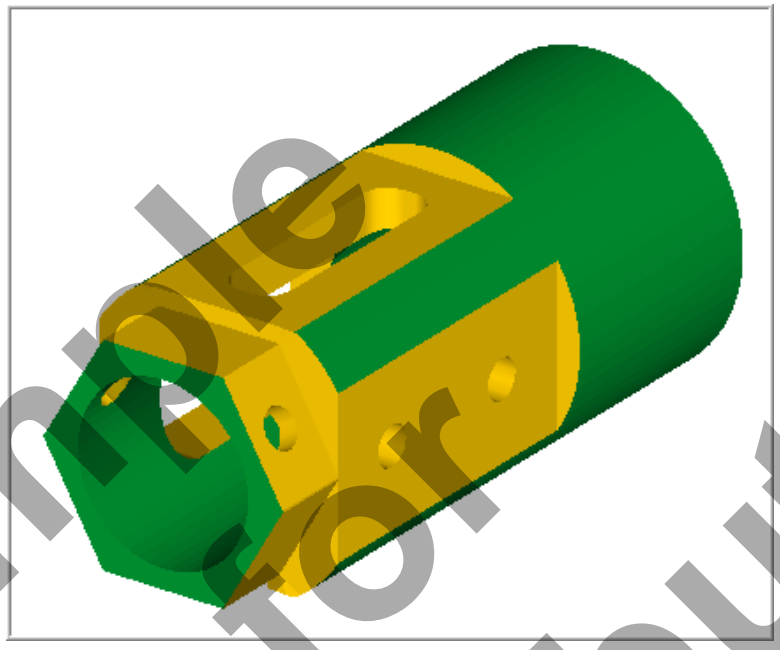
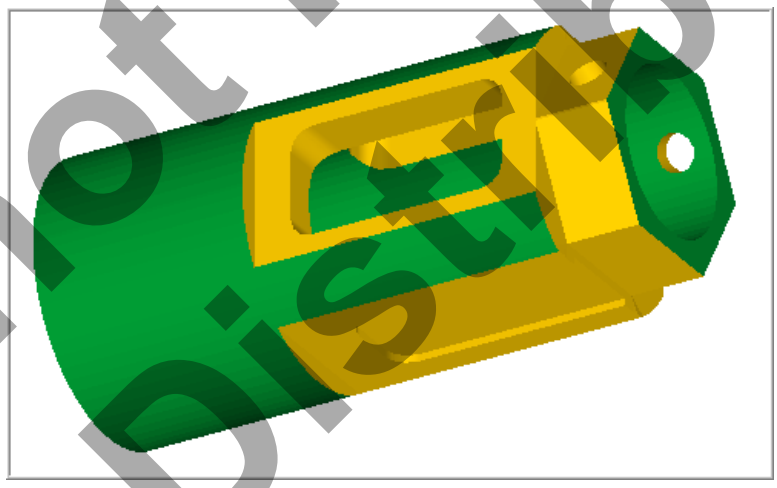
Mastercam X7

TRAINING GUIDE





Sample not for Distribution

FOUR-AXIS-LESSON-3

camInstructor

Objectives

In **Four-Axis-Lesson-3** you will generate the toolpaths to machine the part on a CNC vertical milling machine with four-axis capability. The outside diameters have already been completed.

The part will be held in a rotary indexer as shown below. These types of indexes can utilise collets or a three-jaw chuck. This Lesson covers the following topics:

➲ **Open an existing file containing:**
The solid geometry for the part

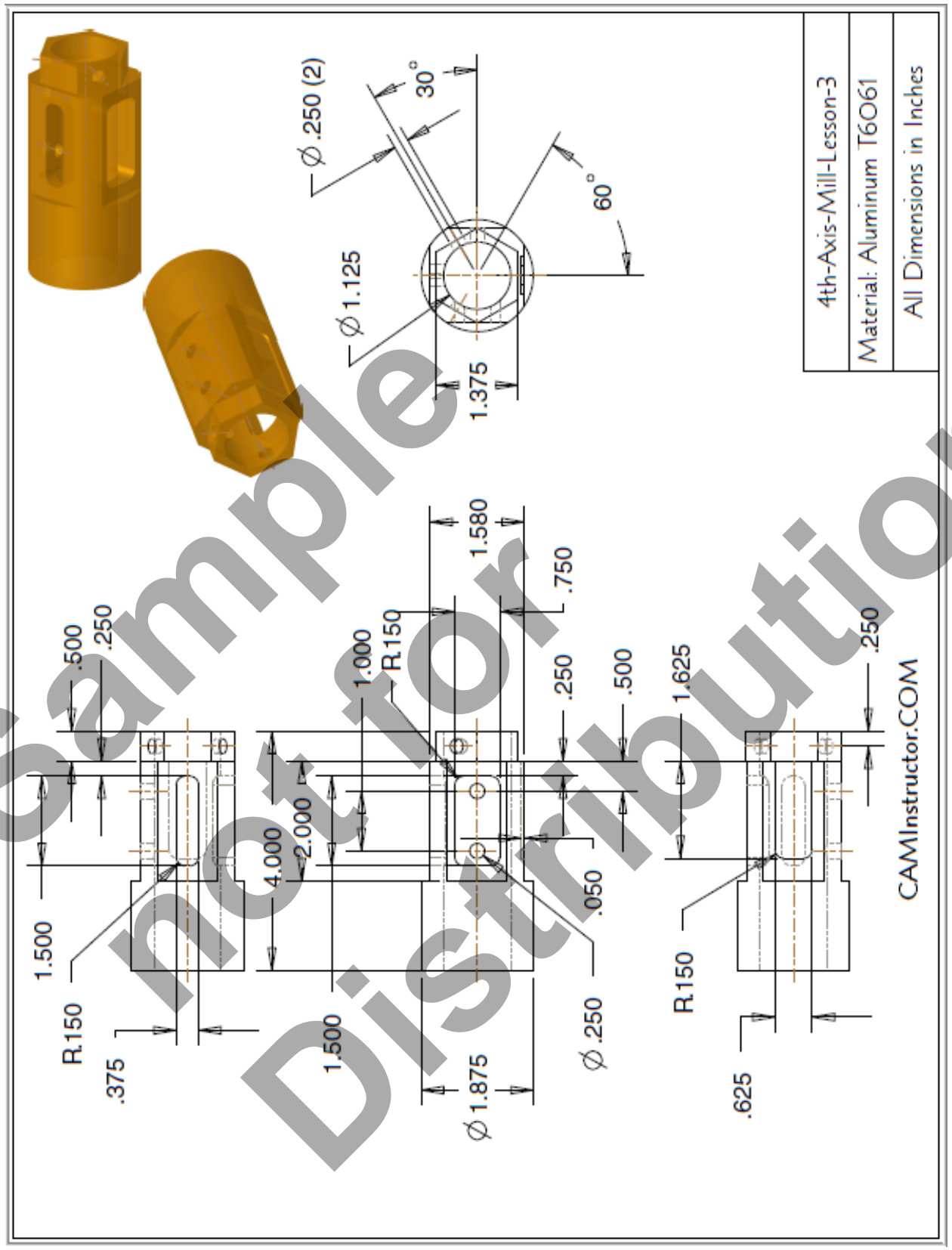
➲ **Establish Stock Setup settings:**
Use a supplied StereoLithography (STL) file for toolpath verification
Material for the part
Feed calculation

➲ **Generate milling toolpath consisting of:**
Using the View Manager
Using Views, Levels, Tool Planes and Construction Planes
Create Contour, Pocket and Drill toolpaths
Using Toolpath Transform to rotate toolpaths

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

Sample not for Distribution

FOUR-AXIS-LESSON- 3 DRAWING



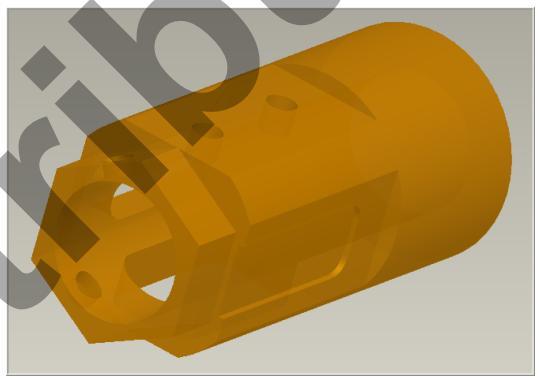
Sample not for Distribution

TOOL LIST

- **Two cutters will be used to create this part.**
- **0.75 diameter flat end mill**
- **0.25 diameter flat end mill**

FOUR-AXIS-LESSON-3 - THE PROCESS

TASK 1: Setting the environment
TASK 2: Introduction - Watch the video
TASK 3: Open an existing file
TASK 4: Identify the entities on each level
TASK 5: Machine the first face of the hexagon
TASK 6: Machine remaining faces of hexagon using Transform Toolpath
TASK 7: Machine the first face of the square
TASK 8: Machine remaining faces of square using Transform Toolpath
TASK 9: Machine Pocket on BACK face
TASK 10: Machine Open Pocket on BOTTOM face
TASK 11: Machine Pocket on TOP face
TASK 12: Drill holes on FRONT face
TASK 13: Drill hole on Hexagon at front
TASK 14: Drill hole on Hexagon at back using Transform Toolpath
TASK 15: Verify toolpaths using a supplied STL file
TASK 16: Post and create the CNC code file



➲ Getting Started

Before working through these lessons you may find it useful to review the material in the **Tips and Techniques** section of the multimedia DVD that came with this text covering **Work Coordinate Systems (WCS)**.

If you are using the Online Course content the Work Coordinate Systems (WCS) material is in the Tips and Techniques section of the main menu.

Sample not for Distribution

TASK 1:
SETTING THE ENVIRONMENT

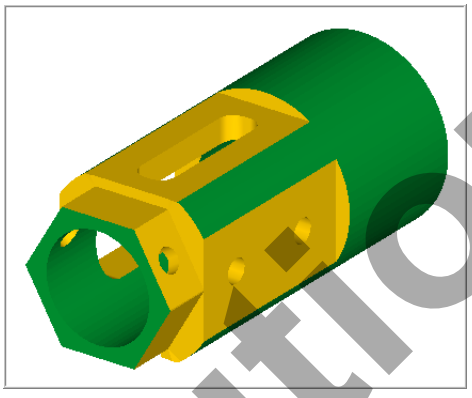
Before starting the geometry creation you should set up the grid, toolbars and machine type as outlined in the **Setting the environment** section at the beginning of this text:

1. Set up the **Grid: Size= 1.0, X & Y Spacing= 0.25.** This will help identify the location of the origin.
2. Customize the toolbars to machine a **4-5 Axis part**.

TASK 2: INTRODUCTION – WATCH THE VIDEO

1. Before you start to work on this Lesson review the Introduction video on the multimedia DVD that came with this text. You will find the video in the **"Four-Axis-Lesson-3**" section, it is entitled **Introduction - 2 Minutes**. If you are using the Online Course content the video is available in this Lesson on the online content screen.

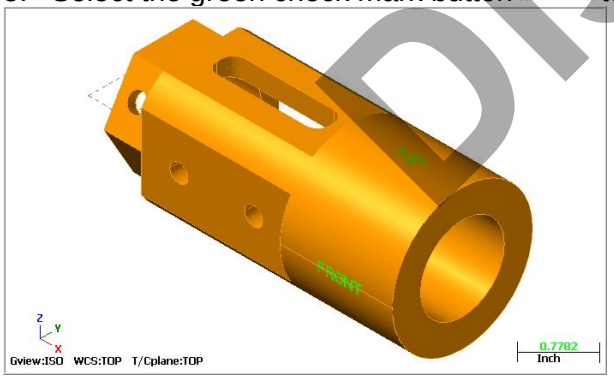
➲ The video will review the techniques that will be used to generate the toolpaths utilizing the indexing table.



TASK 3:
OPEN AN EXISTING FILE

➲ **If you are using a Training Guide to go through this Lesson the required file is on the multimedia DVD that came with this text in a folder called Mastercam Files. If you are using the Online Course content the file is available in this Lesson on the online content screen. The part is already setup for a: GENERIC HAAS 4 AXIS MILL.**

1. Select **File>Open> Four-Axis-Lesson-3.** File is shown below in shaded view.
2. Select **File>Save as…**
3. In the File name box, type **Four-Axis-Lesson-3.**
4. Save to an appropriate location**.**
5. Select the green check mark button to save the file and complete this function.



Sample not for Distribution

TASK 4:
IDENTIFY THE ENTITIES ON EACH LEVEL

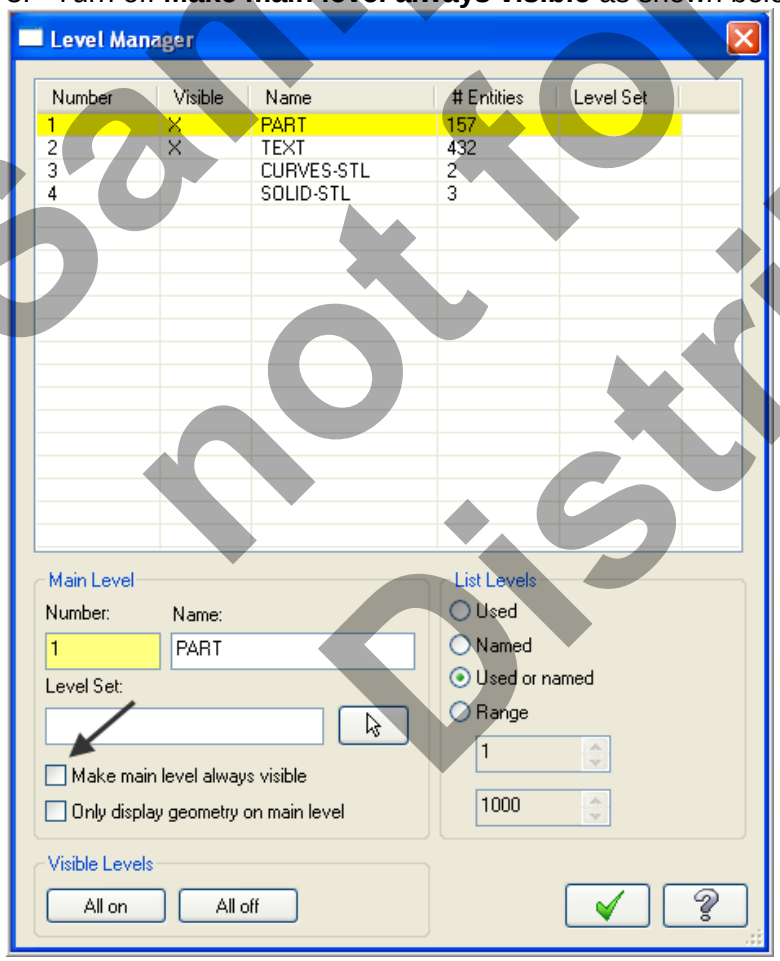
➲ **Before you start to work on the toolpaths lets first identify the entities on each Level.**

➲ Levels are a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities, and toolpaths. ➲ By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and which parts are selectable so that you do not inadvertently make changes to areas of the drawing you do not want to change.

1. Activate a shaded view by selecting the icon at the top of the screen. .
2. From the **Status** bar at the bottom of the screen select **Level**.

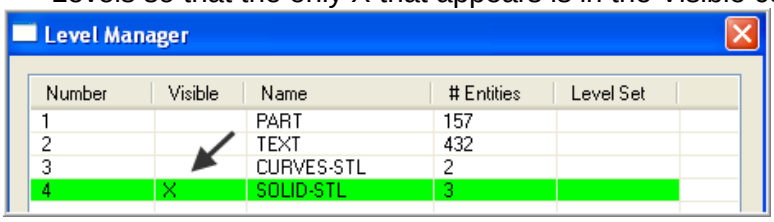
➲ **The Level Manager dialog window will now appear.**
➲ **As you can see in the picture below there are four Levels in this part.**
➲ **Level 1 contains the Part geometry**
➲ **Level 2** contains the **Text**, **TOP, BOTTOM, BACK and FRONT**. This is on the part to help you with the orientation of the part as you work through creating the toolpaths
➲ **Level 3 contains the Curves that were previously created for the STL file geometry**
➲ **Level 4 contains the Solid geometry that was previously created for the STL file geometry**
➲ **Level 1 and 2** are visible as noted by the X in the Visible column.

3. Turn off **Make main level always visible** as shown below.



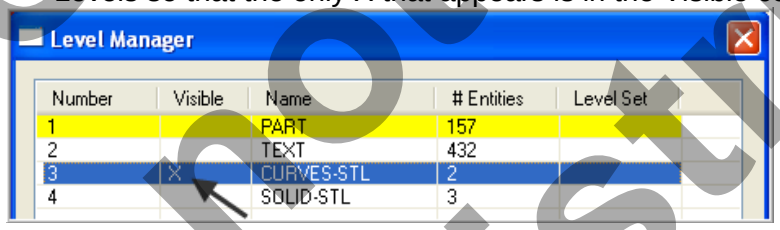
Sample not for Distribution

4. Now make visible **only Level 4 SOLID-STL.** Click in the **Visible column** for the various Levels so that the only X that appears is in the Visible column for Level 4.



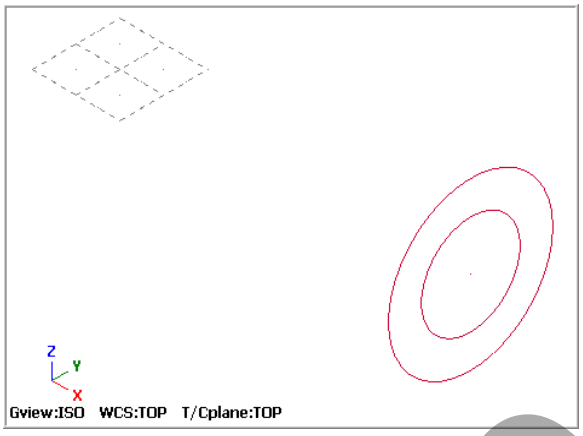
5. Select the OK button to exit the **Level Manager** dialog box. On the screen is the Solid Extrude used to for the STL file. This STL file will be used to verify the toolpaths.

6. From the **Status** bar at the bottom of the screen select **Level**.
7. Now make visible **only Level 3 CURVES-STL.** Click in the **Visible column** for the various Levels so that the only X that appears is in the Visible column for Level 3.

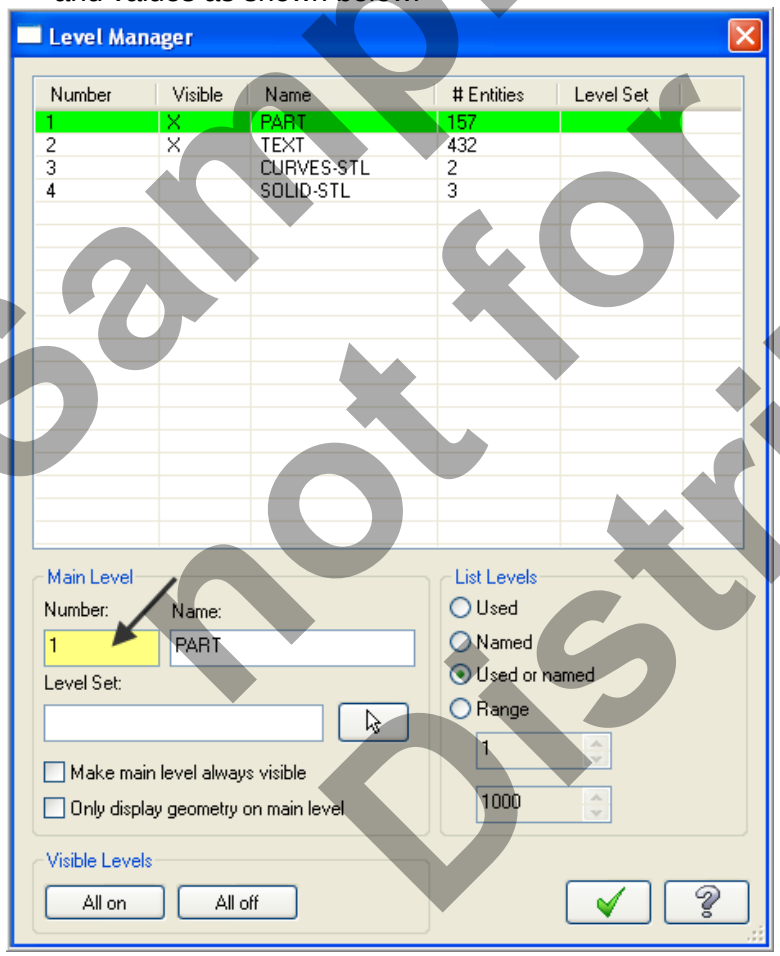


Sample not for Distribution

8. Select the OK button to exit the **Level Manager** dialog box. There are only two circles on this level. These two circles were used to create the Solid Extrude used to for the STL file.



9. From the **Status** bar at the bottom of the screen select **Level**.
10. Now make visible **only Level 1 and 2.** Click in the **Visible column** for the various Levels so that the only X that appears is in the Visible column for Level 1 and 2. Set remaining options and values as shown below.

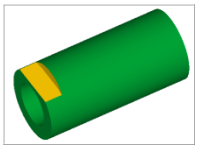


11. Select the OK button to exit the **Level Manager** dialog box.

Sample not for Distribution

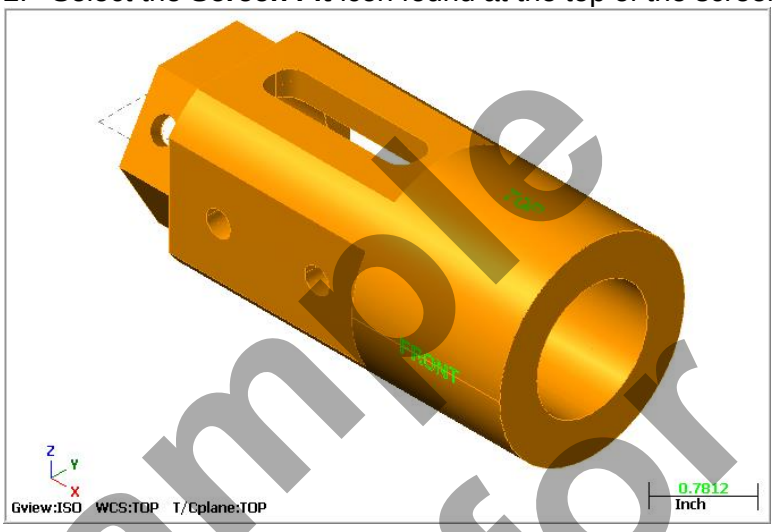
TASK 5:
MACHINE THE FIRST FACE OF THE HEXAGON

➲ In this task you will use the 0.75" diameter flat end mill to rough and finish the first face of the hexagon using Toolpath> Contour. In the next task you will use Toolpath>Transform to machine the remaining faces of the hexagon.



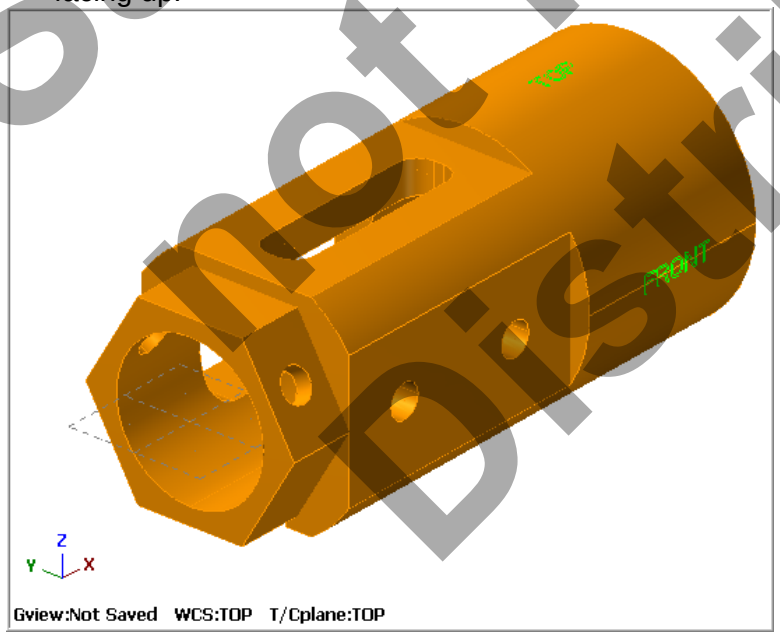
1. Set the graphics view to **Isometric View**.

2. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .



Note
At the bottom of the graphics screen the status of the **Tool and Construction** plane is set to **TOP.**

3. Now dynamically rotate the display similar to the display shown below. The text **TOP** is facing up.



Tool plane (Tplane)
For most typical toolpaths, this is the plane in which cutting moves (G1/G2/G3) take place.

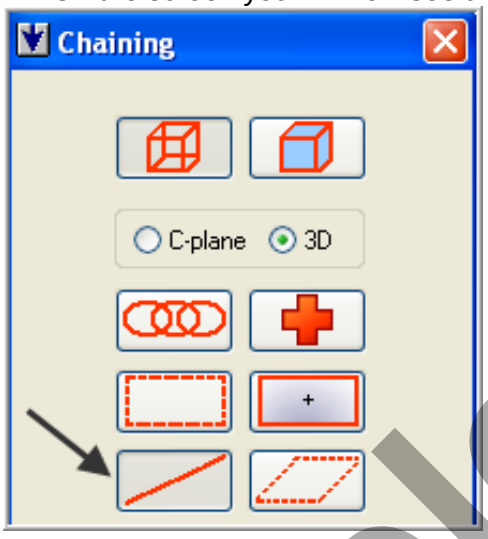
The tool typically approaches and retracts from the part normal to the tool plane in the Z axis.

You can align the Tplane with any of the standard planes or a custom orientation.

4. Activate a wireframe view by selecting the icon at the top of the screen. .

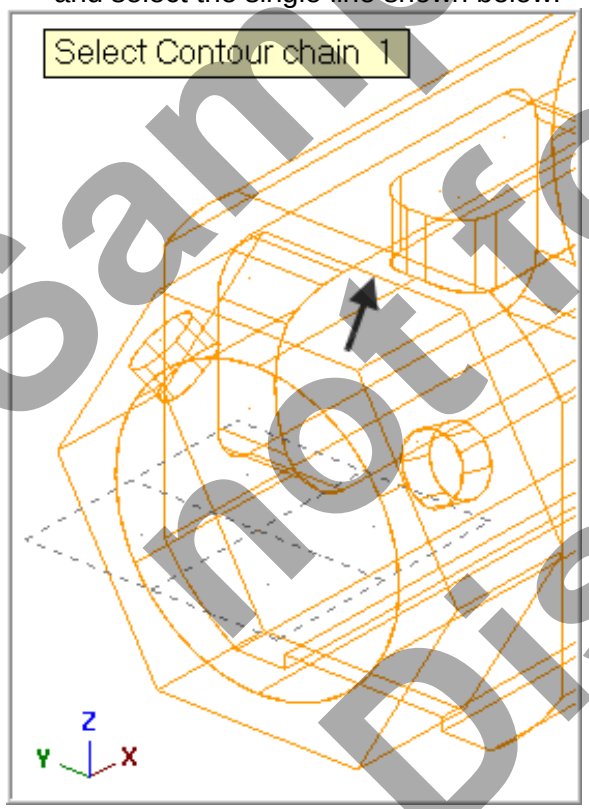
Sample not for Distribution

5. From the menu bar select **Toolpaths>Contour…**
6. When prompted to **Enter new NC name** ensure **FOUR-AXIS-LESSON-3** is displayed and then select the OK button .
7. On the screen you will now see the **Chaining dialog box.** Select **Single chain**.



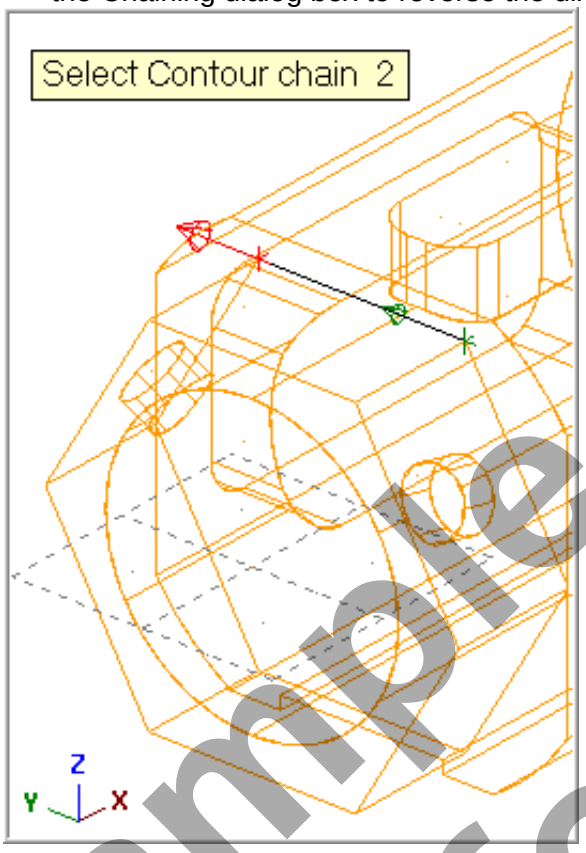
Single
Creates a chain out of a single line, arc, or spline.

8. On the graphics screen is now a prompt to **Select contour chain 1**. Zoom in as required and select the single line shown below:

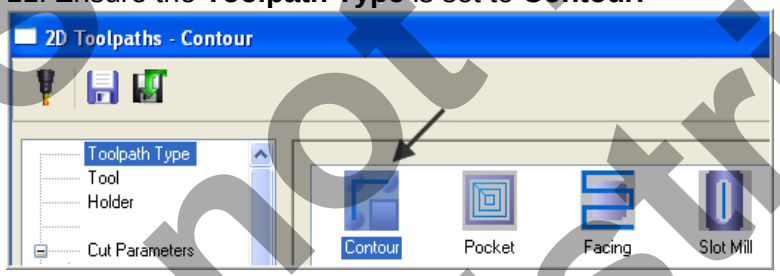


Sample not for Distribution

9. After selecting the line your graphics screen should look like the screenshot below, with the **green arrow pointing left**. If the arrow is not pointing to the left select the arrow from the Chaining dialog box to reverse the direction.

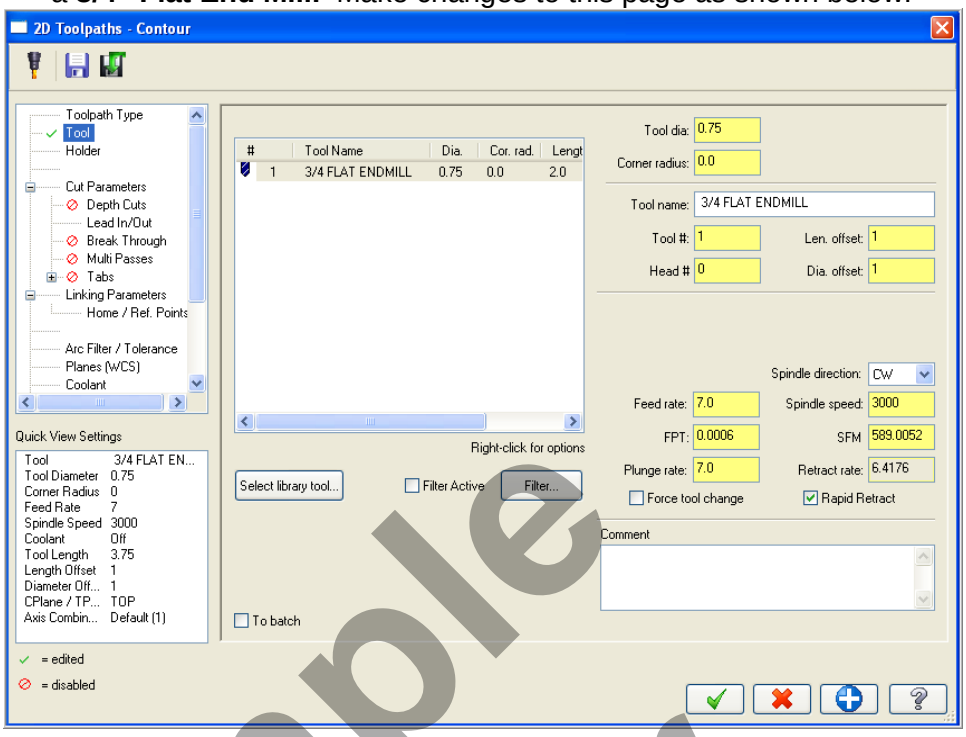


10. After the contour has been successfully chained select the OK button at the bottom of the Chaining dialog box.
11. Ensure the **Toolpath Type** is set to **Contour.**

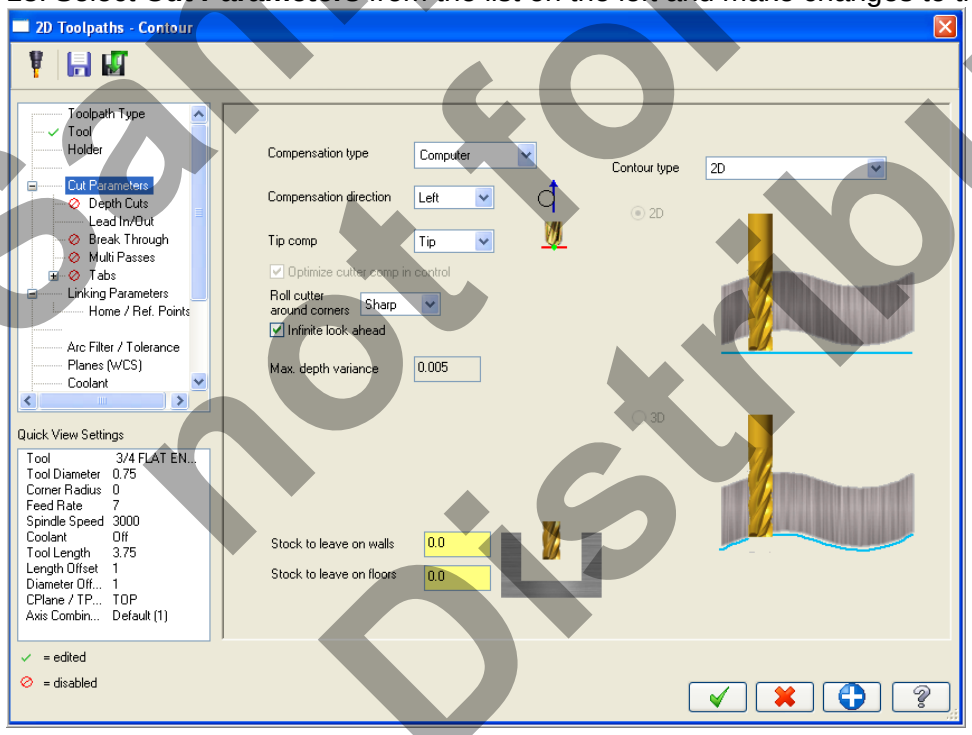


Sample not for Distribution

12. On the left of the dialog box select **Tool**. Click on the **Select library tool** button and choose a **3/4" Flat End Mill.** Make changes to this page as shown below:



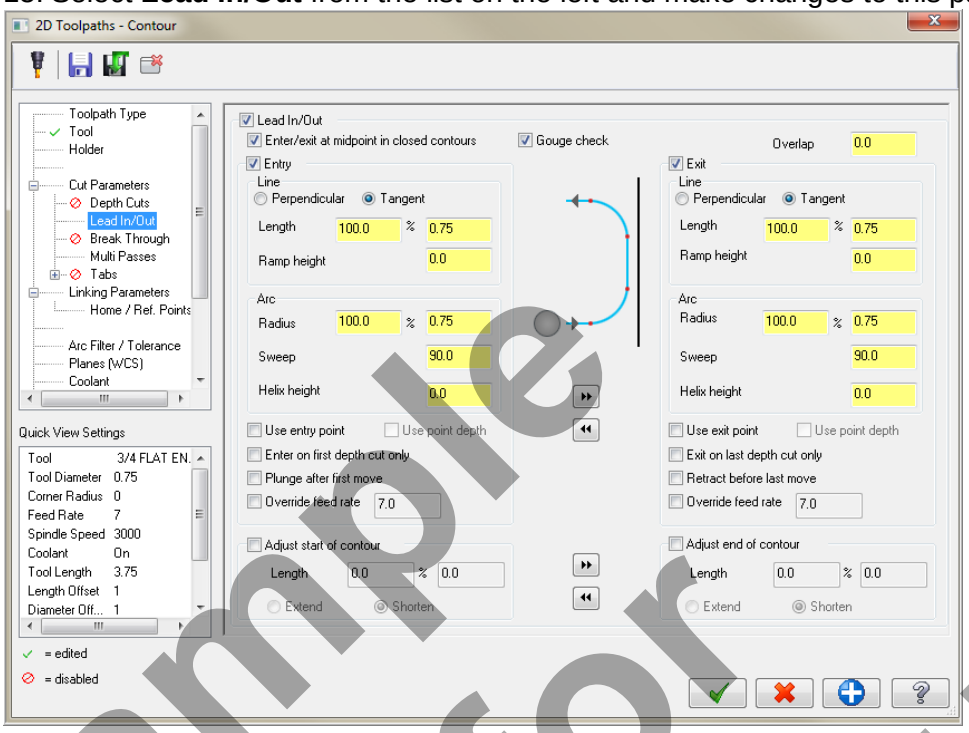
13. Select **Cut Parameters** from the list on the left and make changes to this page if required.



Sample not for Distribution

14. **Depth Cuts** should not be activated.

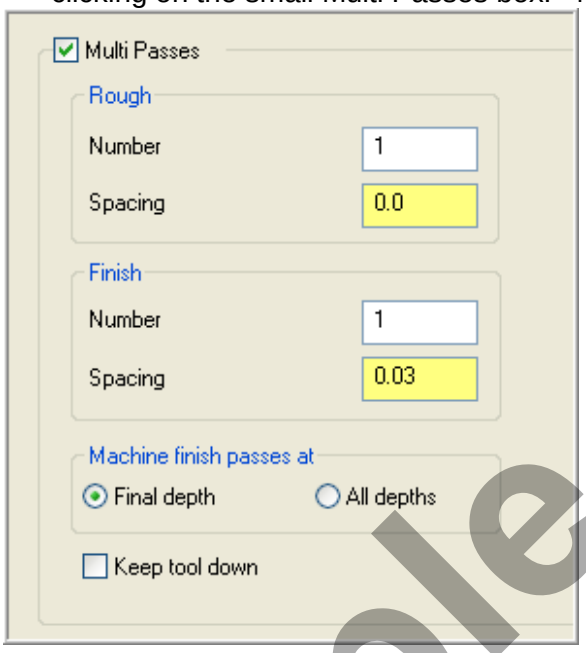
15. Select **Lead In/Out** from the list on the left and make changes to this page as shown below:



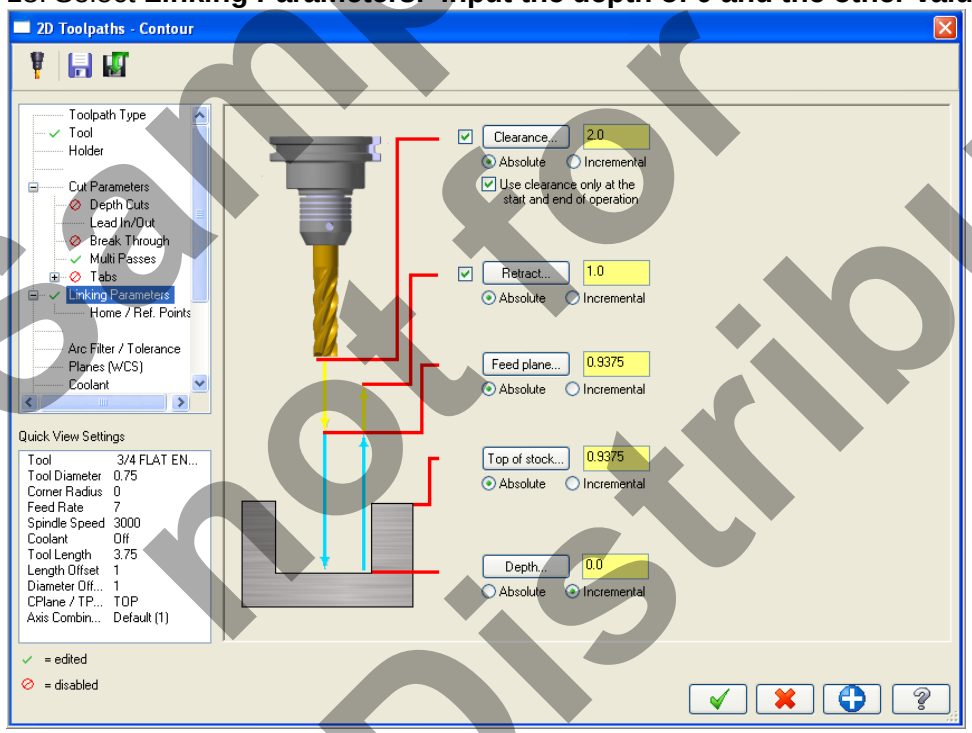
16. **Break Through** should not be activated.

Sample not for Distribution

17. Select **Multi Passes** from the list on the left and ensure **Multi Passes** are activated by clicking on the small Multi Passes box. Make changes to this page as shown below:

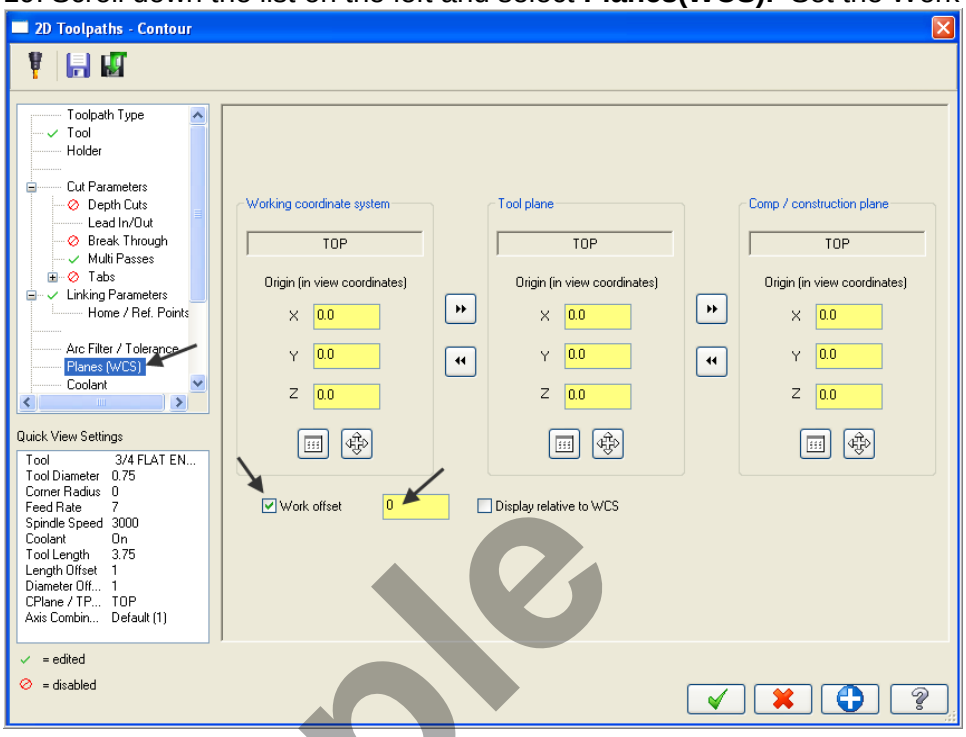


18. Select **Linking Parameters. Input the depth of 0 and the other values as shown below:**



Sample not for Distribution

19. Scroll down the list on the left and select **Planes(WCS).** Set the Work offset to **Zero**.



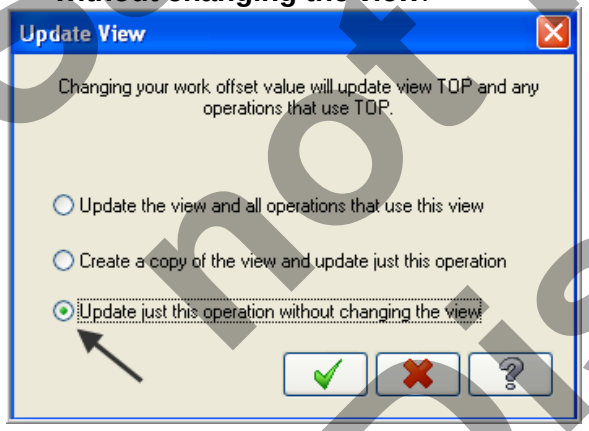
Work Offset # Assigns a work offset code to the view. When you use the view for a Tplane and create a toolpath.

Mastercam will automatically output the work offset when you post the operation. Number the offsets sequentially from zero, or enter –1 to not assign any offset to the view.

Work Offset # Setting this value to **Zero** will out a **G54** in the CNC code file.

```
N1 G20
N2 G0 G17 G40 G49 G80 G90
N3 T1 M6
N4 G0 G90 G54 X.1875 Y-.005 A0. S4500 M3
N5 G43 H1 Z2.
N6 M8
```

20. When confronted with the **Update View** dialog box activate **Update just this operation without changing the view**.

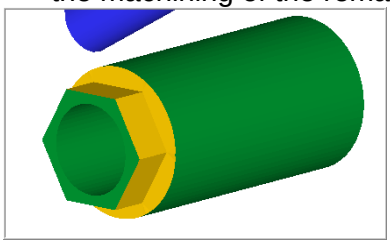


21. Click on the OK button when done .
22. Select Coolant from the list on the left**.** Open up the drop down menu for Flood and set it to On.
23. Select the OK button to exit Contour parameters.
24. Save your Mastercam file.
25. Backplot this toolpath.

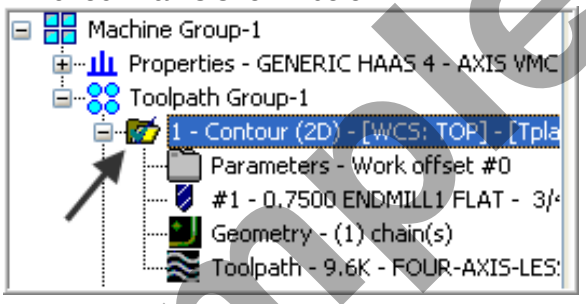
Sample not for Distribution

TASK 6:
MACHINE REMAINING FACES OF HEXAGON USING TRANSFORMTOOLPATH

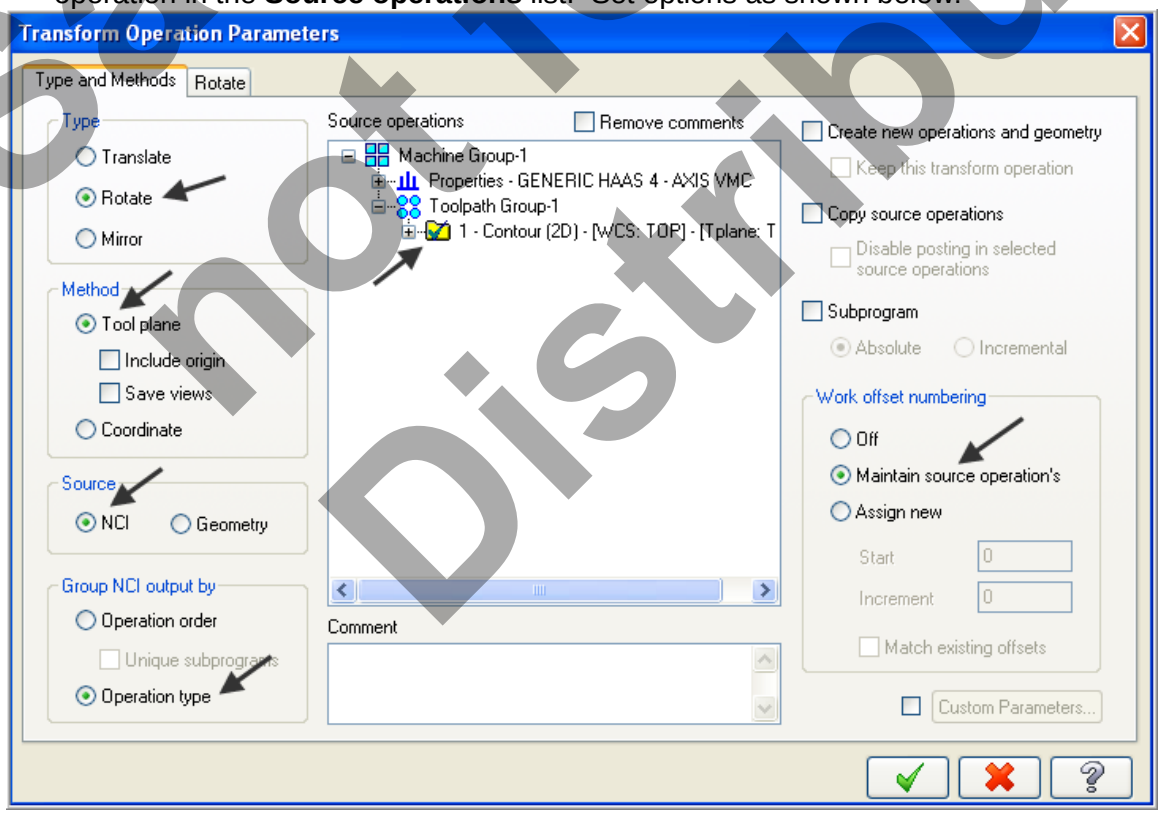
➲ In this task you will use Transform Toolpath to rotate the first contour toolpath to complete the machining of the remaining faces of the hexagon.



1. In the Toolpaths manager **select only the first contour** operation, as indicated by the blue checkmarks shown below.

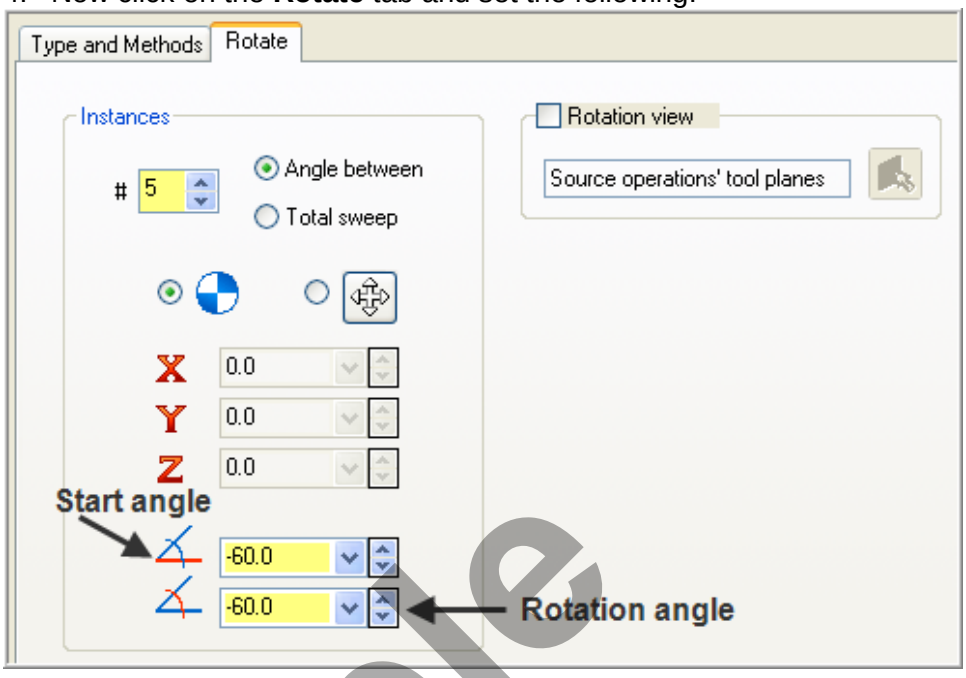


2. From the menu bar select **Toolpaths>Transform…**
3. On the **Types and Methods** page set the following. **Ensure** there is a check for the contour operation in the **Source operations** list. Set options as shown below:



Sample not for Distribution

4. Now click on the **Rotate** tab and set the following:



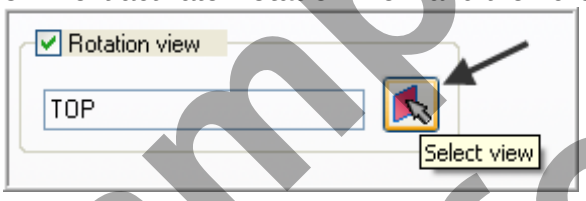
Start angle

Sets the beginning angle for the rotate toolpath.

Rotation angle

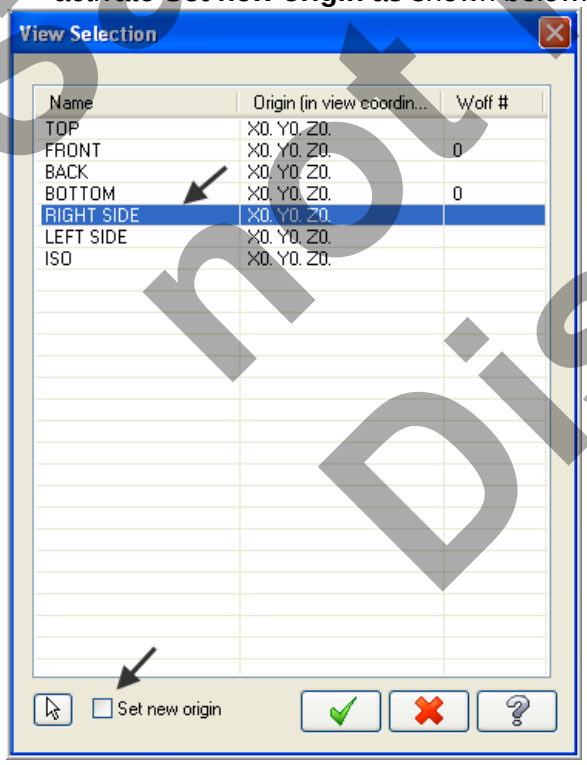
Sets the angle of rotation for the transform toolpath.

5. Next activate **Rotation view** and then click on the **Select view** icon.



Rotation view
Select this option to rotate the transformed toolpaths in a different plane than the original. If this option is inactive, the toolpath rotates in the original tool plane.

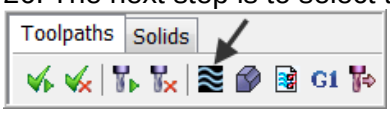
6. You will be confronted with the **View Selection** dialog box, select **RIGHT SIDE** and de-activate **Set new origin** as shown below:



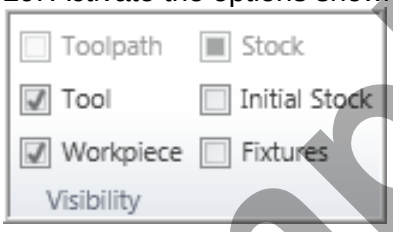
Sample not for Distribution

7. Select the OK button to exit the **View Selection** dialog box.
8. Select the OK button to exit the **Transform Operations Parameters** dialog box.
9. **Save** your Mastercam file.
10. Activate a shaded view by selecting the icon at the top of the screen.
11. Backplot all the toolpaths.
12. To pick all the operations to Backplot pick the **Select All** icon circled below:

26. The next step is to select the **Backplot selected operations** icon shown below:

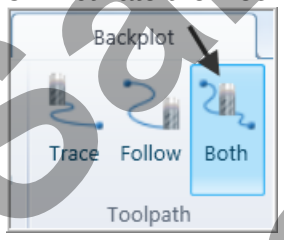


27. **Maximize** the Backplot/Verify window if required.
28. Select the **Home** Tab if required.
29. Activate the options shown below in the **Visibility** section of the Home tab.



30. Click on the **Backplot** tab at the top left of the screen.

31. Activate the **Both** option in the Toolpath section of the Backplot tab.



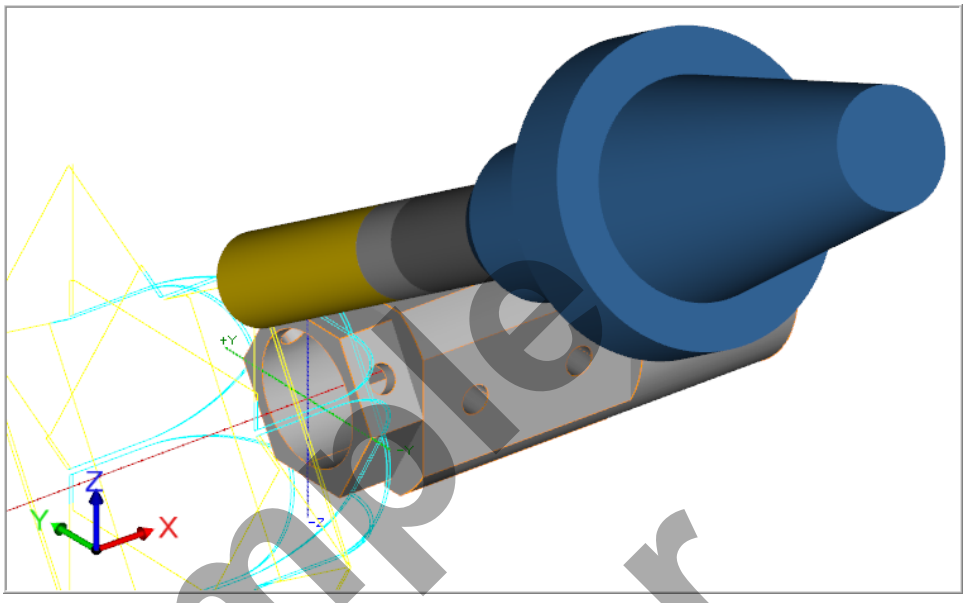
32. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.

Sample not for Distribution

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

34. After reviewing the Backplot of the toolpath use the **Close** button to exit Backplot.

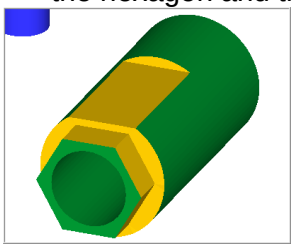
➲ Your part should look similar to the screen shot below.



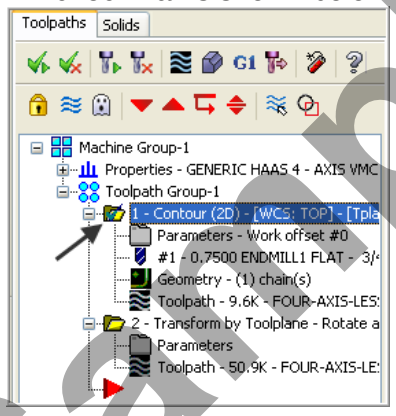
Sample not for Distribution

TASK 7:
MACHINE THE FIRST FACE OF THE SQUARE

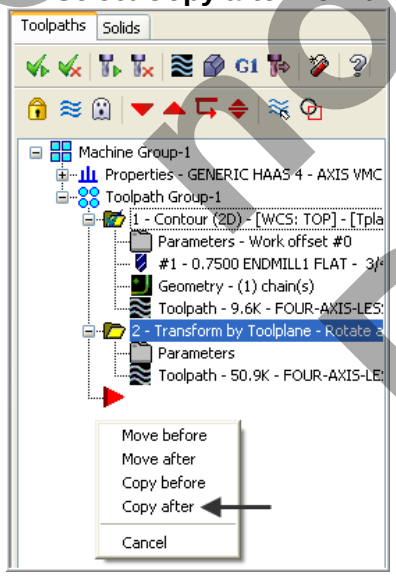
- In this task you will use a 0.75" diameter flat end mill to rough and finish the first face of the square.
- For this example you will copy the first contour operation used to machine the first face of the hexagon and then modify the copied operation to machine the first face of the square.



1. Activate a wireframe view by selecting the icon at the top of the screen. .
2. In the Toolpaths manager **select only the first contour** operation, as indicated by the blue checkmarks shown below. The contour operation for the hexagon is the first operation.

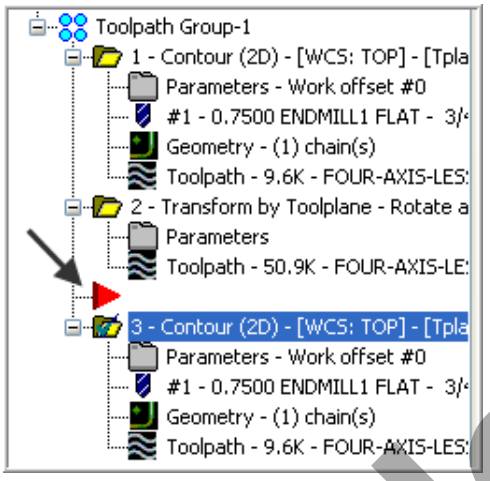


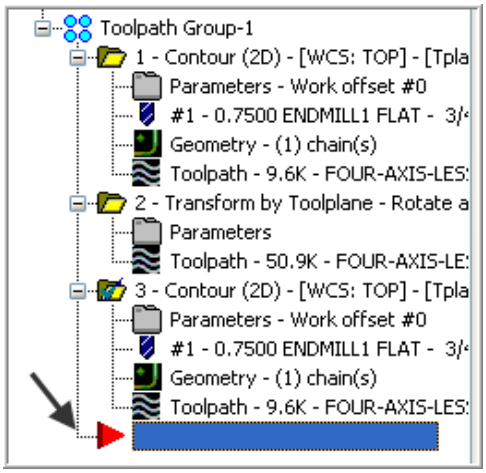
3. To copy this operation **Hold down the Right mouse button over the folder for this operation**. **Keep holding the right mouse button** down and drag below the red arrow and release.
4. Select **Copy after** from this menu.



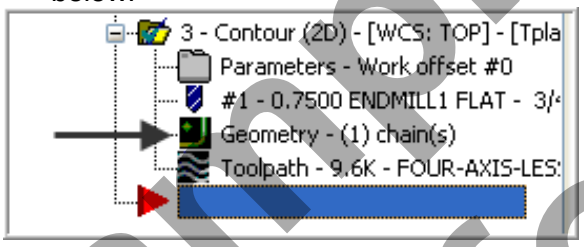
Sample not for Distribution

5. Your screen will look similar to the screenshot below on the left. Move the Insert arrow to the bottom of the list of operations by clicking the icon on the Toolpaths Manager toolbar:

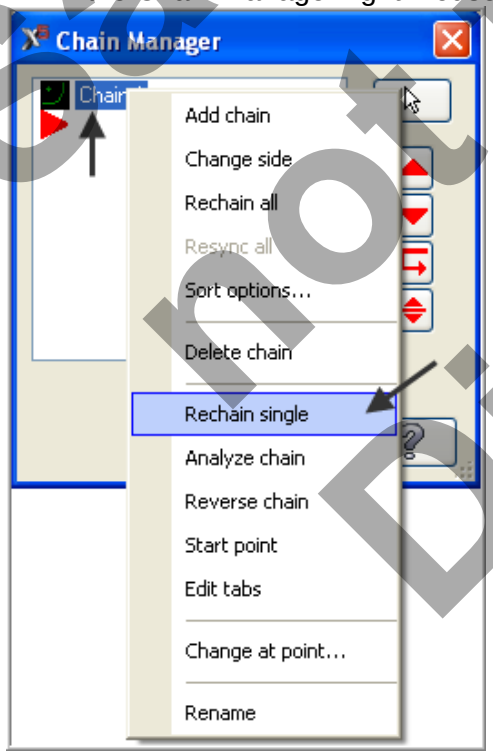




6. In the **Toolpaths Manager** click on the Geometry icon for the **third** operation as shown below:

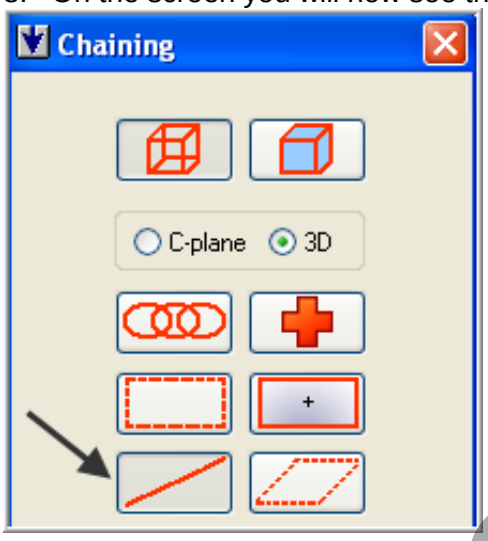


7. In the Chain Manager right mouse click on **Chain 1** and select **Rechain Single**.

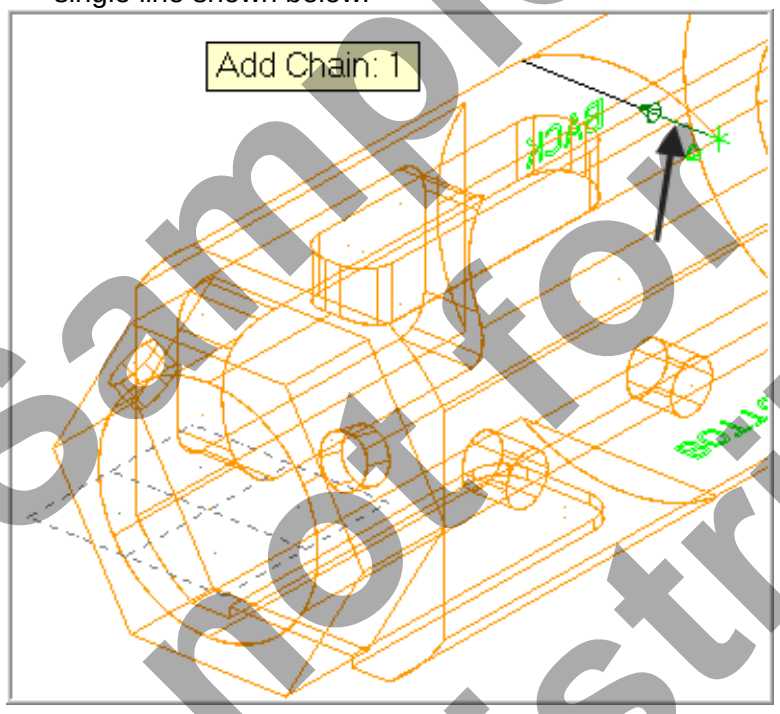


Sample not for Distribution

8. On the screen you will now see the **Chaining dialog box.** Select **Single chain**.



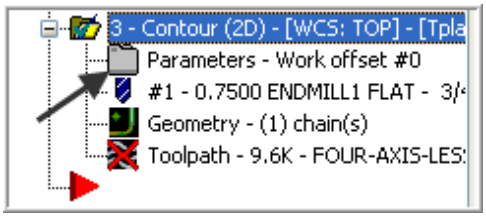
9. On the graphics screen is now a prompt to **Add chain 1**. Zoom in as required and select the single line shown below:



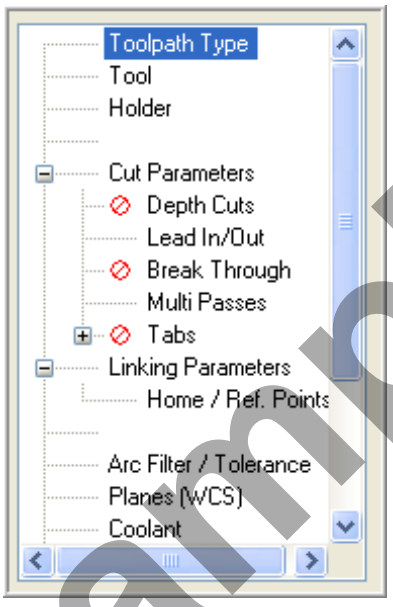
10. After selecting the line your graphics screen should look like the screenshot above, with the **green arrow pointing left**. If the arrow is not pointing to the left select the arrow from the Chaining dialog box to reverse the direction.
11. After the contour has been successfully chained select the OK button at the bottom of the Chaining dialog box.
12. Select the OK button at the bottom of the Chain Manager box.

Sample not for Distribution

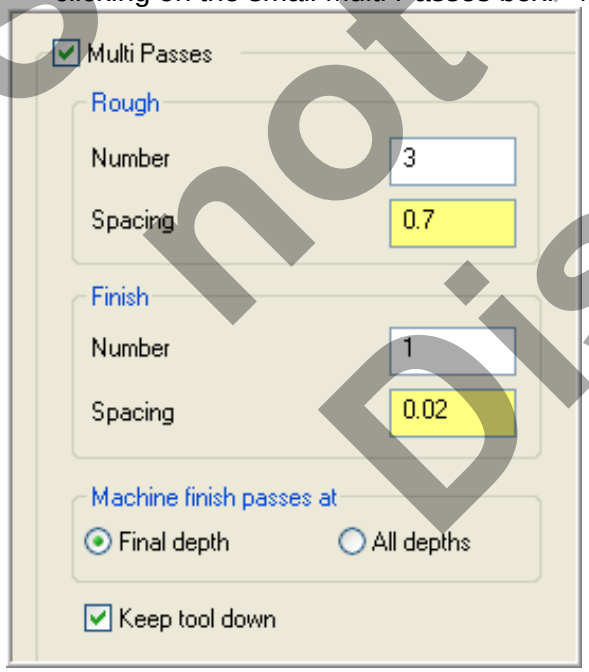
13. In the **Toolpaths Manager** click on the folder **Parameters** for the **third** operation as shown below:



14. All the parameters and values you set in the first contour operation will still be set. **No changes required for**: Tool, Cut Parameters, Lead/In/Out, Linking Parameters, Planes and Coolant.



15. Select **Multi Passes** from the list on the left and ensure **Multi Passes** are activated by clicking on the small Multi Passes box. Make changes to this page as shown below:

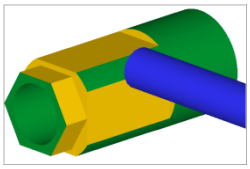


Sample not for Distribution

16. Select the OK button to exit Contour parameters.

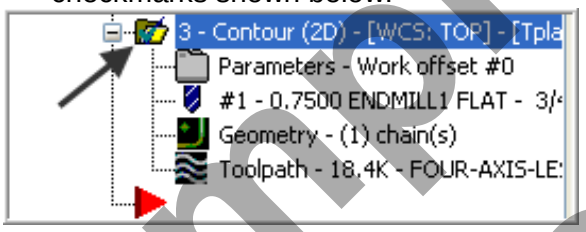
17. Select the **Regenerate all dirty operations** button to remove the red X from the contour operation you have just edited.
18. Save your Mastercam file.
19. Backplot this toolpath.

TASK 8:
MACHINE REMAINING FACES OF SQUARE USING TRANSFORMTOOLPATH

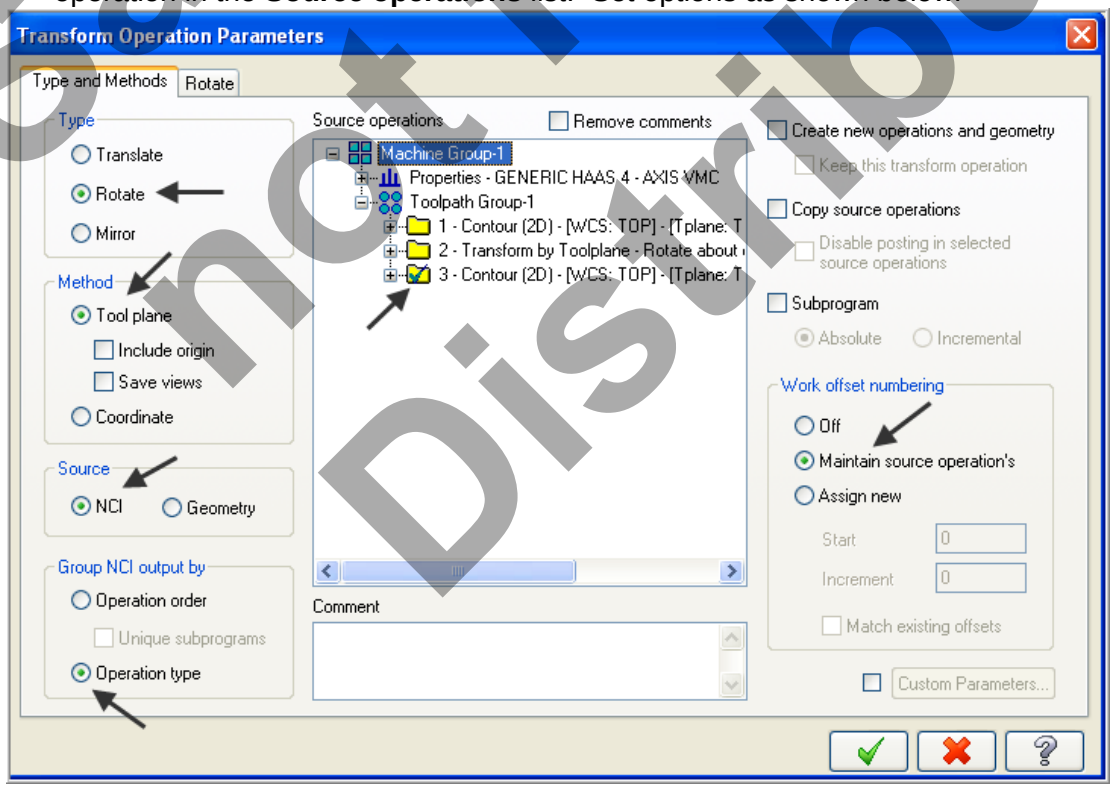


➲ In this task you will use **Toolpath>Transform** to machine the remaining faces of the square.

1. In the Toolpaths manager **select only the third contour** operation, as indicated by the blue checkmarks shown below.

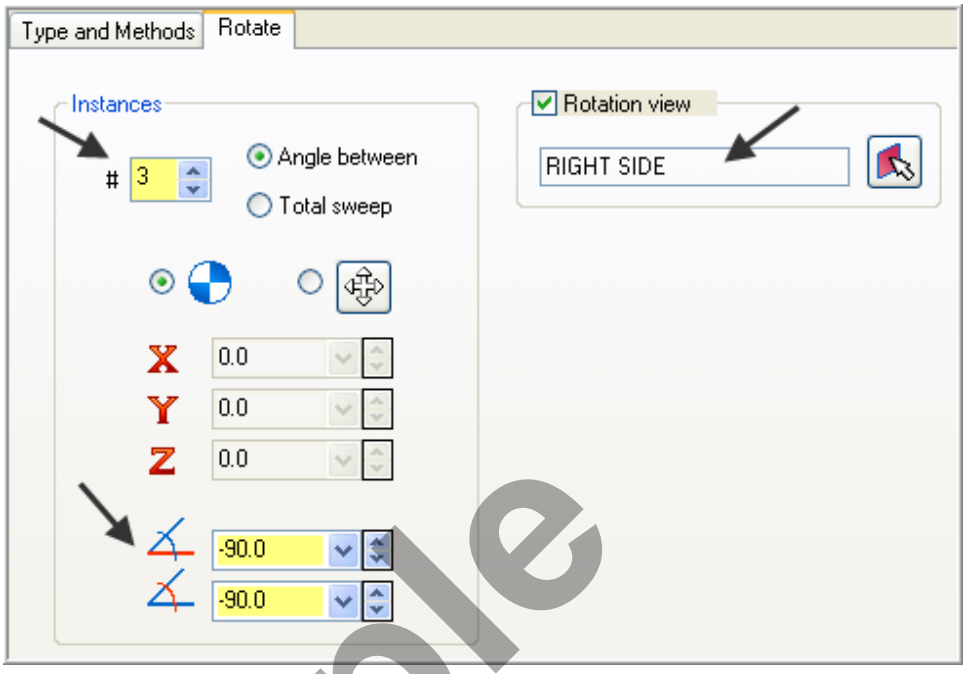


2. From the menu bar select **Toolpaths>Transform…**
3. On the **Types and Methods** page set the following. Ensure there is a check for the contour operation in the **Source operations** list. Set options as shown below:



Sample not for Distribution

4. Now click on the **Rotate** tab and set the following. If required click on the Select view icon and select **RIGHT SIDE**.



Start angle

Sets the beginning angle for the rotate toolpath.

Rotation angle

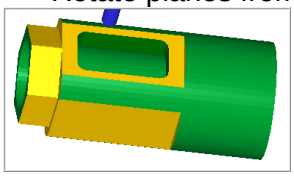
Sets the angle of rotation for the transform toolpath.

5. Select the OK button to exit the **Transform Operations Parameters** dialog box.
6. **Save** your Mastercam file.
7. Activate a shaded view by selecting the icon at the top of the screen. .
8. Backplot all the toolpaths. To pick all the operations to Backplot pick the **Select All** icon **.**

Sample not for Distribution

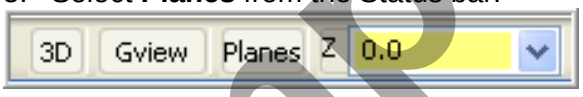
TASK 9:
MACHINE POCKET ON BACK FACE

- In this task you will use a 0.25" diameter flat end mill to rough and finish the pocket on the BACK face using Toolpath>Pocket.
- In this task you will not use the standard **BACK Tplane**, but will create a new Tplane using Rotate planes from the Planes menu on the Status bar.

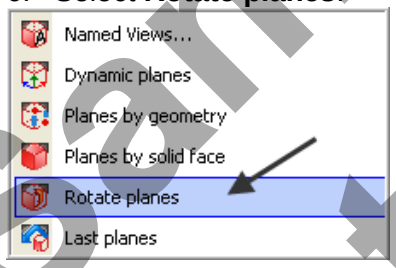


1. Select **Alt-T** on your keyboard to **turn off** the display of toolpaths.
2. Set the graphics view to **Isometric View**.
3. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .
4. Activate a shaded view by selecting the icon at the top of the screen.

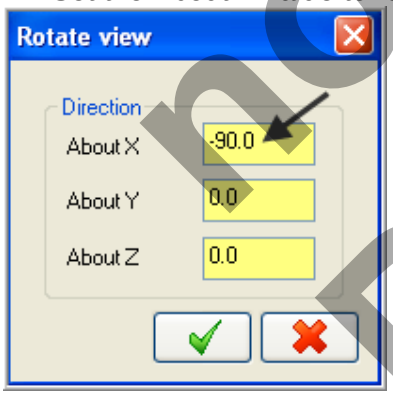
5. Select **Planes** from the Status bar.



6. Select **Rotate planes**.



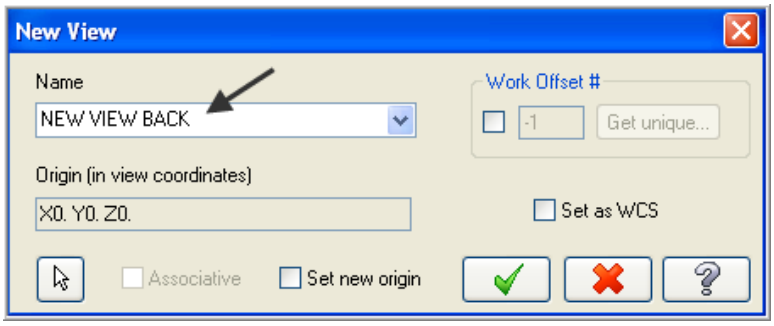
7. Set the About X value to -90.



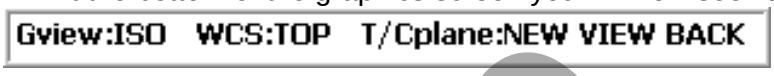
8. Click on the OK button when done .

Sample not for Distribution

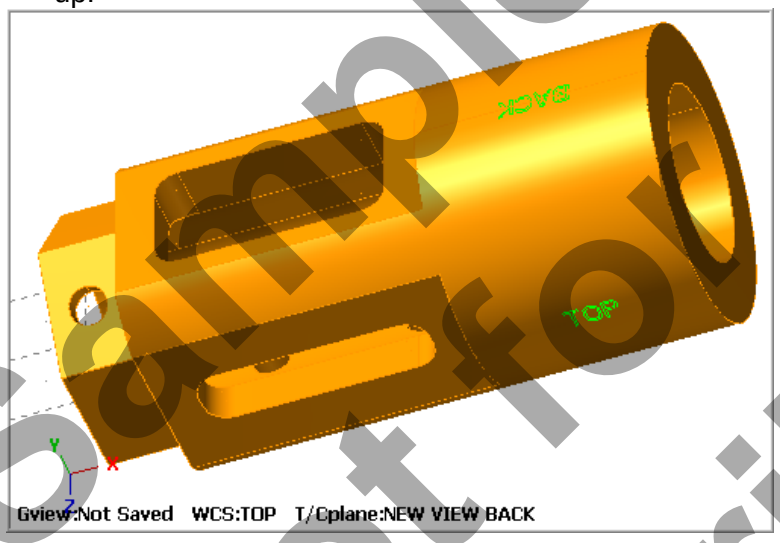
9. In the New View dialog box change the name to **NEW VIEW BACK**. Make changes as shown below:



10. Click on the OK button when done .
11. At the bottom of the graphics screen you will now see **T/Cplane: NEW VIEW BACK**

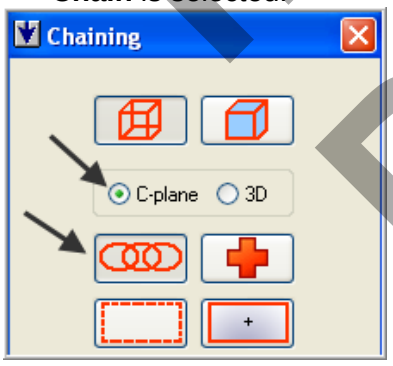


12. Dynamically rotate the display similar to the display shown below. The text **BACK** is facing up.



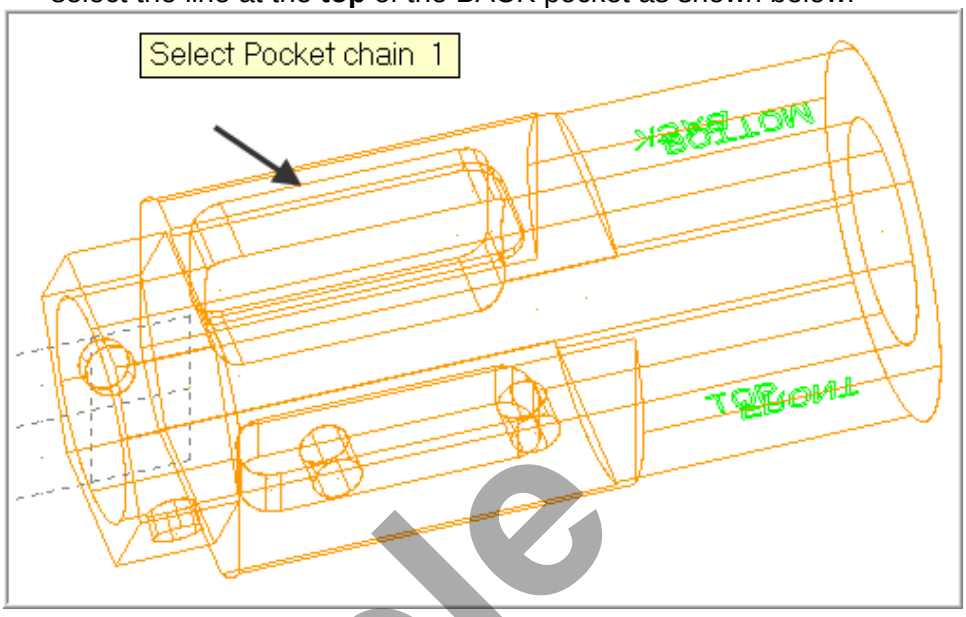
Note
At the bottom of the graphics screen the status of the Tool and Construction plane is set to **NEW VIEW BACK.**

13. Activate a wireframe view by selecting the icon at the top of the screen. .
14. From the menu bar select **Toolpaths>Pocket…**
15. On the screen you will now see the **Chaining dialog box.** Activate **C-Plane** and ensure **Chain** is selected.

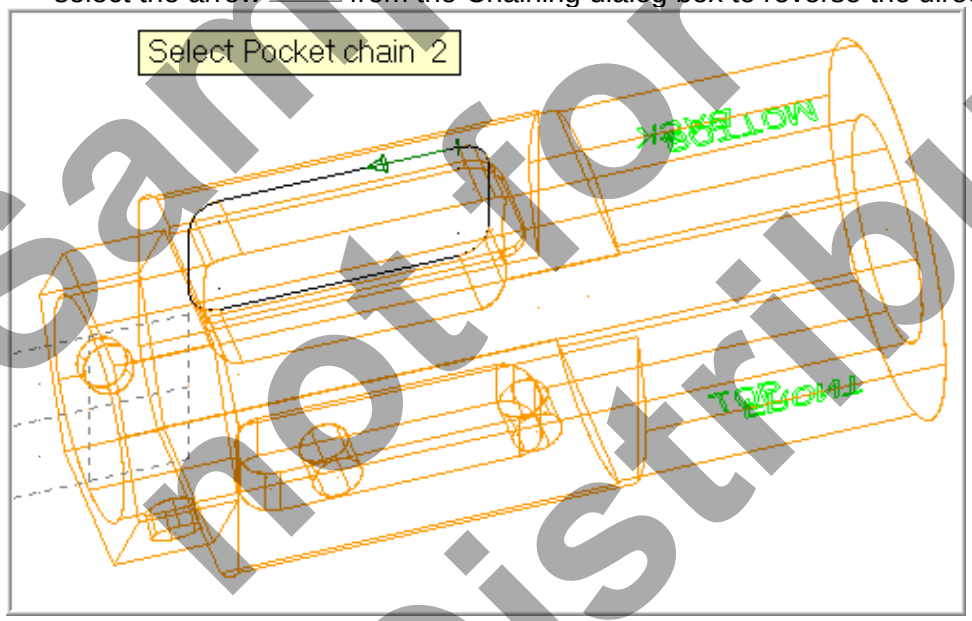


Sample not for Distribution

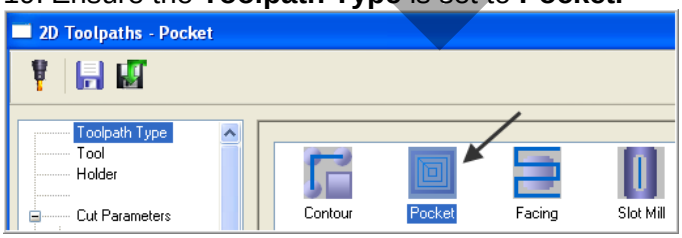
16. On the graphics screen is now a prompt to **Select pocket chain 1** Zoom in as required and select the line at the **top** of the BACK pocket as shown below:



17. After selecting the line your graphics screen should look like the screenshot below, with the **green arrow pointing left** in a clockwise direction. If the arrow is not pointing to the left select the arrow from the Chaining dialog box to reverse the direction.

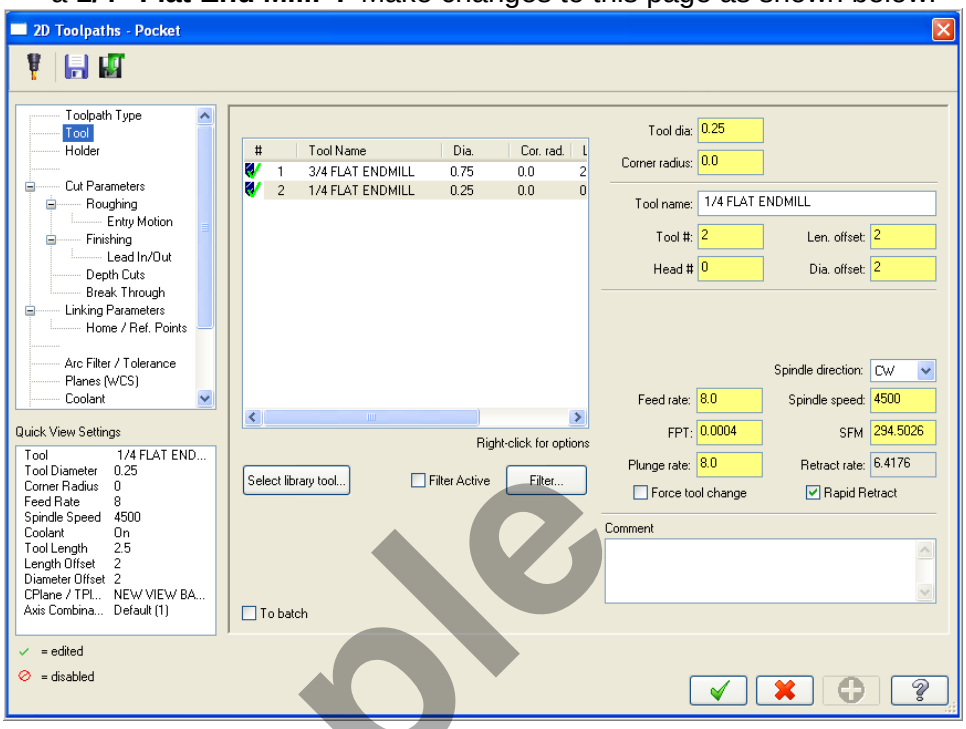


18. After the chain has been successfully chained select the OK button at the bottom of the Chaining dialog box.
19. Ensure the **Toolpath Type** is set to **Pocket.**

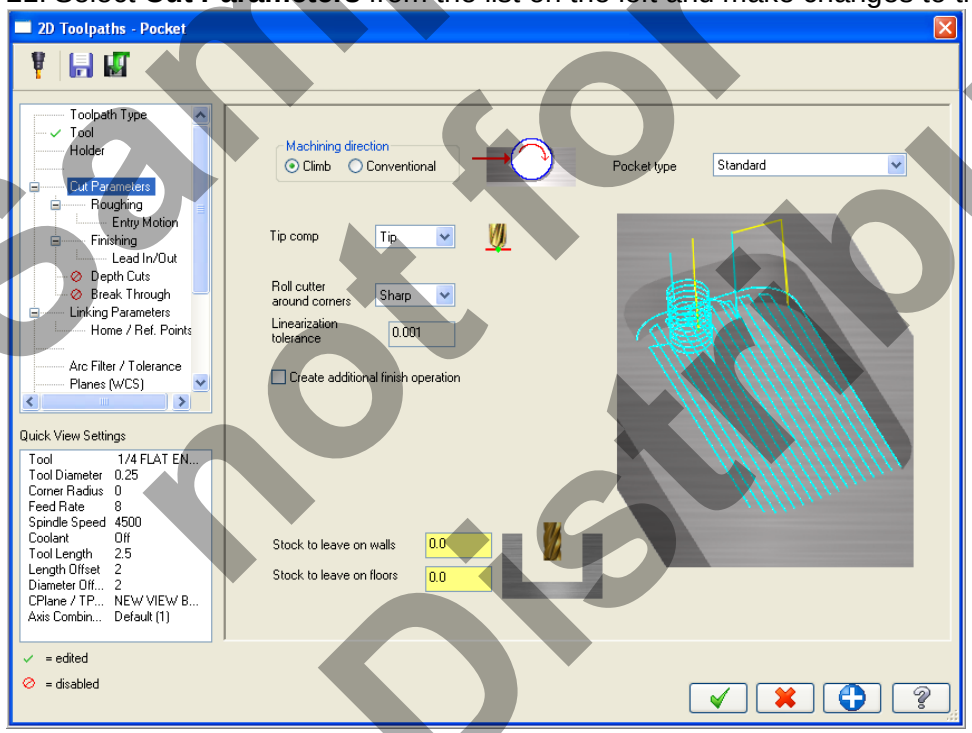


Sample not for Distribution

20. On the left of the dialog box select **Tool**. Click on the **Select library tool** button and choose a **1/4" Flat End Mill. .** Make changes to this page as shown below:

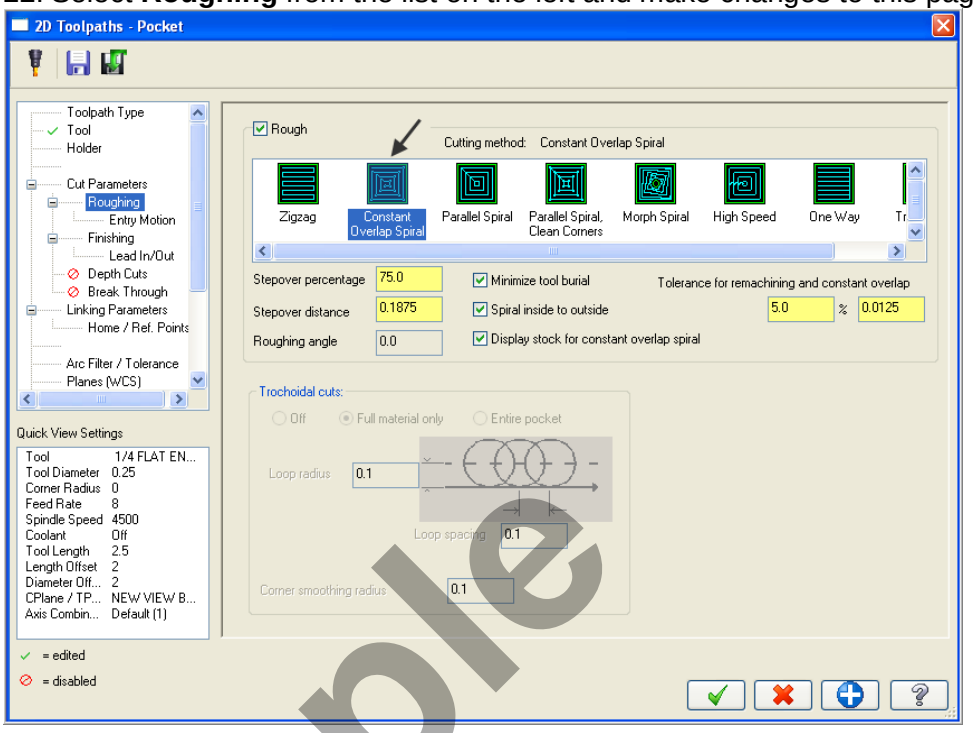


21. Select **Cut Parameters** from the list on the left and make changes to this page.

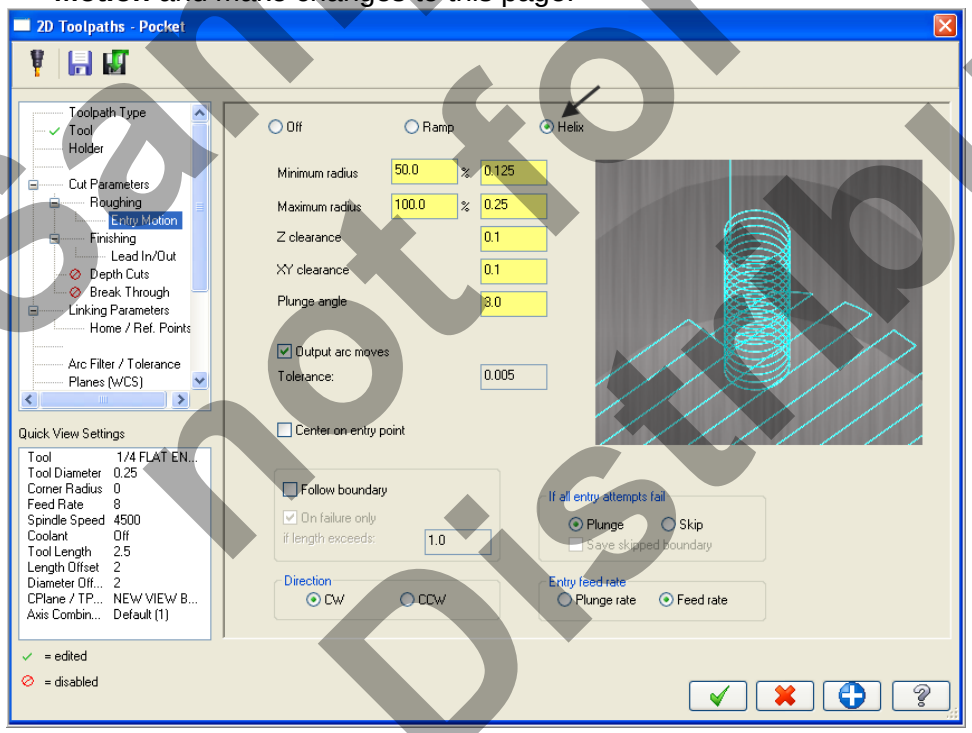


Sample not for Distribution

22. Select **Roughing** from the list on the left and make changes to this page.

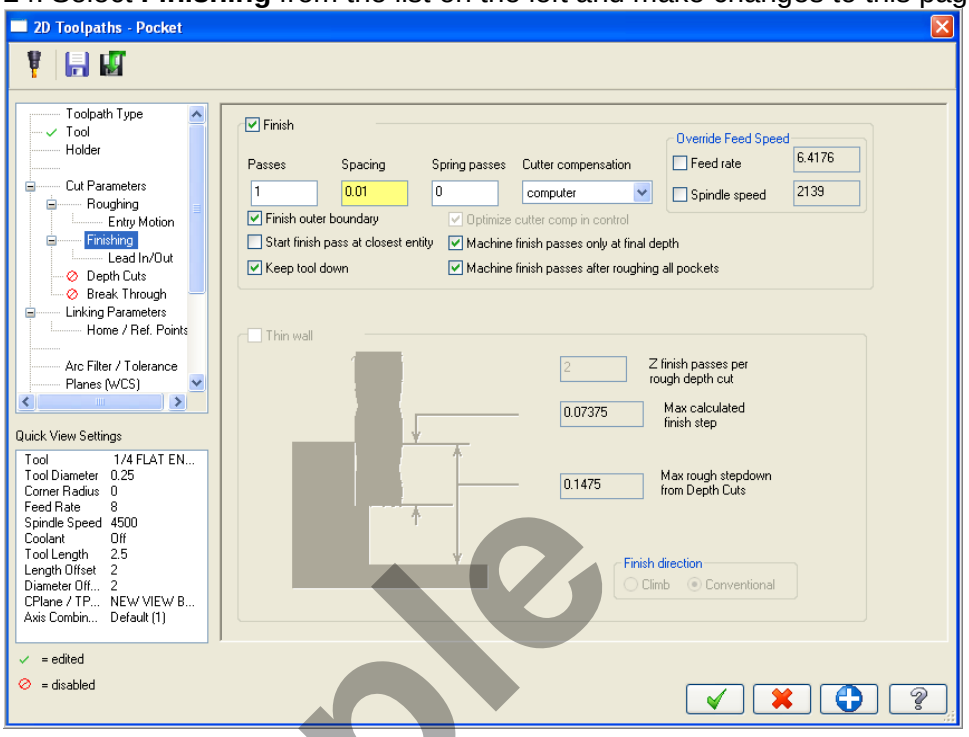


23. Select the plus sign to the left of **Roughing** to expand the list (if required) and click on **Entry Motion** and make changes to this page.

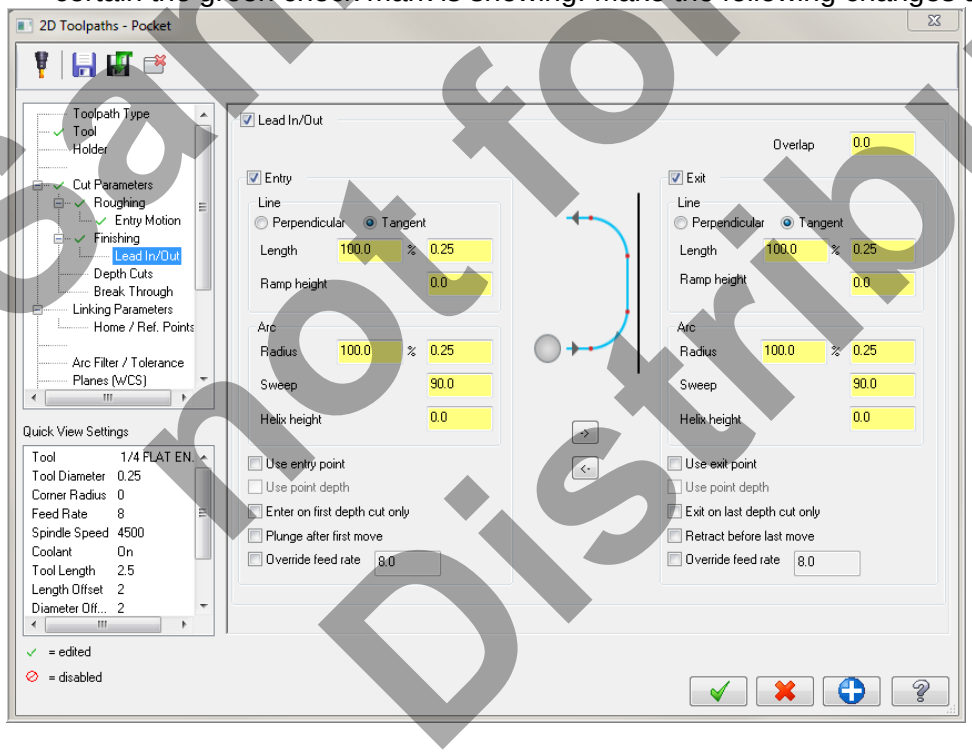


Sample not for Distribution

24. Select **Finishing** from the list on the left and make changes to this page:

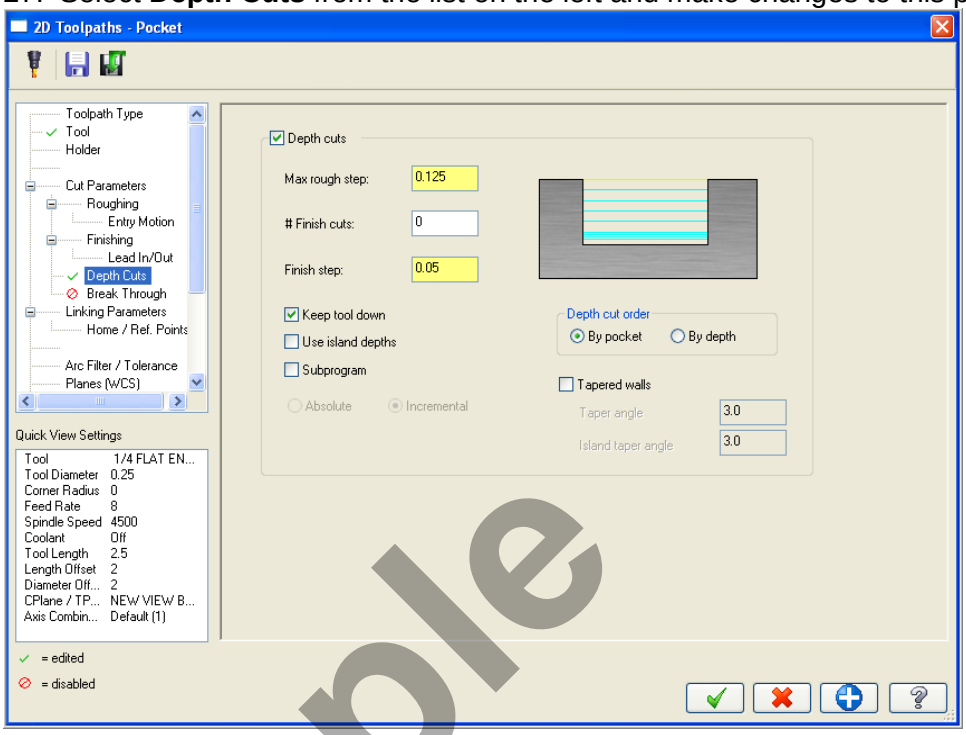


25. Select the plus sign to the left of **Finishing** expanding the list and click on **Lead In/Out.**
26. Ensure the **Lead In/Out** is activated by clicking on the small Lead In/Out box and make certain the green check mark is showing. Make the following changes as shown below:

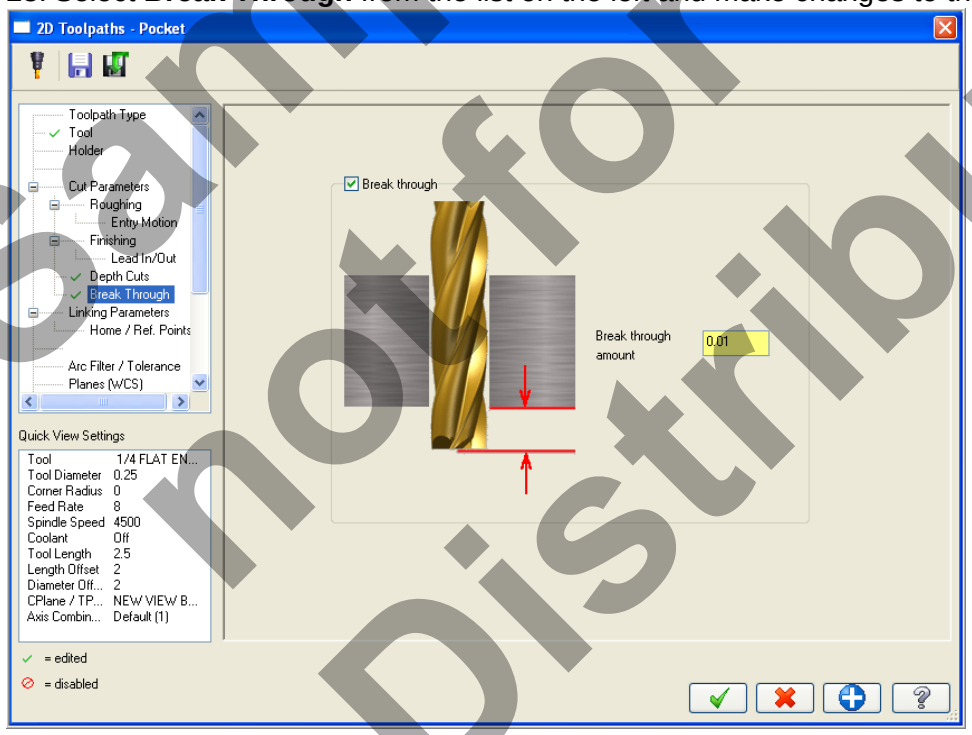


Sample not for Distribution

27. Select **Depth Cuts** from the list on the left and make changes to this page:

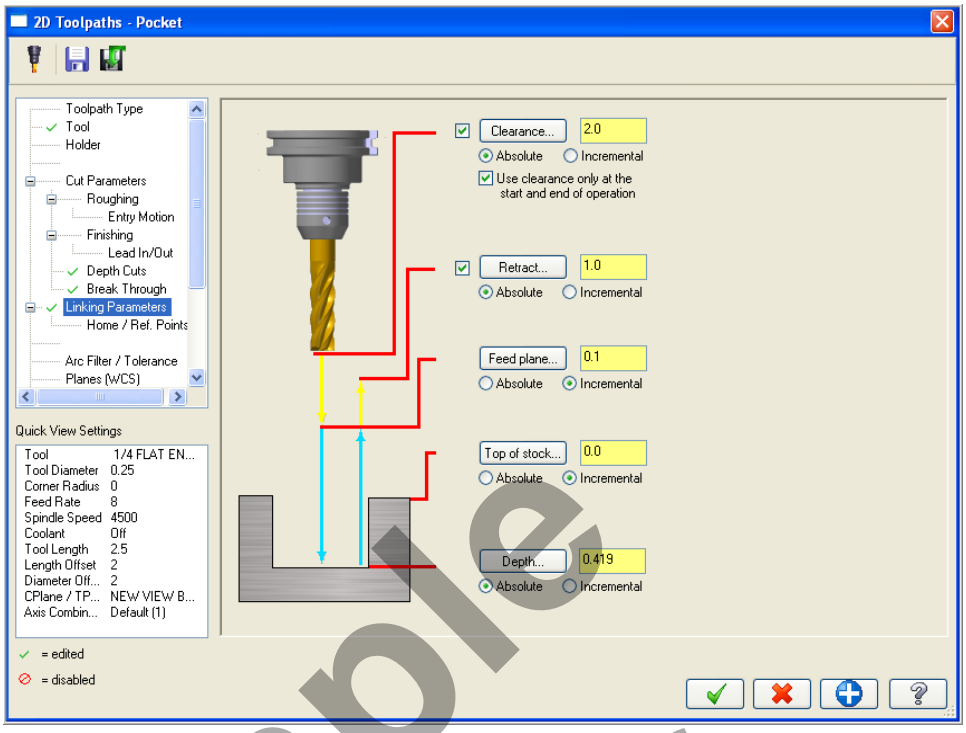


28. Select **Break Through** from the list on the left and make changes to this page:

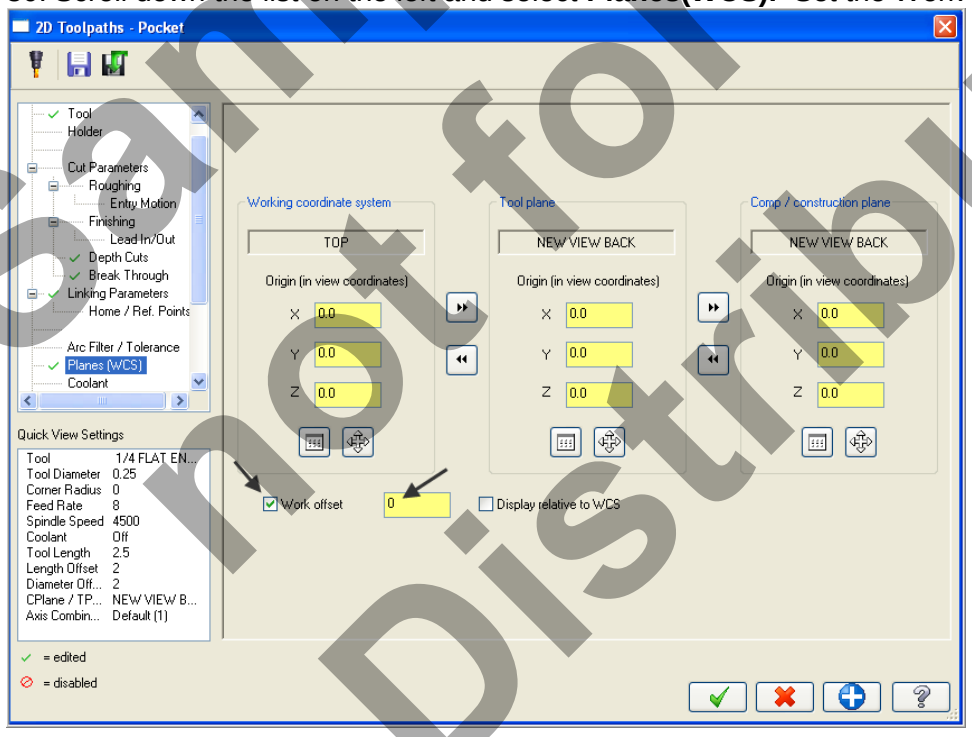


Sample not for Distribution

29. Select **Linking Parameters** from the list on the left and make changes to this page as shown below.



30. Scroll down the list on the left and select **Planes(WCS).** Set the Work offset to **Zero**.

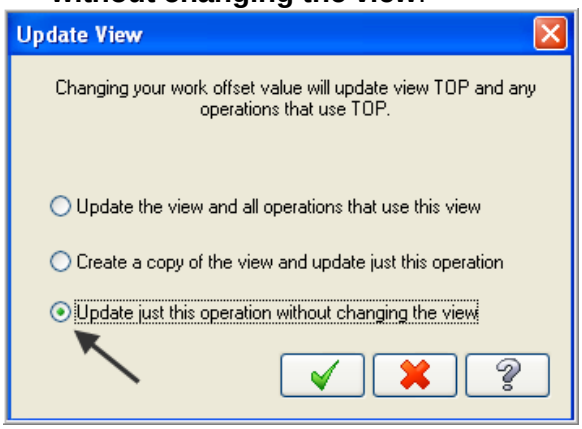


Work Offset # Assigns a work offset code to the view. When you use the view for a Tplane and create a toolpath (or wirepath),

Mastercam will automatically output the work offset when you post the operation. Number the offsets sequentially from zero, or enter –1 to not assign any offset to the view.

Sample not for Distribution

31. When confronted with the **Update View** dialog box activate **Update just this operation without changing the view**.



32. Click on the OK button when done.
33. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.
34. Select the OK button to exit Pocket operation.
35. **Save** your Mastercam file.
36. Backplot this toolpath only.

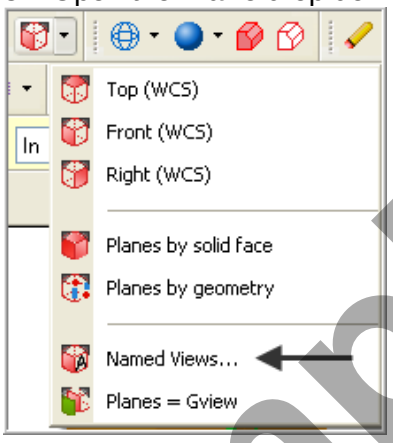
Sample not for Distribution

TASK 10:
MACHINE OPEN POCKET ON BOTTOM FACE

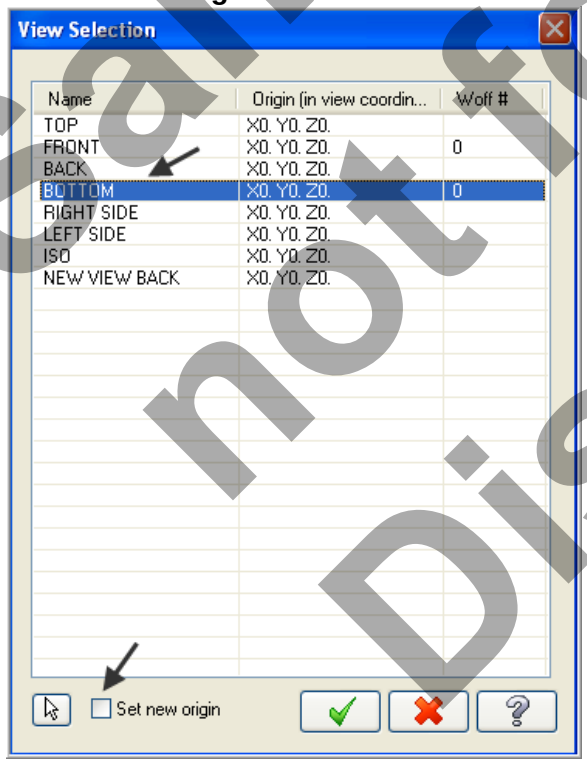
➲ In this task you will use the 0.25” diameter flat end mill to rough and finish the open pocket on the bottom face using Toolpath> Pocket.

1. If required select **Alt-T** on your keyboard to **turn off** the display of toolpaths.
2. Activate a shaded view by selecting the icon at the top of the screen.

3. Open the **Plane** drop down menu on the Top toolbar and select **Named Views...**



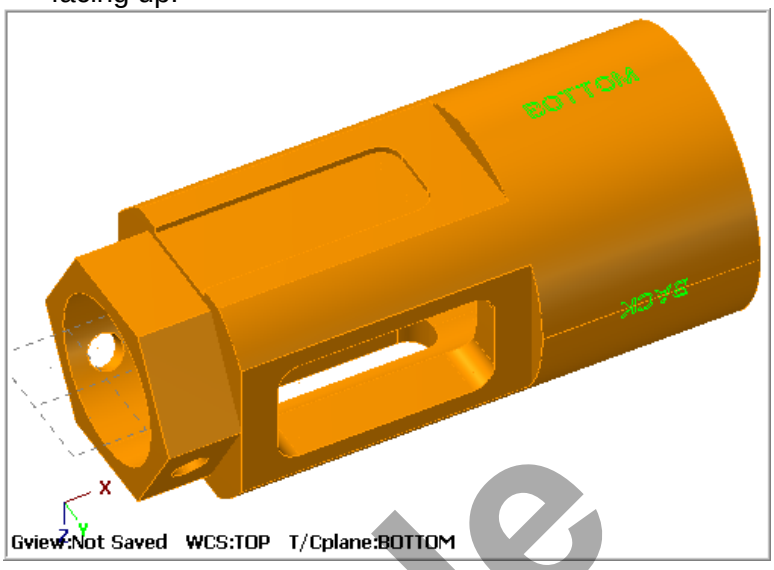
4. You will be confronted with the **View Selection** dialog box, select **BOTTOM** and de-activate **Set new origin** as shown below:



5. Select the OK button to exit the **View Selection** dialog box.

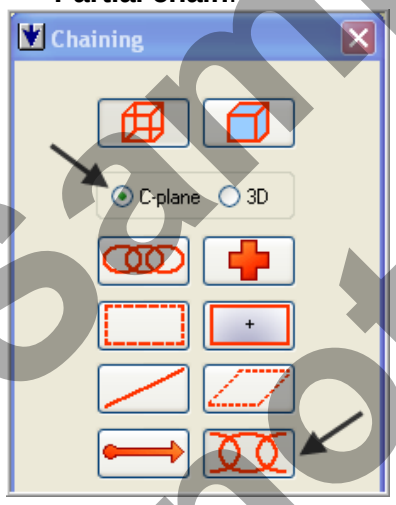
Sample not for Distribution

6. Dynamically rotate the display similar to the display shown below. The text **BOTTOM** is facing up.



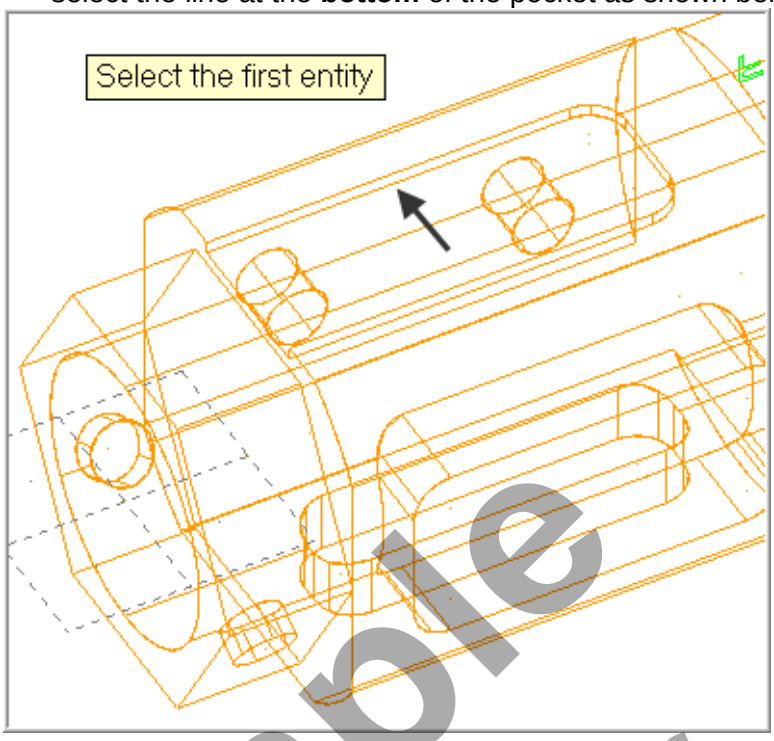
Note
At the bottom of the graphics screen the status of the Tool and Construction plane is set to **BOTTOM.**

7. Activate a wireframe view by selecting the icon at the top of the screen. .
8. From the menu bar select **Toolpaths>Pocket…**
9. On the screen you will now see the **Chaining dialog box.** Activate **C-Plane** and select **Partial chain**.

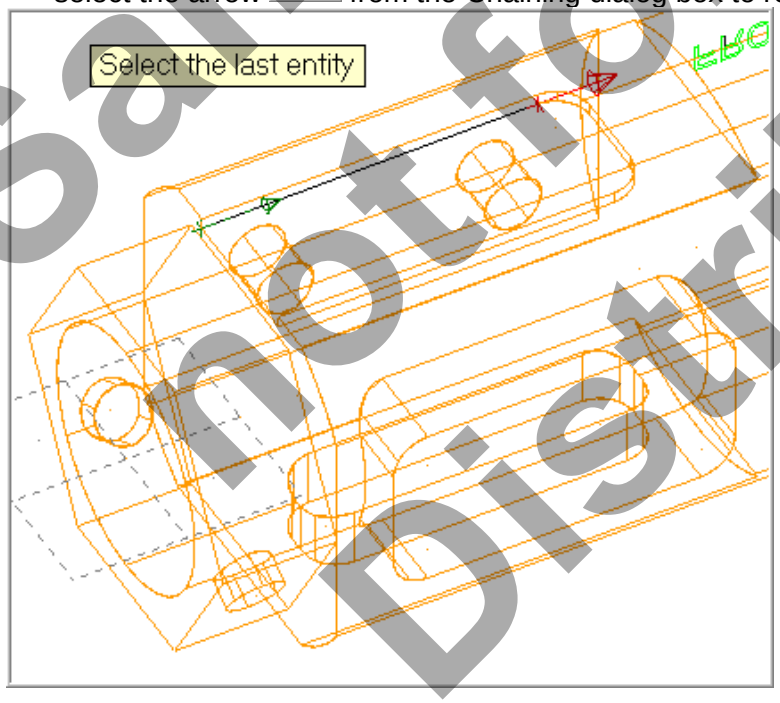


Sample not for Distribution

10. On the graphics screen is now a prompt to **Select the first entity**. Zoom in as required and select the line at the **bottom** of the pocket as shown below:

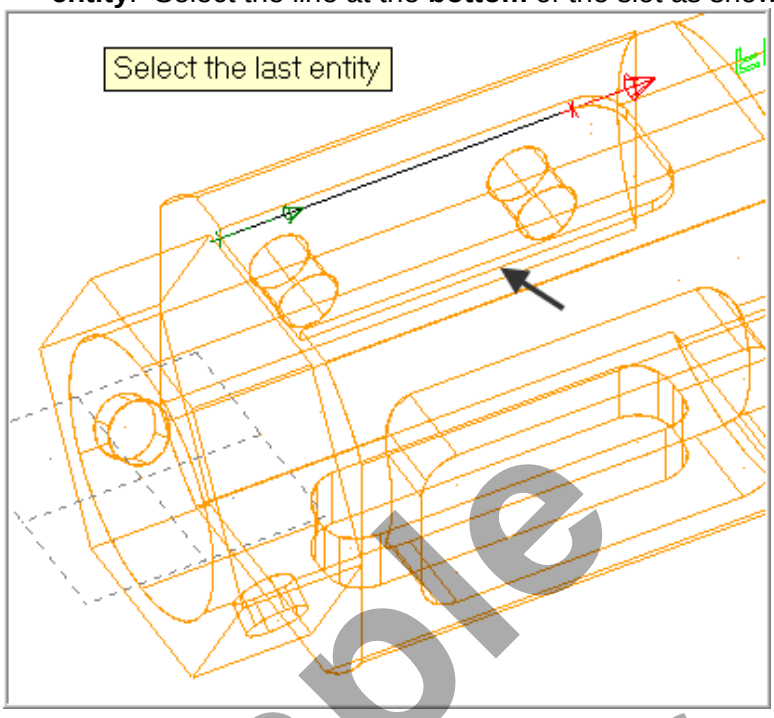


11. After selecting the line your graphics screen should look like the screenshot below, with the **green arrow pointing right** in a clockwise direction. If the arrow is not pointing to the right select the arrow from the Chaining dialog box to reverse the direction.

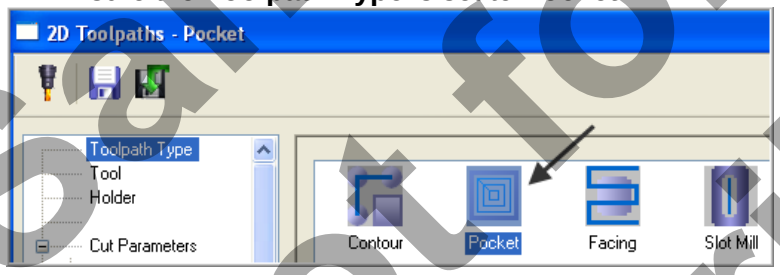


Sample not for Distribution

12. After selecting the first line in the Partial chain you are then prompted to **Select the last entity**. Select the line at the **bottom** of the slot as shown below:

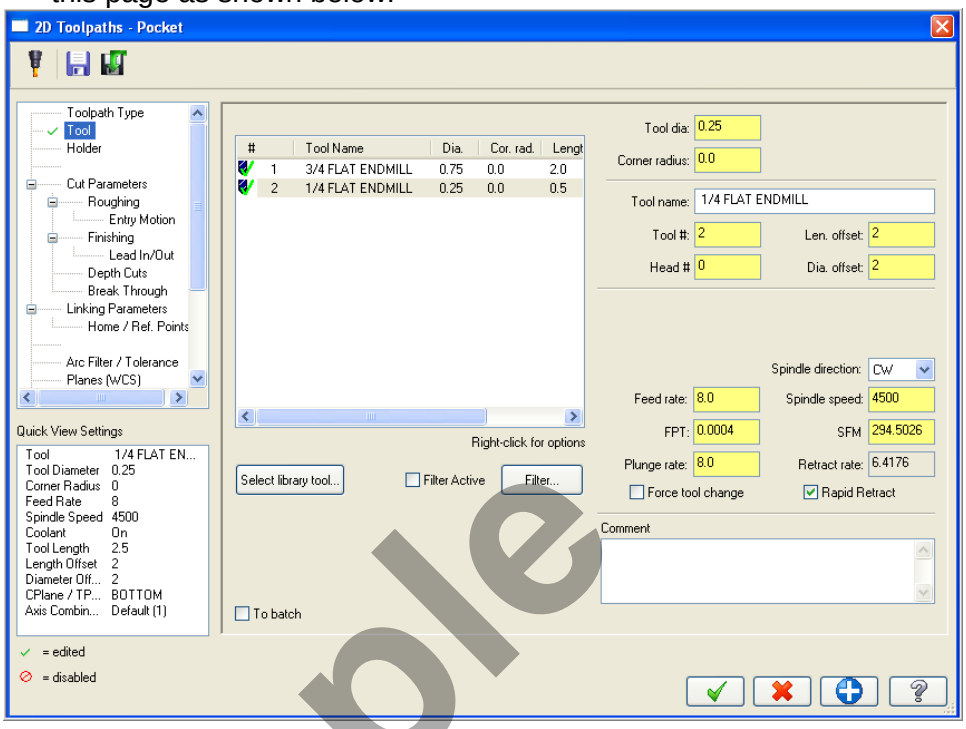


13. After the contour has been successfully chained select the OK button at the bottom of the Chaining dialog box.
14. Ensure the **Toolpath Type** is set to **Pocket.**

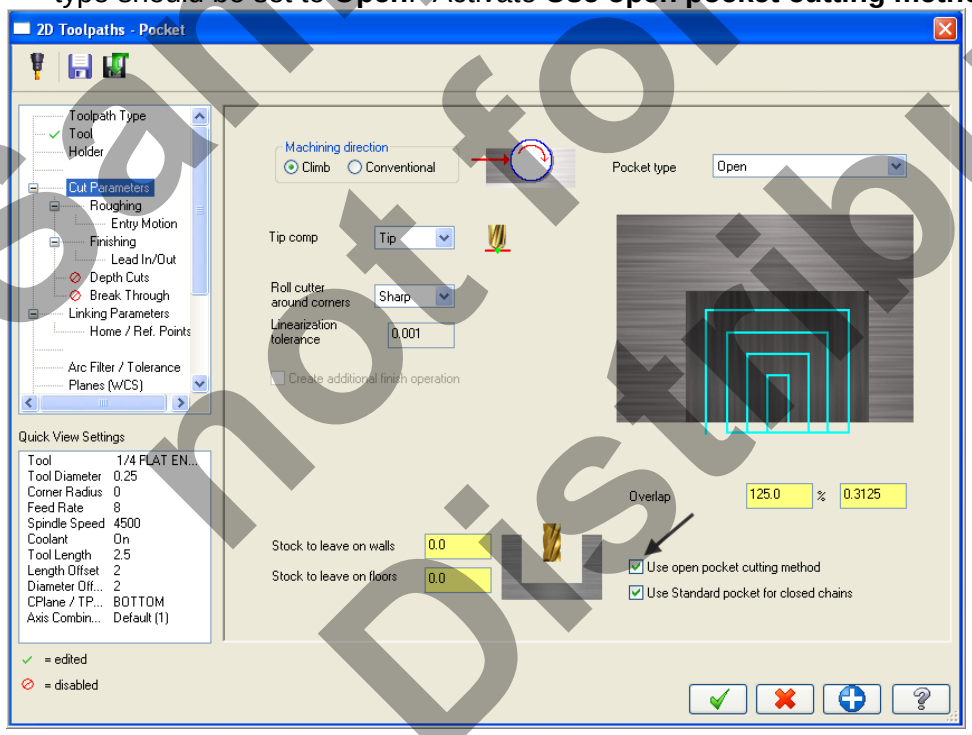


Sample not for Distribution

15. On the left of the dialog box select **Tool**. Select the **1/4" Flat End Mill.** Make changes to this page as shown below:

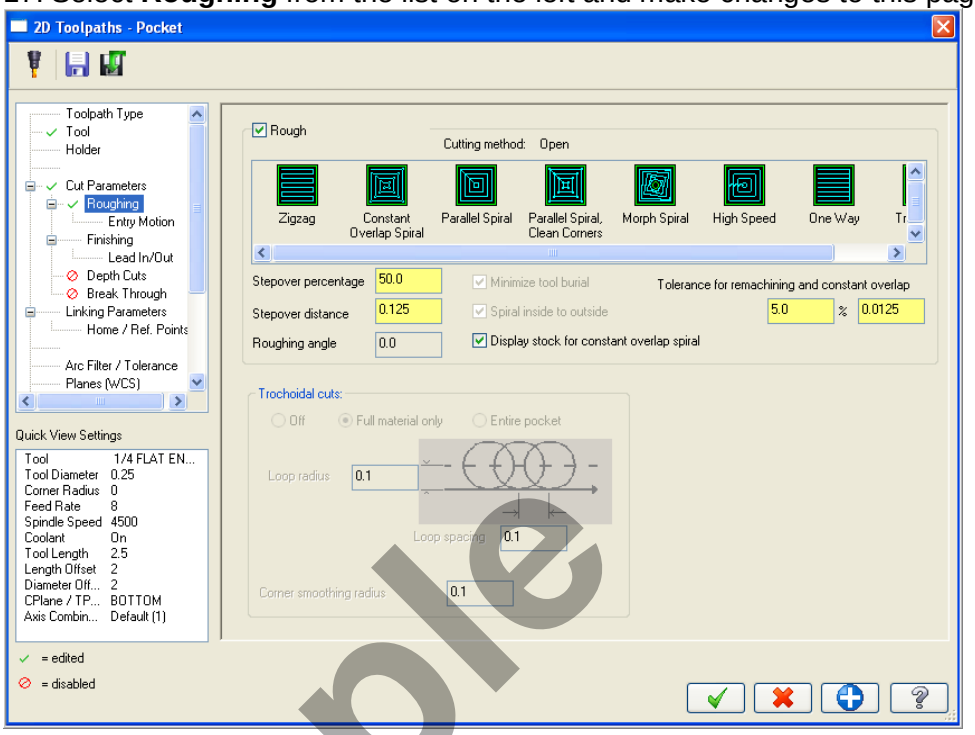


16. Select **Cut Parameters** from the list on the left and make changes to this page. The Pocket type should be set to **Open**. Activate **Use open pocket cutting method**.

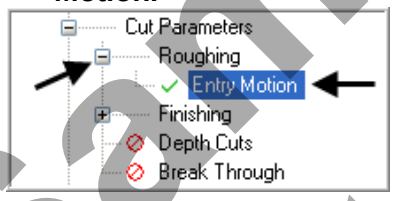


Sample not for Distribution

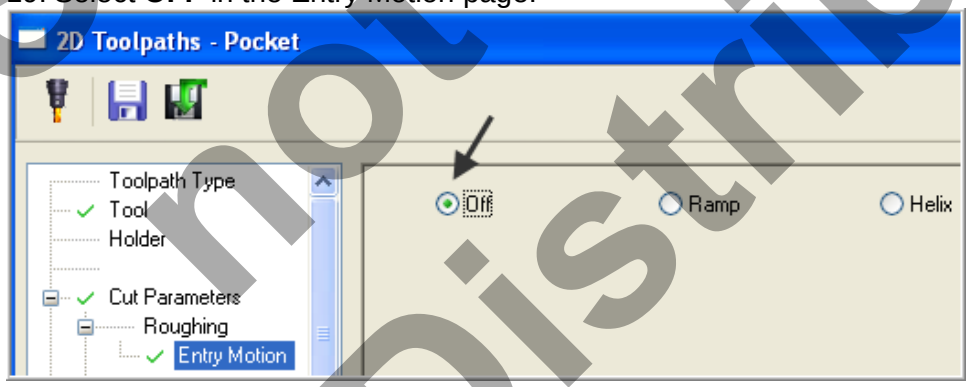
17. Select **Roughing** from the list on the left and make changes to this page.



18. Select the plus sign to the left of **Roughing** to expand the list (if required) and click on **Entry Motion.**

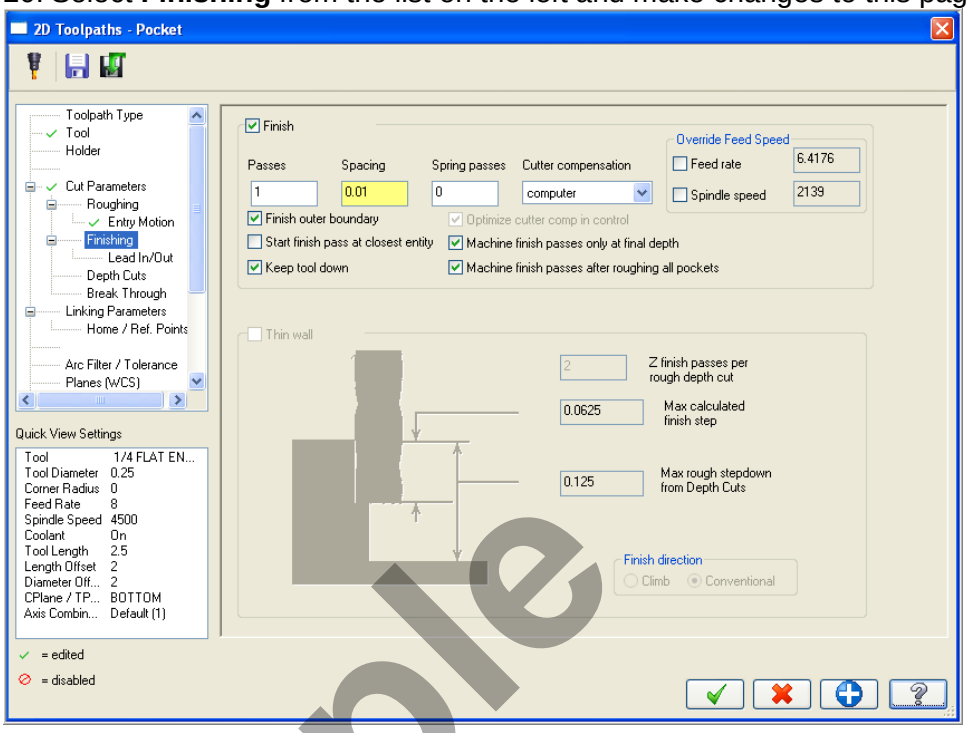


19. Select **OFF** in the Entry Motion page.

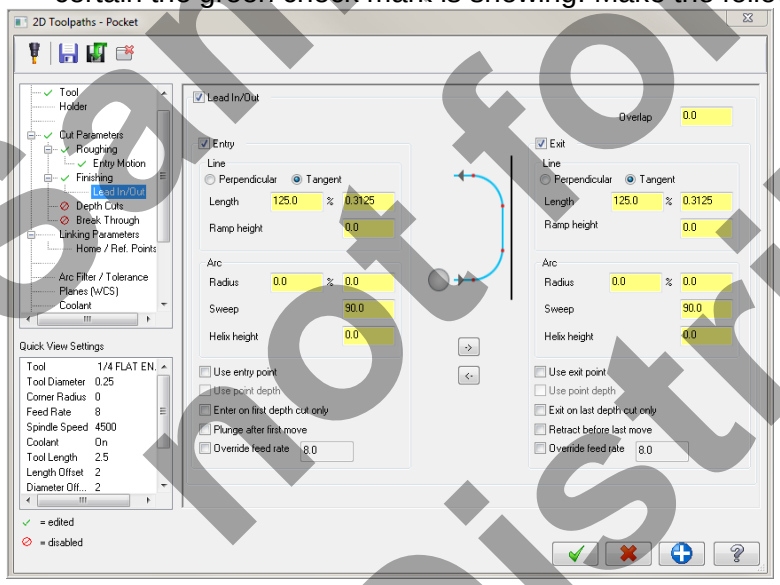


Sample not for Distribution

20. Select **Finishing** from the list on the left and make changes to this page:

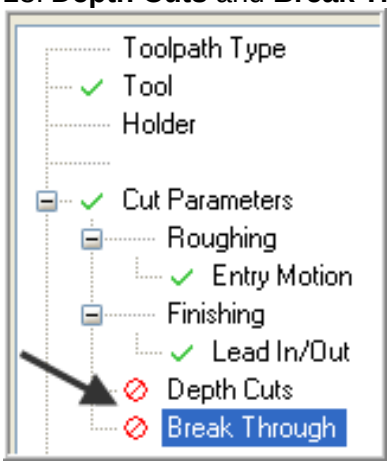


21. Select the plus sign to the left of **Finishing** expanding the list and click on **Lead In/Out.**
22. Ensure the **Lead In/Out** is activated by clicking on the small Lead In/Out box and make certain the green check mark is showing. Make the following changes as shown below:

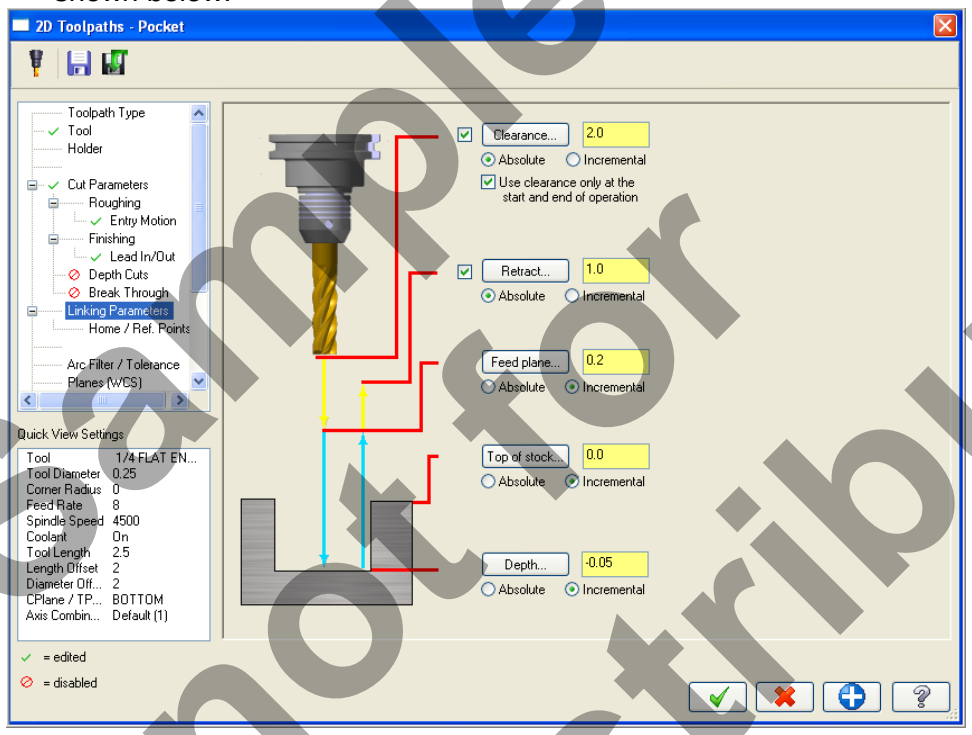


Sample not for Distribution

23. **Depth Cuts** and **Break Through** should **not** be activated.

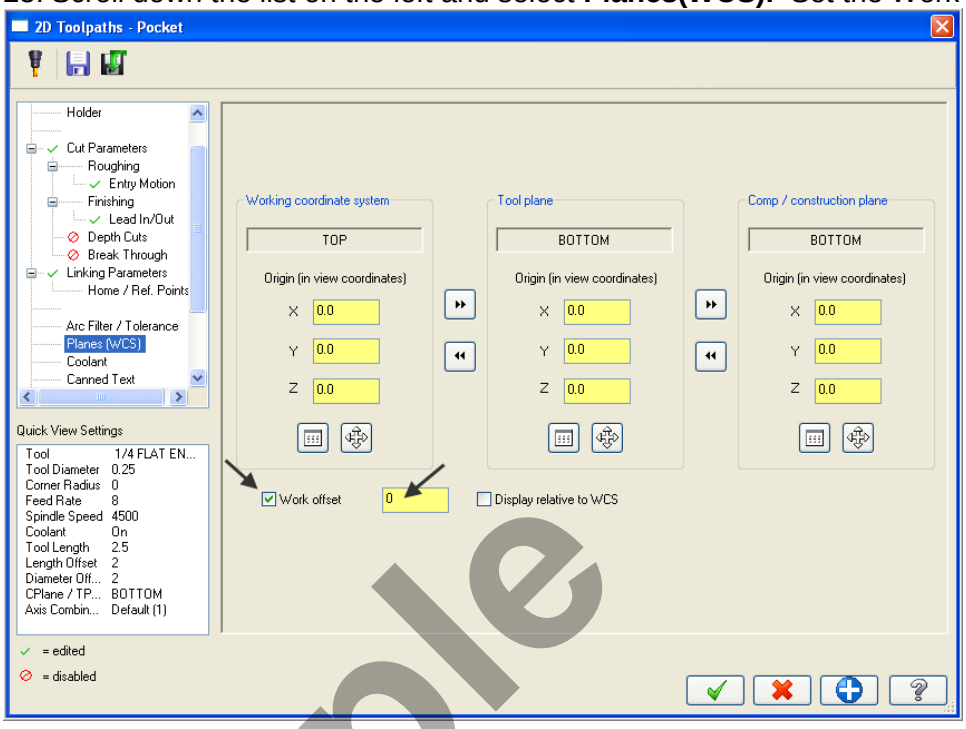


24. Select **Linking Parameters** from the list on the left and make changes to this page as shown below.



Sample not for Distribution

25. Scroll down the list on the left and select **Planes(WCS).** Set the Work offset to **Zero**.



Work Offset #
Assigns a work offset code to the view. When you use the view for a Tplane and create a toolpath (or wirepath),

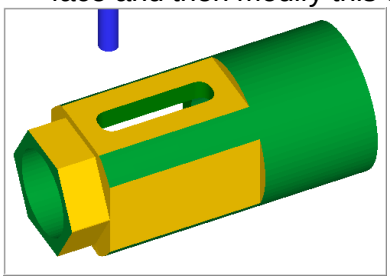
Mastercam will automatically output the work offset when you post the operation. Number the offsets sequentially from zero, or enter –1 to not assign any offset to the view.

26. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and ensure it is set to **On**.
27. Select the OK button to exit Pocket operation.
28. **Save** your Mastercam file.
29. Backplot this toolpath

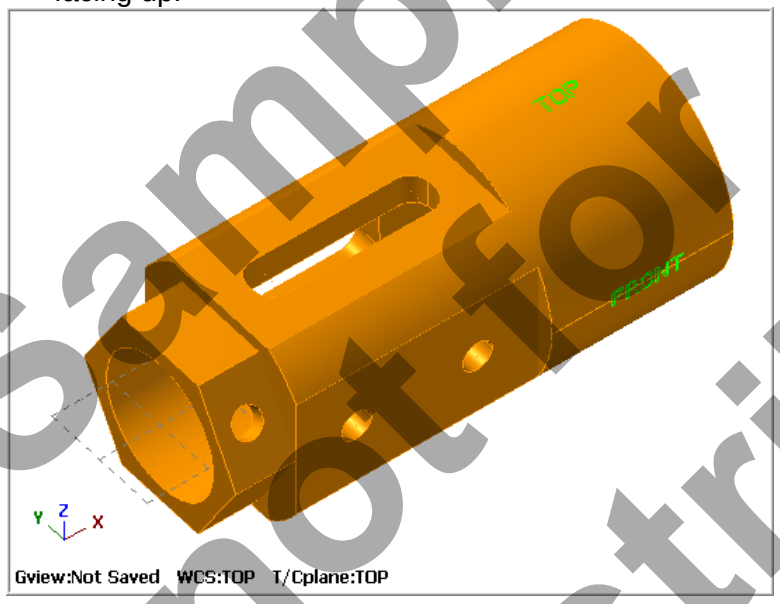
Sample not for Distribution

TASK 11:
MACHINE POCKET ON TOP FACE

- In this task you will use a 0.25” diameter flat end mill to rough and finish the pocket on the **TOP** face
- For this example you will copy the fifth operation used to machine the pocket on the BACK face and then modify this copied operation to machine the pocket on the TOP face.



1. Set the graphics view to **Isometric View**.
2. Activate a shaded view by selecting the icon at the top of the screen. .
3. Now dynamically rotate the display similar to the display shown below. The text **TOP** is facing up.

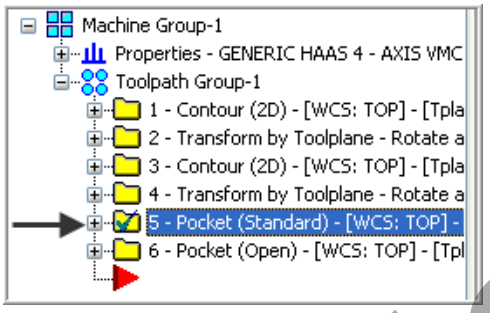


Note
At the bottom of the graphics screen the status of the **Tool and Construction** plane is set to **TOP.**

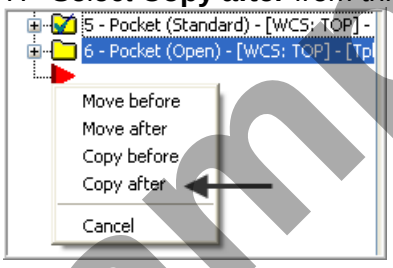
Sample not for Distribution

4. Before moving on, let's organize the operations in the Toolpaths Manager. Click on the minus sign for each operation to collapse the tree structure of these operations.

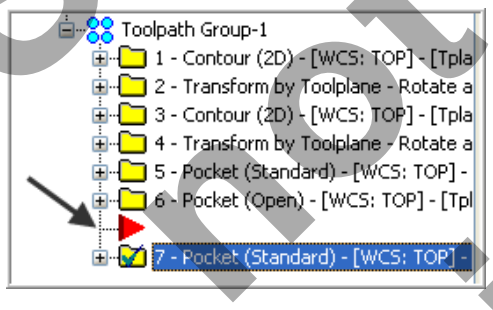
5. In the Toolpaths manager **select only the fifth Pocket** operation, as indicated by the blue checkmarks shown below. This fifth operation is used to machine the pocket on the **BACK** face.

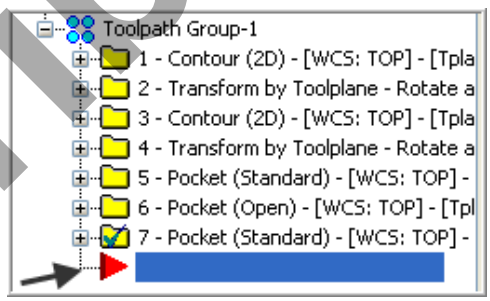


6. To copy this operation **Hold down the Right mouse button over fifth**. **Keep holding the right mouse button** down and drag below the red arrow and release.
7. Select **Copy after** from this menu.



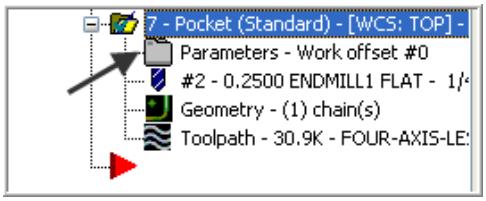
8. Your screen will look similar to the screenshot below on the left. Move the Insert arrow to the bottom of the list of operations by clicking the icon on the Toolpaths Manager toolbar:



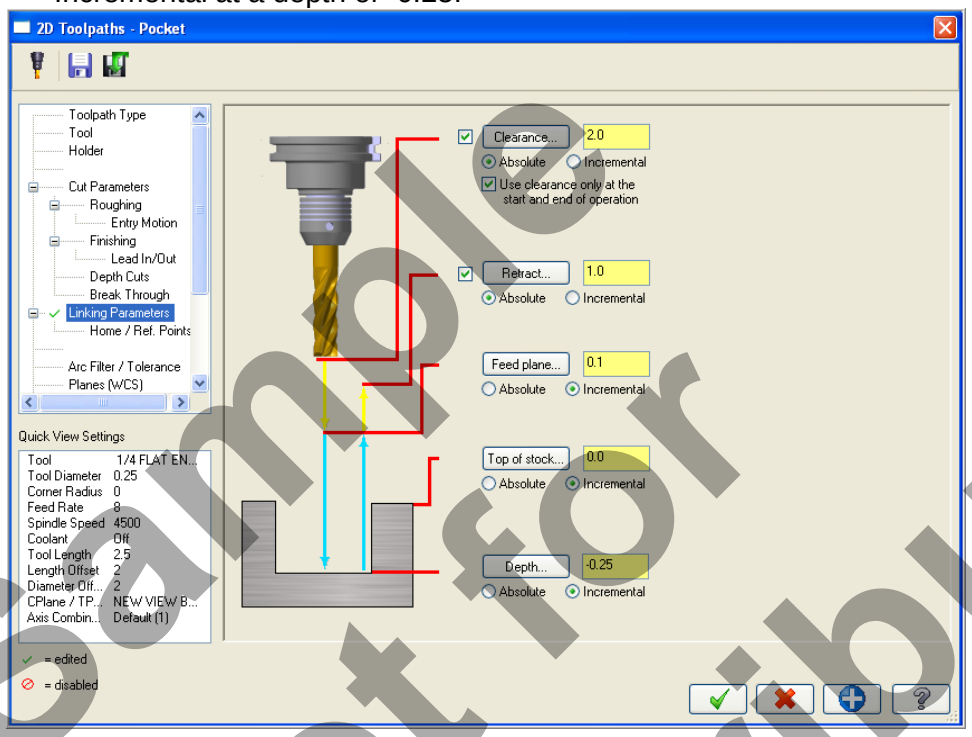


Sample Not for Distribution

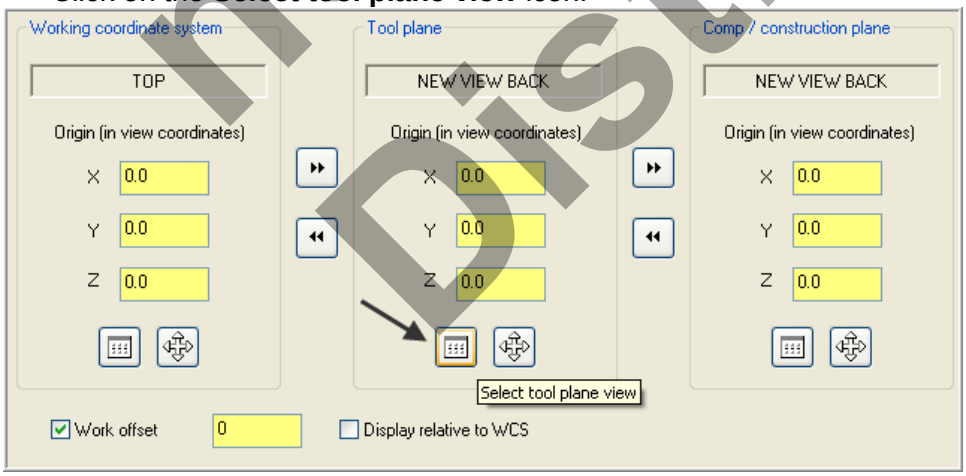
9. In the **Toolpaths Manager** click the plus sign to the left of **operation 7 Pocket** to expand the list. Now click on the folder **Parameters** for the **seventh** operation as shown below:



10. All the parameters and values you set earlier in this copied pocket operation will **still be set**. **The Only changes required are for Linking Parameters and Planes (WCS).** Select **Linking Parameters** and make changes to this page as shown below. The **Depth** is set in Incremental at a depth of -0.25.

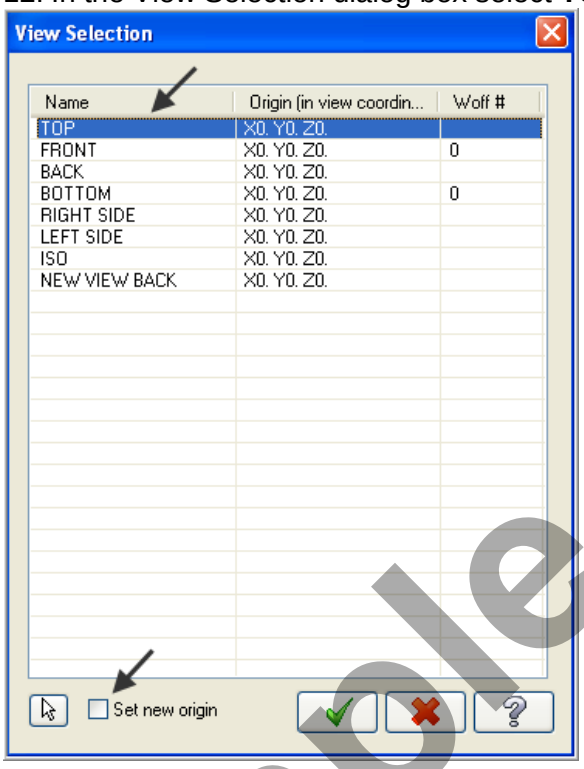


11. Scroll down the list on the left and select **Planes(WCS).** The Tool plane and Construction plane is set to NEW VIEW BACK at present. You need to change the Tool plane to **TOP**. Click on the **Select tool plane view** icon.

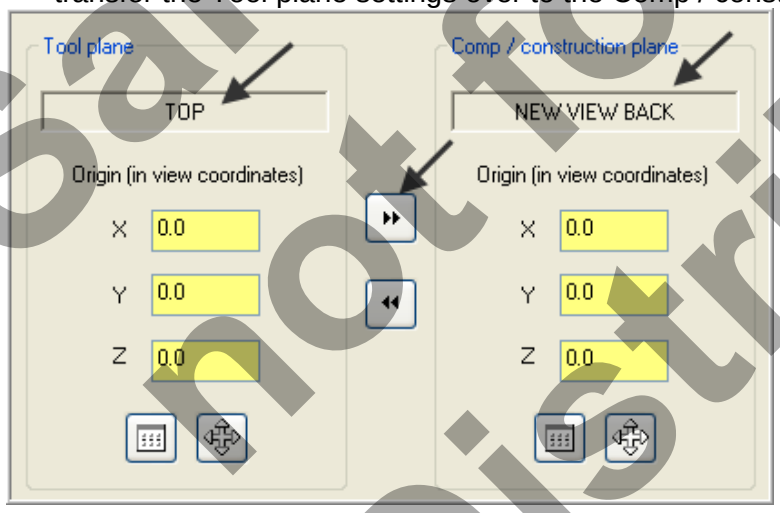


Sample not for Distribution

12. In the View Selection dialog box select **TOP** and **un-check Set new origin**.

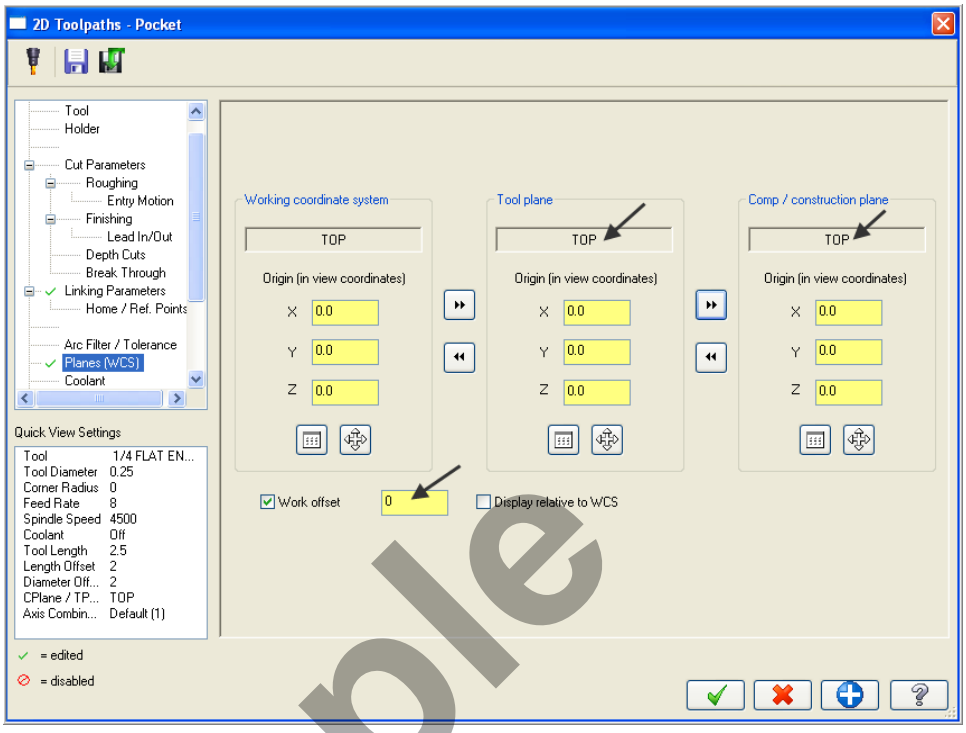


13. Click on the OK button when done.
14. When returned to the Planes page note now that the **Tool plane is set to TOP**, but the Comp / construction plane is set to NEW VIEW BACK. Click on the **right Copy button** to transfer the Tool plane settings over to the Comp / construction plane settings.

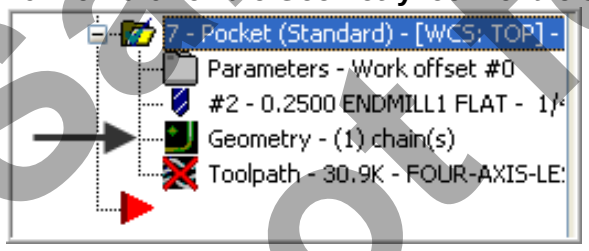


Sample not for Distribution

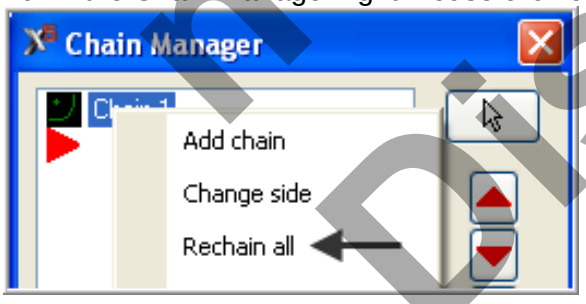
15. Now set the Work offset to **Zero**.



16. When confronted with the **Update View dialog box** activate **Update just this operation without changing the view.**
17. Click on the OK button in the **Update View dialog box** when done .
18. Select the OK button to exit Pocket parameters.
19. Now click on the **Geometry icon** for the **seventh** operation as shown below:

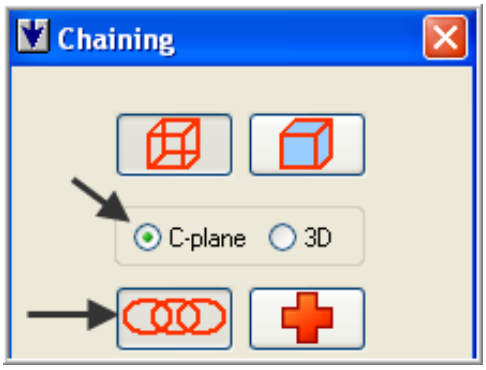


20. In the Chain Manager right mouse click on **Chain 1** and select **Rechain all**.



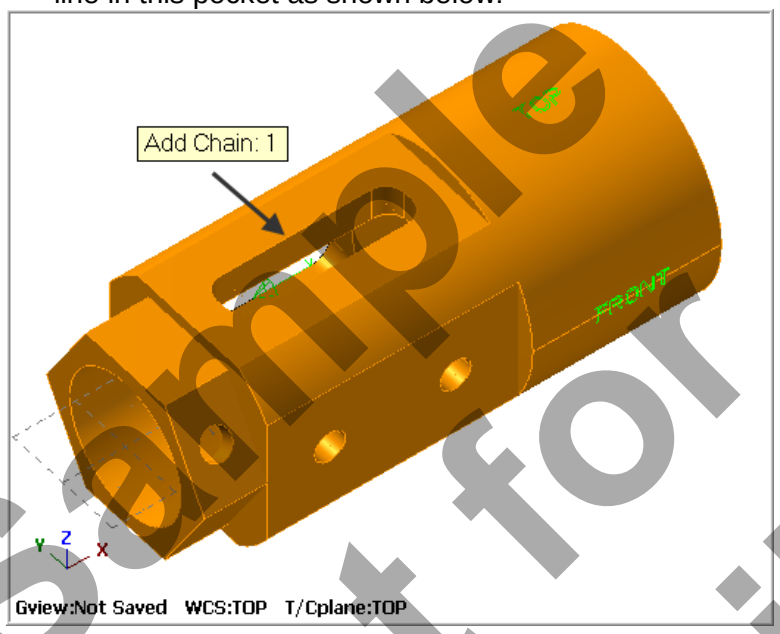
Sample not for Distribution

21. On the screen you will now see the **Chaining dialog box.** Activate **C-Plane** and ensure **Chain** is selected.



Cplane
Chains only entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain. In Window, Area, or Polygon chaining, chains only entities that are on the current construction plane and at the current Z depth.

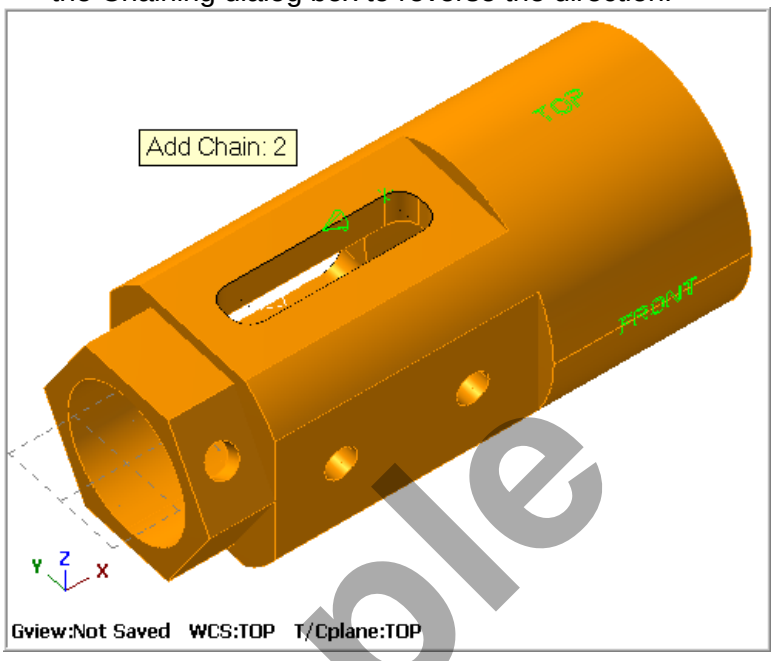
22. On the graphics screen is now a prompt to **Add chain 1**. Zoom in as required and select the line in this pocket as shown below:



Note
At the bottom of the graphics screen the status of the **Tool and Construction** plane is now set to **TOP.**

Sample not for Distribution

23. After selecting the line your graphics screen should look like the screenshot below, with the **green arrow pointing left**. If the arrow is not pointing to the left select the arrow from the Chaining dialog box to reverse the direction.

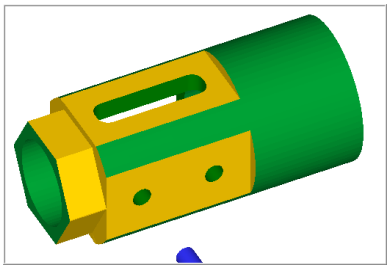


24. After the contour has been successfully chained select the OK button at the bottom of the Chaining dialog box.
25. Select the OK button at the bottom of the Chain Manager box.
26. Select the **Regenerate all dirty operations** button to remove the red X from the contour operation you have just edited.
27. Click on the minus sign to the left of this last operation to collapse it.
28. Save your Mastercam file.
29. Backplot this toolpath.

Sample not for Distribution

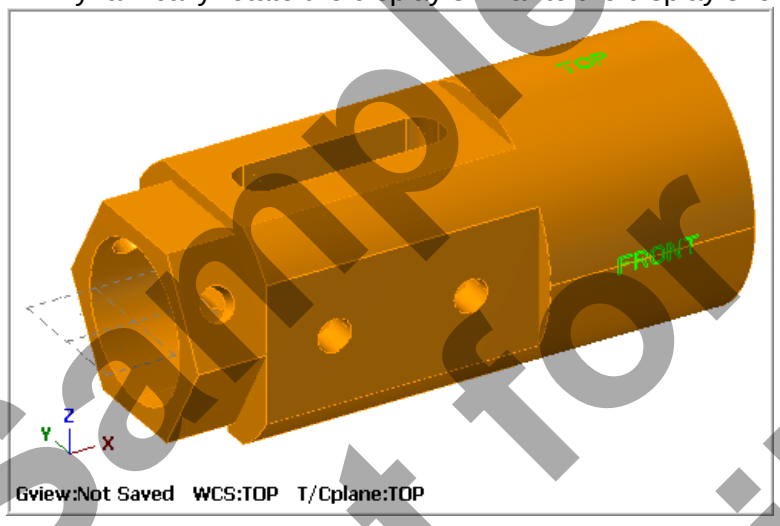
TASK 12:
DRILL HOLES ON FRONT FACE

➲ **In this task you will drill the two holes on the front face with a 0.25" centre cutting Flat End Mill.**



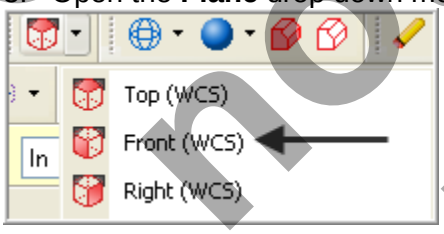
1. Activate a shaded view by selecting the icon at the top of the screen.

2. Dynamically rotate the display similar to the display shown below. The text **TOP** is facing up.



Note
At the bottom of the graphics screen the status of the Tool and Construction plane is set to **TOP.**

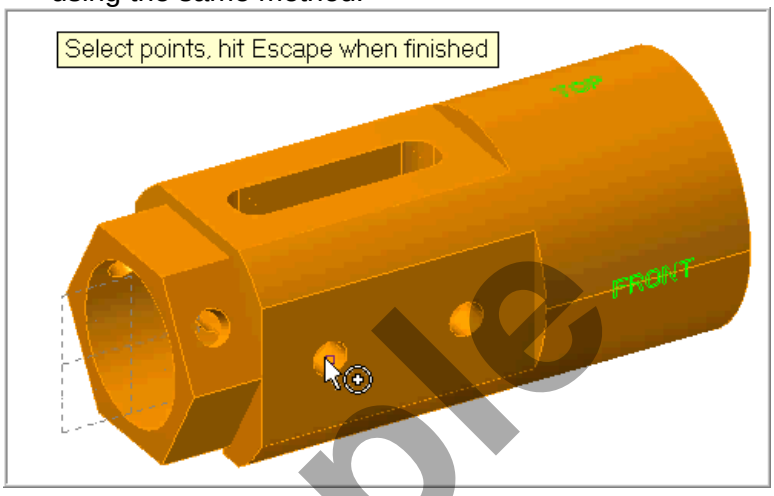
3. Open the **Plane** drop down menu on the top toolbar and select **FRONT (WCS).**



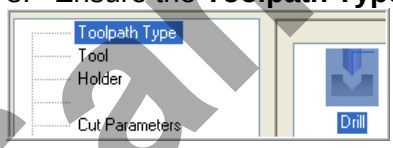
4. At the bottom of the graphics screen you will now see **T/Cplane: FRONT**

Sample not for Distribution

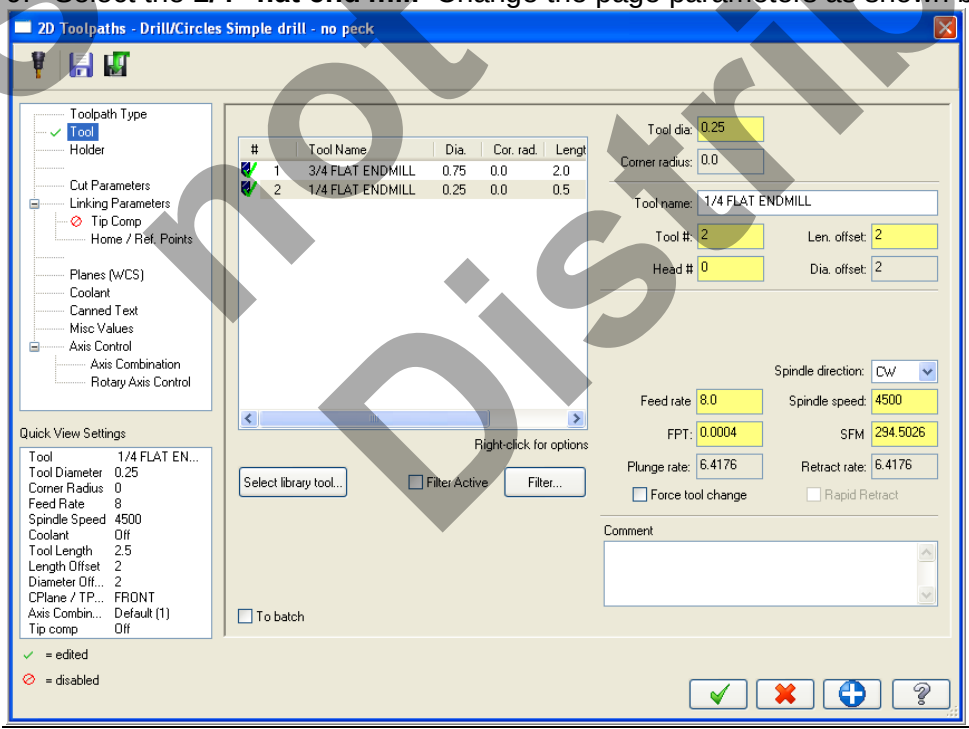
5. From the menu bar select **Toolpaths>Drill…**
6. Now you are prompted to Select points and the Drill Point Selection dialog box appears. All you need to do now is snap to the centre of the two circles. Move the mouse and position the cursor over the **center of the left circle shown below,** you will notice a circle appears as you move closer to the center of the circle, this is the visual cue for Arc center. **Ensure the visual cue for Arc center point appears** before you select the point. Use the left button of your mouse to pick this center point. Now select the centre of the **second** hole using the same method.



7. Select the OK button in the **Drill Point Selection** dialog window to complete the selection of points.
8. Ensure the **Toolpath Type** is set to **Drill.**

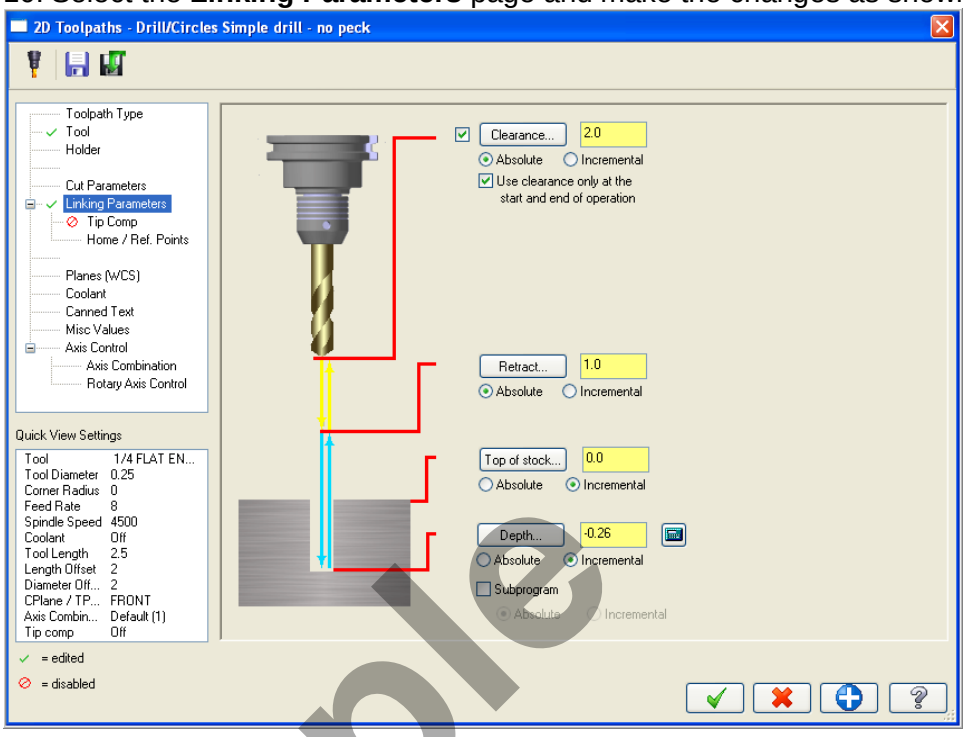


9. Select the **1/4" flat end mill** Change the page parameters as shown below:

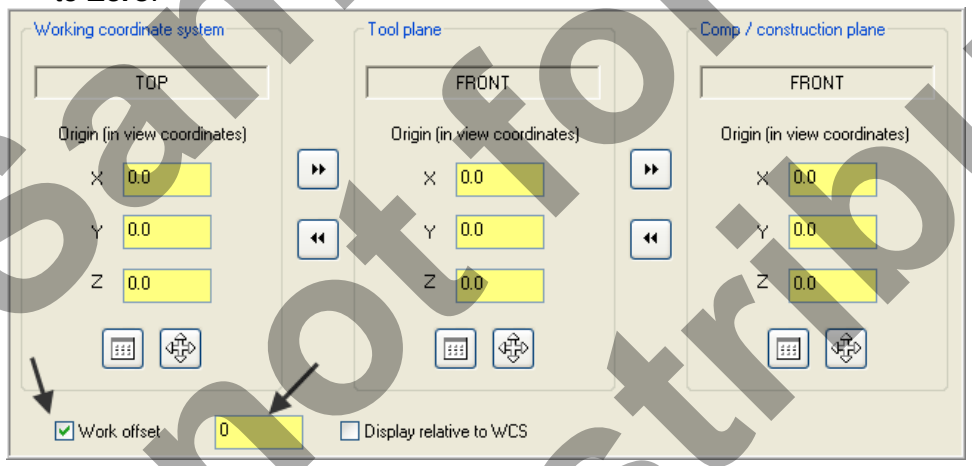


Sample not for Distribution

10. Select the **Linking Parameters** page and make the changes as shown below.



11. Tip Comp **should not** be activated.
12. From the list on the left select **Planes(WCS).** Ensure the values are as below. Work offset to **Zero**.

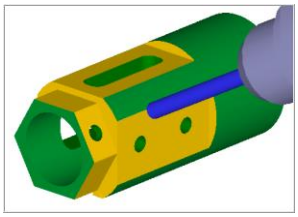


13. Select the **Coolant** page, and set **Flood** to **On**:
14. Select the **OK** button to complete this drilling function.
15. Click on the minus sign to the left of this last operation to collapse it.
16. Save your Mastercam file.
17. Backplot this toolpath.

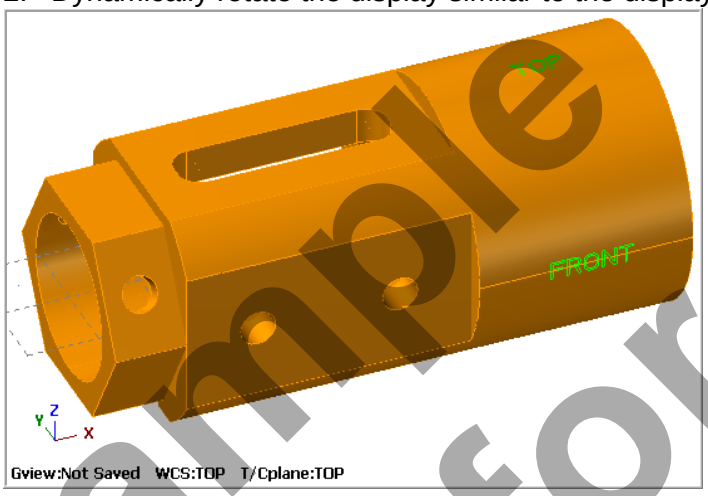
Sample not for Distribution

TASK 13:
DRILL HOLE ON HEXAGON AT FRONT

➲ In this task you will drill the hole on the hexagon at the front with a 0.25” centre cutting Flat End Mill.

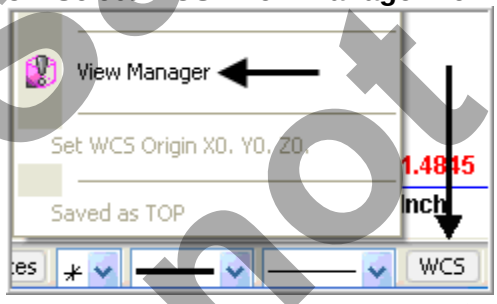


1. Select an Isometric view and then Screen Fit.
2. Dynamically rotate the display similar to the display shown below. The text **TOP** is facing up.

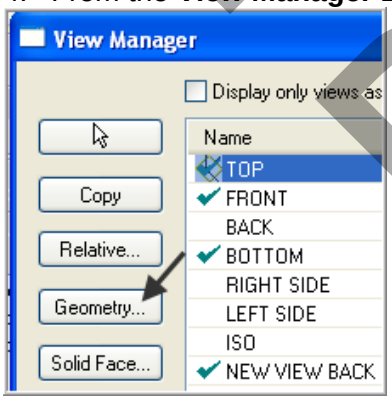


Note
At the bottom of the graphics screen the status of the Tool and Construction plane is set to **TOP.**

3. Select **WCS>View Manager** from the Status bar.



4. From the **View Manager Dialog** box select the **Geometry** button as shown below.

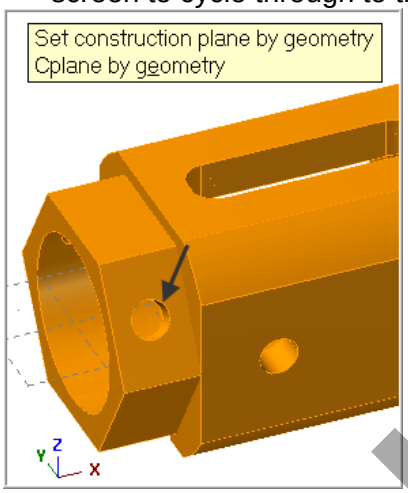


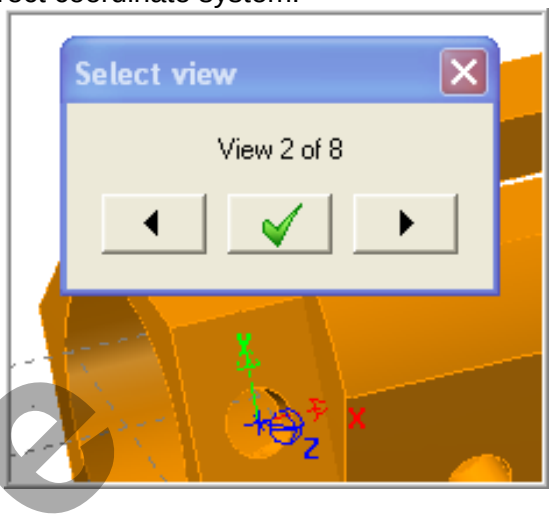
Geometry
Click the Geometry button to create a new view based on geometry in your part.

Mastercam will prompt you for the necessary geometry and other information needed to create the view.

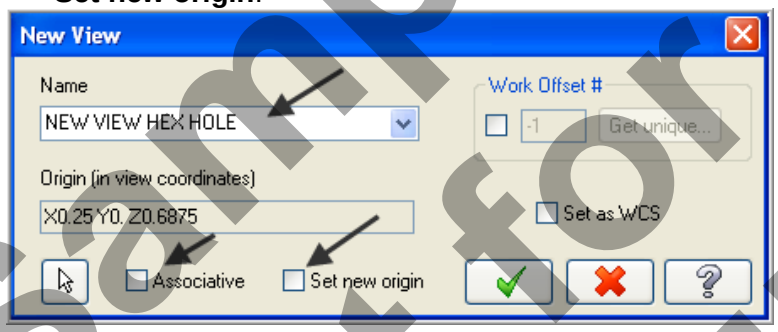
Sample not for Distribution

5. At the graphics screen you will be prompted to: **Set construction plane by geometry Cplane by geometry**. Select the **arc of the hole shown below**. Make sure the **XYZ tripod** is the same as the example below right with the **Z axis** arrow pointing up from the hole, **X axis** pointing to the right as shown below right. Use the **arrow buttons** in the Select view screen to cycle through to the correct coordinate system.

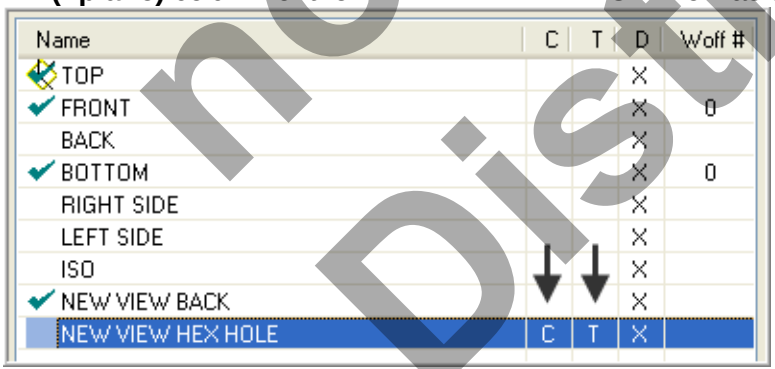




6. Select the green check mark for **OK**.
7. In the **New View** dialog box enter **NEW VIEW HEX HOLE** in the name section. Un-check **Set new origin**.

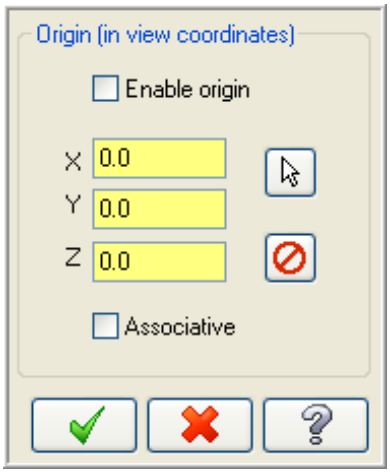


8. Click on the OK icon to complete this feature.
9. Now you are back in the View Manager dialog box pick in the **C (Cplane) column** and **T (Tplane)** column of the **NEW VIEW HEX HOLE** row as shown below.

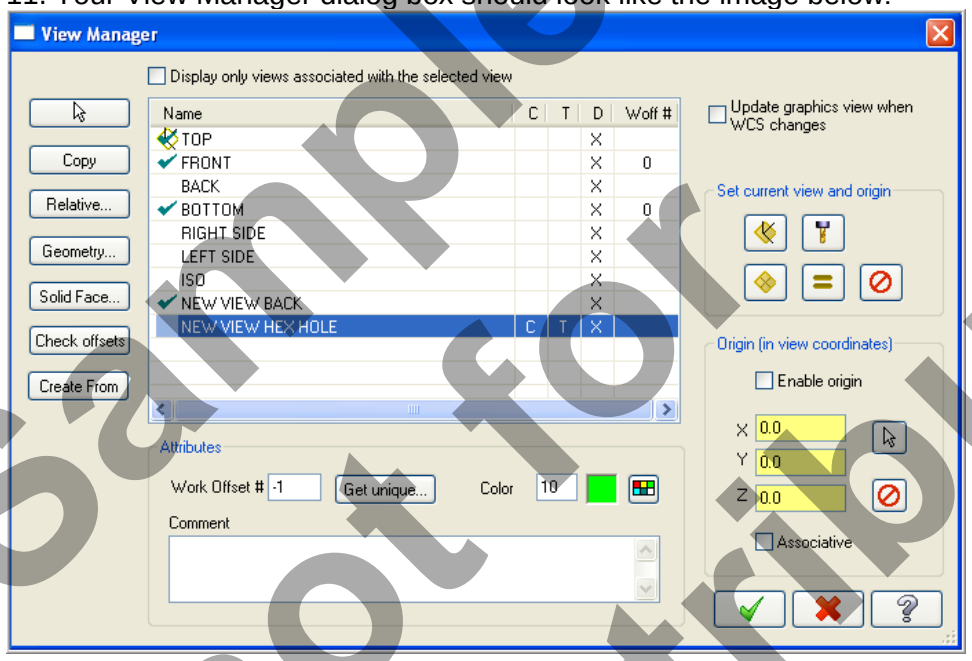


Sample not for Distribution

10. In the lower right of the view manager dialog window change the origin coordinates to **X0 Y0 Z0**.



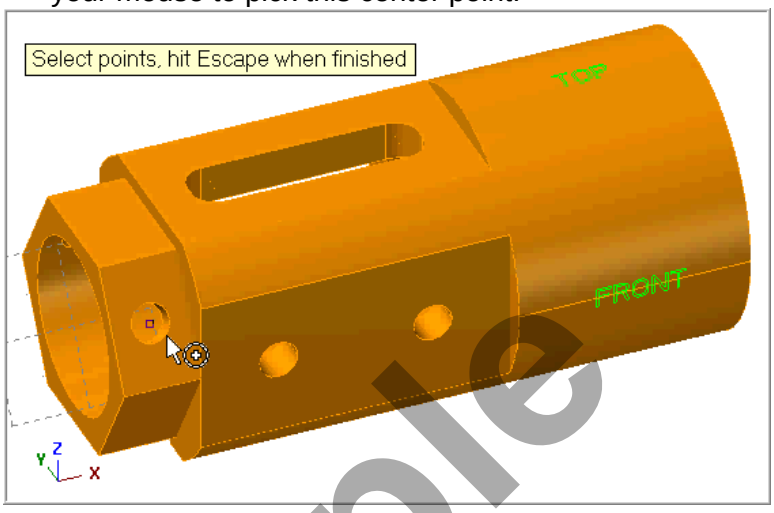
11. Your View Manager dialog box should look like the image below:



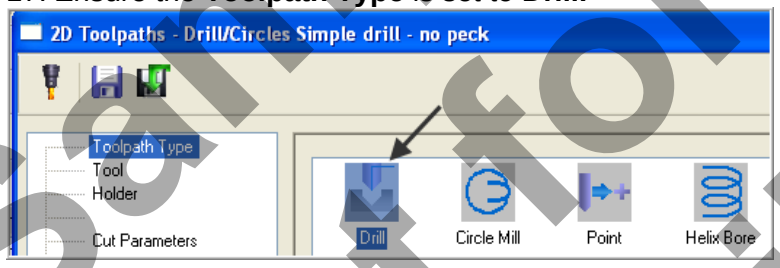
12. Click on the **OK** icon to complete this feature.
13. At the bottom of the graphics screen you will now see **T/Cplane: NEW VIEW HEX HOLE (Not Saved)**

Sample not for Distribution

14. From the menu bar select **Toolpaths>Drill…**
15. Now you are prompted to Select points and the Drill Point Selection dialog box appears. All you need to do now is snap to the centre of the circle. Move the mouse and position the cursor over the **center of the circle shown below,** you will notice a circle appears as you move closer to the center of the circle, this is the visual cue for Arc center. **Ensure the visual cue for Arc center point appears** before you select the point. Use the left button of your mouse to pick this center point.

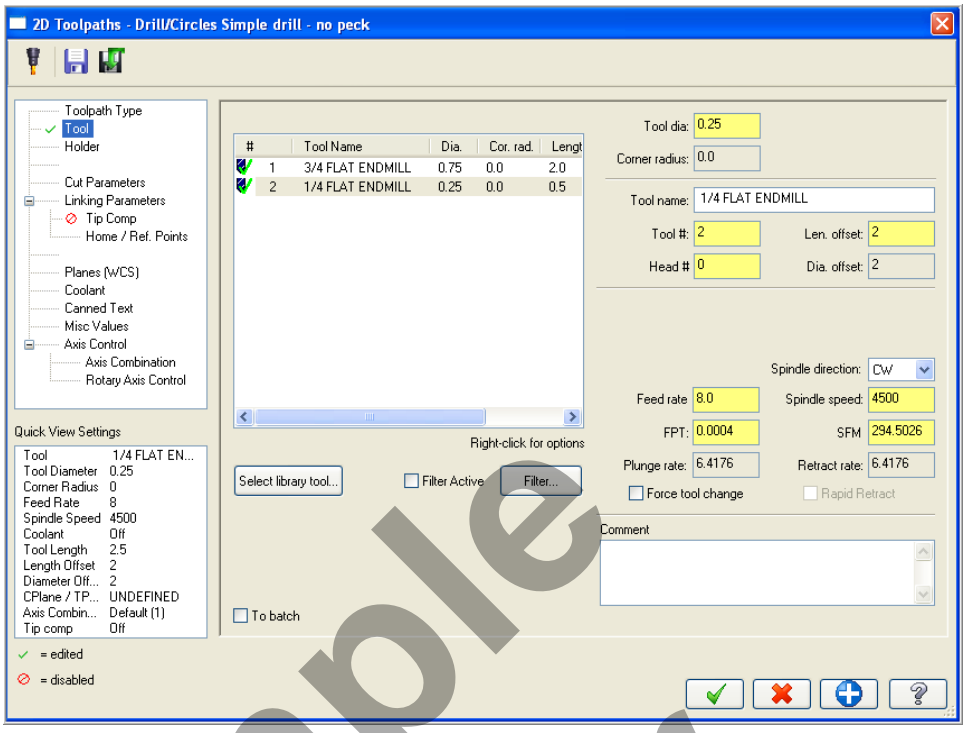


16. Select the OK button in the **Drill Point Selection** dialog window to complete the selection of points.
17. Ensure the **Toolpath Type** is set to **Drill.**

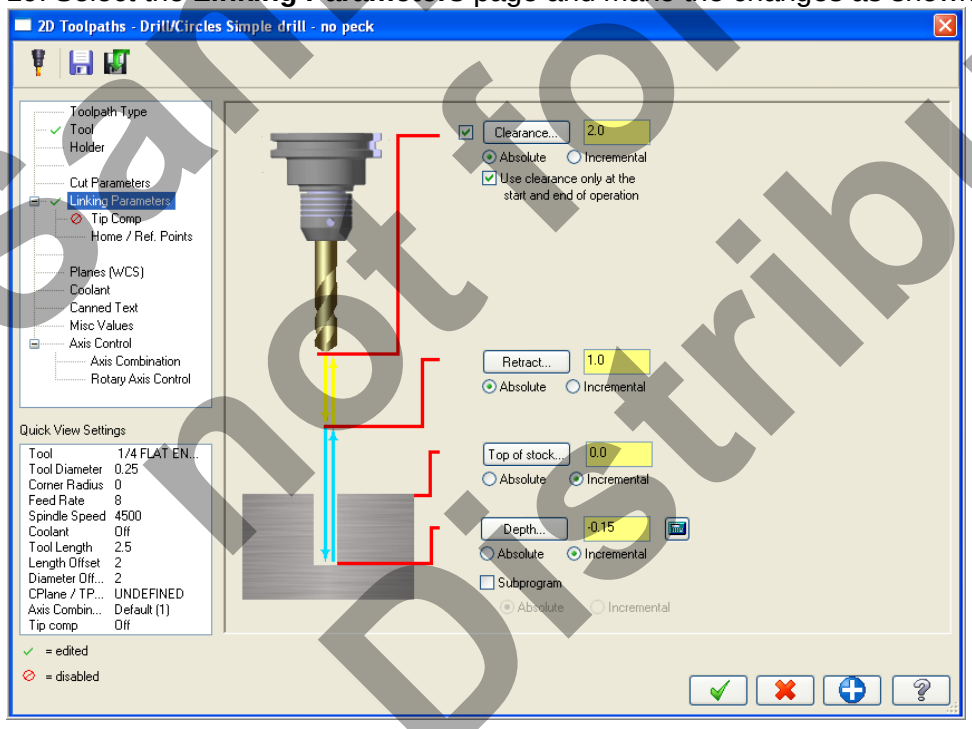


Sample not for Distribution

18. Select the **Tool** page select the **1/4" flat end mill** Ensure the page parameters are as shown below:



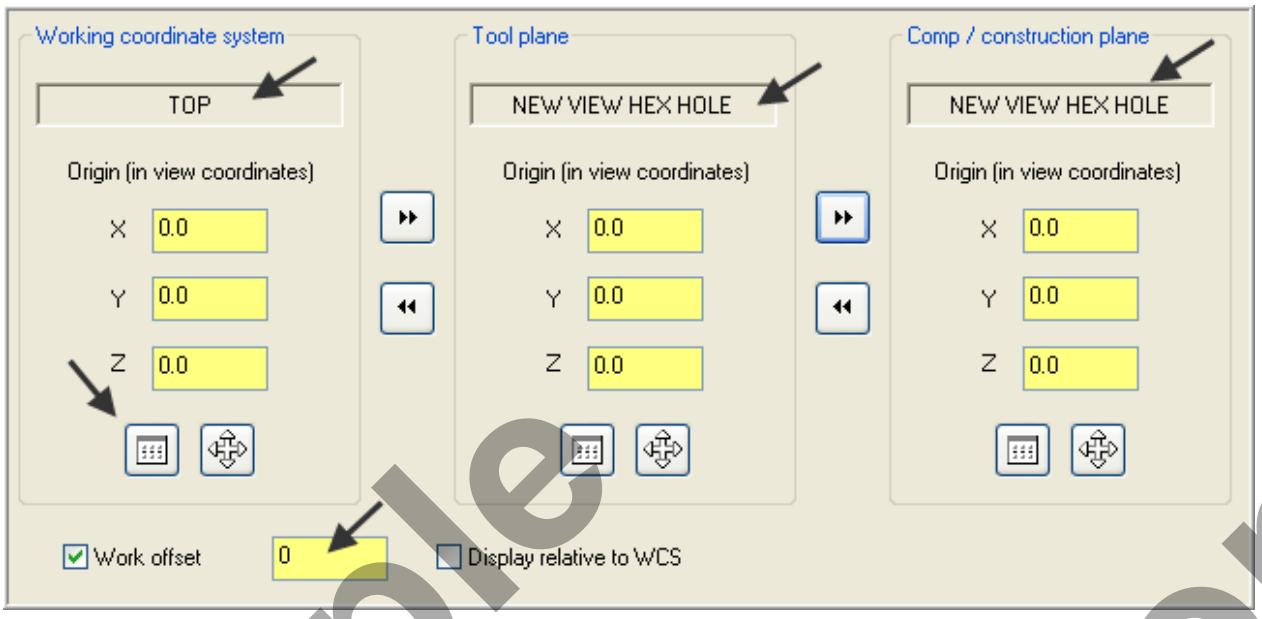
19. Select the **Linking Parameters** page and make the changes as shown below.



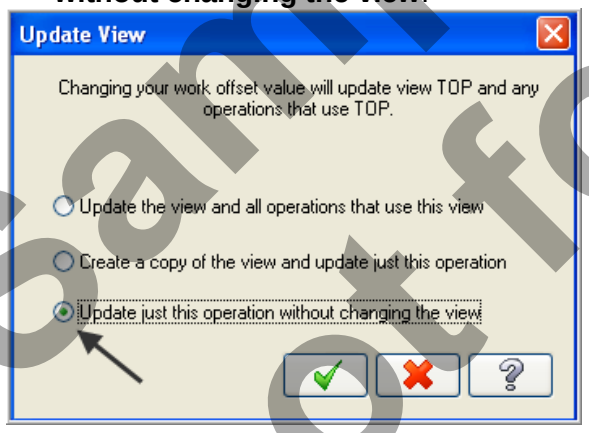
20. Tip Comp **should not** be activated.

Sample not for Distribution

21. From the list on the left select **Planes(WCS).** Ensure the values are as below. Work offset to **Zero**. Click on the View Selection button to select a new view .



22. When confronted with the **Update View** dialog box activate **Update just this operation without changing the view**.

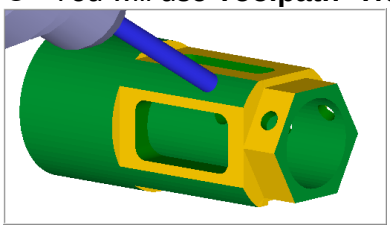


23. Click on the OK button when done .
24. Select the **Coolant** page, and set **Flood** to **On**:
25. Select the **OK** button to complete this drilling function.
26. Click on the minus sign to the left of this last operation to collapse it.
27. Save your Mastercam file.
28. Backplot this toolpath.

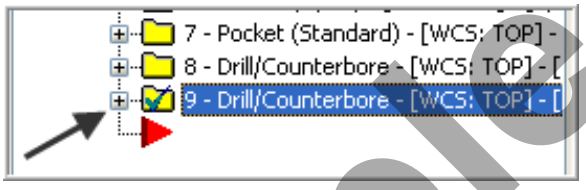
Sample not for Distribution

TASK 14:
DRILL HOLE ON HEXAGON AT BACK USING TRANSFORM TOOLPATH

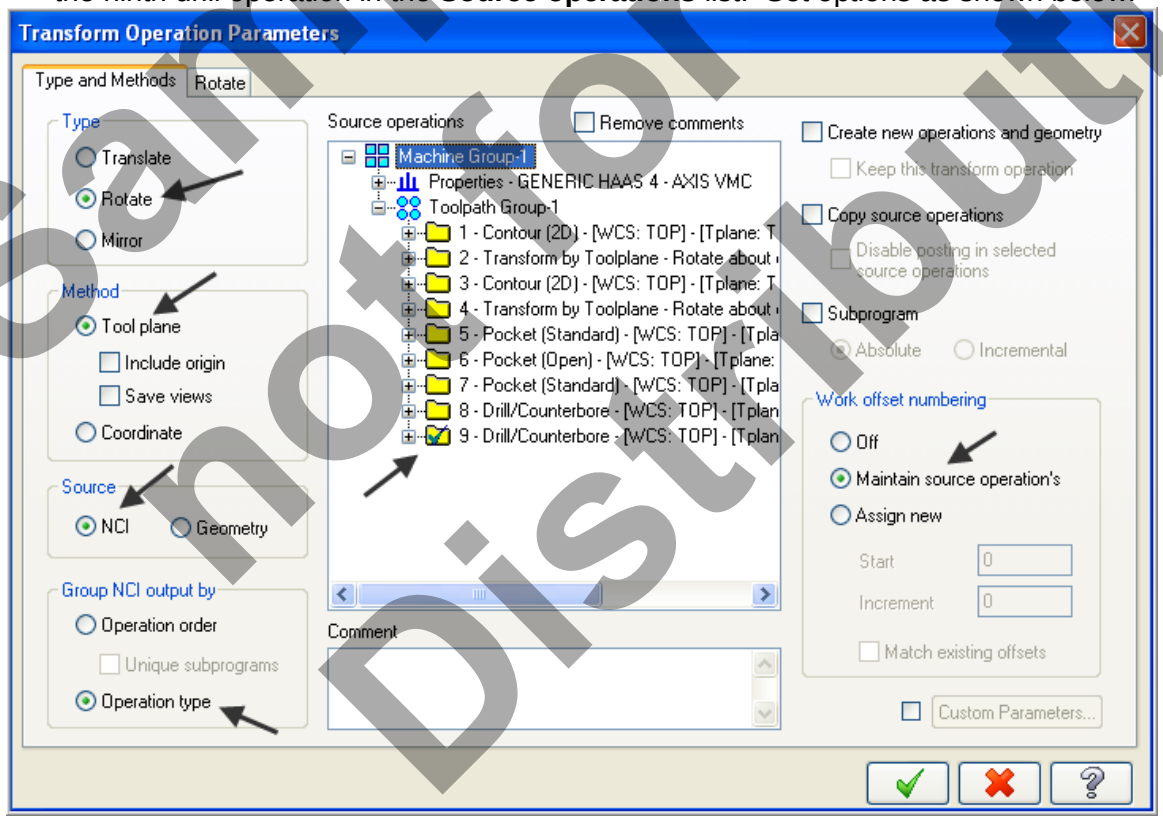
- In this task you will a drill hole on the hexagon with a 0.25” centre cutting Flat End Mill.
- You will use **Toolpath>Transform** to drill the hole on the back face of the hexagon.



1. In the Toolpaths manager **select only the ninth** operation, as indicated by the blue checkmarks shown below.

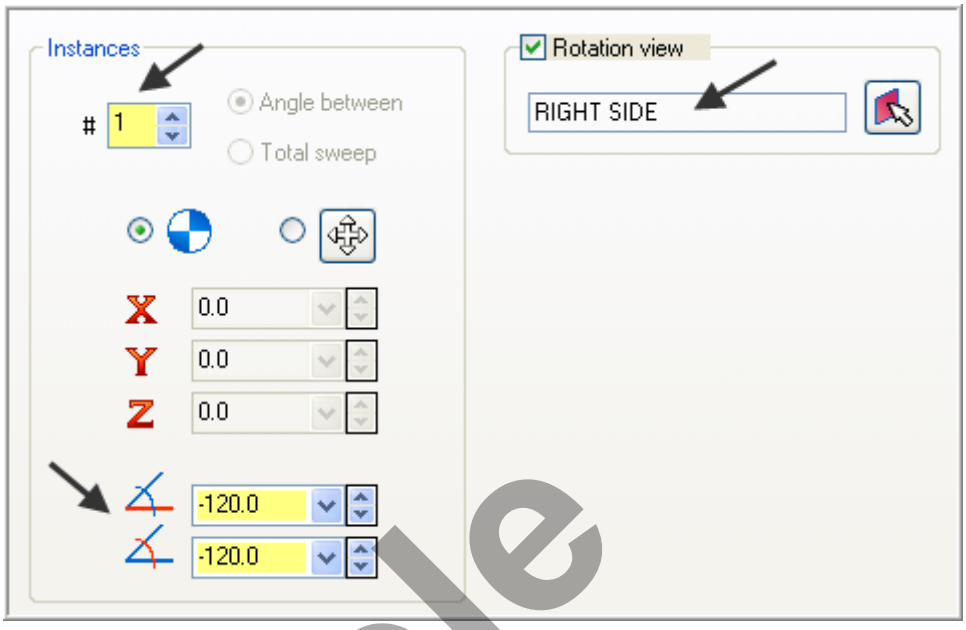


2. From the menu bar select **Toolpaths>Transform…**
3. On the **Types and Methods** page set the following. Ensure there is only a check mark for the ninth drill operation in the **Source operations** list. Set options as shown below:



Sample not for Distribution

4. Now click on the **Rotate** tab and set the following. If required click on the Select view icon and select **RIGHT SIDE**.



Start angle

Sets the beginning angle for the rotate toolpath.

Rotation angle

Sets the angle of rotation for the transform toolpath.

5. Select the OK button to exit the **Transform Operations Parameters** dialog box.
6. Click on the minus sign to the left of this last operation to collapse it.
7. Save your Mastercam file.
8. Backplot this toolpath.

Sample not for Distribution

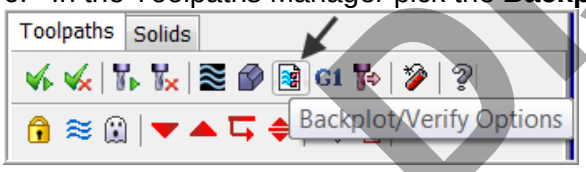
TASK 15:
VERIFY THE TOOLPATHS USING A SUPPLIED STL FILE

- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part.
- If you are using a Training Guide to go through this Lesson the required STL file is on the multimedia DVD that came with this text in a folder called Mastercam Files. If you are using the Online Course content this file is available in Step 2 of this Lesson on the online content screen.

1. Locate the supplied STL file **FOUR-AXIS-LESSON-3.STL** and copy the file to the same folder that your Mastercam files resides in.
2. For a better view of the part use the toolbar at the top of the screen to change the graphics view to Isometric.

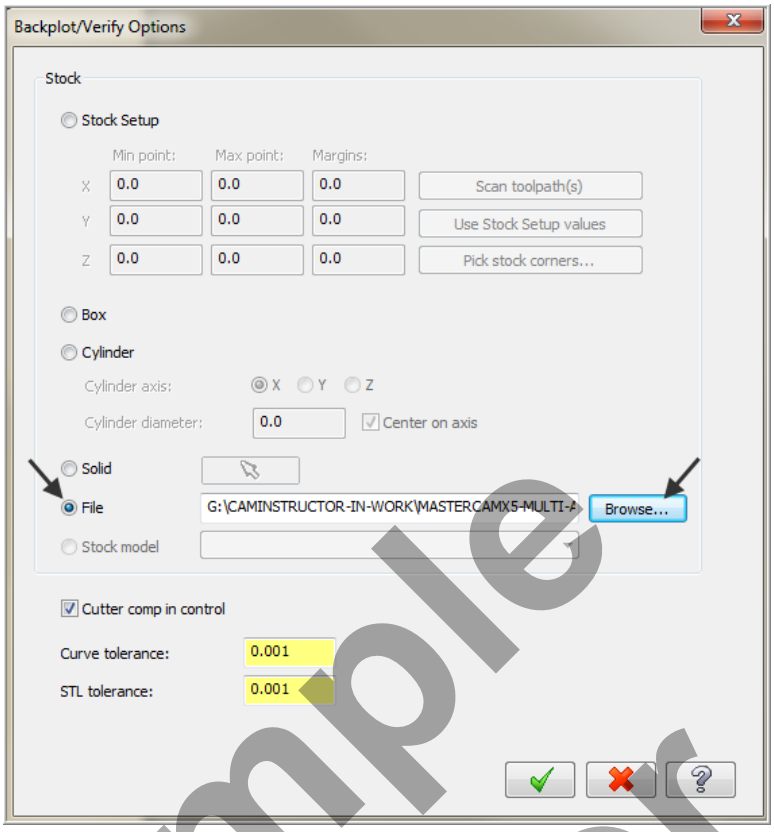
3. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .
4. Now dynamically rotate the display similar to the display shown below. The text **TOP** is facing up.

5. In the Toolpaths Manager pick all the operations to verify by picking the Select All icon
6. In the Toolpaths Manager pick the **Backplot/Verify Options** icon.

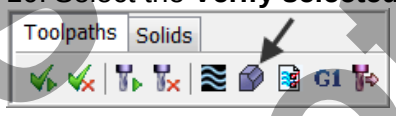


Sample not for Distribution

7. In the **Backplot/Verify Options** window activate the radio button for **File** and then select the **Browse** icon.

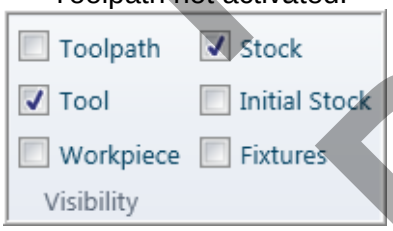


8. Select the **FOUR-AXIS-LESSON-3.STL** file and then click on **Open**. **Please Note:** The STL file is supplied on the multimedia DVD that came with this text in a folder called Mastercam Files.
9. Select the **OK** button to exit **Backplot/Verify Options**.
10. Select the **Verify selected** operations button shown below:



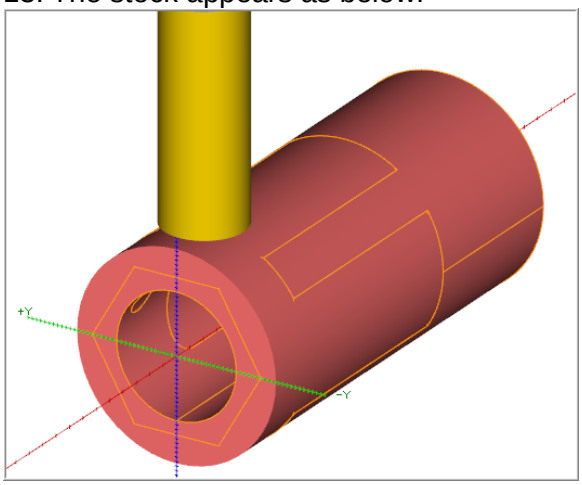
11. **Maximize** the Backplot/Verify window if required.

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

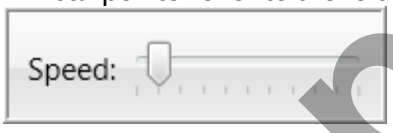


Sample not for Distribution

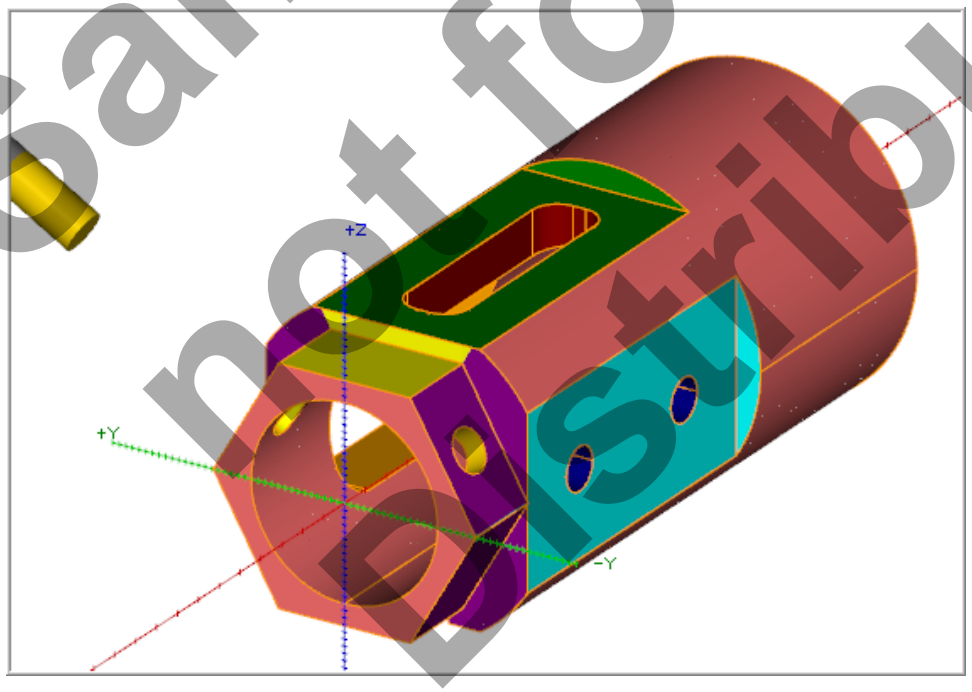
13. The stock appears as below:



14. Activate the **Color Loop** to change the color of the tools for the verified part.
15. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.



16. Now select the Play Simulation button to review the toolpaths.

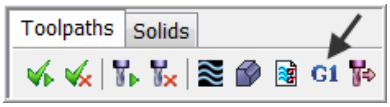


17. Select the **Close** button in the top right hand corner to exit Verify.

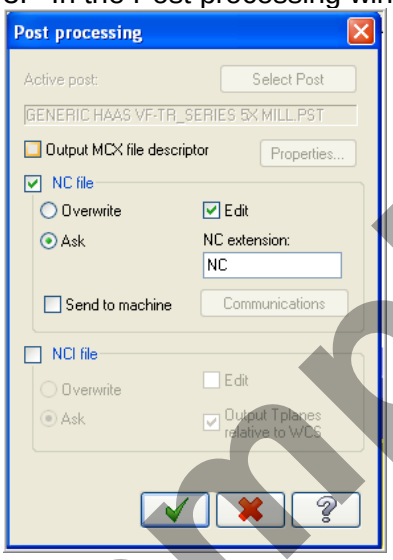
Sample not for Distribution

TASK 16:
POST AND CREATE THE CNC CODE FILE

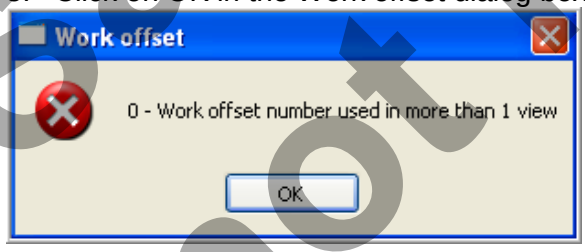
1. Ensure all the operations are selected by picking the **Select All** icon from the Toolpaths Manager.
2. Select the **Post selected operations** button from the Toolpaths Manager.



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



4. Select the OK button to continue.
5. Click on OK in the Work offset dialog box.



6. Enter the same name as your Mastercam part file name in the NC File name field **FOUR-AXIS-LESSON-3**.
7. Select the **Save** button.

Sample not for Distribution

8. Scroll down and review the NC file. Note the various **A-Axis** moves to machine the pockets and holes.

FOUR-AXIS-LESSON-3.NC

```
%
O3
(FOUR-AXIS-LESSON-3)
(DATE=DD-MM-YY - 09-04-13 TIME=HH:MM - 14:31)
(MCX FILE - G:\CAMINSTRUCTOR-IN-WORK\MASTERCAMX5-MULTI-AXIS-LESSONS\FOUR-AXIS-LESSON-3\COMPLETED\FOUR-AXIS-LESSON-3.MCX-7)
(NC FILE - C:\USERS\NEW\DOCUMENTS\MY MCAMX7\MILL\NC\FOUR-AXIS-LESSON-3.NC)
(MATERIAL - ALUMINUM INCH - 6061)
(T1| 3/4 FLAT ENDMILL|H1|D1|TOOL DIA. - .75)
(T2| 1/4 FLAT ENDMILL|H2|D2|TOOL DIA. - .25)
N1 G20
N2 G0 G17 G40 G49 G80 G90
N3 T1 M6
N4 G0 G90 G54 X-1.405 Y-1.1469 A0. S3000 M3
N5 G43 H1 Z2.
N6 M8
N7 Z.9375
N8 G1 Z.6875 F7.
N9 X-.655
```

9. Select the in the top right corner to exit the CNC editor.
10. This completes **Four-Axis-Lesson-3**.

FOUR-AXIS-LESSON-3 EXERCISES

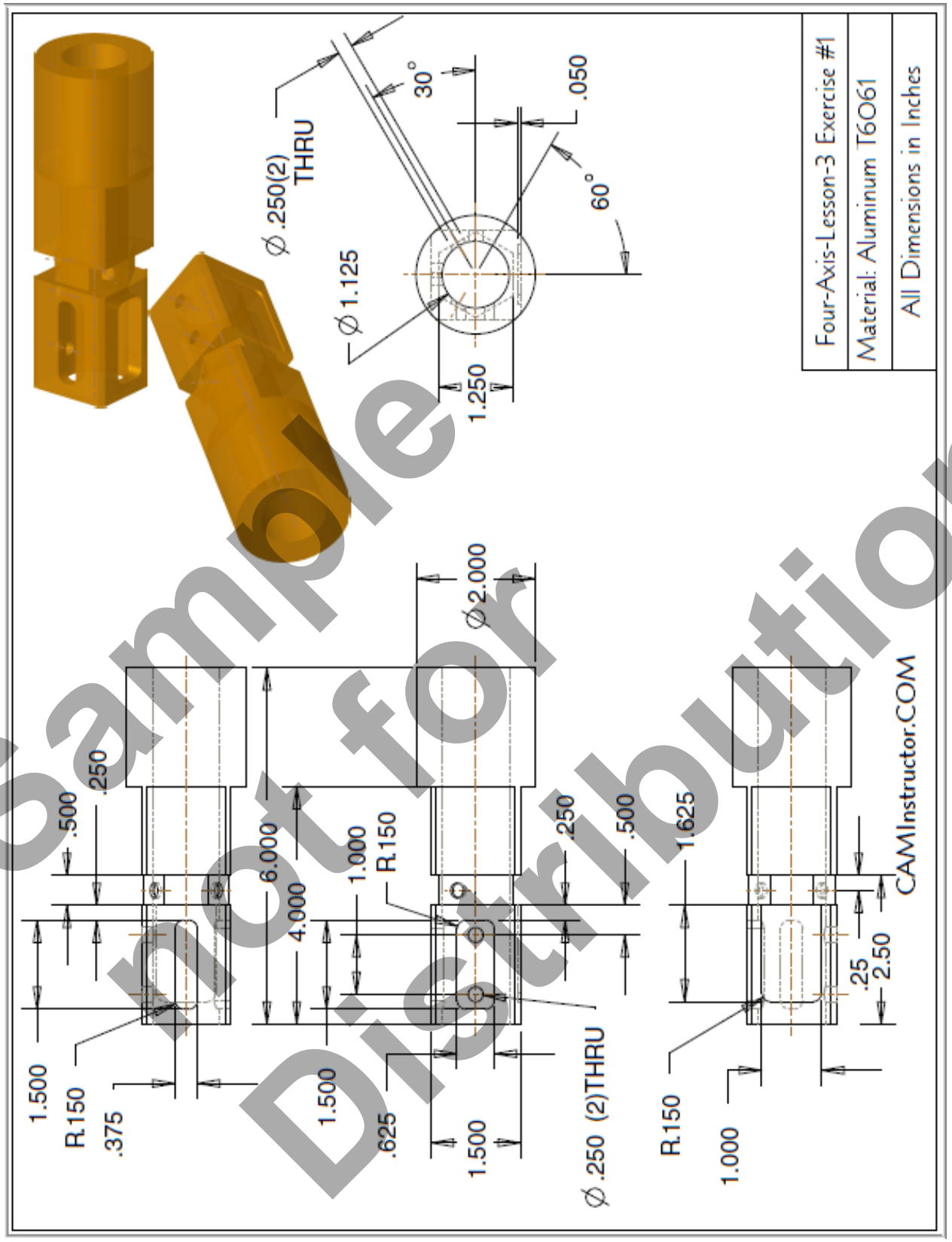
➲ If you are using a Training Guide the required exercise files are on the multimedia DVD that came with this text in a folder called Mastercam Files. If you are using the Online Course content the exercise files are available in this Lesson on the online content screen.

FOUR-AXIS-LESSON-3 EXERCISE #1

FOUR-AXIS-LESSON-3 EXERCISE #2

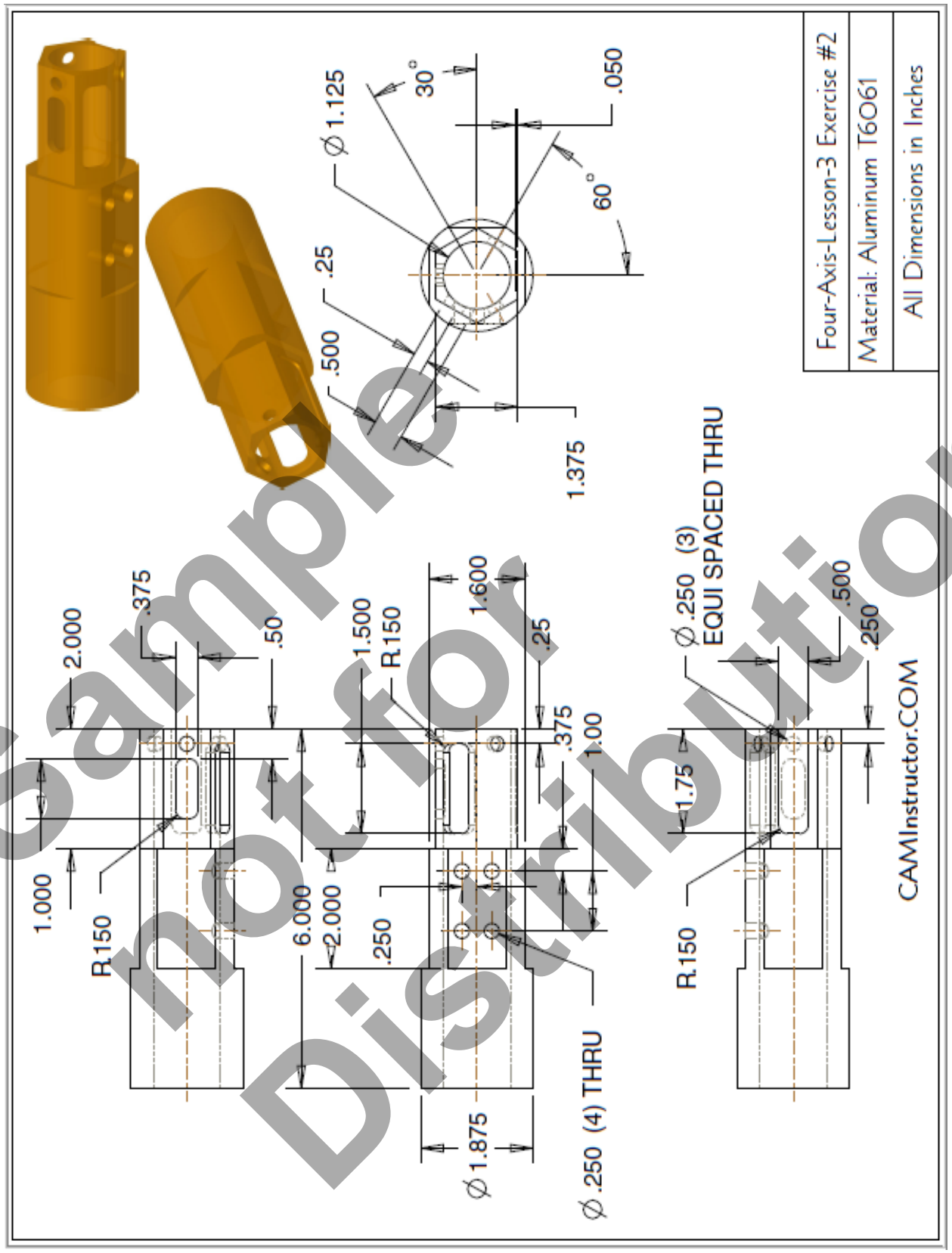
Sample not for Distribution

FOUR-AXIS-LESSON-3 EXERCISE #1



Sample not for Distribution

FOUR-AXIS-LESSON-3 EXERCISE #2



Sample not for Distribution