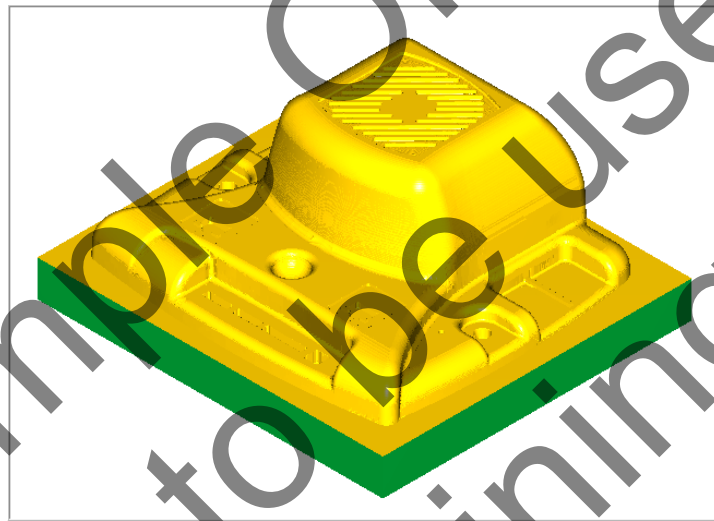


Mastercam X⁴

TRAINING

GUIDE



MILL-LESSON-16

**CORE ROUGHING, REST
ROUGHING, HORIZONTAL,
WATERLINE, AND SURFACE FINISH
LEFTOVER**

camInstructor

Objectives

You will use a provided model for Mill-Lesson-16, and 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.

➤ **Prepare model for cutting by:**

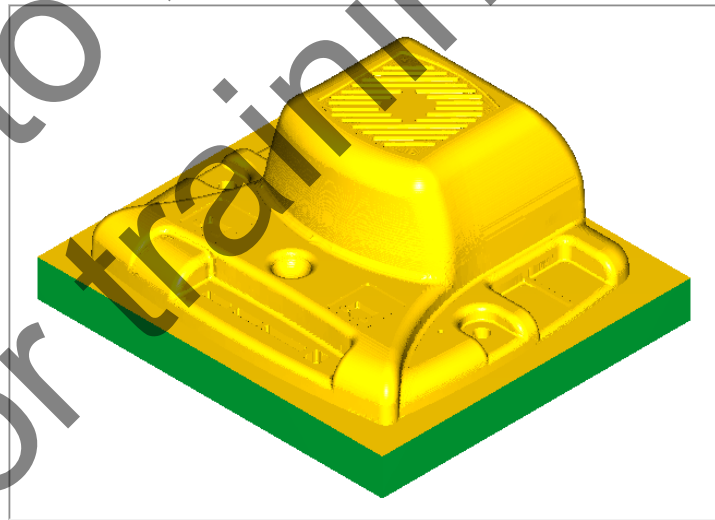
Creating surfaces from solid faces.
Using Silhouette Boundary for containment boundaries.
Creating flat boundary surfaces.

➤ **Generate 3-dimensional milling toolpaths consisting of:**

Importing toolpaths
Rest Roughing
Finishing Horizontal Area


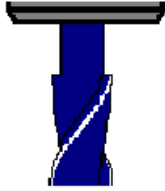
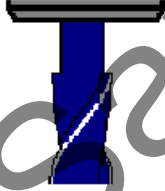



➤ **Inspect the toolpath using Mastercam's Verify by:**

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

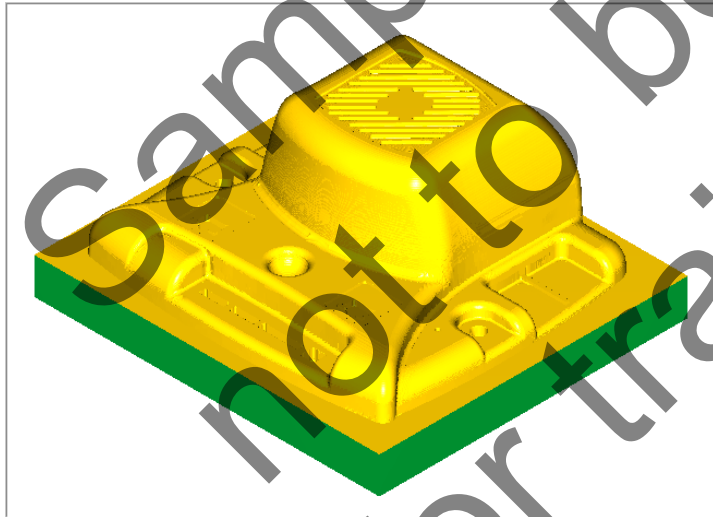
- ☉ 1.000 diameter flat end mill to rough and finish machine.
- ☉ .250 diameter flat end mill to rough and finish machine.
- ☉ .0625 diameter flat end mill to finish machine
- ☉ .500 diameter ballnose to finish machine
- ☉ .125 diameter ballnose to finish machine
- ☉ .0313 diameter ballnose to finish machine

Tool List of MILL-LESSON-16.MCX					
Proj./Part No. : 0	Date	: 06/23/09			
Drawing No. : 1	Customer	: 1 -			
Prog. No. : 16	Programmer	: 1			
	Tool type	: 0.0625 Endmill1 Flat 1/16 FLAT ENDMILL			
	Manufact.code	:			
	Chuck	:			
	Tool Number	: 230	Feedrate	: 18.7757	
	Diameter	: 0.0625	RPM	: 19558	Plunge feed r.:
	Corner radius	: 0	Tip angle	: 0	Diam. offset
	Flute length	: 0.375	Material	: ALUMINUM ...	Length offset
	Overall length:	2.25	No flutes	: 4	
	Tool type	: 0.25 Endmill1 Flat 1/4 FLAT ENDMILL			
	Manufact.code	:			
	Chuck	:			
	Tool Number	: 235	Feedrate	: 19.556	
	Diameter	: 0.25	RPM	: 4889	Plunge feed r.:
	Corner radius	: 0	Tip angle	: 0	Diam. offset
	Flute length	: 0.5	Material	: ALUMINUM ...	Length offset
	Overall length:	2.5	No flutes	: 4	
	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.:
	Corner radius	: 0	Tip angle	: 0	Diam. offset
	Flute length	: 2	Material	: ALUMINUM ...	Length offset
	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.:
	Corner radius	: 0.0156	Tip angle	: 0	Diam. offset
	Flute length	: 0.375	Material	: ALUMINUM ...	Length offset
	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.:
	Corner radius	: 0.0625	Tip angle	: 0	Diam. offset
	Flute length	: 0.375	Material	: ALUMINUM ...	Length offset
	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.:
	Corner radius	: 0.25	Tip angle	: 0	Diam. offset
	Flute length	: 1	Material	: ALUMINUM ...	Length offset
	Overall length:	3	No flutes	: 4	

MILL-LESSON-16 - 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:** Prepare geometry for roughing and rest roughing
- TASK 5:** Importing the Core Roughing toolpath
- TASK 6:** Rest Roughing using Surface High Speed (Rest Roughing)
- TASK 7:** Apply Mastercam High Feed functionality
- TASK 8:** Finish flat surfaces using Surface High Speed (Horizontal)
- TASK 9:** Importing Waterline and Surface Finish Leftover toolpaths
- TASK 10:** Verify the toolpath and compare to STL file
- TASK 11:** Save the updated MCX file
- TASK 12:** Post and create the CNC code file
- TASK 13:** 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.

Please Note:

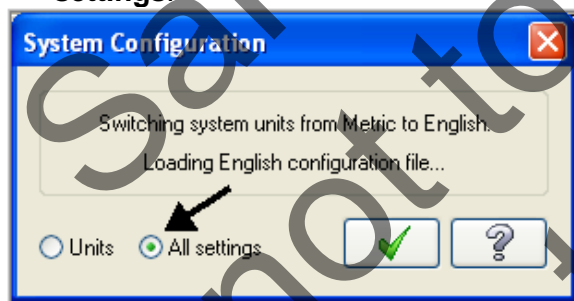
Review the videos on the CD **before** working through each task.

There is some intricate geometry selection techniques used in this lesson and reviewing the procedures on the CD will make it easier for you to complete this lesson.


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-16.MCX**.
 2. If confronted with the System Configuration dialog box activate the radio button for **All settings**.



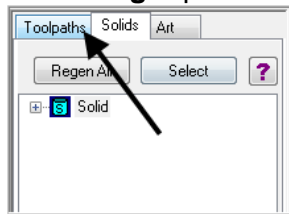
- **System Configuration (switch units) dialog box**
- When you open a part file that uses different units (English or metric) from those currently in use, Mastercam automatically displays this dialog box, which informs you that Mastercam is switching units and loading an alternate default configuration 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.

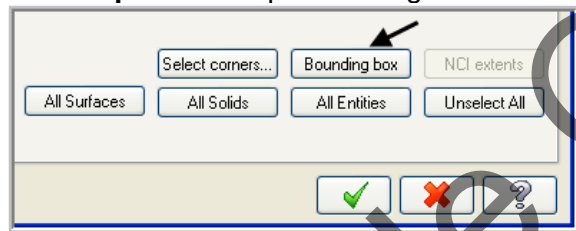


TASK 3: DEFINE THE ROUGH STOCK USING STOCK SETUP

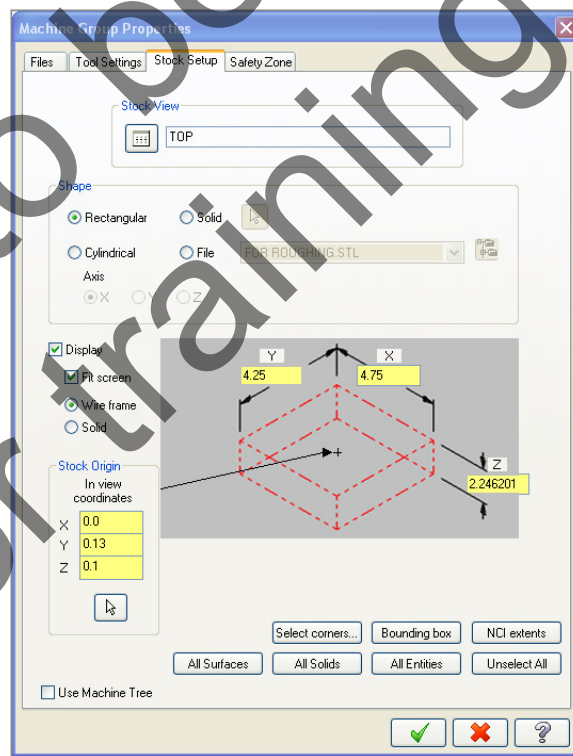
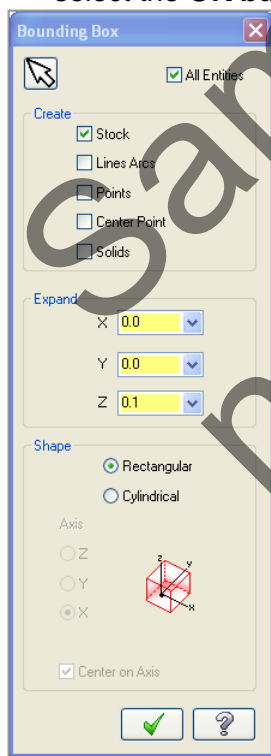
1. Click on the Toolpaths Tab as shown below: Note; **Alt-O** will **Show/hide** the **Operations Manager** pane.



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:

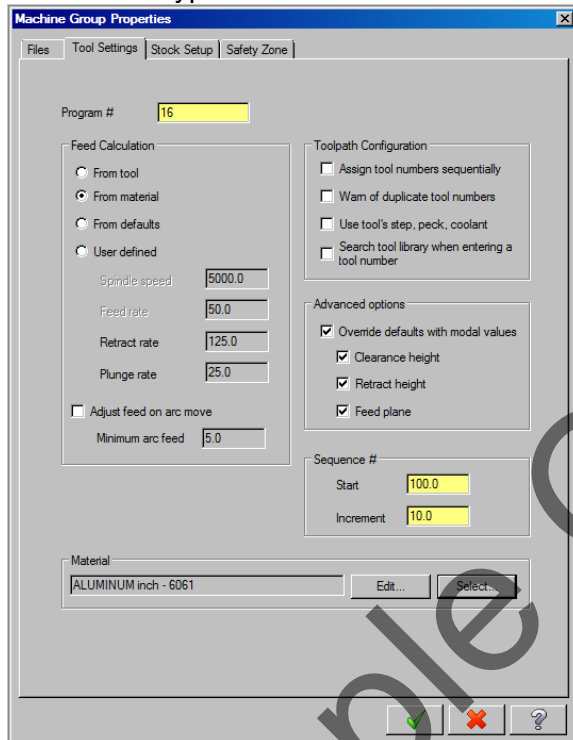


3. Set the parameters to match the **Bounding Box** screenshot in the left image below, then select the **OK** button.




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**.

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

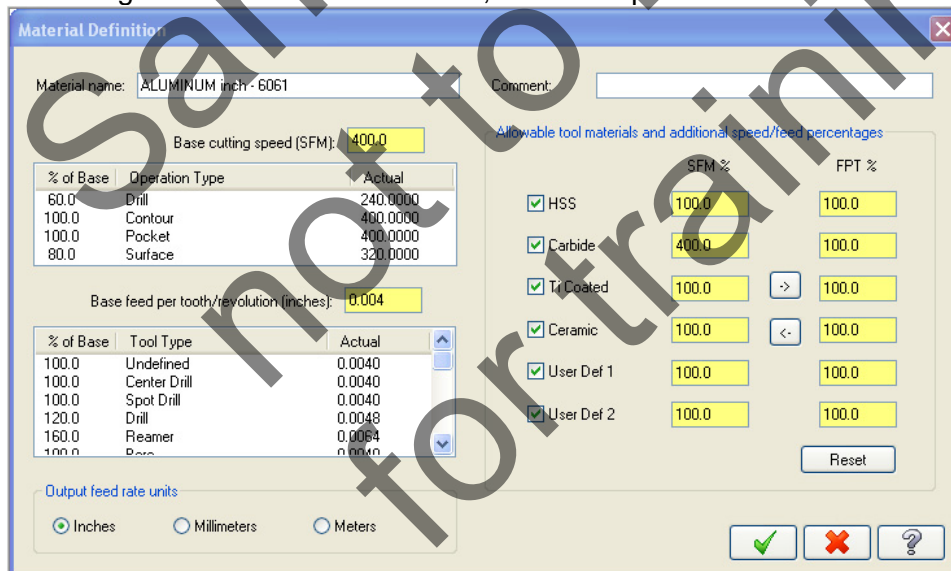


To change the **Material** type to **Aluminium 6061** pick the **Select** button at the bottom of the **Tool Settings** page.

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 .

- 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.

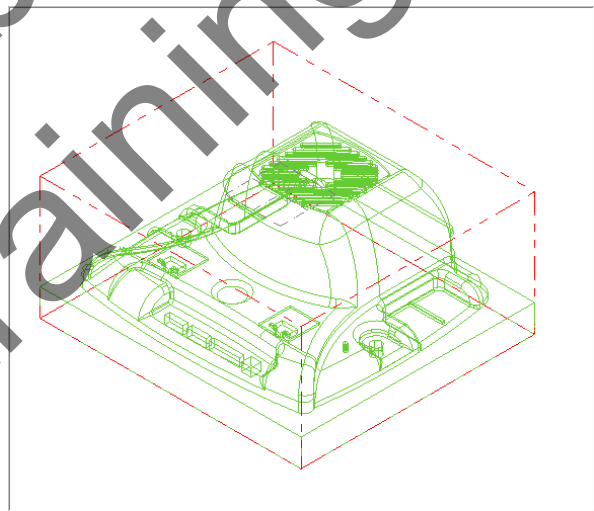
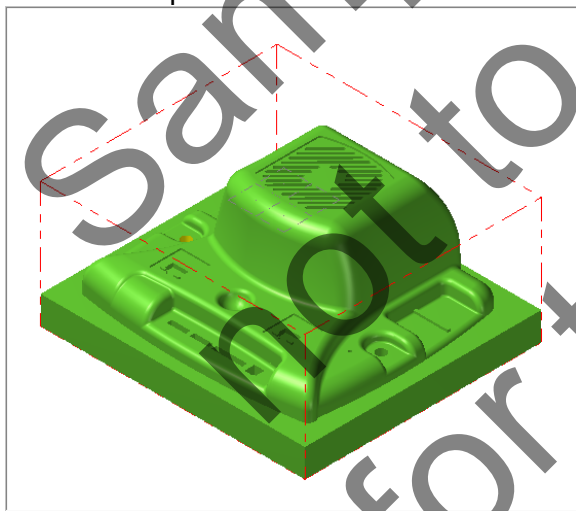
% of Base by Tool Type allows the user to vary feed per tooth by the tool. Eg. endmills will typically have a much higher feed per tooth (FPT) than a ballnose tool will.

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 4**.

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: PREPARE GEOMETRY FOR ROUGHING

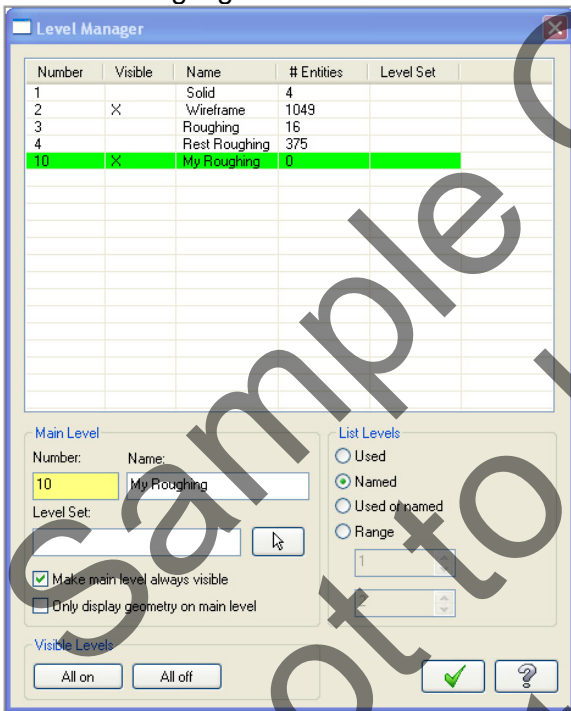
- In this task you will use create surfaces and containment boundaries for machining.
- The first step will be to cover areas of the core that will later be machined by electrode.

Please Note:

Review the videos on the CD **before** working through each task.

There is some intricate geometry selection techniques used in this lesson and reviewing the procedures on the CD will make it easier for you to complete this lesson.

1. From the bottom Status menu bar select **Level**.
2. In the **Number** field type **10**, and in the **Name** field type **My Roughing**. Level 10 will become highlighted to indicate that it is the main level.






The **Level Manager** is a good way to organize your geometry for easy reference.

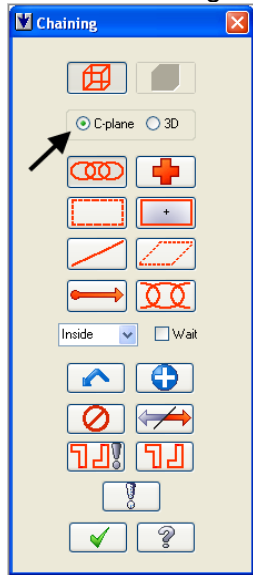
The highlighted level is the main level. Any geometry created always goes onto the current main level. To select a level as main simply click on the number of the level in the level list.

The X under the column called **Visible** indicates whether or not the geometry on a certain level is showing. The main level is always visible.

It is possible to leave the **Level Manager** open at all times.

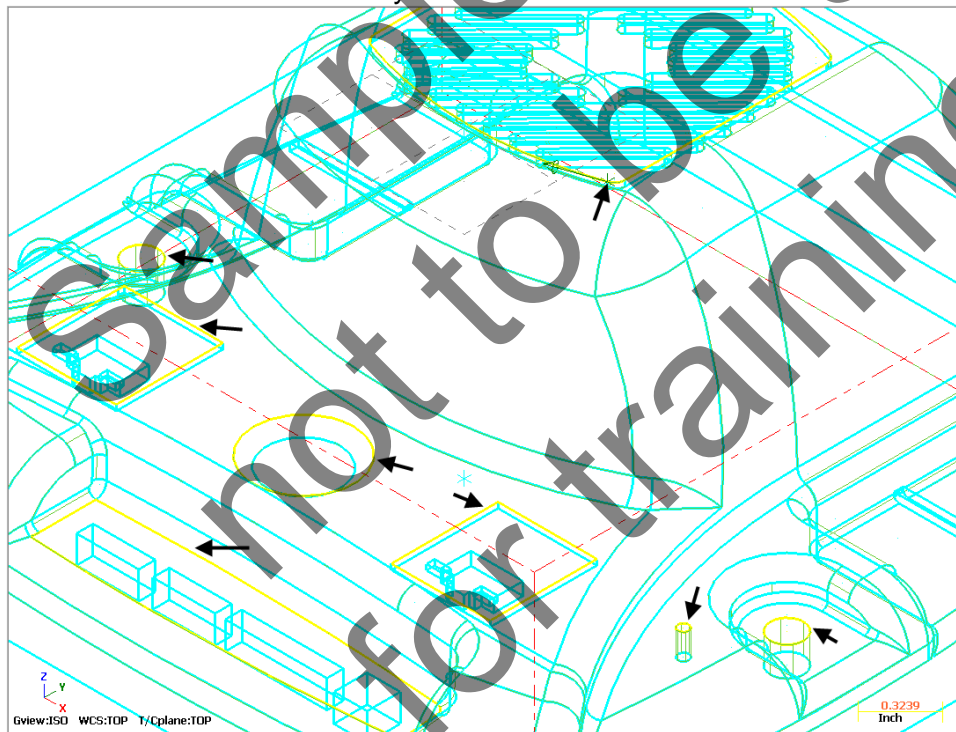
3. Click on the X under the visible column to hide the other levels except 2. Be sure to keep the new level as the main. Click the **10** to do so.
4. Select the **OK button**  to exit.
5. Select a bright colour by left clicking on the **System Colour** in the bottom Status menu bar.
6. Select colour **137**, , then select the **OK button**  to exit.
7. From the top menu bar select **Create>Surface>Flat Boundary...**

8. The chaining dialogue will appear. Select **C-Plane**.



9. Unshade the model using **Alt+S**.

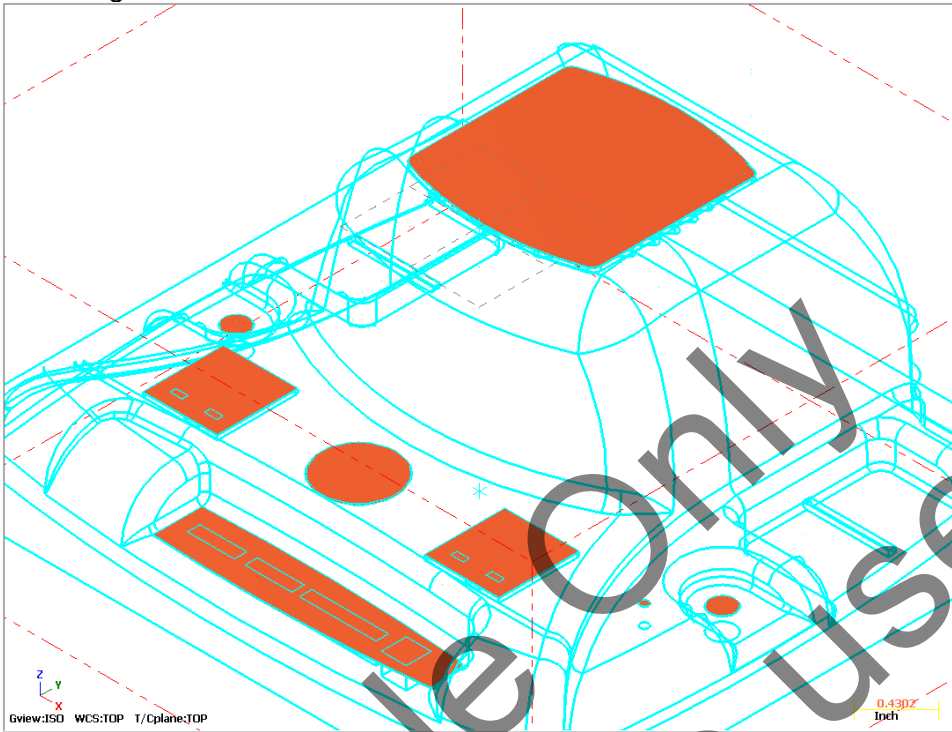
10. Select the wireframe in any order as shown below:



11. Click on **OK** to exit **Chaining**.

12. Select the **OK** button to create the surfaces and exit.

13. Using **Alt+S** to shade the model will show the flat surfaces created:

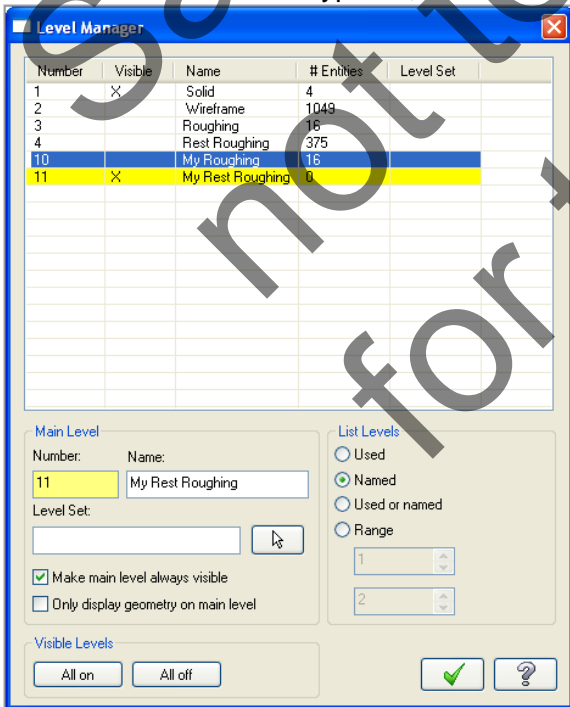


14. Now that you know how to turn levels on and off, check your work by comparing with the surfaces on level 3.

➔ Next we will create surfaces to be used for machining and creating containment boundaries.

15. From the bottom Status menu bar select **Level**.

16. In the **Number** field type **11**, and in the **Name** field type **My Rest Roughing**.



- Click on the X under the visible column to hide all of the levels except 1. Be sure to keep the new level as the main. Click the **11** to do so.
- Select the **OK button** to exit.

Please Note:

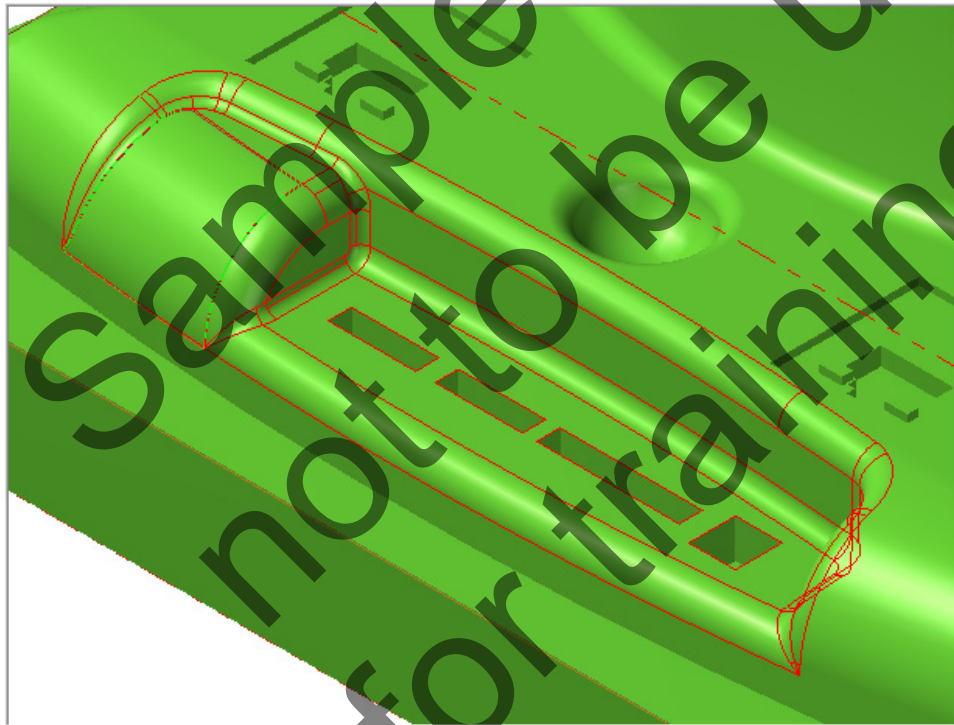
Review the videos on the CD **before** working through each task.

There is some intricate geometry selection techniques used in this lesson and reviewing the procedures on the CD will make it easier for you to complete this lesson.

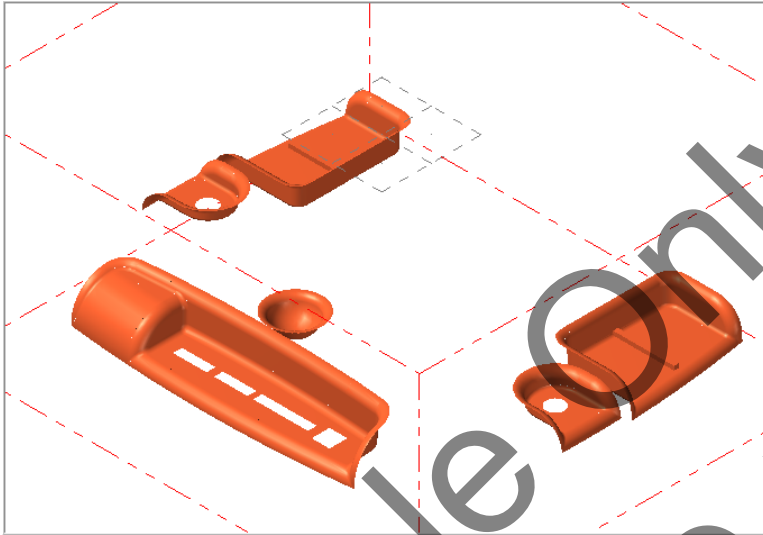
- Select **Create>Surface>From Solid...**
- Be sure that only the **Solid Face** selection method is active.



- To begin carefully select all of the surfaces in the front pocket. There are some very small ones so carefully zooming in to each area will help. If you make an incorrect selection, selecting the same surface again will unselect it.

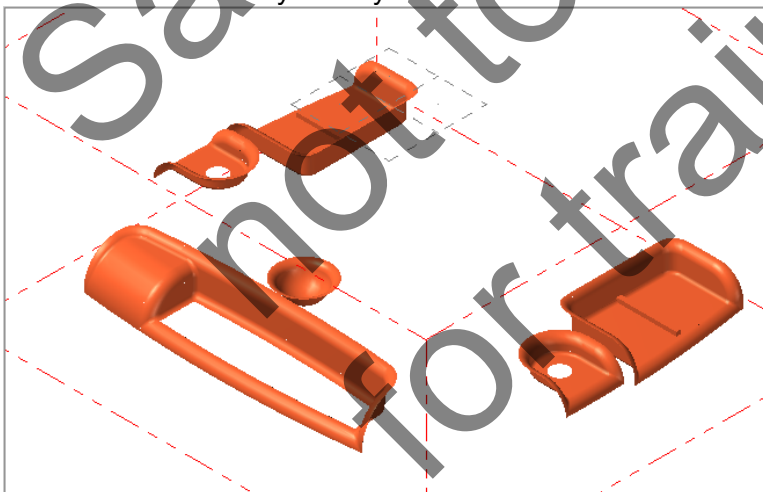


22. Check your work by comparing with the front surface on level 4. There are 44 surfaces in total. If you have missed a surface, repeat steps 19 through 21 adding only the surfaces that you have missed. **Do not worry about the floor surface with the holes in it. We will fix this later.**
23. Repeat **steps 19 through 21** to create surfaces from the solid faces on the other pockets.
24. Again, check your work by comparing with level 4 and the picture below. **Do not worry about the floor surface with the holes in it.**



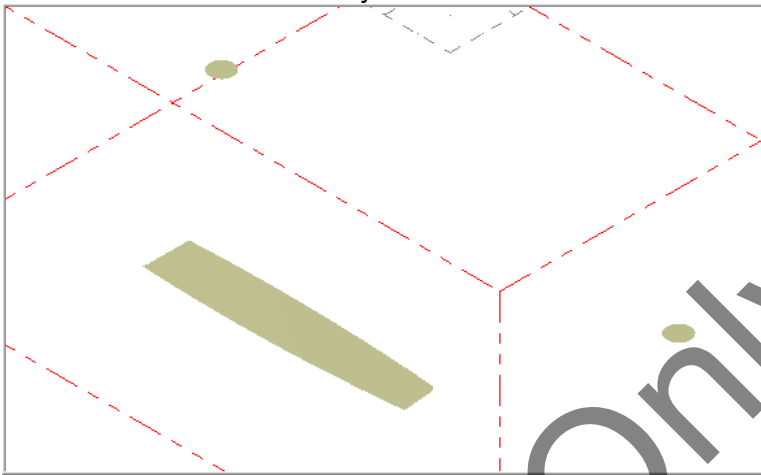
In order to avoid machining areas that will be burned by electrode, we will fill in the floors of our rest roughing surfaces just as we did for the roughing. Clean, continuous, untrimmed geometry will always give better toolpath processing results.

25. Delete the floor surface at the front of your part as shown below. Click on it then hit the **Delete** button on your keyboard.

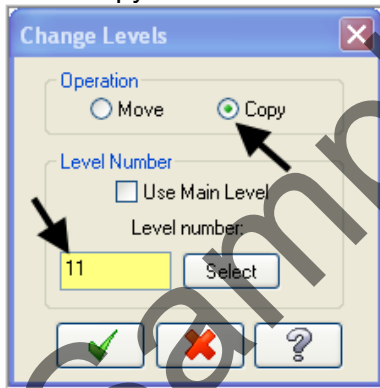


26. Open the **Level Manager** and make level **10** visible while keeping level **11** as the main.

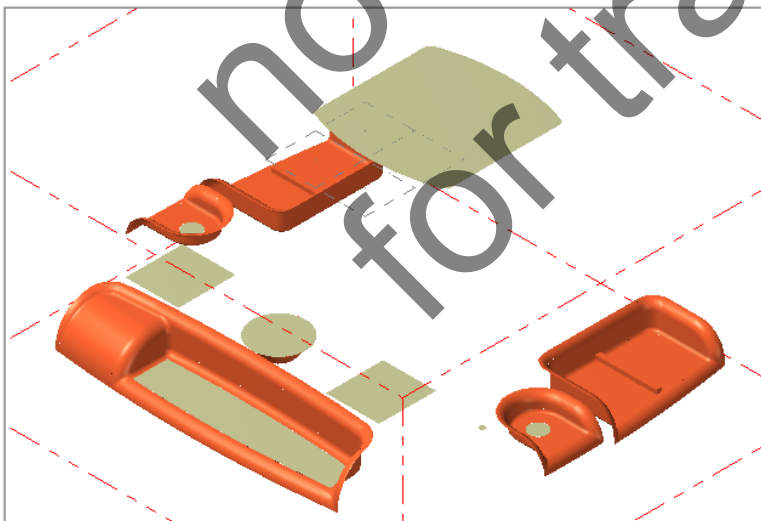
27. Pre-select the **three surfaces** that will fill the holes in your rest roughing surfaces. They are shown below: **Please Note:** In the screen shot below some of the surfaces have been hidden the sake of clarity.



28. Right click on **Level** in the bottom Status menu bar then make the selections shown below to copy the surfaces to level 11 for rest roughing.



29. Press **OK** to exit.
30. Your screen should now look like this:



➤ Next we will create containment boundaries.

31. Enter the **Level Manager** and ensure that level 11 is the main level and all others are not visible.

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

33. Select 2D and set the Z level for geometry creation as 0.



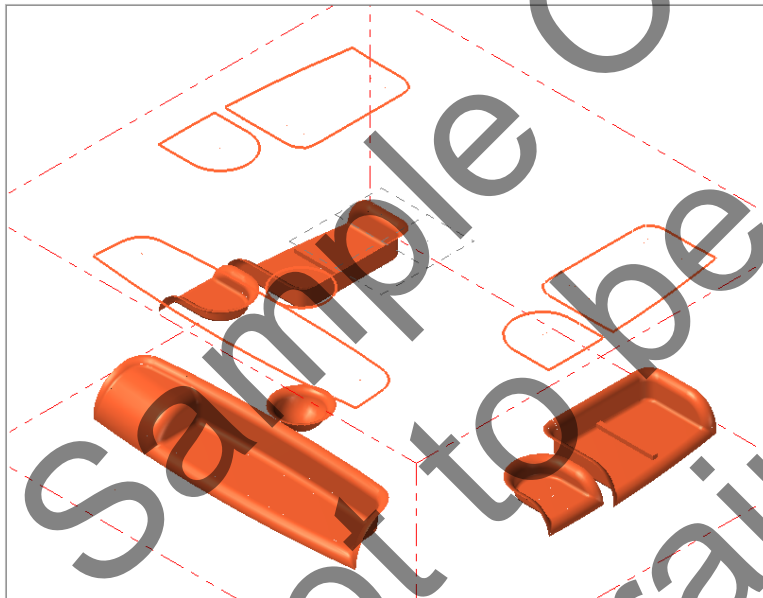
34. Select **Create>Silhouette Boundary...**

35. You are prompted to **Select solids, solid faces, or surfaces.**

36. Select all of the surfaces on the screen. To window select the geometry click in the top left corner of the graphics window and drag the mouse to the bottom right before clicking again.

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

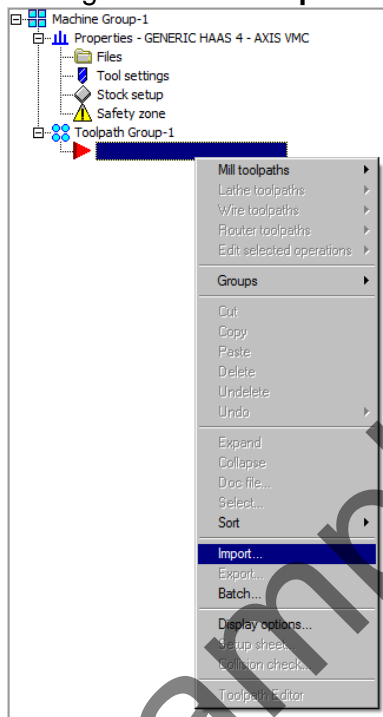
38. Select the **OK button** to select the default **Silhouette Boundary** options. Your screen should look the same as below:



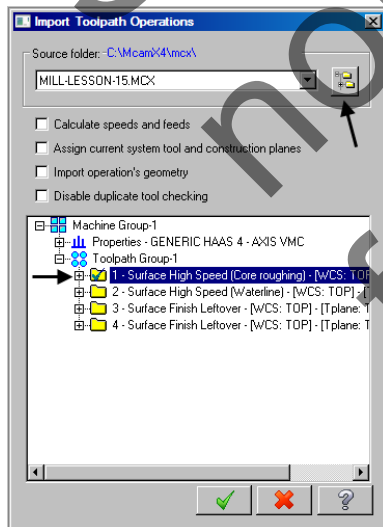
TASK 5: IMPORTING THE CORE ROUGHING TOOLPATH

➤ In this task you will import the core roughing toolpath used in Lesson 15 and alter it to rough the part.

1. Enter the **Level Manager** and ensure that level 1 is the main level and all others are not visible
2. Right click in the **Operations Manager** and select **Import** from the list as shown below:

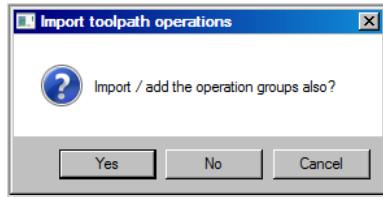



3. The **Import Toolpath Operations** window will now appear. Using the **Select Operations file** button browse and select the source folder that has the **Mill-Lesson-15.mcx** file stored within it and select the first operation, **Surface High Speed (Core Roughing)**.



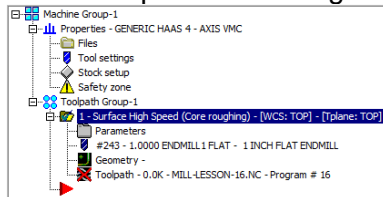
4. Select the **OK button**  to import the operation.

5. You will be asked whether you want to Import/add the operation groups also. Select **No**:



6. A message will appear indicating 1 operation has successfully been imported. Hit the **Cancel button**  to return to the operations manager.

7. Your operation manager should now have one dirty operation as shown below:



8. Select **Geometry** from the first operation.

9. First select the **Drive Surfaces** select button. When prompted to **Select Drive surfaces**, window select all of the entities on the screen.

10. Click the **End Selection** icon .

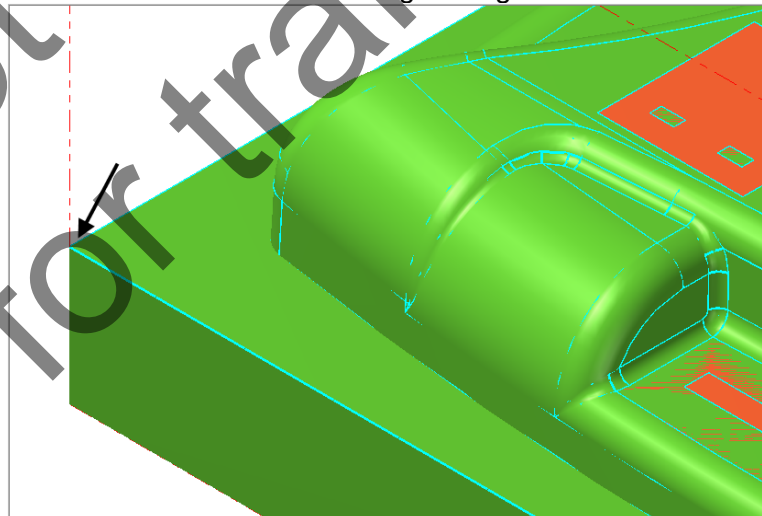
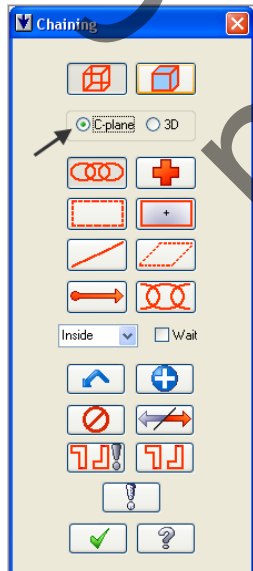
11. Click on the **Containment** Button.



12. The wireframe level is not currently active to select the containment boundary. Select **Level** from the bottom Status menu bar. Then make **level 2** visible.

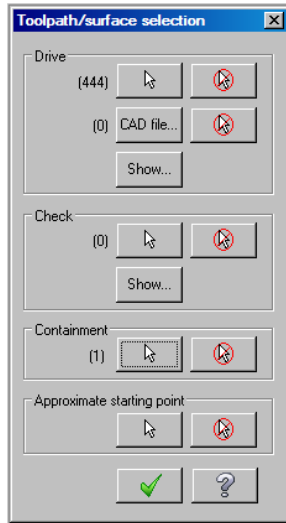
13. Exit with the **OK button** .


14. 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:



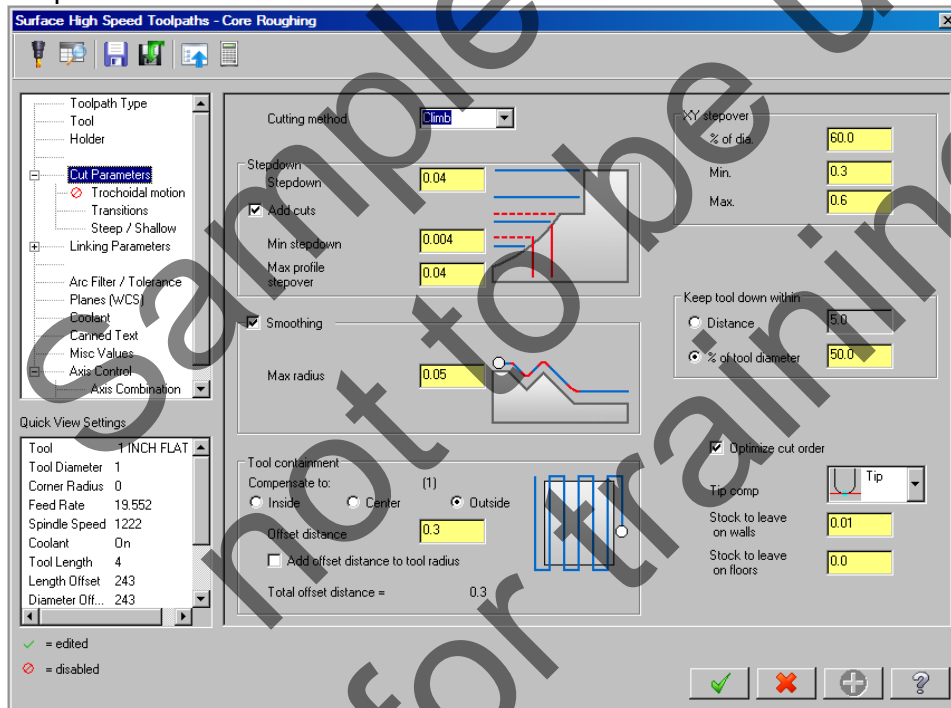
15. Click on **OK**  to exit Chaining.

16. Confirm the number of drive surfaces as shown below:



17. Click the **OK button**  to exit the Toolpath/surface selection.

18. Click on the Parameters folder and move into the **Cut Parameters** page and enter the parameters as shown:



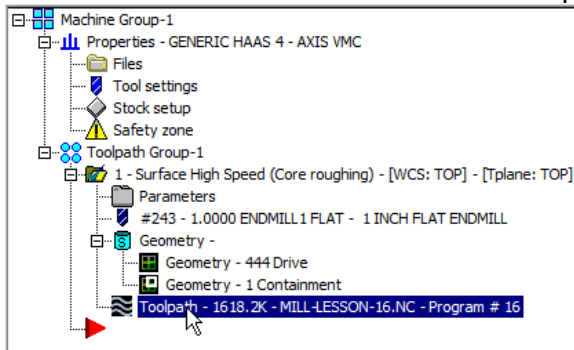
19. Click the **OK button**  to complete the operation.

20. Select the **Regenerate all dirty operations** icon



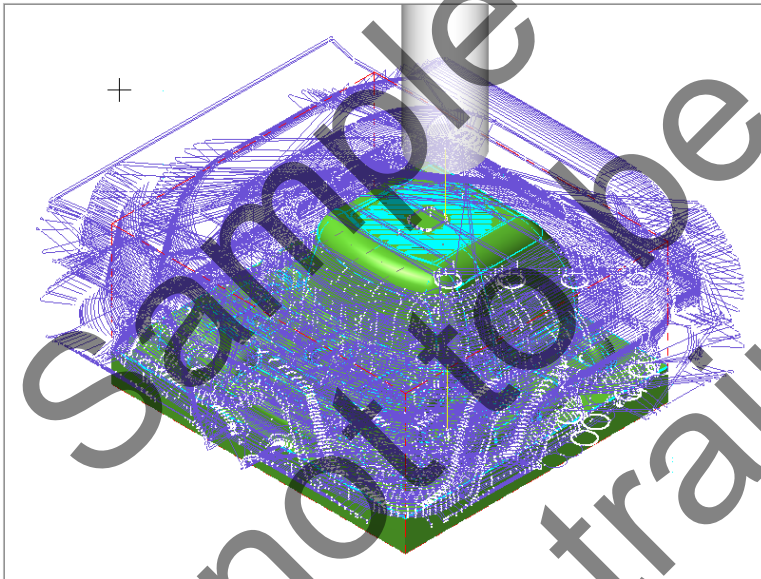
➤ Now you will inspect the toolpath using Mastercam's **Backplot** function.

21. In the operations manager, click on the **+** sign to the left of the toolpath to expand it. Then, left click on the bottom line with the toolpath information.



22. Press the **Next Stop** button  to move to the end of the toolpath.

In the backplot you have the ability to jump to a particular location in a toolpath just by clicking on it. This helps during inspection of large toolpaths.

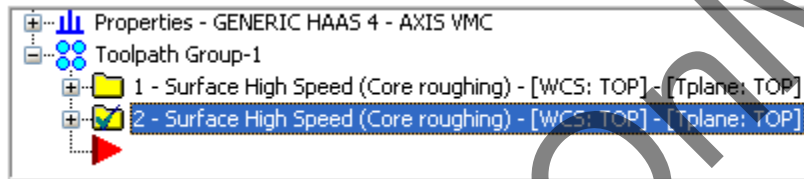


23. Exit the backplot by clicking the **OK** button .

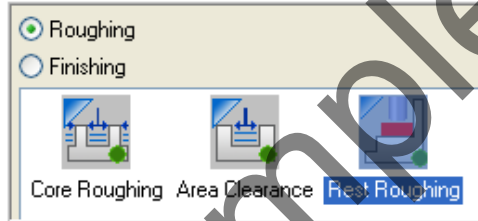
TASK 6: REST ROUGHING USING SURFACE HIGH SPEED (REST ROUGHING) TOOLPATHS

➤ In this task you will use a .250 diameter end mill to continue roughing the part.

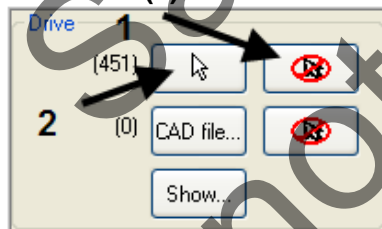
1. Open the **Level Manager** and make **level 11** the main level while making all other levels invisible.
2. On the status bar at the bottom of the screen select **2D/3D** to toggle to **3D construction**.
3. In the **Operations Manager**, copy the operation by selecting it with your mouse then pressing **Ctrl+C**. To paste the operation, select the red arrow with your mouse then press **Ctrl+V**.



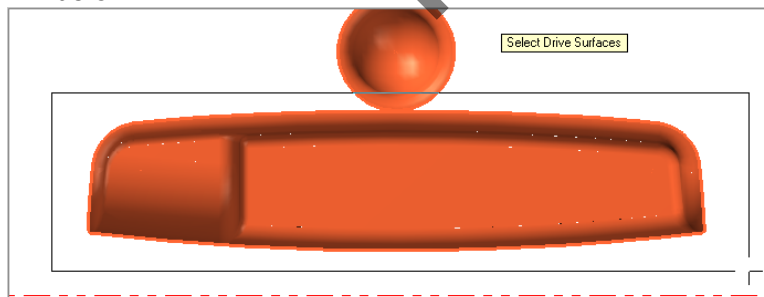
4. Expand the information for the newly created operation by pressing the **+** button, and then select the **Parameters** button.
5. Change the **Toolpath Type** to **Rest Roughing**.




6. Select the Remove selected drive surfaces button (1) then select the Select drive surfaces button (2).



7. You are prompted to **Select Drive Surfaces**.
8. Press the  button to move to the **Top View**.
9. Left click on the screen and Window select the pocket at the front of the part as shown below:

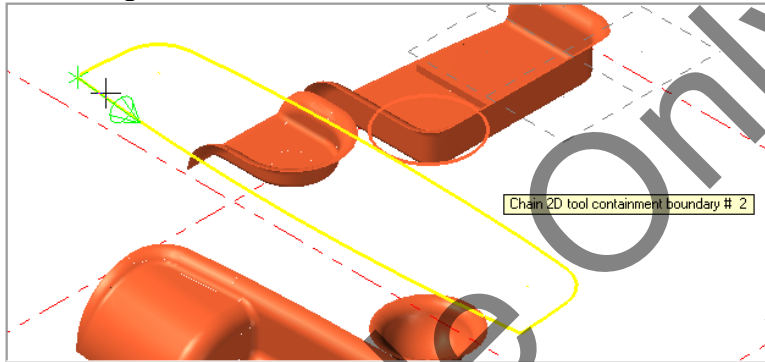


By default Mastercam will only select surfaces that are completely inside the window.

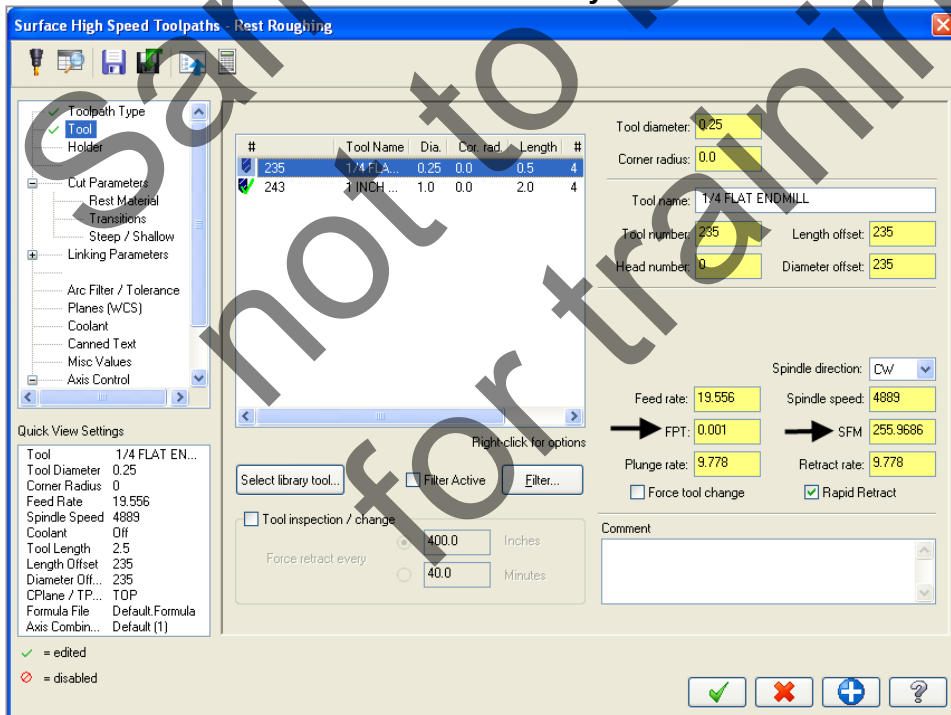
10. Click the **End Selection** icon . You will see 44 drive surfaces selected.
11. Similarly, clear the existing containment boundary (1), then click to select a new one (2).



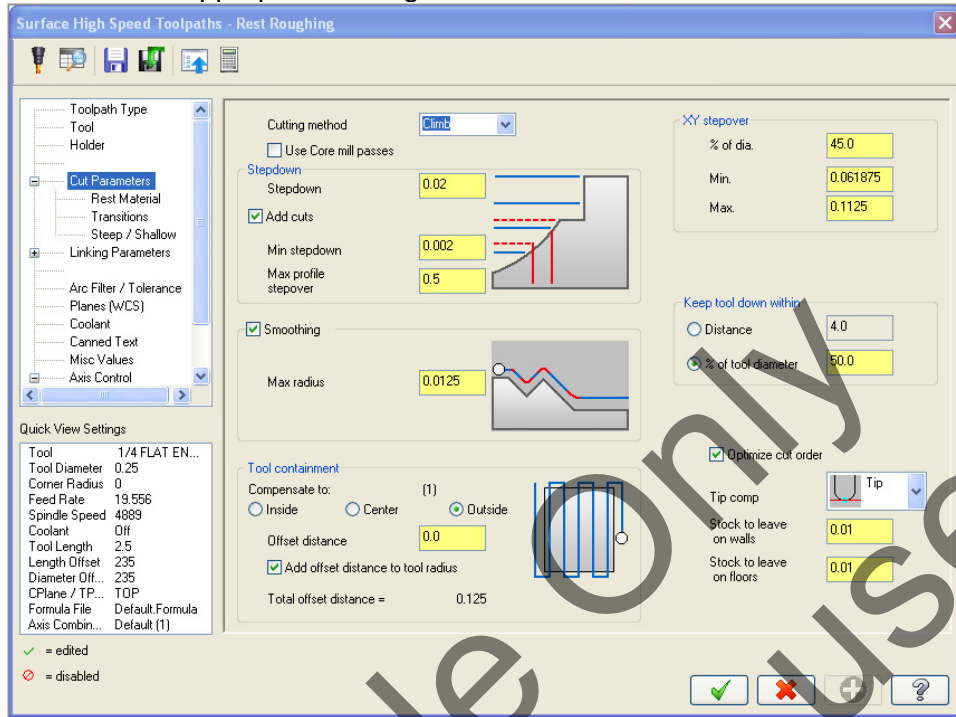
12. You will be prompted to **Chain 2D tool containment boundary # 1**.
13. Change to the **Isometric View** then select the chain as shown below:



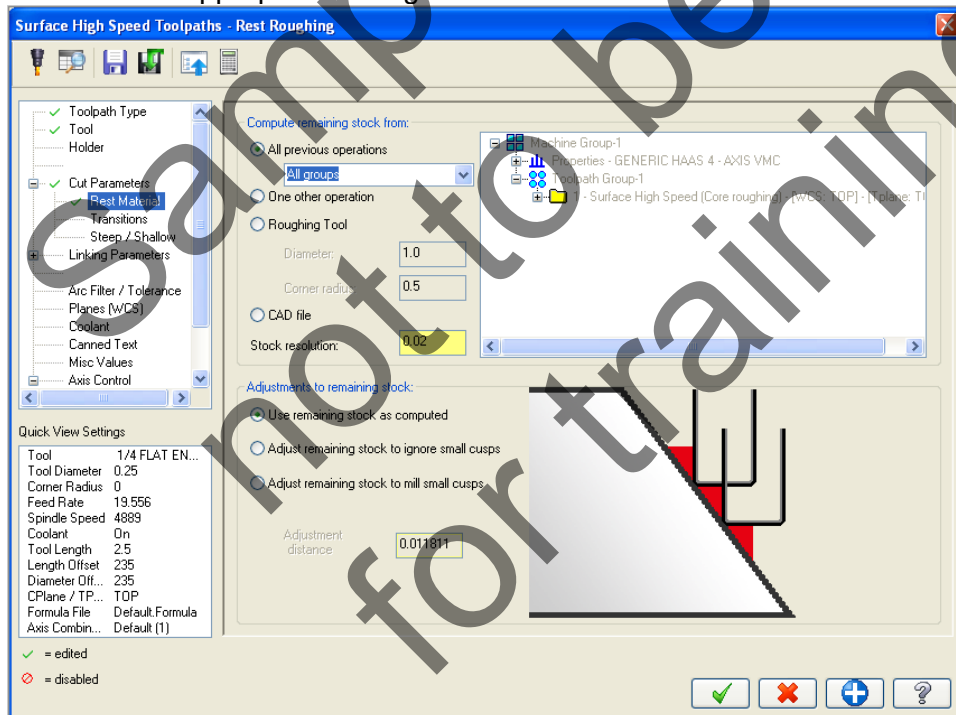
14. Click on **OK** to exit Chaining.
15. Move to the **Tool** page, press the **Select library tool...**, and select a **1/4 Flat Endmill** and click the OK button to return to the **Tool** page. Notice that the **FPT** and **SFM** set themselves based on the **Material Library** selection:



16. Make the appropriate changes as shown in the **Cut Parameters** tab below:



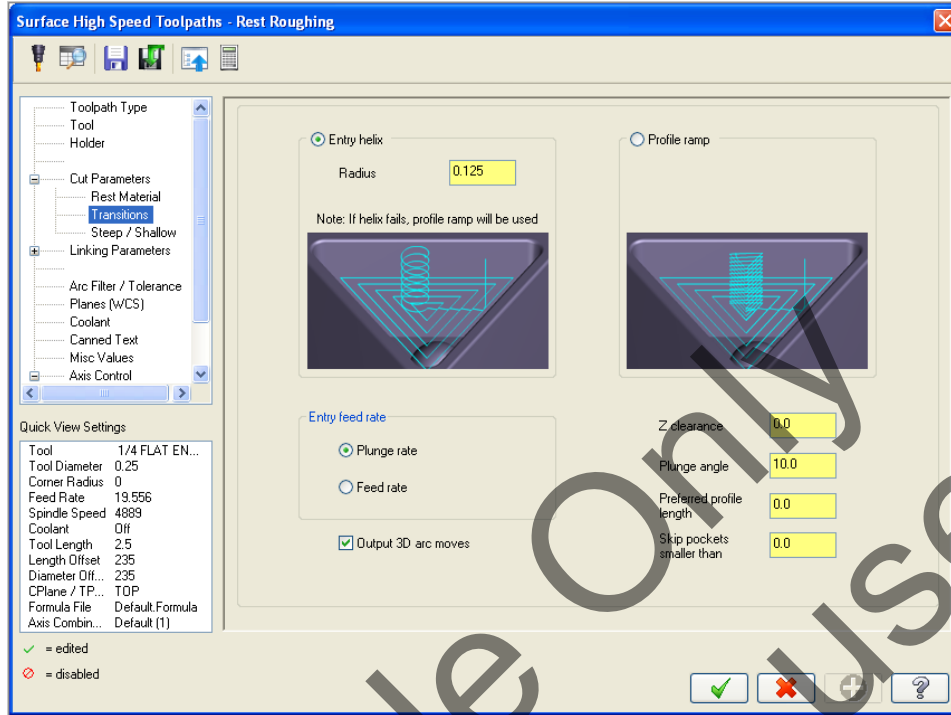
17. Make the appropriate changes as shown in the **Rest Material** tab below:



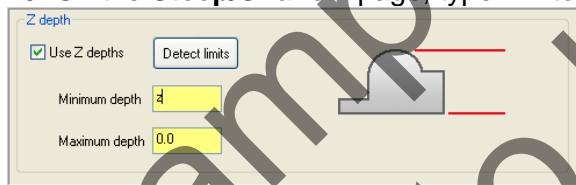
Compute remaining stock from:

Use the dimensions of the roughing tool to create the stock to be machined by the toolpath.

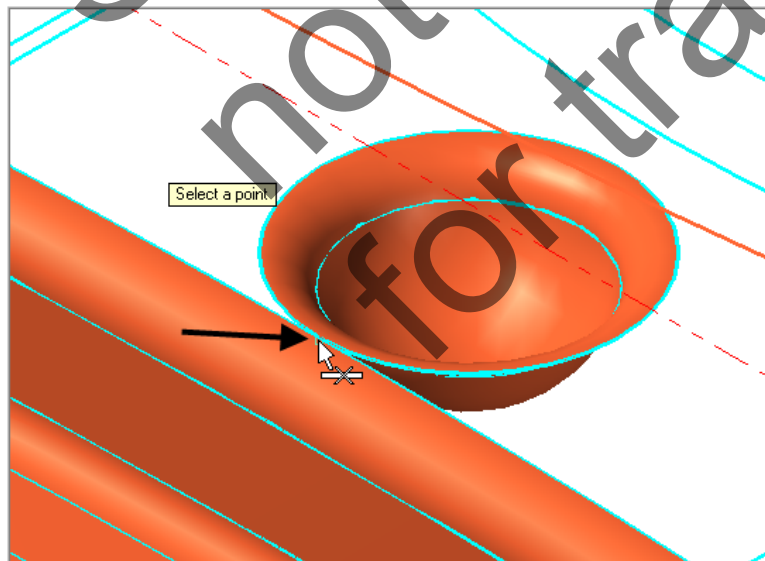
18. Make the appropriate changes to the **Transitions** tab as shown below:



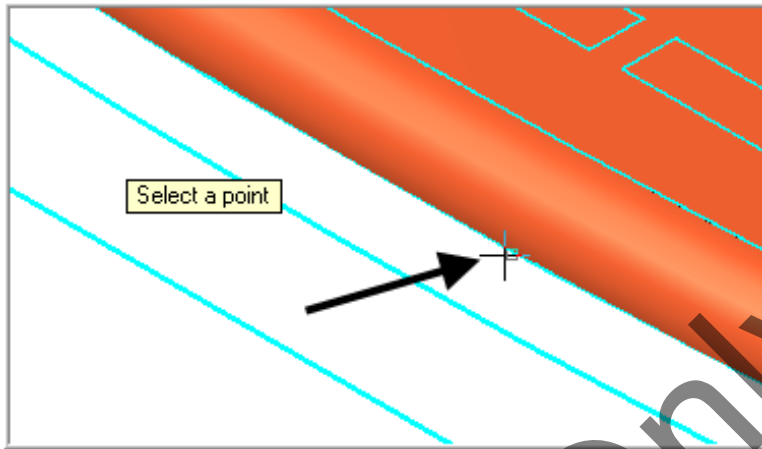
19. On the **Steep/Shallow** page, type **Z** into the **Minimum depth** field and press enter.



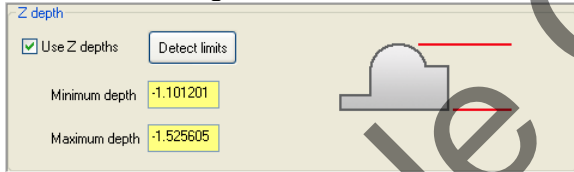
20. You will be prompted to **Select a point**. Open the **Level Manager** and make **level 2** (wireframe) visible. Then, select a point along the chain that runs along the top of the surface:



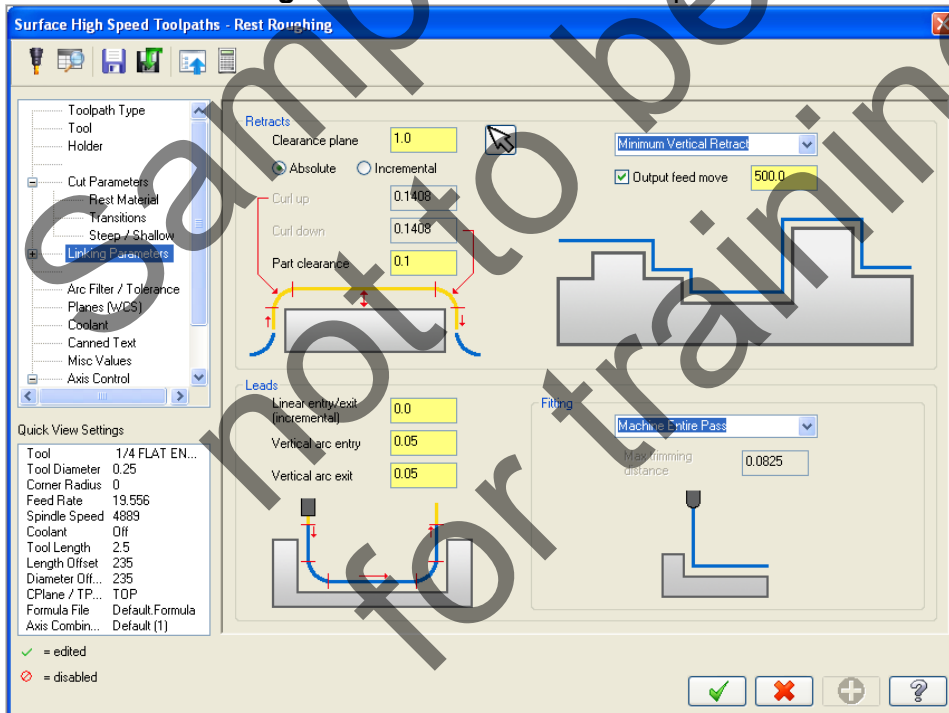
21. Repeat the process for the **Maximum depth** field and select a point on the front pocket as shown below:



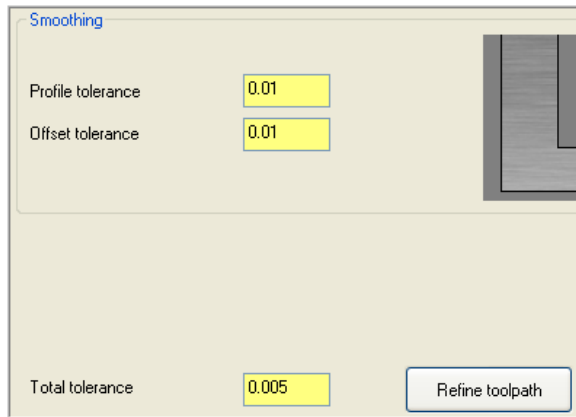
22. The following values should be entered for the **Minimum** and **Maximum depth**:



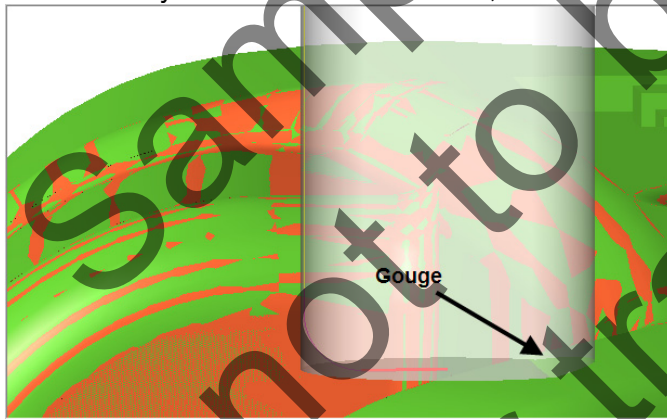
23. Select the **Linking Parameters** tab and set the parameters as shown below:



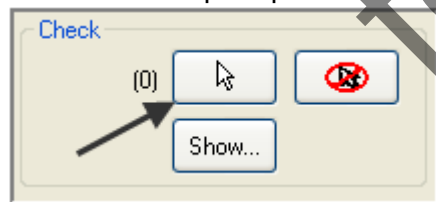
24. Select the **Arc Filter/Tolerance** tab and adjust the parameters as shown below:



25. Press the **OK button** to complete the toolpath.
 26. Regenerate the operation by selecting the **Regenerate all dirty operations** icon in the **Operations Manager**.
 27. Turn on **levels 1 and 11** then backplot the operation. Position your **Graphics View** so you can see the tool lead-in to the initial cut.
 28. Click the mouse anywhere inside the graphics area then press the **S** button on your keyboard to step through the toolpath.
- ☞ The tool is gouging the part on the right hand side. The gouge is on an adjacent part of the solid outside of the **containment boundary**. The tool must be allowed outside of the boundary to cut the entire surface, so check surfaces must be added to avoid gouging.



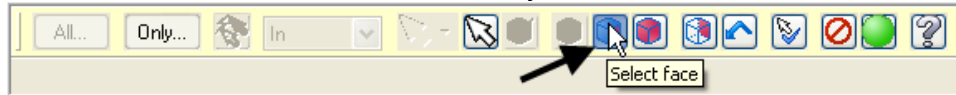
29. Select **Parameters** under the second operation in the **Operation Manager**. Return to the **Toolpath Type** page of the operation and click the **Select Check Surfaces** button.
30. You will be prompted to **Select Check Surfaces**.



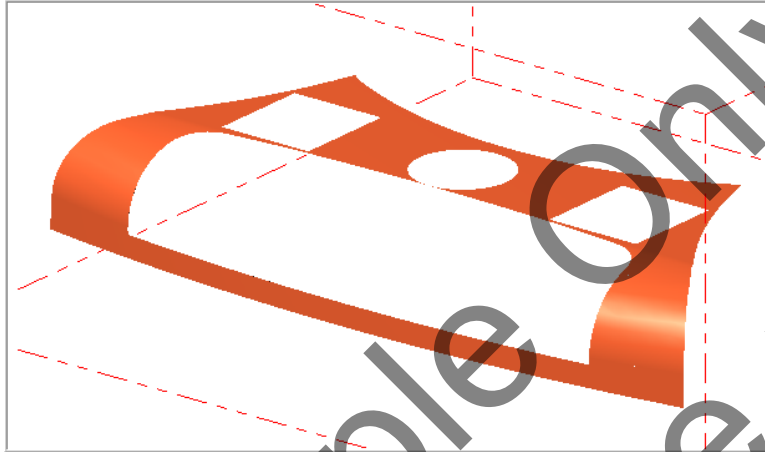
31. Change the selection method to **Active solid selection**.



32. Make sure the **Select face** is the only method that is active.



33. Select all of the solid faces that are adjacent to the drive surfaces as shown below:



34. Click on the **end selection button** 


35. Hit the **OK button**  to exit the toolpath.

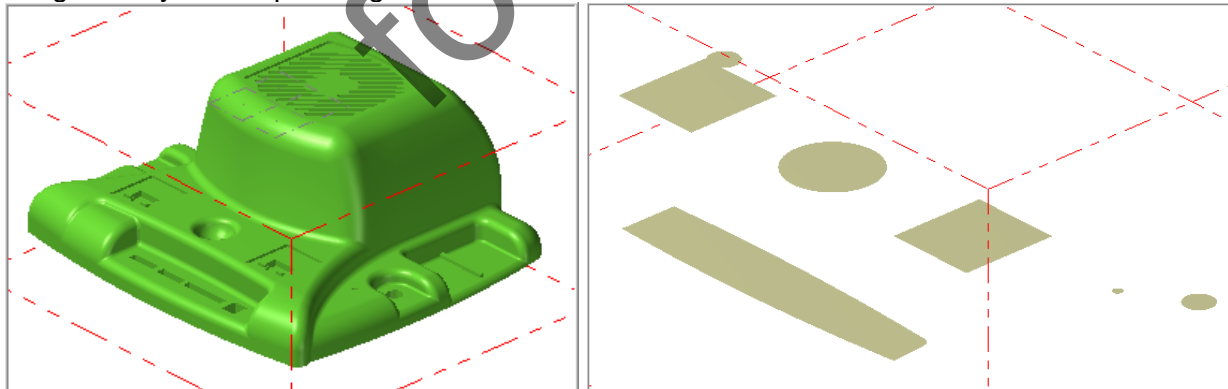
36. Regenerate the toolpath and **Backplot** the toolpath to confirm the gouge has been removed. Accept any warning messages.

It is generally advisable to use **Check Surfaces** whenever you have surfaces that are adjacent to the **Drive Surfaces**. If your toolpath **Drive Surfaces** include all of the surfaces or solid faces in or around the cutting area then **check surfaces** are not necessary.

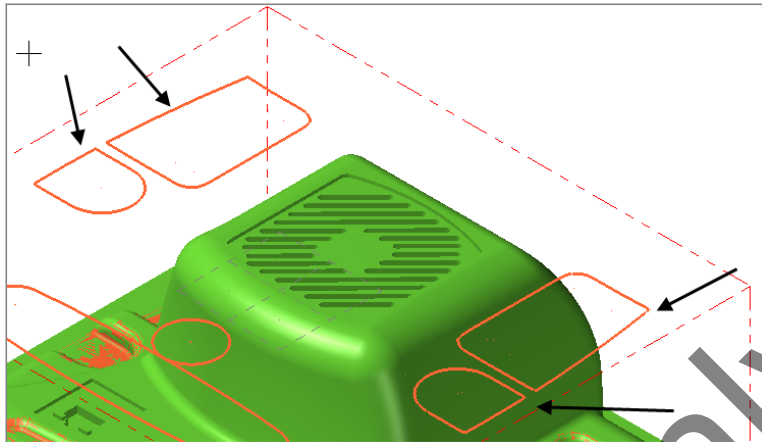
➤ Next you will rough out the pockets on the sides of the part.

37. In the **Operations Manager**, copy and paste **Operation 2** to create **Operation 3**.

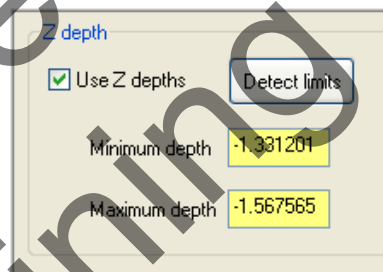
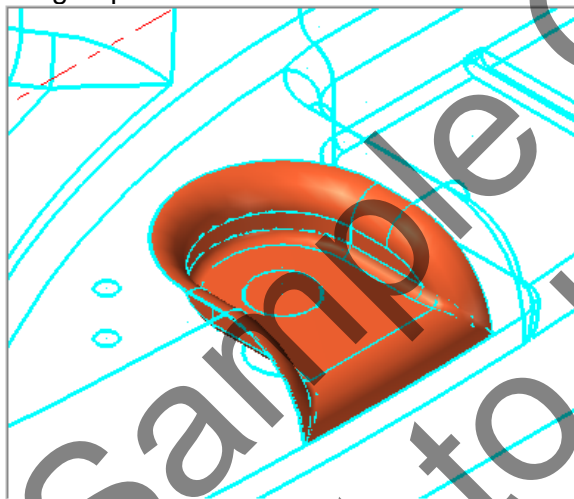
38. On the **Toolpath Type** page of the new operation, remove the drive geometry. Add the solid on **level 1** as shown below left, and the surfaces on **level 10** as shown below right (all except the one on the top of the part) as the new drive geometry. You must select all of the geometry before pressing the **End Selection** icon . You should have 445 entities.



39. Add the 4 Chains shown below as **Containment Boundaries**:

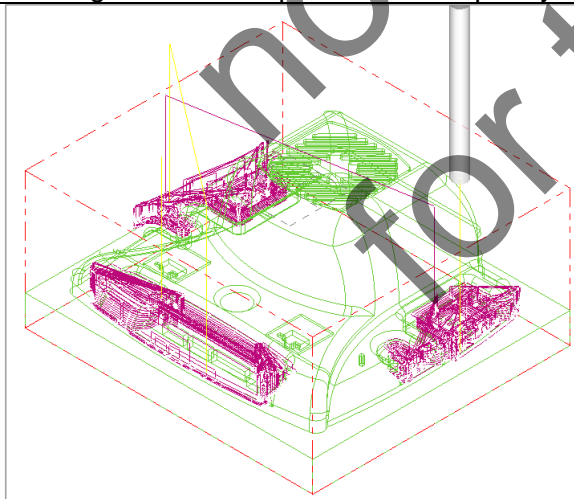


40. On the **Steep/Shallow** page, repeat steps 19 through 22 to select the top and bottom of the group of solid faces shown in the following **Isometric View**:



41. Complete the toolpath by selecting the **OK button** .

42. Regenerate the operation. Accept any warning messages and **Backplot** to view the results.



You have just completed two examples of rest roughing. Each method has its own advantages as explained below.

The first one used only the surfaces/faces of the pocket as the **Drive Surfaces**. As a result, you needed to select **Check Surfaces** to protect the adjacent geometry. Toolpath processing is typically faster when you select a minimal number of **Drive** and **Check Surfaces**.

The second example involved selecting the entire solid for machining. Toolpath processing will typically take longer in this case. **Check Surfaces** typically do not have to be selected in this case. The advantages of this method will be further illustrated as we copy and paste this toolpath to rough out the other pockets.

TASK 7: APPLY MASTERCAM HIGH FEED FUNCTIONALITY

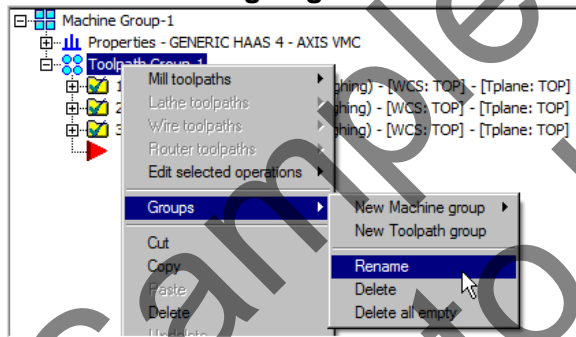
➡ Next you will apply the High Feed function to the roughing toolpaths.

The **High Feed** function is used to adjust the feed rate of the tool on the fly to keep a constant volumetric material removal rate. This typically results in a decrease in cycle time and can greatly increase machining productivity.

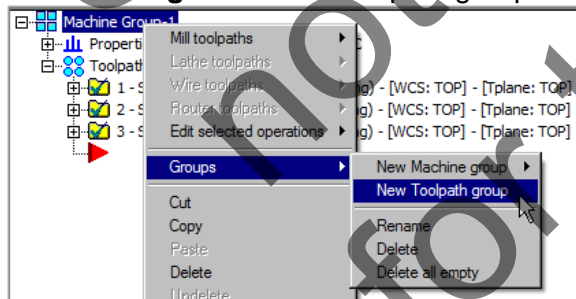
The function will also ease the tool in and out of the material between cuts to avoid shocking the tool and inserts. This typically results in an increase in cycle time but can greatly increase tool life.

In your operations, you set a base feed rate for the particular tool and operation. **High Feed** will allow you to supply a minimum and maximum feed rate for the tool as well as a feed rate for air cuts. The calculation will vary the tool's feed rate on each based on the location of the cutter with respect to the material and the volume of material being removed during the move.

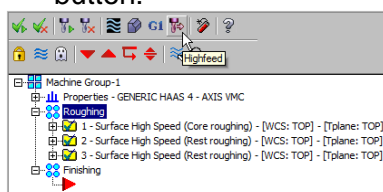
1. In the **Operations Manager**. Right click on **Toolpath Group-1**. Select **Groups>Rename** and enter **Roughing** as the new name.



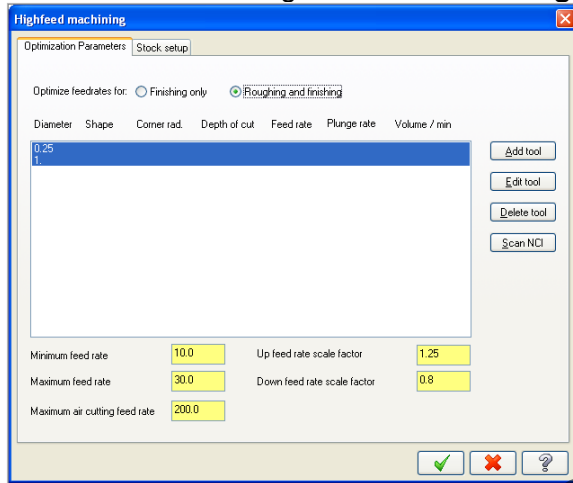
2. Right Click on **Machine Group-1**. Select **Groups>New Toolpath group** and enter **Finishing** as the new toolpath group name.



3. Select all of the roughing operations by selecting **Roughing** then select the **High Feed** button:



4. Make the following selections in the **Highfeed machining** dialogue box:



The **Minimum feed rate** is the low end of the adjustment range and will be used when a high volumetric rate of material removal occurs or when the tool is entering the stock.

The **Maximum feed rate** is used when the tool is cutting very little material.

Maximum air cutting feed rate is for positioning between cuts. It is possible to increase this to the highest allowable feed rate by your CNC controller.

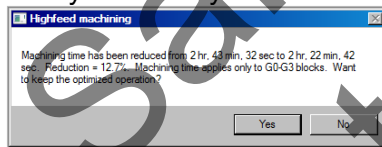
Up and Down feed rate scale factors adjust the cutting when the tool is lifting or plunging.

The **Stock Setup** tab allows you to setup stock in the same way that you would with **Verify**. The default stock comes from the **Stock Setup** that you completed in **TASK 2**.

5. Exit the dialogue box then select the **Run** button in the ribbon bar.



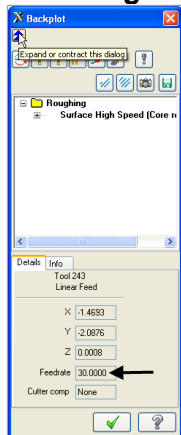
➤ The **Highfeed machining** results will be displayed after calculating. We have reduced the cycle time by 12.7%. Select **Yes** to keep the results.



6. Notice that in the Operations Manager, Mastercam has locked operations for editing now.

Each operation has a lock  next to the operation.

7. To illustrate the results, backplot any of the toolpaths, select the **Expand or contract this dialogue** button and watch the feed rate vary as you step through the toolpath.



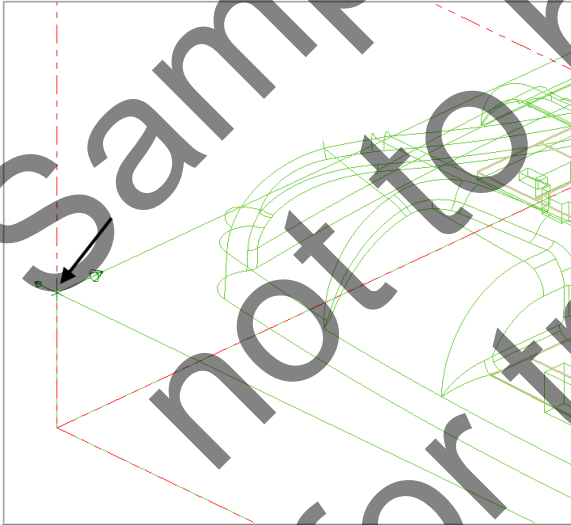
TASK 8: FINISH FLAT SURFACES USING SURFACE HIGH SPEED (HORIZONTAL)

- Next you will finish the flat portions of the part using the Surface High Speed (Horizontal) toolpath.

The **Horizontal** toolpath strategy will automatically find flat, or horizontal, surfaces to machine. 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.

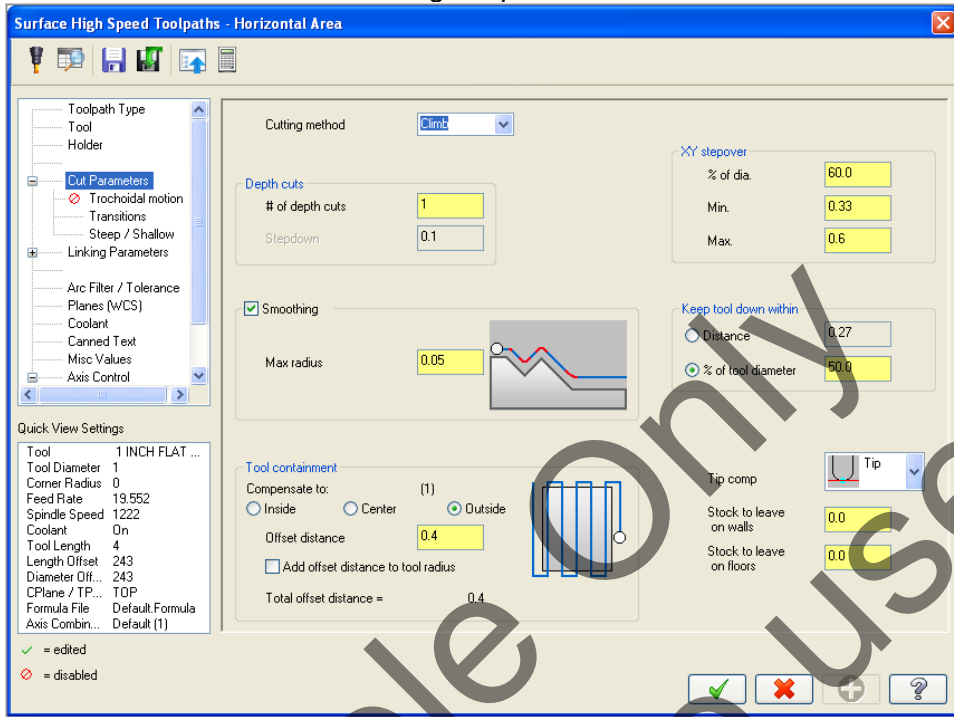
Using the bottom of a flat endmill or fly cutter will give a better surface finish on floor surfaces than a ballnose endmill will because there will be no scallops. Alternatively, a ballnose tool will provide a much better surfaces finish for shape profiling.

1. In the **Operations Manager** confirm that the red arrow used to locate new operations is in the **Finishing Toolpath Group**. 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. Add **Drive Surfaces**, and select all of the entities on the **Solid (1)** and **Roughing (10)** levels.
4. Turn on **level 2** and add the top edge of the mold base as the **Containment Boundary**.

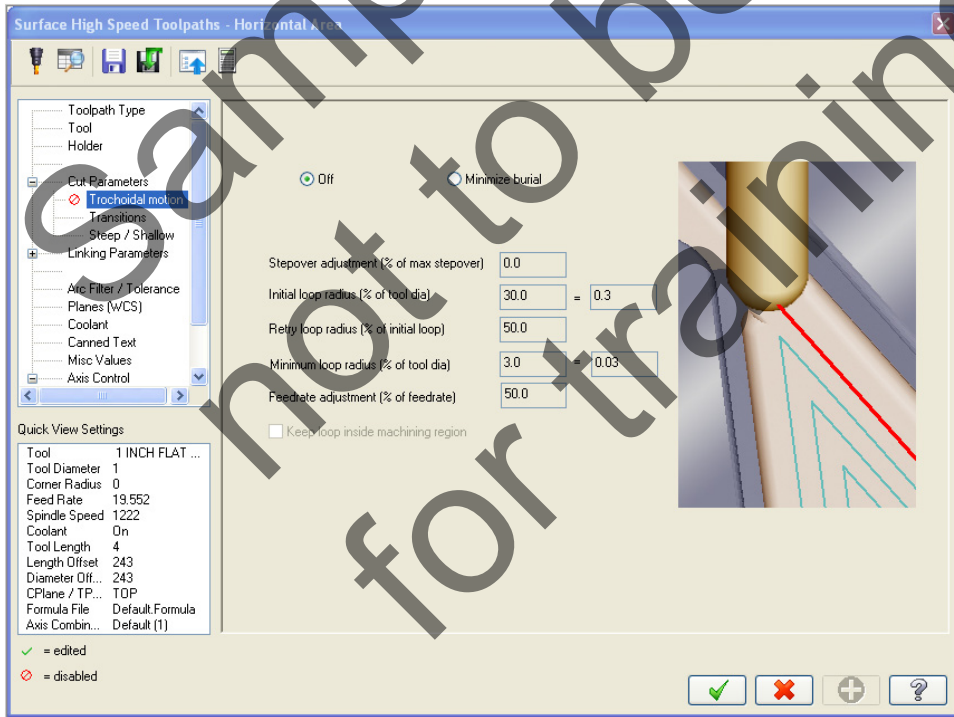


5. Select the **OK** button to exit.
6. Change the **Toolpath Type** to **Finishing Horizontal Area**.
7. Navigate to the **Tool** page and select the **1.000 Flat Endmill** that was previously used for roughing.

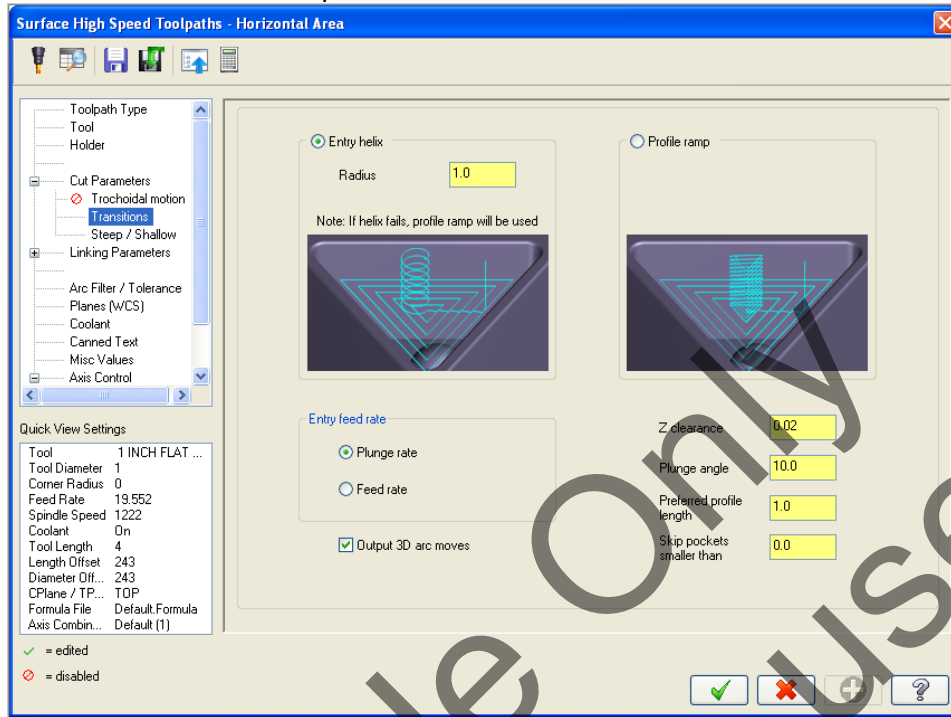
8. Make the appropriate selections on the **Cut Parameters** page shown below. Note that the **Stock to leave** for all finishing toolpaths will be 0.



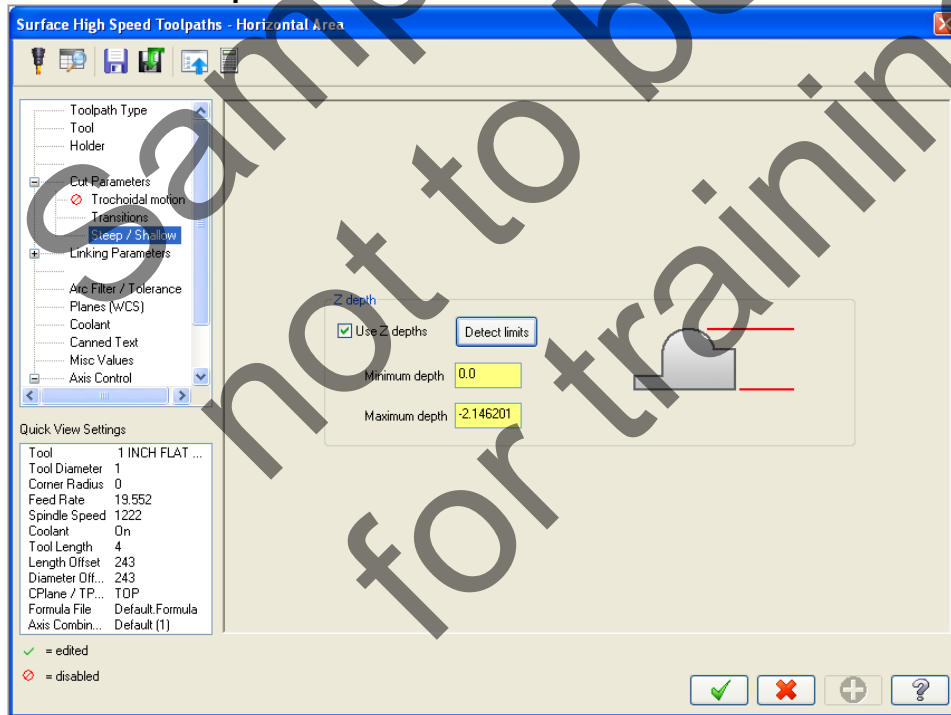
9. Turn Off the **Trochoidal motion** as shown below:



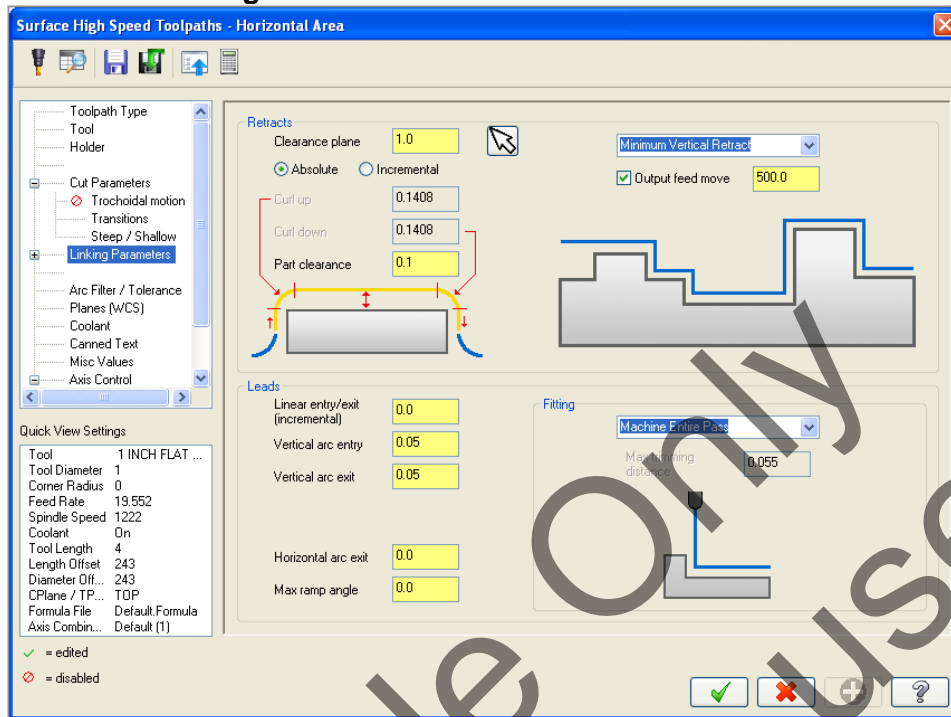
10. Set the **Transitions** parameters as shown below:



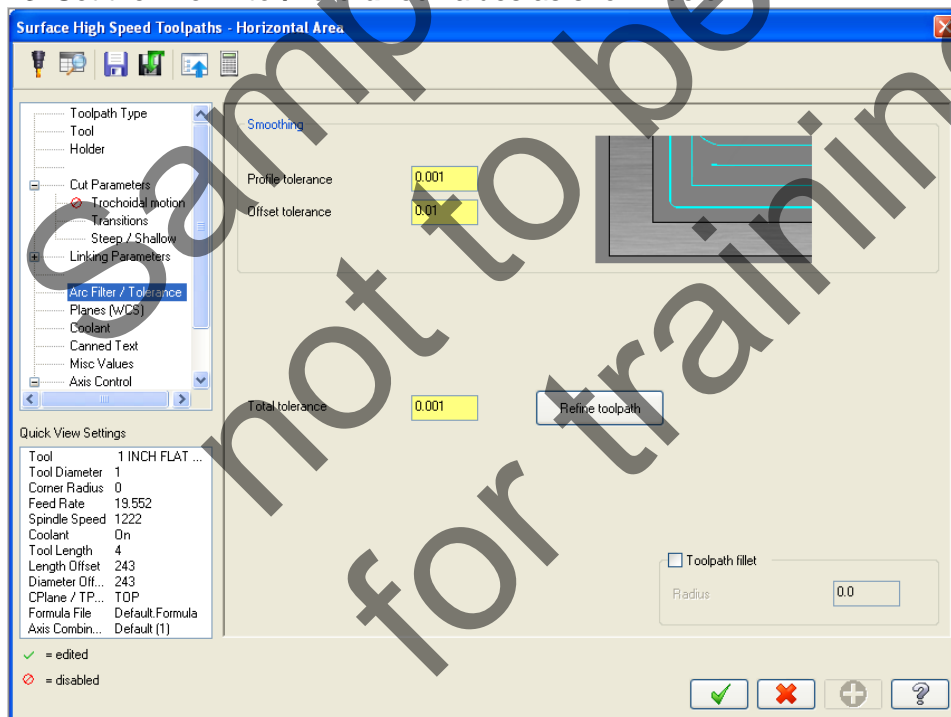
11. On the **Steep/Shallow** page select the **Detect limits** button to fill in the **Minimum** and **Maximum depth**:



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



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





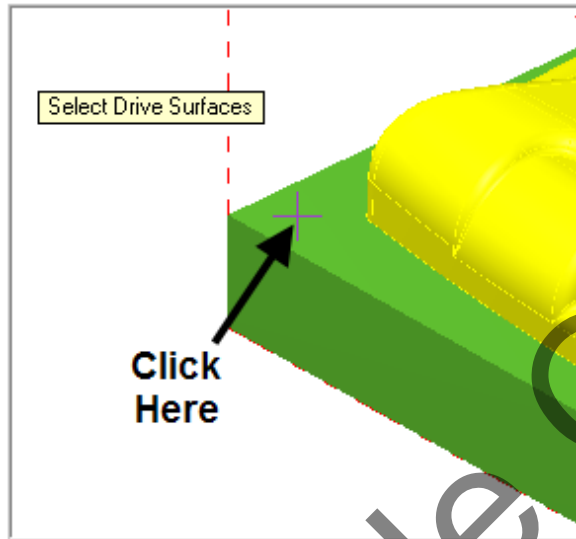
14. Navigate to the **Coolant** tab and turn the **Flood** coolant on.

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

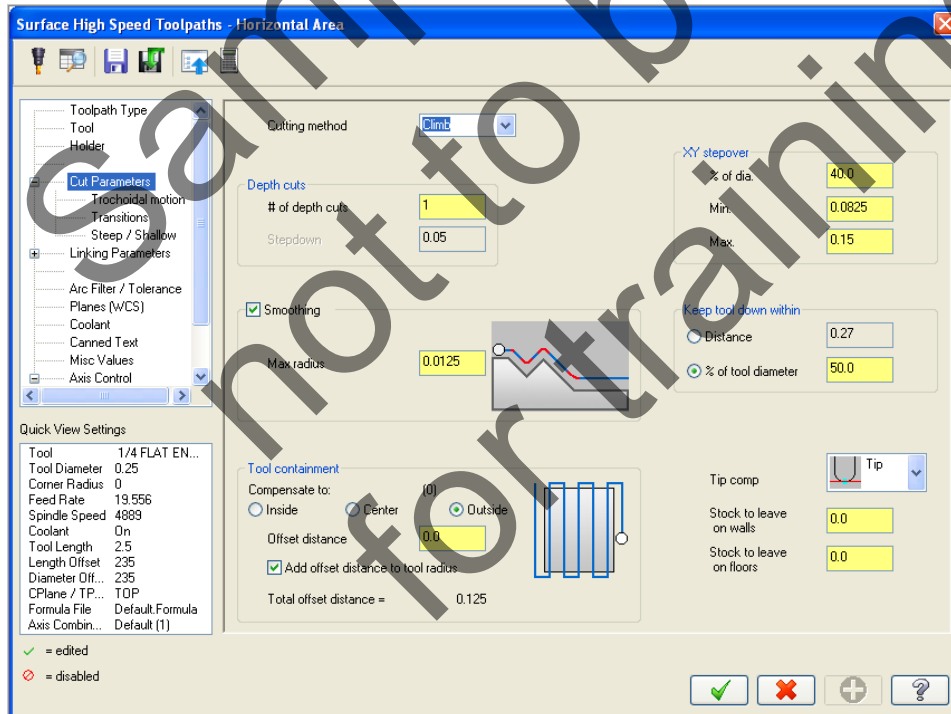
16. In the **Operations Manager**, copy **Operation 4**, which was just created, and paste the operation directly after, to create **Operation 5**.

17. Select the **Parameters** option under **Operation 5**.

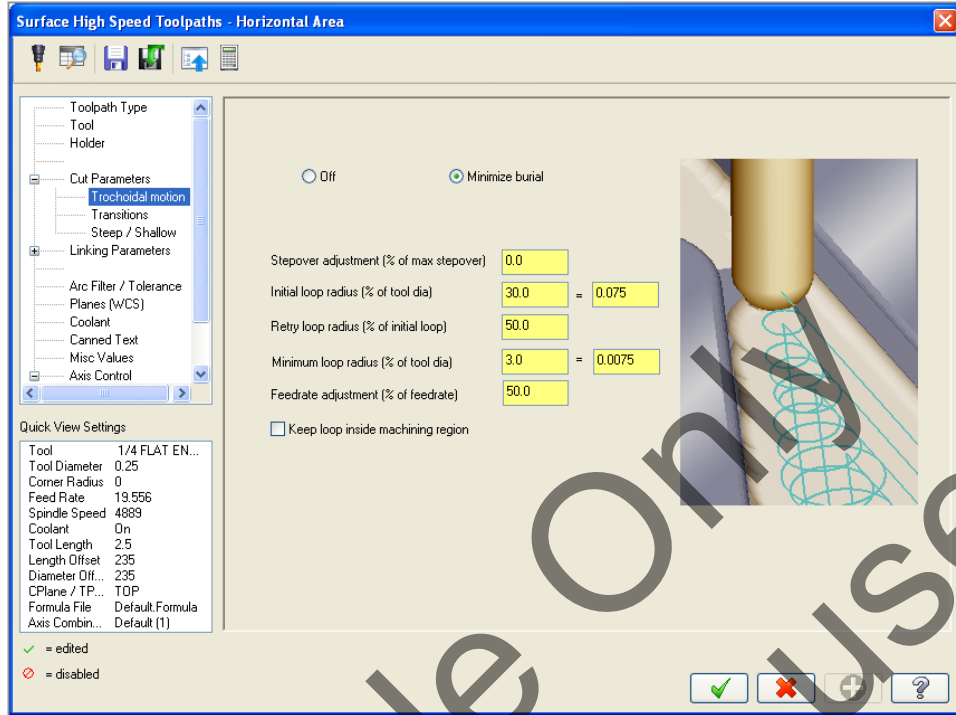
18. On the **Toolpath Type** tab, select the  button in the **Drive** window. You will be prompted to **Select Drive Surfaces**. All of the **Drive Surfaces** from the previous toolpath are currently highlighted. Unselect the solid base of the part as shown below and then click the  **End Selection** icon.



19. On the **Tool** page, Change the tool to the **1/4 Flat Endmill** used previously.
20. Select the **Cut Parameters** page and make the necessary changes to match the screen shown below:

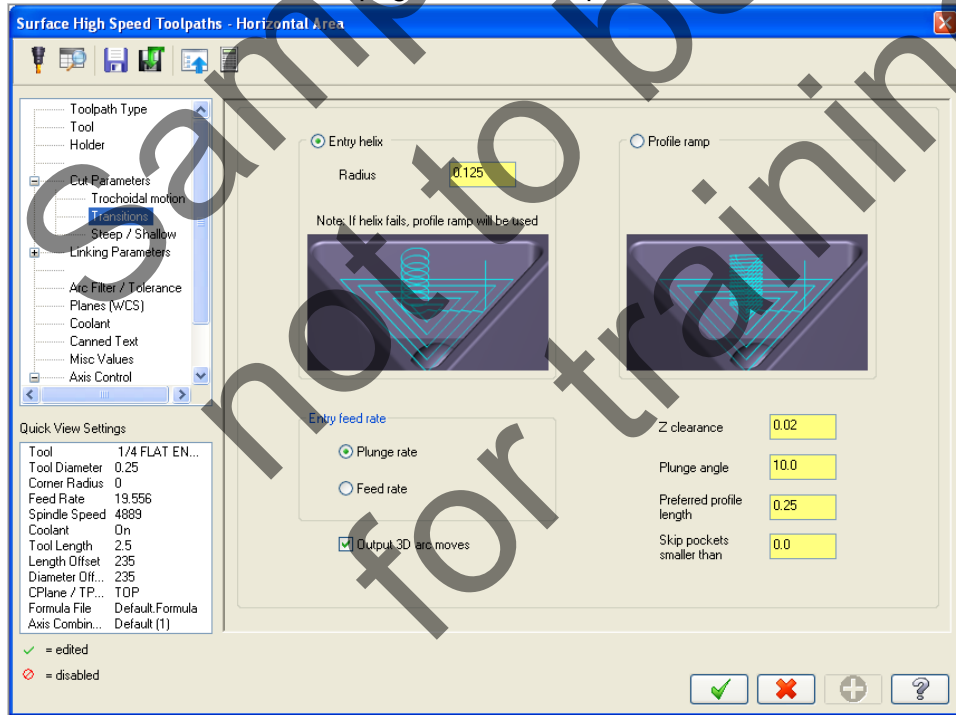


21. Select the **Trochoidal motion** page and enter the parameters shown below:

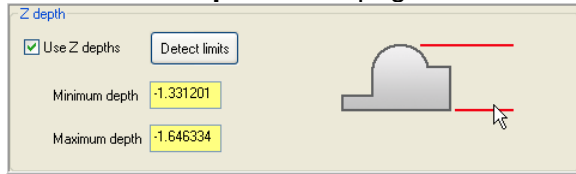



Minimize burial will apply trochoidal motion to the toolpath anywhere that the tool is moving into an area where it's leading edge profile would fully loaded. The trochoidal (circular) motion that is applied allows the tool to use a **stepover** that is close to the one programmed on the **Cut Parameters** page. You can control the circular motion's **radius** and vary **Stepover** as well.

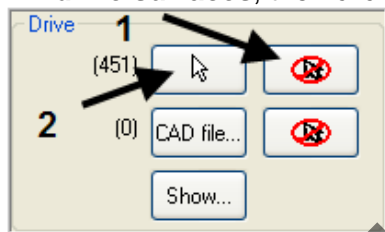
22. Select the **Transitions** page and set the parameters as shown below:



23. On the **Steep/Shallow** page enter the following values for **Z depth**:



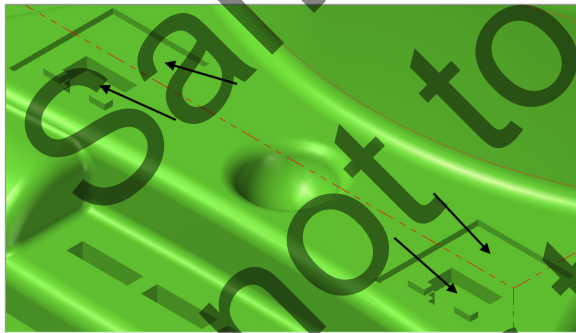
24. Select the **OK button**  to complete the toolpath.
25. Regenerate the operation by selecting the **Regenerate all dirty operations** icon in the **Operations Manager**.
26. Copy **Operation 5** and paste the operation below to create **Operation 6**.
27. Click on the **Parameters** for **Operation 6**.
28. On the **Toolpath Type** page, clear the **Drive surfaces** by selecting (1) **Remove selected drive surfaces**, then click on (2) **Select drive surfaces**.



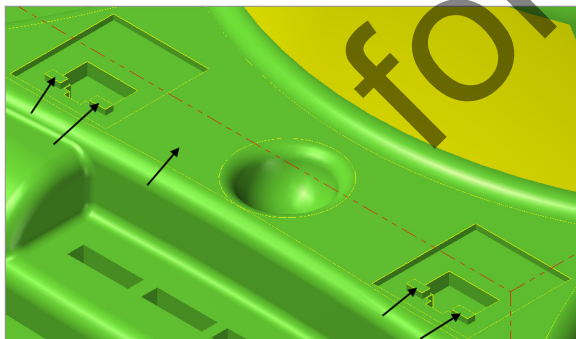
29. Make sure the **Select face** is the only method that is active.



30. Select the 4 surfaces from the solid shown below and click on the **End Selection** button:

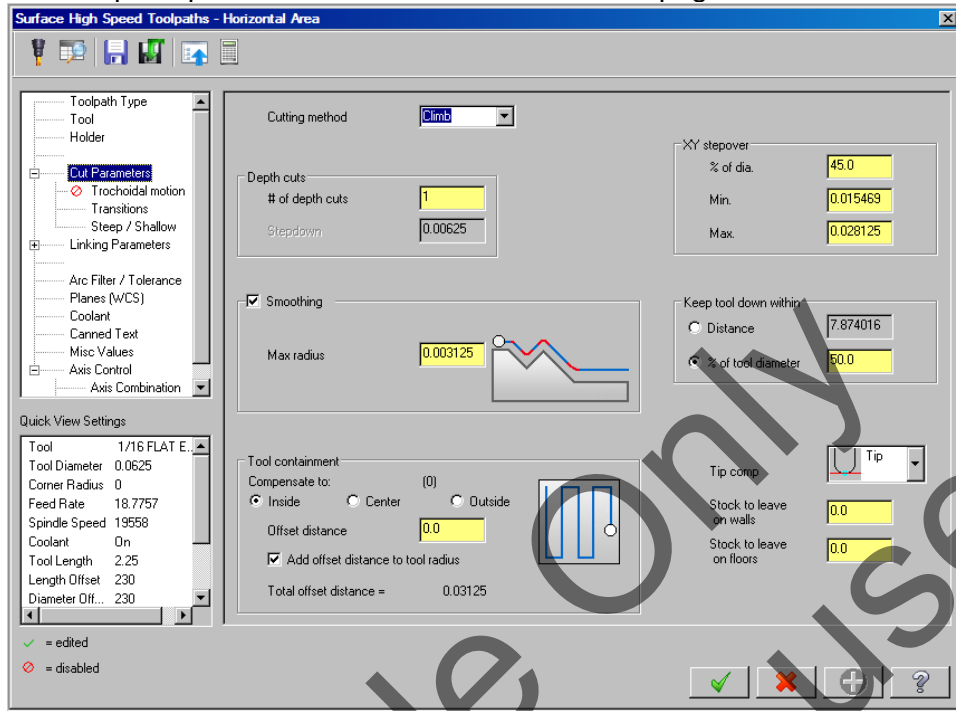


31. Select the **Select check surfaces** button and add the surfaces shown below:



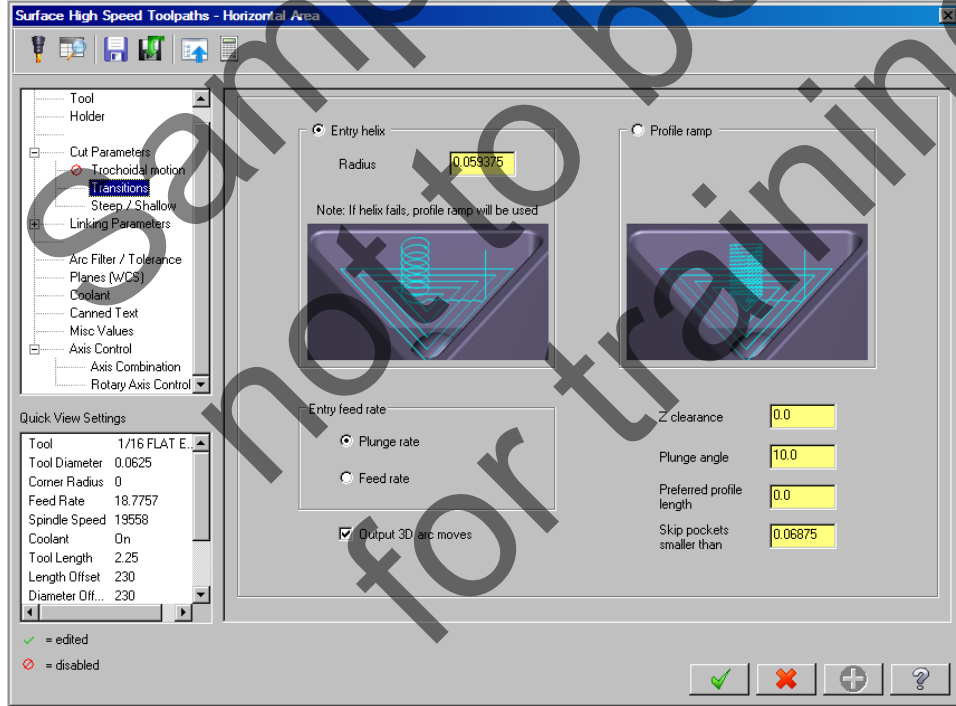
32. Click on the **Remove selected containment boundary** button.
33. On the **Tool** page, select a **1/16 Flat Endmill**.

34. Setup the parameters on the **Cut Parameters** page as shown below:

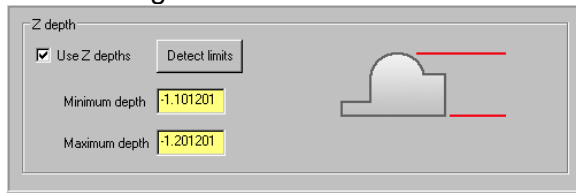


35. Turn **Off** the **Trochoidal motion**.

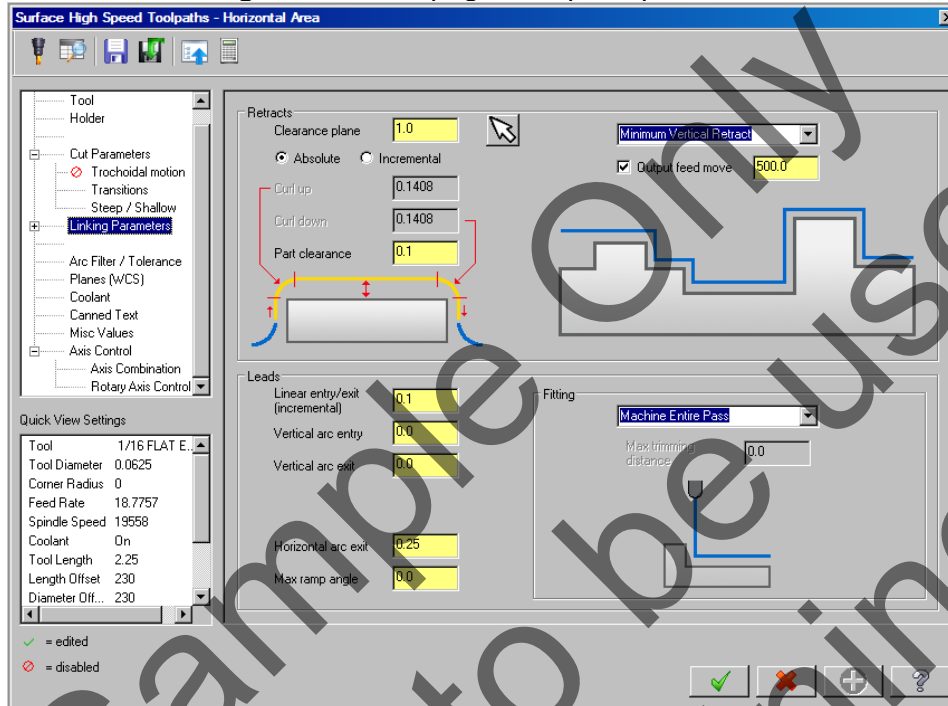
36. On the **Transitions** page, setup the parameters as shown:



37. On the **Steep/Shallow** page, select the **Detect limits** button and ensure the values match the image below:



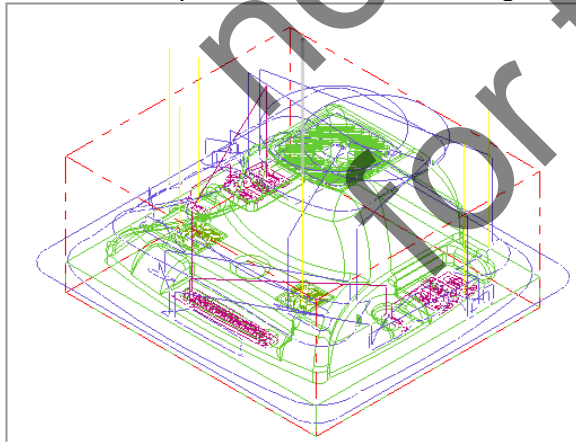
38. On the **Linking Parameters** page, setup the parameters as shown below:



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

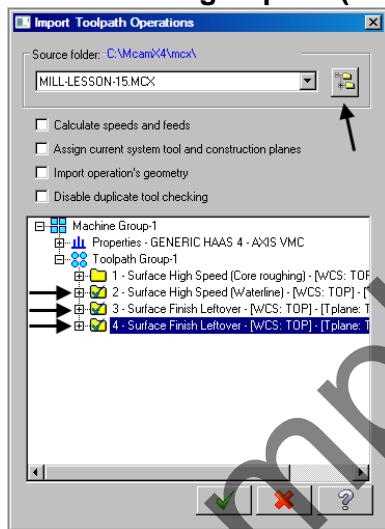
40. Click on the **Regenerate all dirty operations** icon in the **Operations Manager**. Accept any warning messages.

41. Review Operations 4, 5 and 6 using **Backplot**:

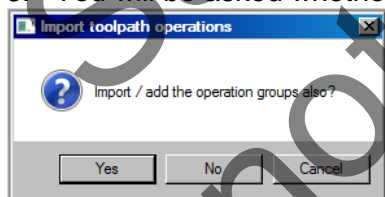


TASK 9: IMPORTING WATERLINE AND SURFACE FINISH LEFTOVER TOOLPATHS

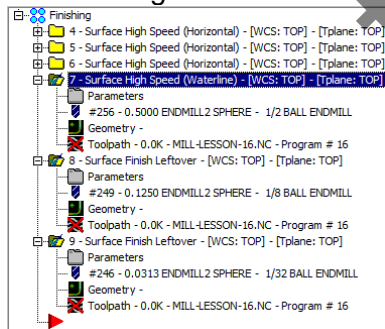
- Next you will import the waterline and surface finish leftover toolpaths from Lesson 15.
- 1. Ensure the red arrow is at the bottom of your operations list in the finishing toolpath group. Select the red arrow and drag it to the bottom if it is not there already.
- 2. Right click in the **Operations Manager** and select **Import**.
- 3. The **Import Toolpath Operations** window will now appear. Select the source folder that has the **Mill-Lesson-15.mcx** file stored within it and select the final three operations, **Surface High Speed (Waterline)** and the two **Surface Finish Leftover** operations:



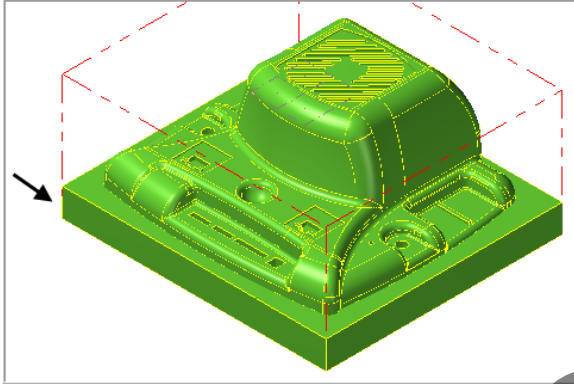
- 4. Select the **OK button** to import the operation.
- 5. You will be asked whether you want to Import/add the operation groups also. Select **No**:



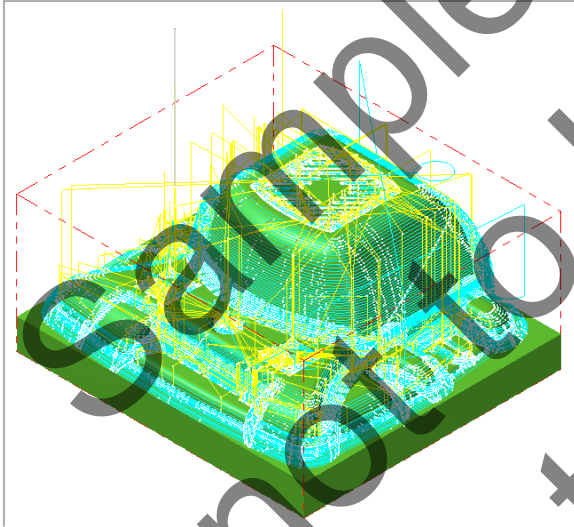
- 6. You will be prompted letting you know that 3 operations have successfully been imported. Hit the **Cancel button** to return to the operations manager which should now look like the image below:



7. Select **Geometry** in Operation 7.
8. Click on **Select Drive Surfaces** and window select all the solids on level 1, then click on **End Selection**.
9. Click on **Select Check Surfaces** and select the solid body shown below:



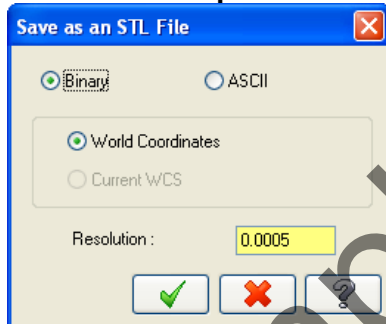
10. Click the **OK button** to complete the toolpath.
11. Repeat steps 7 through 10 for **Operations 8 and 9**.
12. Click on the **Regenerate all dirty operations** icon in the **Operations Manager**. Accept any warning messages.
13. Review Operations 7, 8 and 9 using **Backplot**:



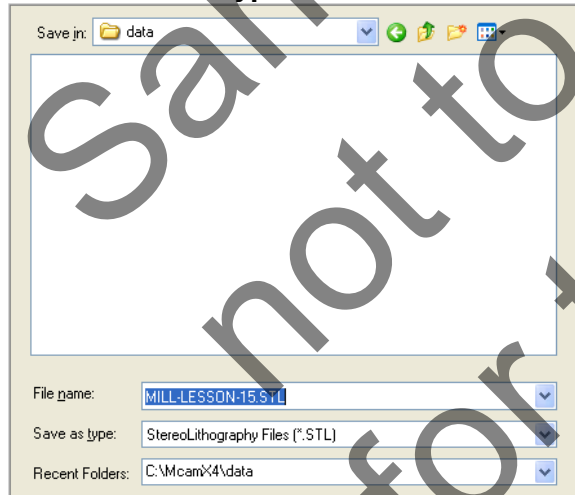
TASK 10: VERIFY THE TOOLPATH AND COMPARE TO STL FILE

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 the entities to save**. Select the two solids (the main part and the base). End the selection.
3. The **Save As** dialogue box will open. Browse to your Mastercam directory, then the **data** sub directory.
4. Select the **Options** button and make the selections below.

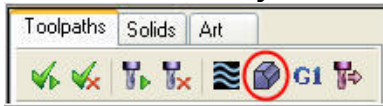


5. For **Save as type** select **.STL file** then save the file with the name **MILL-LESSON-16.STL**.



6. Select all of the operations you have completed so far by picking the **Select All** icon .

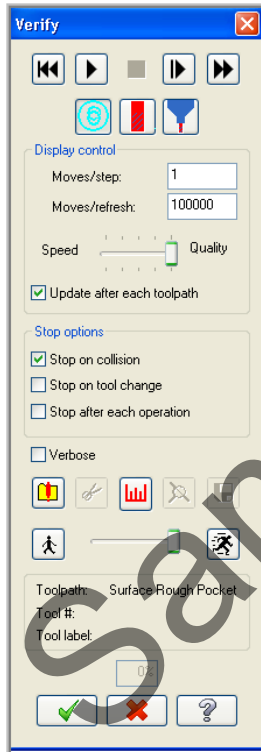
7. Select the **Verify selected operations** button circled below:



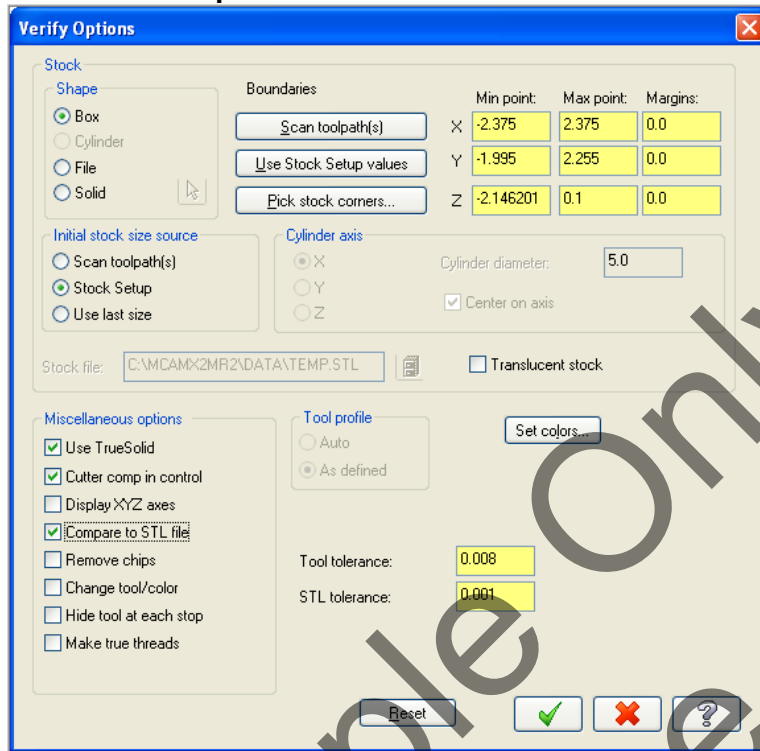
8. Select the **Turbo** button. Turbo does not display the tool or holder and does not perform live updates of the stock, which is updated at the end of the verification. Turbo is generally the fastest option for verification.



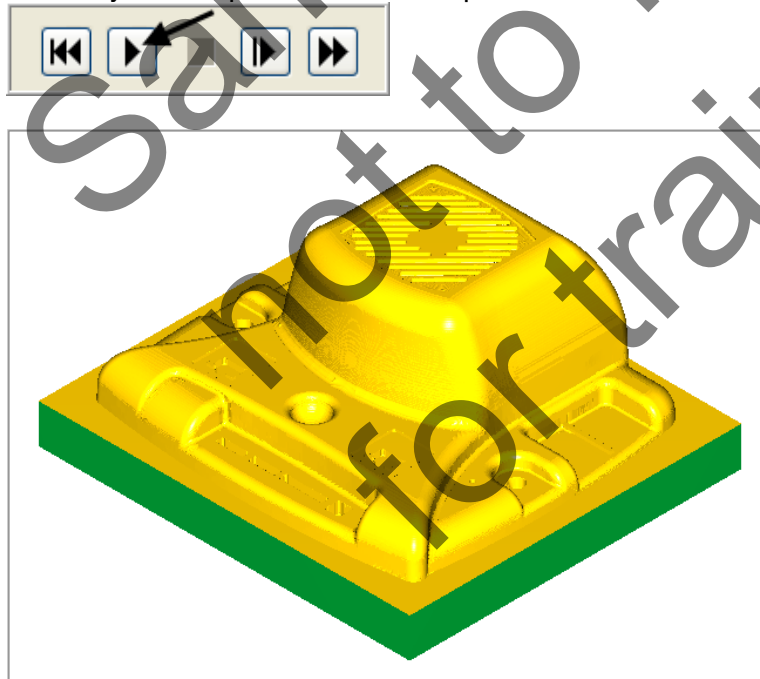
9. Check mark **Stop on collision** and set values as shown below:




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

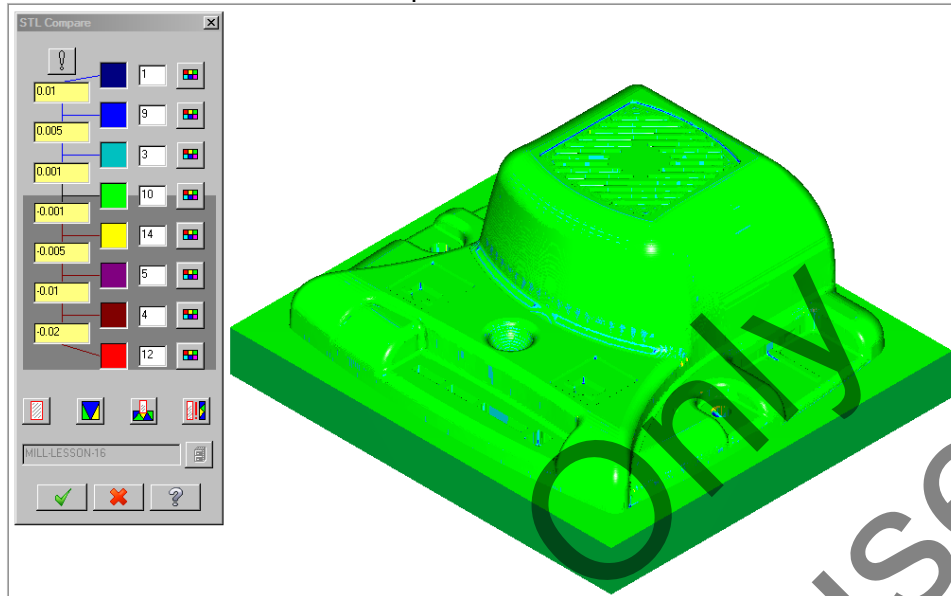


11. Accept the selections by pressing the **OK** button  then select the **machine** button to verify the toolpath. The verified part is shown below:



12. The **STL Compare** dialogue box is now open. Select the **File** button  and select the .stl file that you saved in step 3.

13. 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!

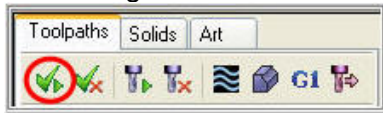
14. Exit the **STL Compare** and **Verify** functions by clicking on the **OK button** .

TASK 11: SAVE THE UPDATED MCX FILE

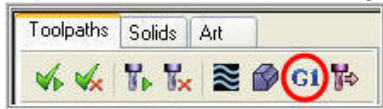
1. Select the save icon from the toolbar .

TASK 12: POST AND CREATE THE CNC CODE FILE

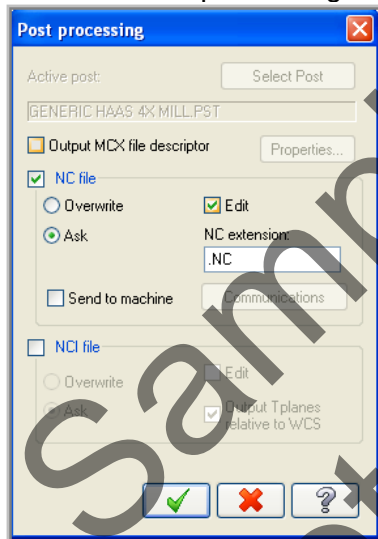
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.





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 manager window and expand the window to the right.



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

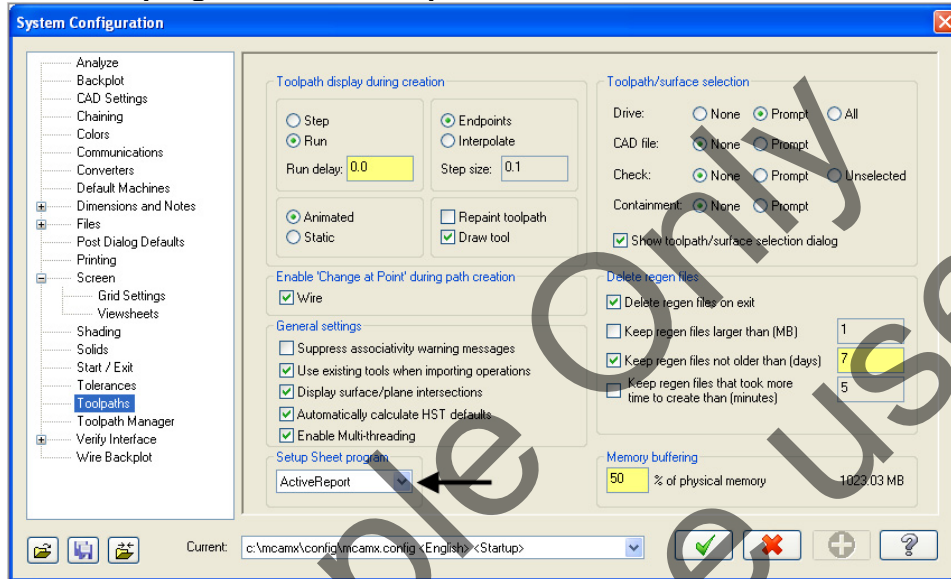




4. Select the **OK** button  to continue.
5. Enter the same name as your Mastercam part file name in the NC File name field **Mill-Lesson-16**.
6. Select the **Save** button.
7. The CNC code file opens up in the default editor.
8. Select the  in the top right corner to exit the CNC editor

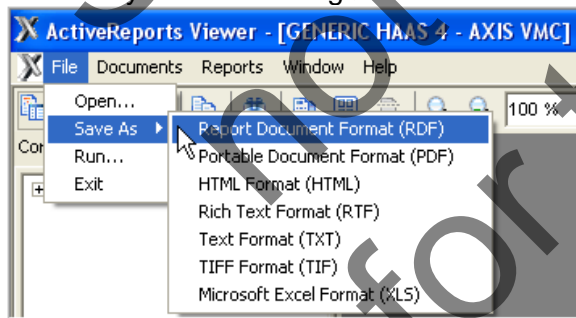
TASK 13: CREATE ACTIVEREPORT

➤ 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. You have the option of saving the report as any of the following file formats.



This completes Mill-Lesson-16.

MILL-LESSON-16 EXERCISE

- The **Mill-Lesson-16-Exercise** file can be found on the accompanying DVD in the Mastercam-Files folder and is called **Mill-Lesson-16-Exercise.mcx**.

Use the information learned in Lesson 16 to create a toolpath for the Mill-Lesson-16-Exercise.

