this lesson. Do this by selecting **Alt-T** on your keyboard to hide/show the toolpath display.

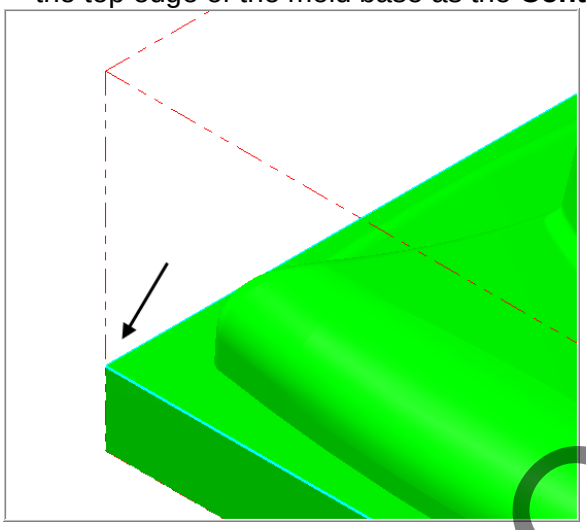
For programming, it is efficient to use Mastercam's various functions such as **Horizontal**, **Waterline** and **Rest Material** to isolate part features and machine them with the best strategy and tools.
Waterline toolpaths are best suited for surfaces whose angles are between 30 and 90 degrees. This is because the distance between passes is measured along the tool axis. Where the surfaces are shallower, material typically won't be removed as efficiently. However, you can configure the toolpath to generate extra cuts in shallow or flat areas.

1. In the **Operations Manager** confirm that the red arrow used to locate new operations is in the **Finishing Toolpath Group,** just after the first operation. If it is not, simply grab it with the left mouse button, and drag it to the desired location.
2. Create a new operation, by right clicking in the **Operations Manager** window, select **Mill toolpaths>Surface high speed toolpath...**.
3. Window select all elements on the screen when prompted to **Select Drive Surfaces** and click on **End Selection**  .
4. Select the **Check Surfaces** button and select the base solid as shown below. **Solid Selection turned on**.

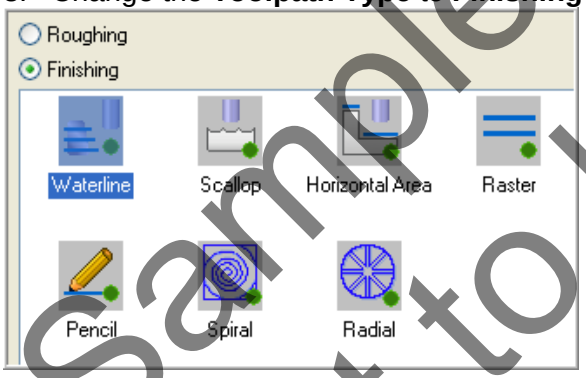


5. Click on **End Selection**  .

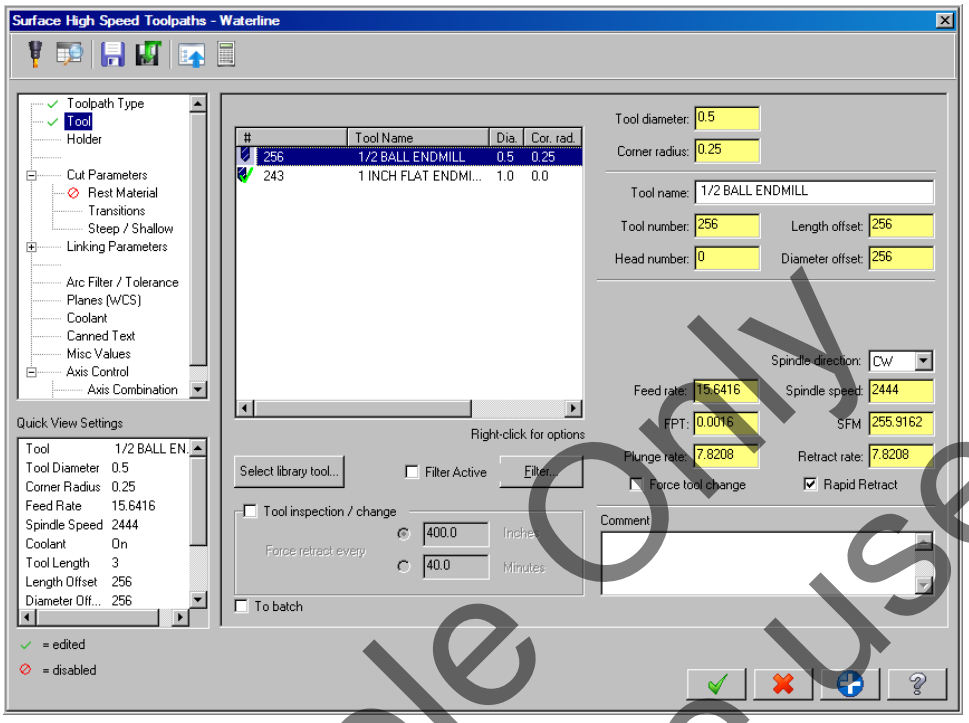
6. Now select the **Containment** button and select **C-plane** in the Chaining window. Next add the top edge of the mold base as the **Containment Boundary**.



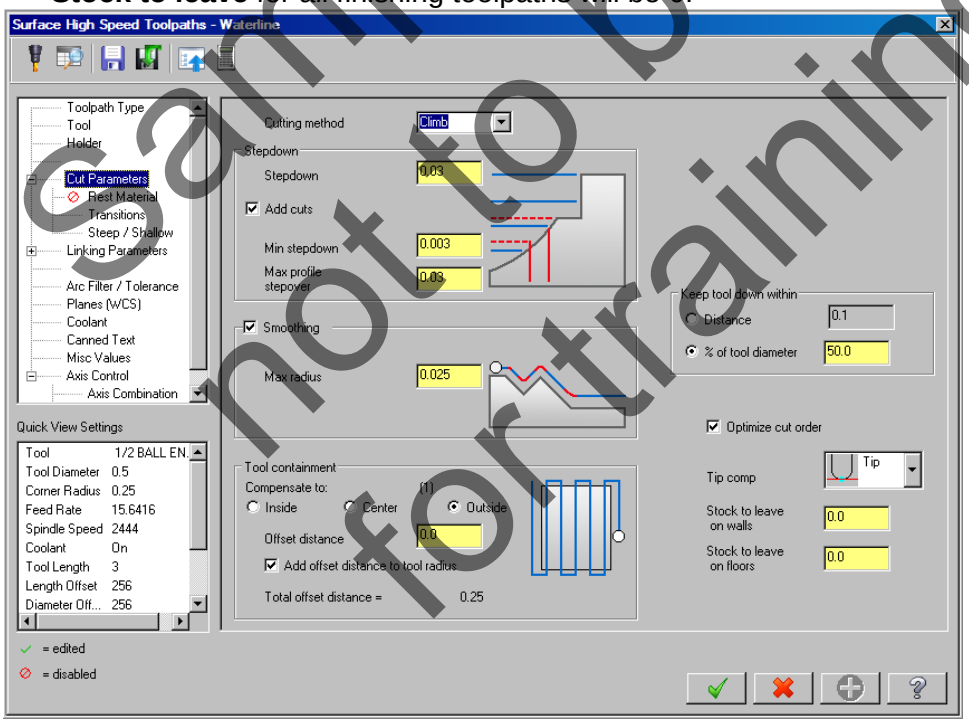
7. Select the **OK button** to exit Chaining and select the **OK button** to exit Toolpath/surface selection.
8. Change the **Toolpath Type** to **Finishing** and select **Waterline**.



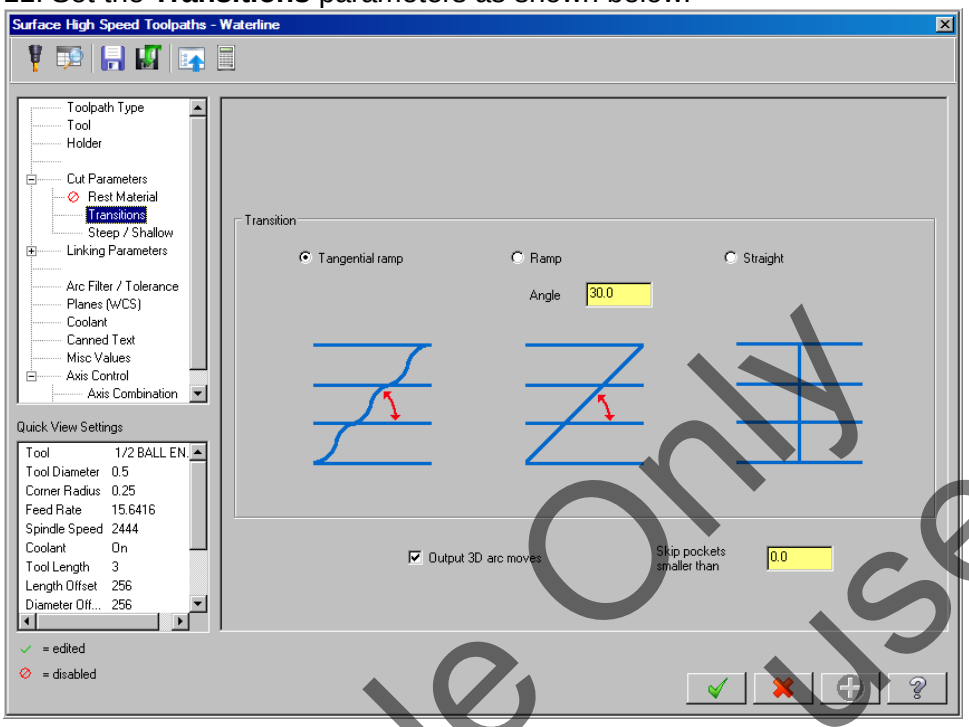
9. Navigate to the **Tool** page and use the **Select Library tool...** icon to select a 0.500 **Ball Endmill**.



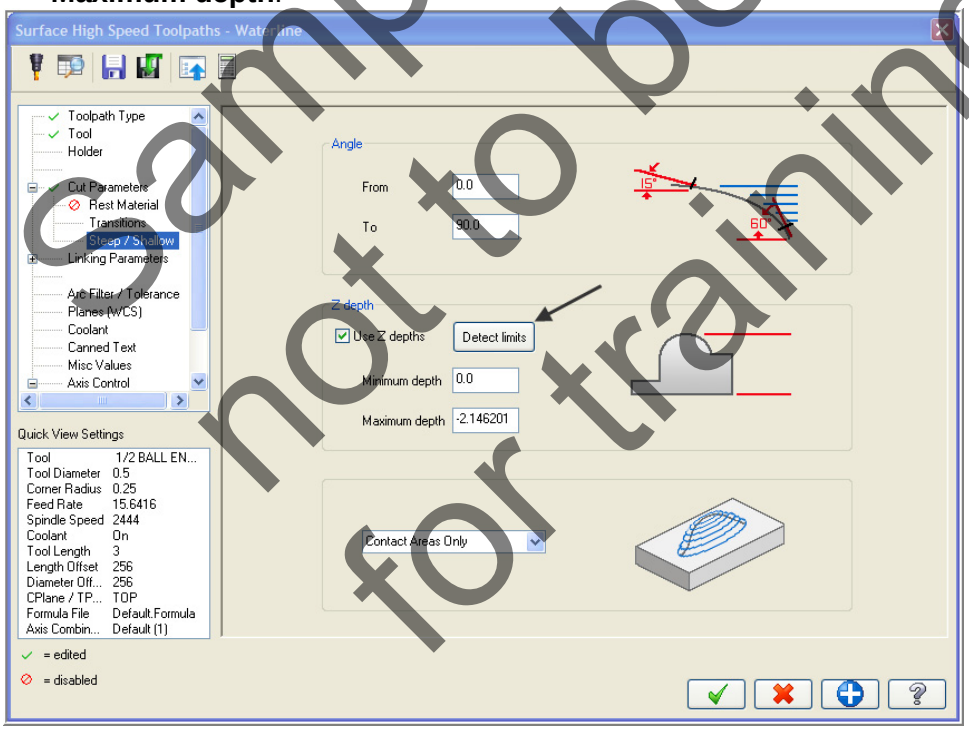
10. Make the appropriate selections on the **Cut Parameters** page shown below. Note that the **Stock to leave** for all finishing toolpaths will be 0:



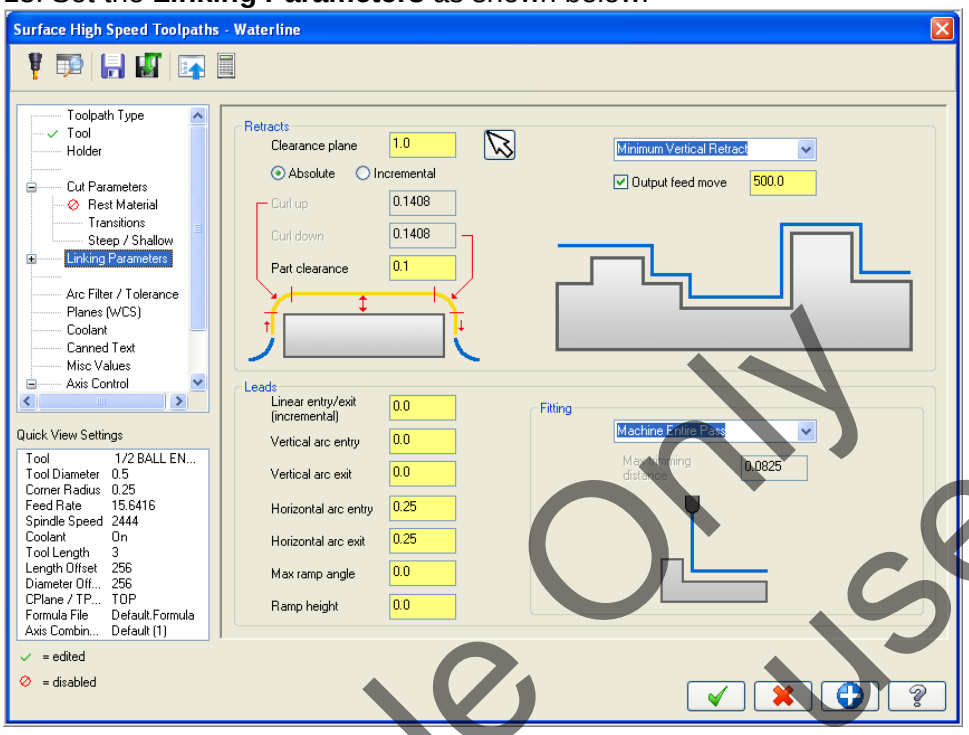
11. Set the **Transitions** parameters as shown below:



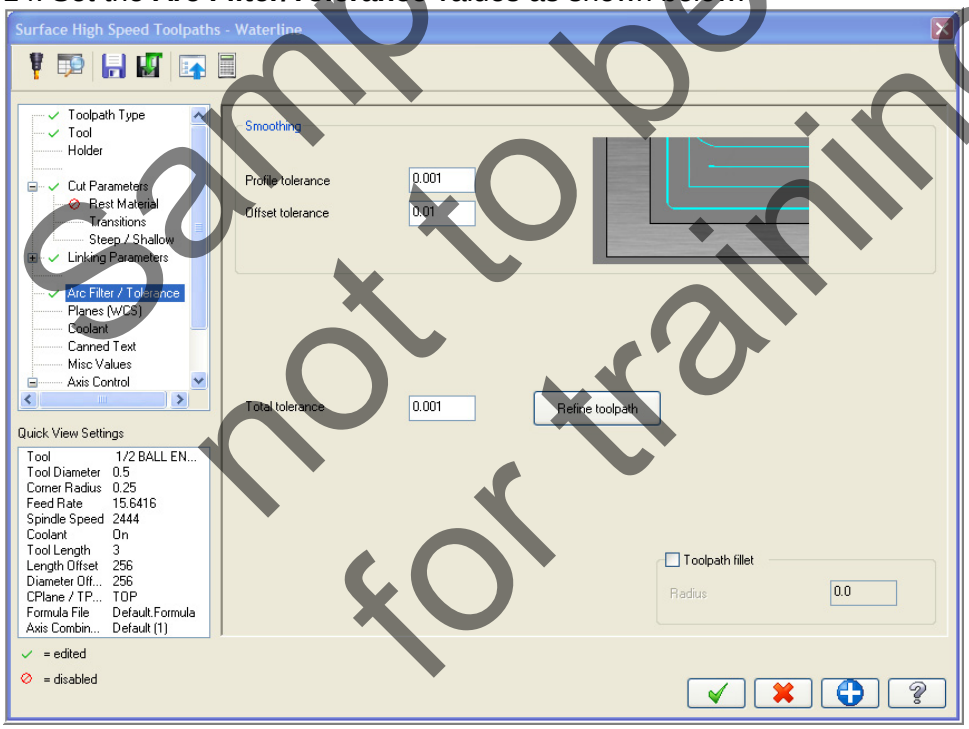
12. Select **Steep/Shallow** and select the **Detect limits** button to fill in the **Minimum** and **Maximum depth**:



13. Set the **Linking Parameters** as shown below:



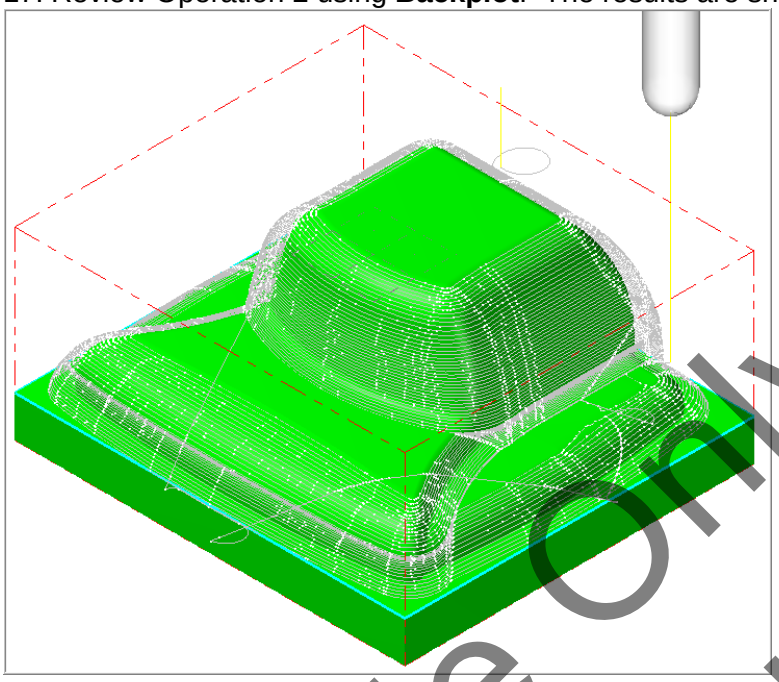
14. Set the **Arc Filter/Tolerance** values as shown below:



15. Navigate to the **Coolant** tab and turn the **Flood** coolant on.

16. Select the **OK button**  to complete the toolpath.

17. Review Operation 2 using **Backplot**. The results are shown below:

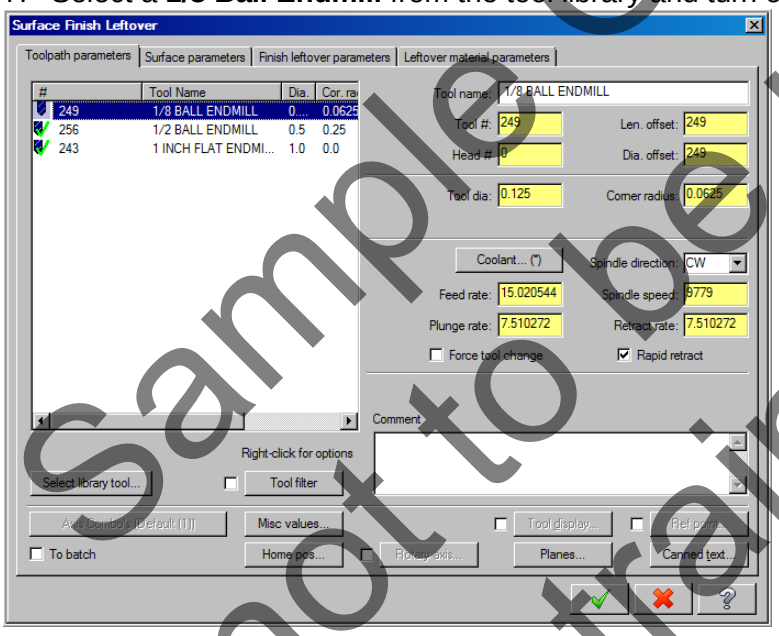


TASK 6:
FINISH ALL REMAINING STOCK USING SURFACE FINISH LEFTOVER

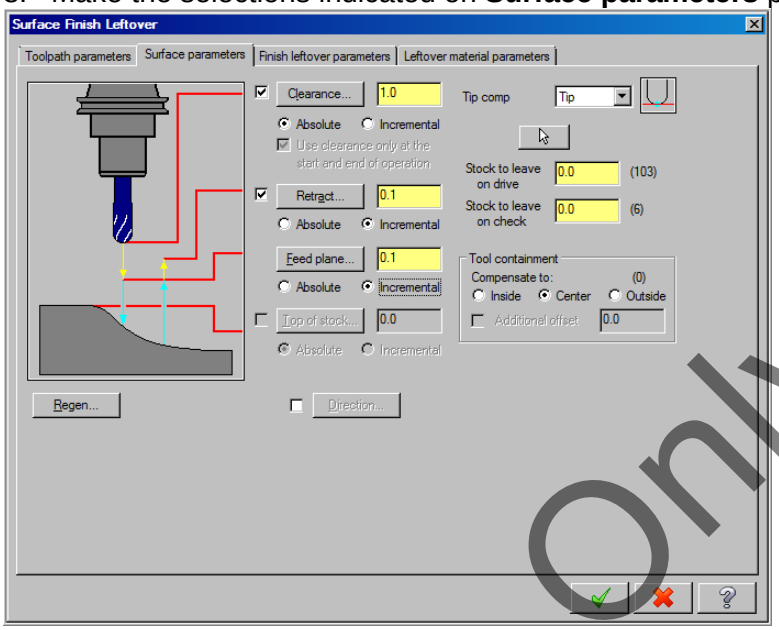
➲ Next you will finish leftover stock with a 0.125” ballnose tool.

Great care must be taken to run the next toolpath on a machine. The toolpath uses a very small ballnose tool which is difficult to setup and machine due to tool length, rigidity, and strength.

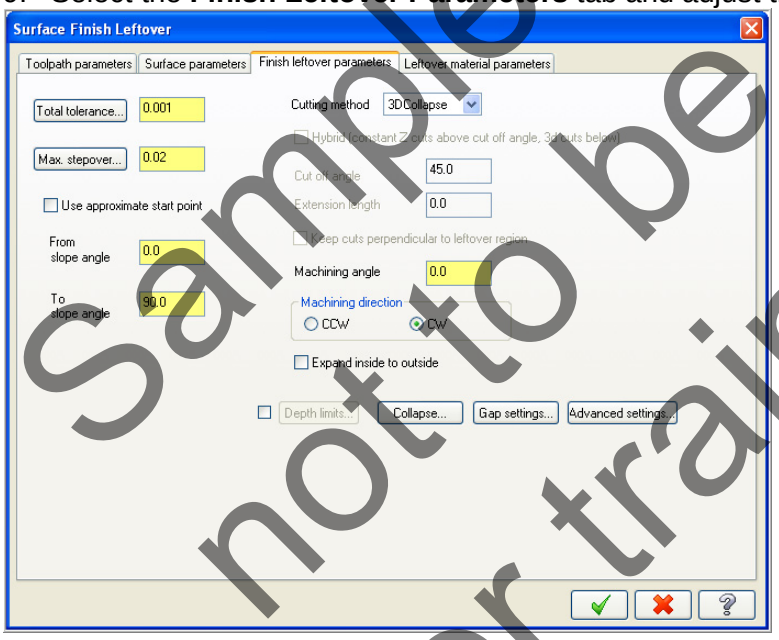
1. Select **Toolpaths>Surface Finish>Leftover**.
2. Select the **mold core solid** on **level 1** as the **drive surface**, leaving the base unselected.
3. Click on End Selection .
4. Select the **Check** select button and add the **lower mold base solid** on **level 1** as a **check surface**.
5. Click on End Selection .
6. Select the **OK button** to continue.
7. Select a **1/8 Ball Endmill** from the tool library and turn on the **Flood** coolant on.



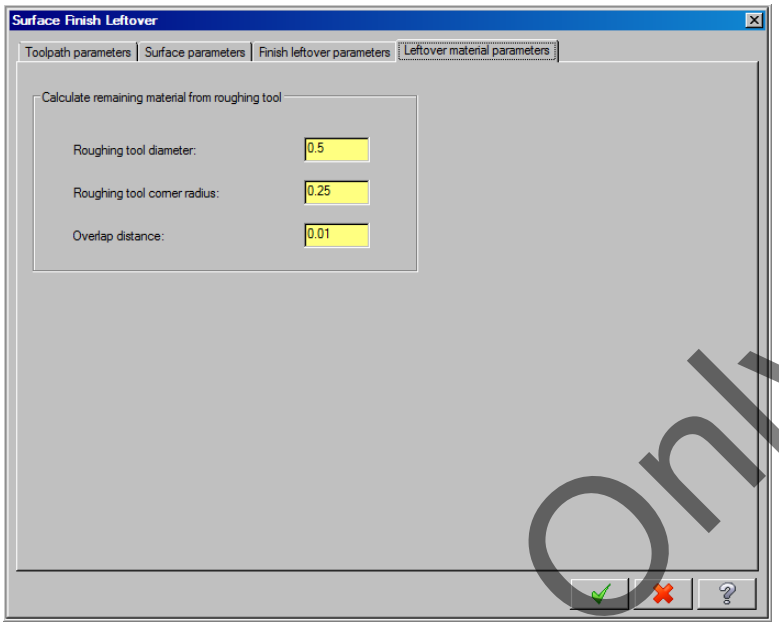
8. Make the selections indicated on **Surface parameters** page shown below:



9. Select the **Finish Leftover Parameters** tab and adjust the parameters as shown below:



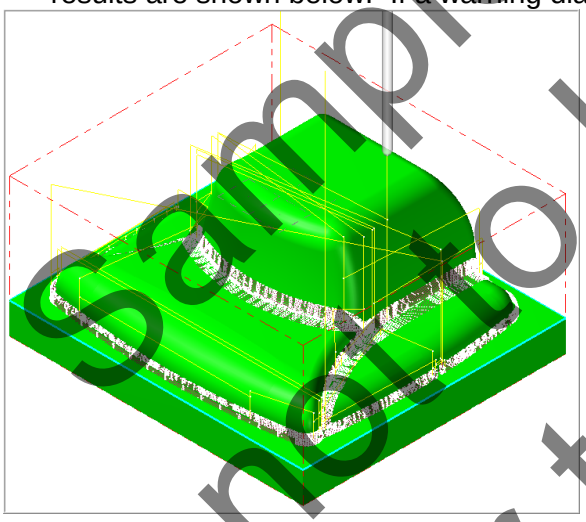
10. Select the **Leftover Material Parameters** tab and set the parameters as shown below:

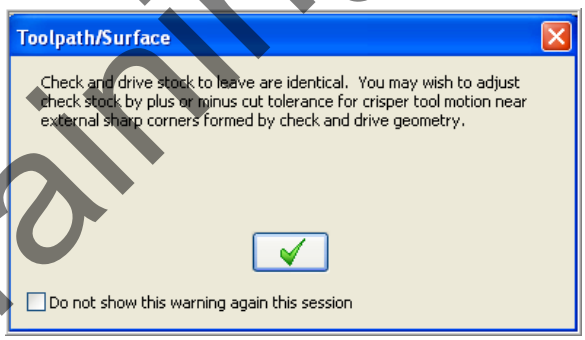


Similar to the **Surface High Speed** toolpaths this tab is used as a rough way of determining the stock left for machining.

The **Overlap distance** is used to calculate the stock based on a larger tool size to make the computed stock to machine increase.

11. Select **OK** to create the toolpath then **Backplot** after generation is complete. The results are shown below. If a warning dialog box appears select the OK to exit.

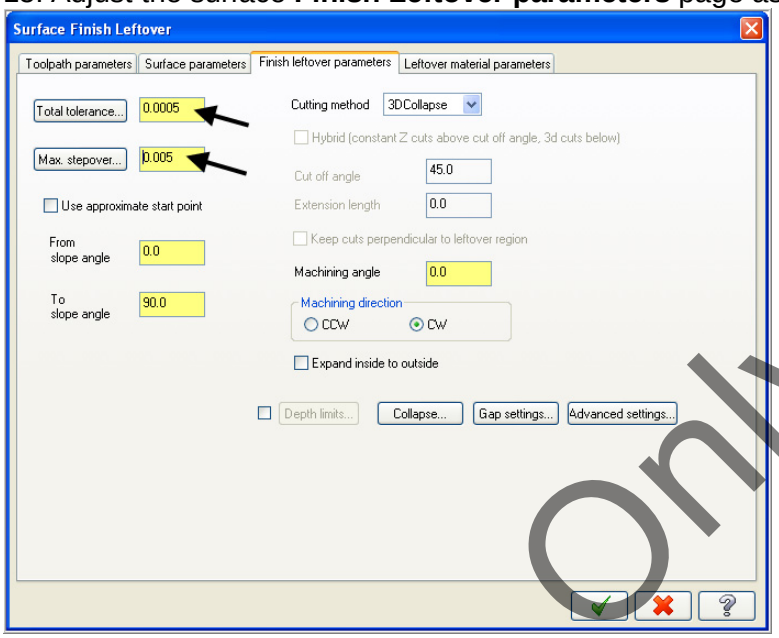




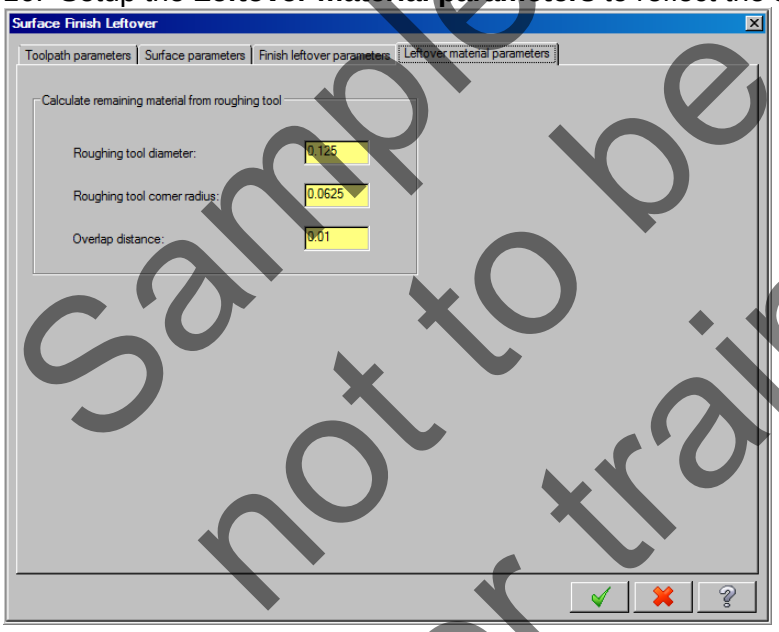
- Typically a final toolpath would be required to finish the smallest fillets. The process would require a smaller tool, and processing the toolpath with a tighter tolerance to get the desired results. As processing time will be high, please view the creation of this next toolpath as an optional exercise.

12. Copy the last toolpath, **Operation 3**, and paste it below in the **Operations Manager** creating **Operation 4**.
13. Select the **Parameters** option for **Operation 4**.
14. Change the tool to a **1/32 Ball Endmill**.

15. Adjust the surface **Finish Leftover parameters** page as shown below:

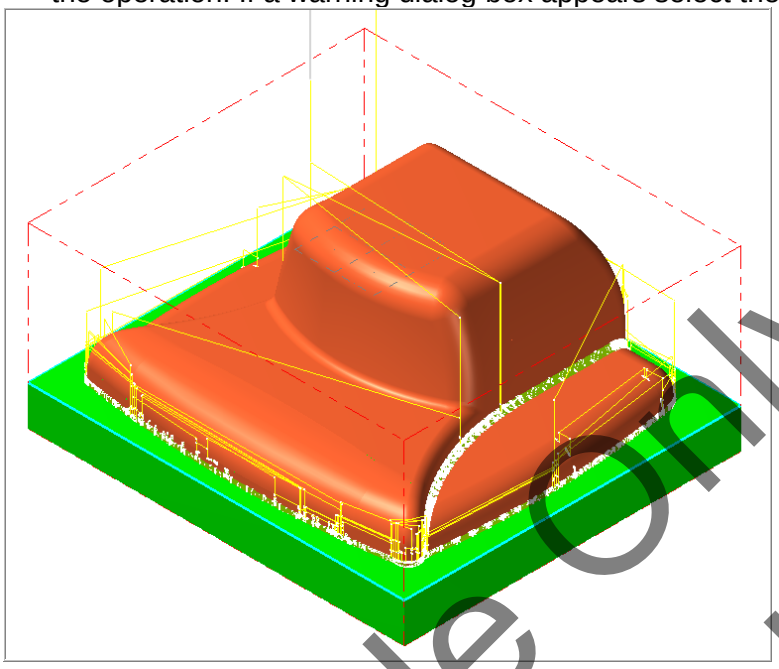


16. Setup the **Leftover material parameters** to reflect the size of the previous tool used:



17. Select **OK**  to exit the **Toolpath Parameters**.

18. Regenerate the toolpath by clicking on the **Regenerate all dirty operations** and **Backplot** the operation: If a warning dialog box appears select the OK to exit.

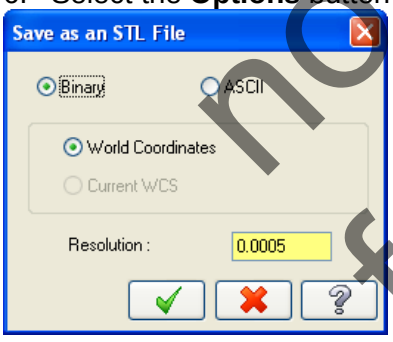


TASK 7:
VERIFY THE TOOLPATH AND COMPARE TO STL FILE

Using Mastercam's **Compare to STL file** function will allow us to identify areas on the part that require further machining as well as areas that may have been gouged. It is important to keep in mind that tolerances play a very large role in the use of this function. Accuracy of a comparison can only be as accurate as your machining tolerance.

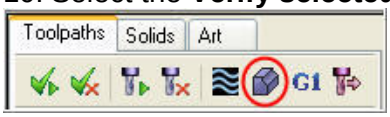
1. Use the **Alt+Z** shortcut to get to the **Level Manager**, turn on **level 1**, make all other levels invisible, exit the manager.
2. In the top menu bar, select **File>Save Some**. You will be prompted to **Select entities to save**. Select the two solids (the main part and the base).
3. Click on End Selection  .
4. The **Save As** dialogue box will open. Browse to your Mastercam install directory, then the **data** sub directory.
5. For **Save as type** select .STL file then save the file with the name **MILL-LESSON-15.STL**.

6. Select the **Options** button and make the selections below.



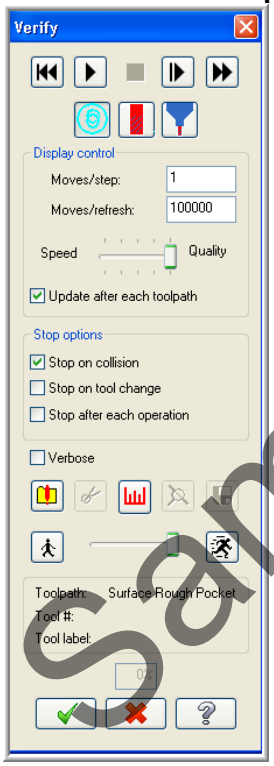
7. Select **OK**  to exit Options.
8. Select **OK**  to exit Save As
9. Select all of the operations you have completed so far by picking the **Select All** icon  .

10. Select the **Verify selected operations** button circled below:

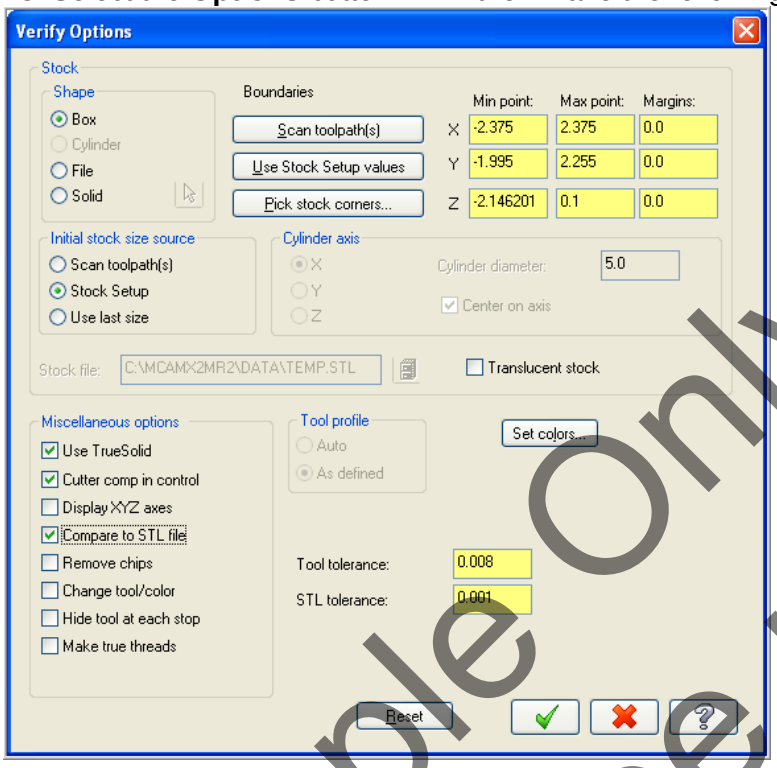


11. Select the **Turbo** button. Turbo does not display the tool or holder and does not perform live updates of the stock, which is updated at the end of the verification. Turbo is generally the fastest option for verification.

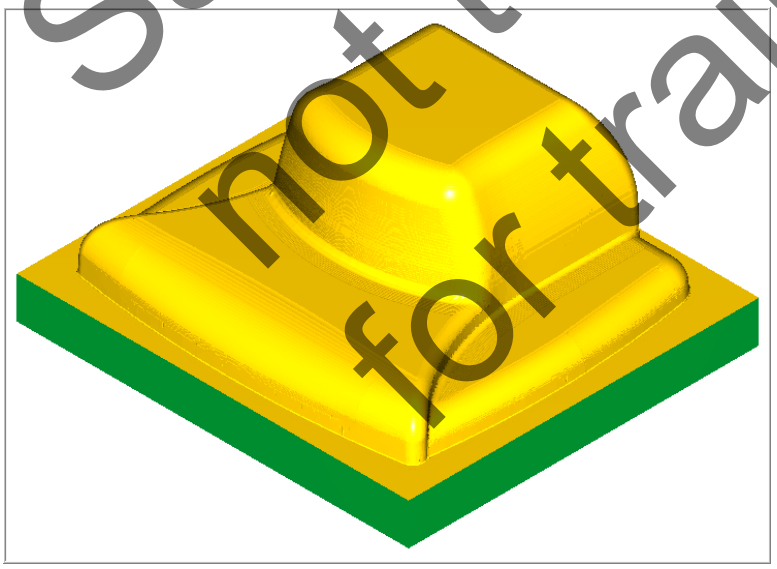
12. Check mark **Stop on collision** and set values as shown below:



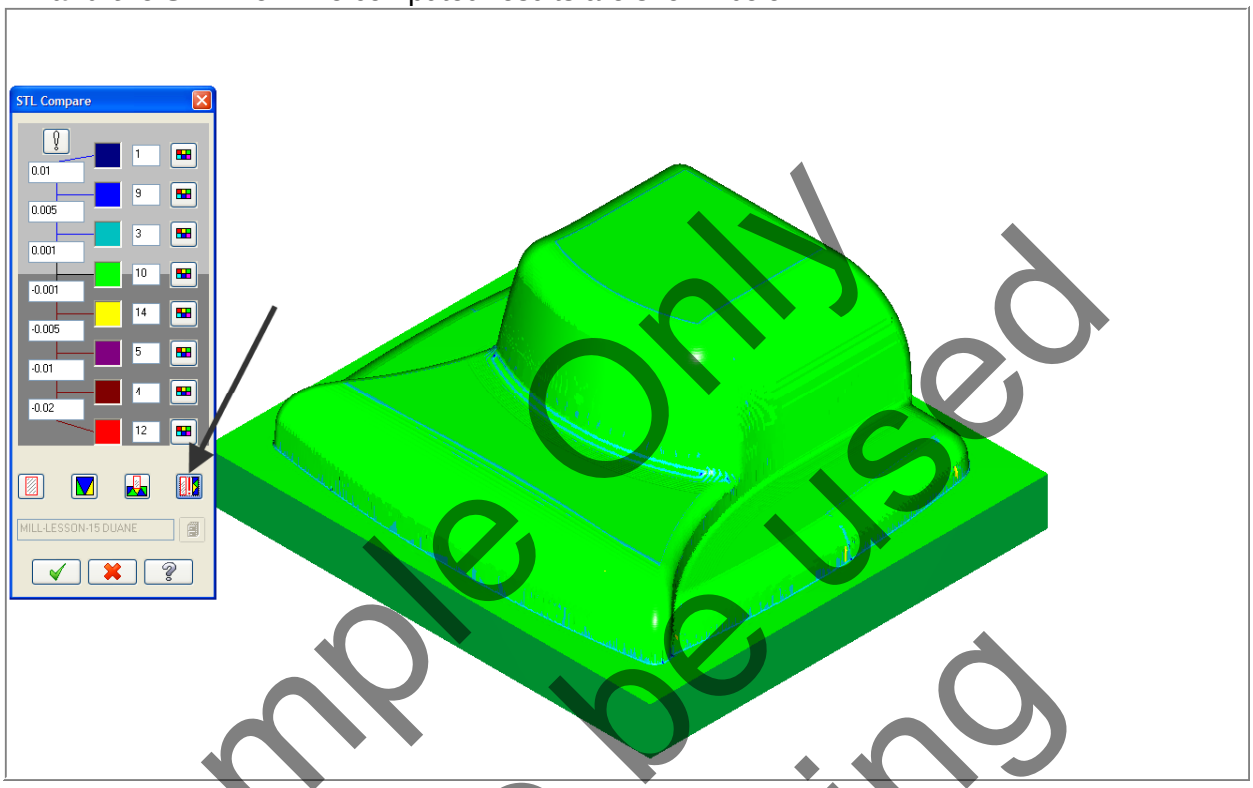
13. Select the **Options** button  then make the following selections:



14. Accept the selections by pressing the **OK button**  then select the **machine** button to verify the toolpath. The verified part is shown below:



15. The **STL Compare** dialogue box is now open. Select the **File** button  and select the .stl file that you saved in the earlier step.
16. The comparison stock model is now loaded. Select the **Compare the machined stock and the STL file**. The computed results are shown below:



Compare the colours on the model with those on the chart at the left.

The light to dark blue shaded areas indicate additional stock to remove.

The finishing tolerance was .001 so anything between light blue and yellow will be acceptable and considered complete.

Purple to red shaded areas would indicate part gouges and areas in previous toolpaths that need to be addressed!

17. Exit the **STL Compare** and **Verify** functions by clicking on the **OK button**  .

TASK 8:
SAVE THE UPDATED MCX FILE

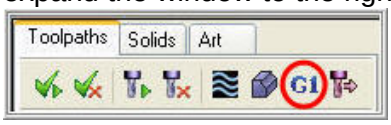
10. Select the save icon from the toolbar  .

TASK 9:
POST AND CREATE THE CNC CODE FILE

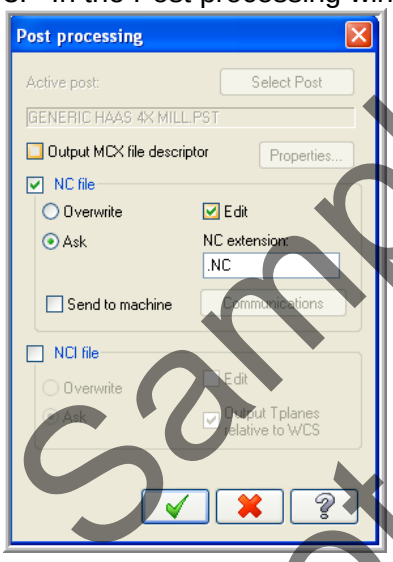
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.

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:

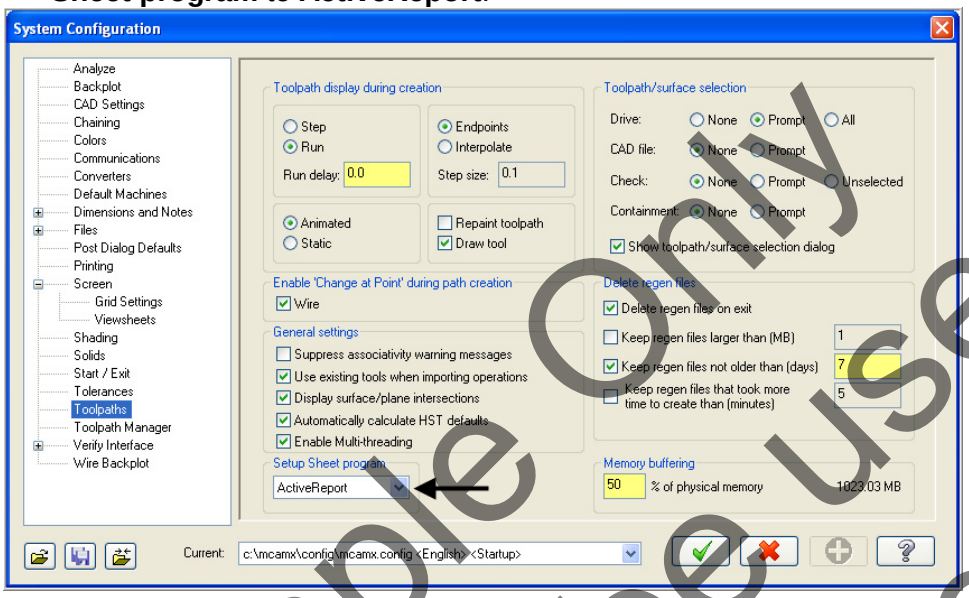


4. Select the **OK button**  to continue.
5. Enter the same name as your Mastercam part file name in the NC File name field **Mill-Lesson-15**.
6. Select the **Save** button.
7. The CNC code file opens up in the default editor.
8. Select the  in the top right corner to exit the CNC editor

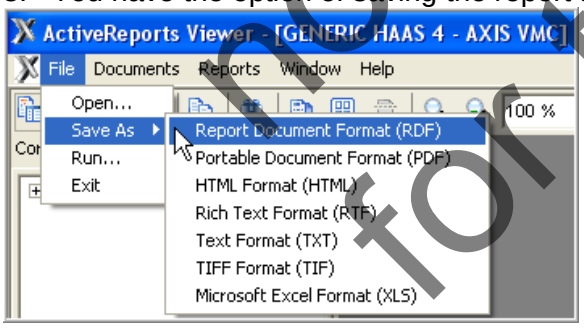
TASK 10:
CREATE ACTIVEREPORT

➲ Finally, you will create a report to help with part setup at the machine.

1. In the top menu bar, select **Settings>Configuration>Toolpaths** then change the **Setup Sheet program** to **ActiveReport**.



2. Select the **OK button** to exit.
3. Setup the screen so level 1 is the main and no other levels are visible. Exit the **Level Manager** and shade the solid.
4. Right click inside the **Operations Manager** window and select **Setup Sheet...**.
5. Select the **OK button** to generate the report.
6. The **ActiveReports Viewer** will load automatically. Note it may take a while to load.
7. Print the report and go through the various pages comparing the report information to the toolpaths in the **Operations Manager**.
8. You have the option of saving the report as any of the following file formats.



This completes Mill-Lesson-15.

MILL–LESSON-15 EXERCISE

➲ The **Mill-Lesson-15-Exercise** file can be found on the accompanying DVD in the Mastercam-Files folder and is called **Mill-Lesson-15-Exercise.mcx**.

Use the information learned in Lesson 15 to create a toolpath for the Mill-Lesson-15-Exercise.