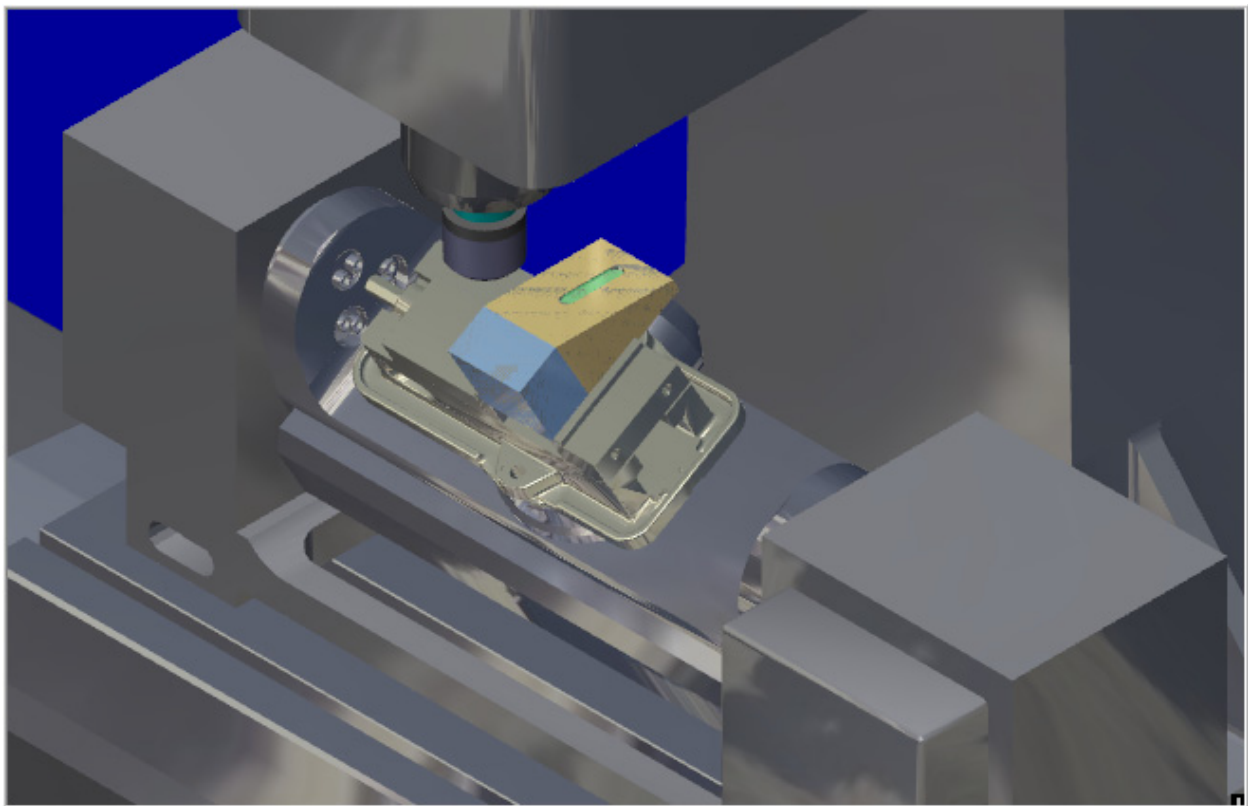


Mastercam X⁴

TRAINING

GUIDE



MULTI-AXIS-LESSON-1

FACE AND POCKET

TRUNNION-TABLE – 3+2 POSITIONAL

camInstructor

Objectives

For **Multi-Axis-Lesson-1** you will generate the toolpaths to machine the part on a CNC vertical milling machine with multi-axis capability. The part will be held in a machine vise and a Trunnion table as shown below will be utilised to machine the angled face and pocket. This Lesson covers the following topics:

➤ **Open an existing file containing:**

The solid geometry for the machine vise and Wireframe geometry for the part

➤ **Establish Stock Setup settings:**

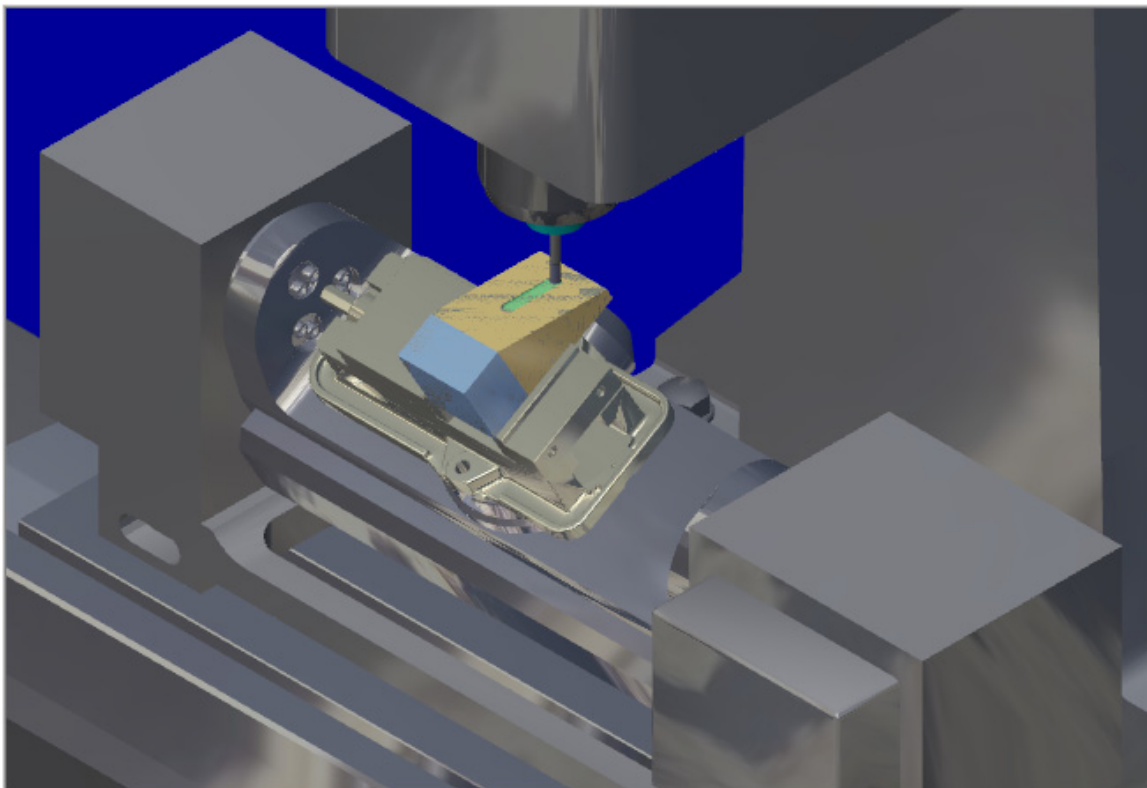
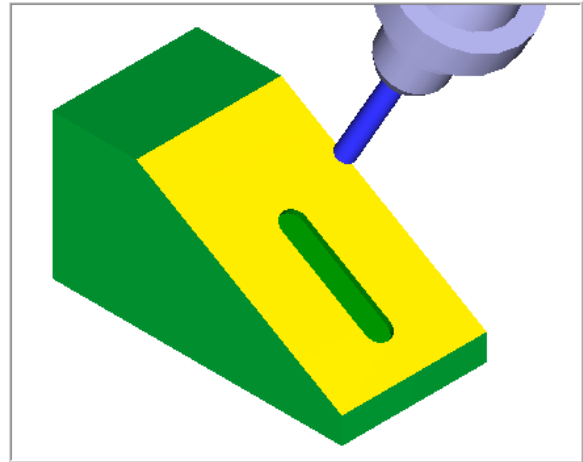
Stock size
Create points to be used for Trim toolpath
Material for the part
Feed calculation

➤ **Generate milling toolpath consisting of:**

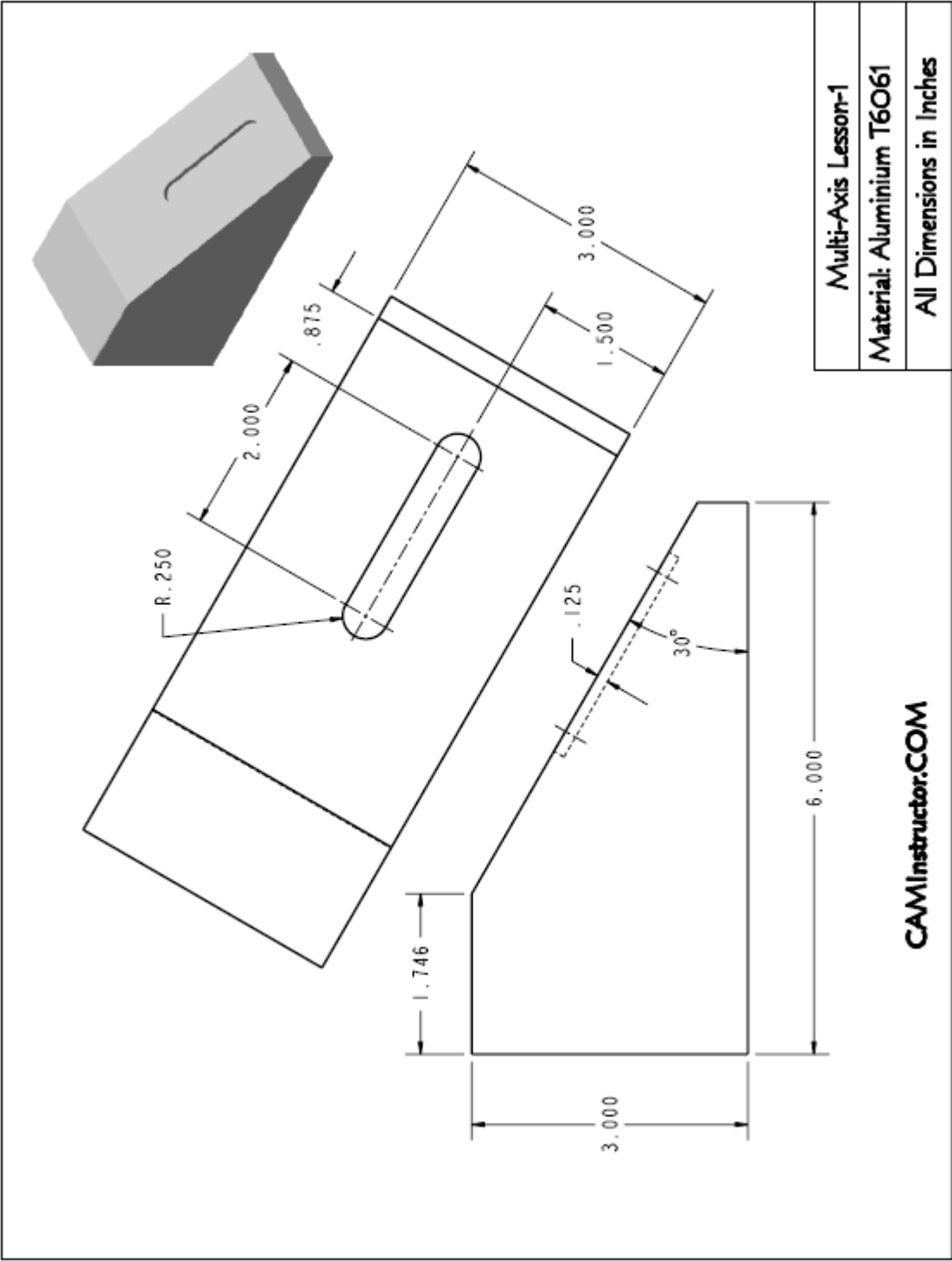
Using the View Manager
Using Views, Tool Planes and Construction Planes
Create Face and Pocket toolpaths

➤ **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


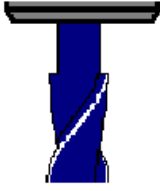


MULTI-AXIS-LESSON-1 DRAWING



TOOL LIST

- ☞ Two cutters will be used to create this part.
- ☞ 3.0 diameter face mill to machine the 30 degree angled face.
- ☞ 0.375 diameter flat end mill to machine the pocket. The pocket depth is 0.125.

Tool List of COMPLETED-MULTI-AXIS-LESSON-1.MCX									
Proj./Part No.:	0	Date	:	11/21/08					
Drawing No.:	1	Customer	:	-					
Prog. No.:	1	Programmer	:	1					
<hr/>									
	Tool type	:	2	Face mill 2" Face Mill					
	Manufact.code	:							
	Chuck	:							
	Tool Number	:	1		Feedrate	:	20		
	Diameter	:	2	RPM	:	2500	Plunge feed r.:	20	
	Corner radius	:	0	Tip angle	:	45	Diam. offset	:	1
	Flute length	:	1.575	Material	:	ALUMINUM ...	Length offset	:	1
	Overall length:		2.1	No flutes	:	2			
	Tool type	:	0.375	Endmill11 Flat 3/8 FLAT ENDMILL					
	Manufact.code	:							
	Chuck	:							
	Tool Number	:	2		Feedrate	:	10		
	Diameter	:	0.375	RPM	:	3500	Plunge feed r.:	8	
	Corner radius	:	0	Tip angle	:	0	Diam. offset	:	2
	Flute length	:	0.75	Material	:	ALUMINUM ...	Length offset	:	2
	Overall length:		2.5	No flutes	:	4			

MULTI-AXIS-LESSON-1 - THE PROCESS

Toolpath Creation

- TASK 1:** Setting the environment
- TASK 2:** Introduction - Watch the video
- TASK 3:** Open an Existing file from the Multimedia CD
- TASK 4:** Define the rough stock using stock setup and create points that will be used for Trim toolpath
- TASK 5:** Use View manager to set up new Cplanes and Tplanes
- TASK 6:** Face Mill 30 Degree angle
- TASK 7:** Use Trim toolpath to edit the facing toolpath
- TASK 8:** Pocket slot on 30 degree face. The pocket depth is 0.125.
- TASK 9:** Backplot the toolpath
- TASK 10:** Verify toolpaths
- TASK 11:** Post and create the CNC code file

EXAMPLES OF HAAS ROTARY TRUNNION TABLES



Toolpath Creation

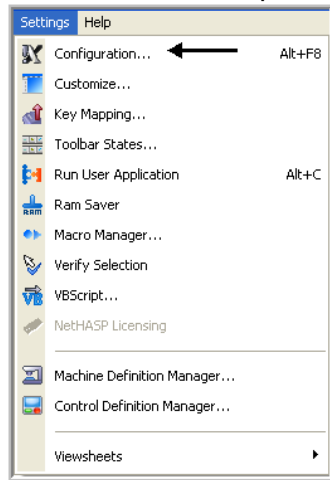
TASK 1: SETTING THE ENVIRONMENT


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:

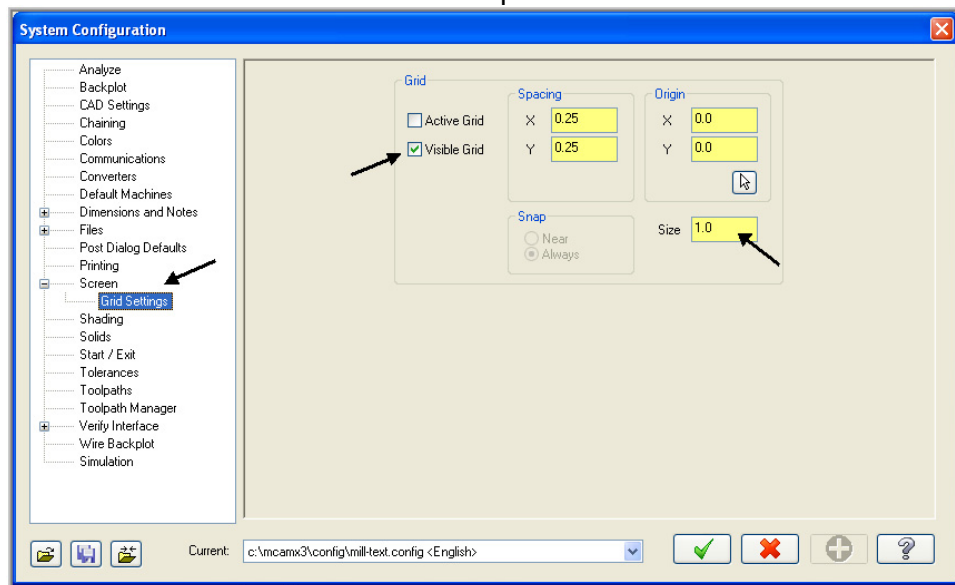
1. Set up the Grid. This will help identify the location of the origin.
2. Customize the toolbars to machine a **4-5 Axis part**.

SET THE DISPLAY OF THE GRID:

1. Launch Mastercam.
2. Select from the pull down menu **Settings>Configuration**.



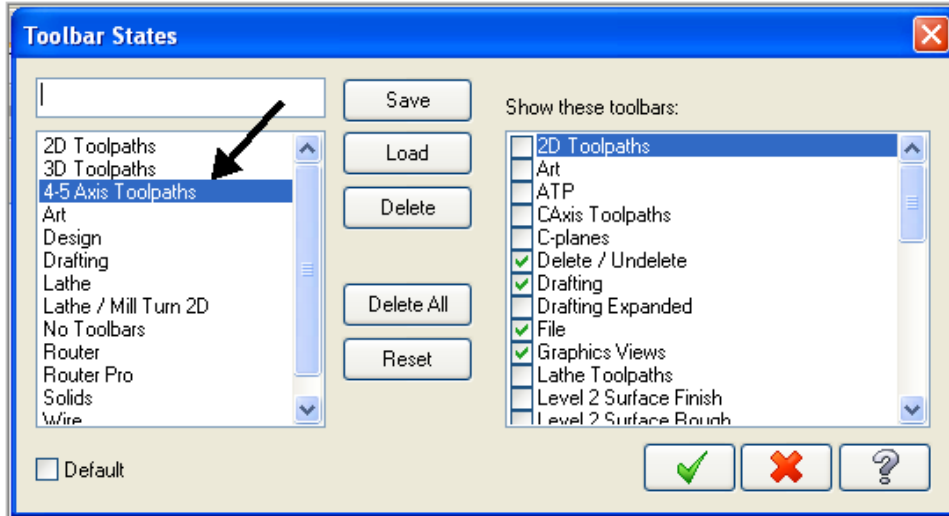
3. From the window on the left side of this window expand the Screen topic by selecting the + sign and then select **Grid Settings**.
4. Enable the **Visible Grid** and change the **Size** to 1.
5. Select the OK button  to complete this function.



6. When prompted to “**Save settings to configuration file**” select **Yes**.

SET THE TOOLBARS REQUIRED FOR A 4-5 AXIS PART:

1. Select from the pull down menu **Settings>Toolbar States**.
2. Select **4-5 Axis Toolpaths**.
3. Then select the Load button.

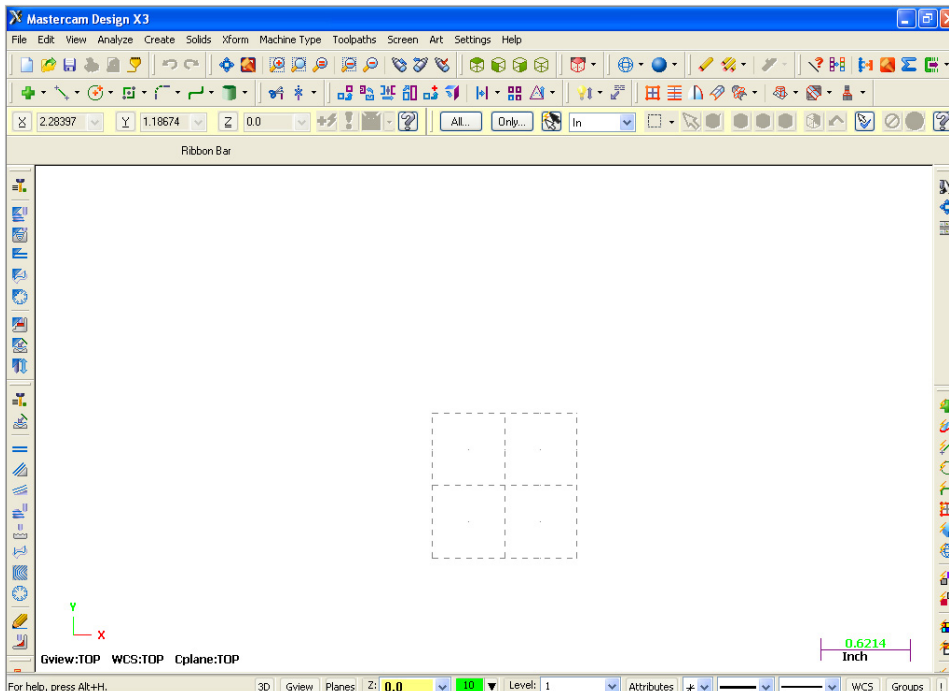


4. Select the OK button  to accept the settings.

☞ **To toggle on or off the display of the Operations Manager.**

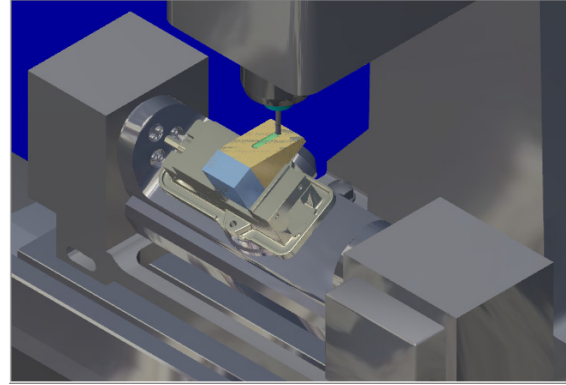
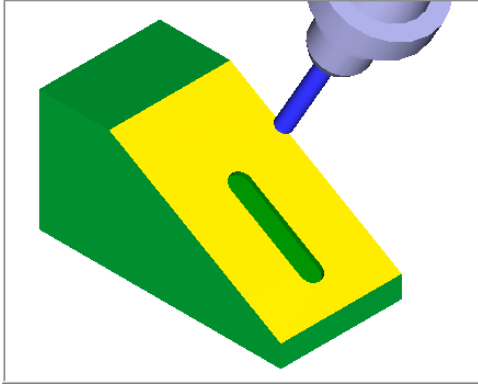
☞ To Show or Hide the **Operations Manager** pane on the left of the screen press **Alt + O**. Pressing **Alt + O** acts like a toggle switch between Show and Hide. For more information on the **Operations Manager** see the **Tips and Techniques** section on the multimedia CD supplied with this text.

5. The Mastercam interface will be displayed as shown below when the operations Manager is hidden.



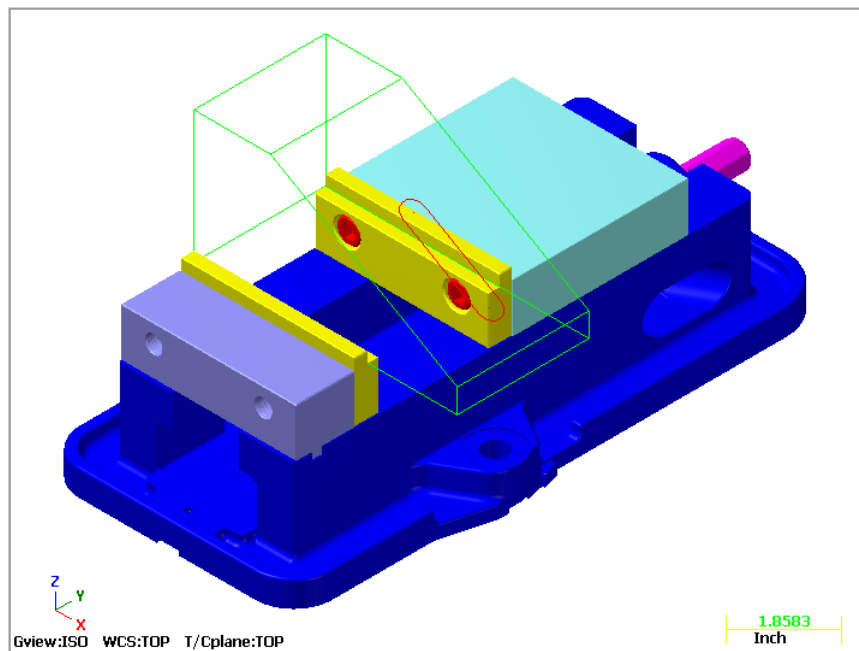
TASK 2: INTRODUCTION – WATCH THE VIDEO

1. Before you start to work on this Lesson review the Introduction video on the multimedia CD that came with this text. You will find the video in the “**Lesson-1**” section, it is entitled **Introduction - 10 Minutes**.
- The video will review the techniques that will be used to generate the two toolpaths for the face and pocket toolpaths utilizing the Trunnion table.



TASK 3: OPEN EXISTING FILE FROM THE MULTIMEDIA CD

- On the multimedia CD that came with this text is a folder called **Mastercam-Files**. The file will contain the wireframe geometry of the part and the vise constructed as a solid. The part is already setup for a: **GENERIC HAAS VF-TR_SERIES 5X MILL**.
1. Select **File>Open> Multi-Axis-Lesson-1.MCX**.
2. Activate a shaded view by selecting the icon at the top of the screen. The file should appear as below. The vise is made up of solid features, the part is wireframe geometry.



TASK 4:

DEFINE THE ROUGH STOCK USING STOCK SETUP

- Define the Rough Stock sizes and material type using stock setup.
- Before you use Stock Setup the Levels that contain the vice will be turned off, it will make it a lot easier to select the entities in this next set of instructions.
- During this lesson you will turn the visibility of Levels off and on.

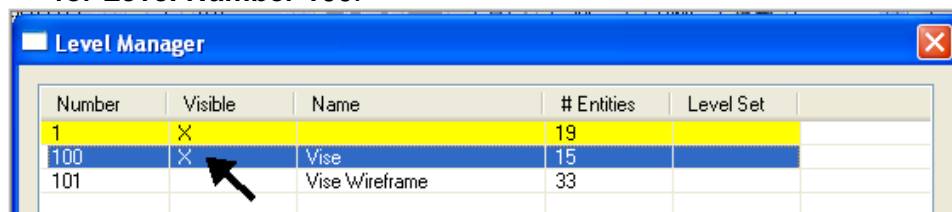
➤ Levels are a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities, and toolpaths.

➤ 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.

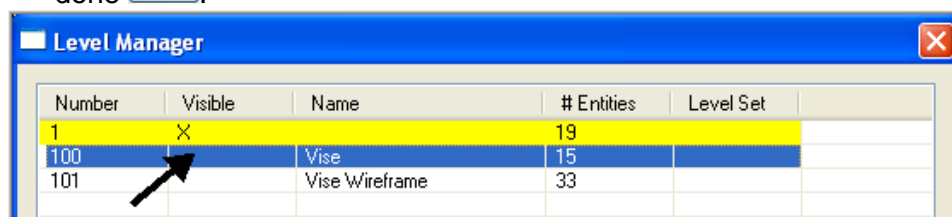
1. Activate a wireframe view by selecting the icon at the top of the screen.
2. From the **Status** bar at the bottom of the screen select **Level**.



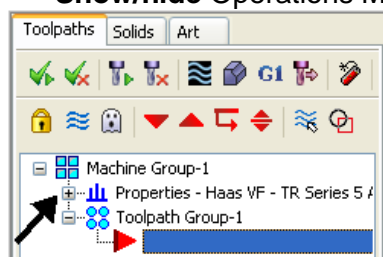
3. The **Level Manager** dialog window will now appear. In the **Visible** column **click on the X for Level Number 100**.



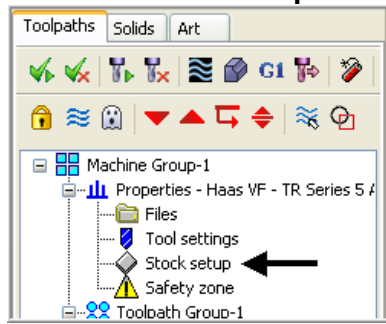
4. Ensure the Level Manager dialog window appears as below, with the **X for Level Number 100 removed**. This makes the entities on that level **invisible**. Click on the OK button when done .



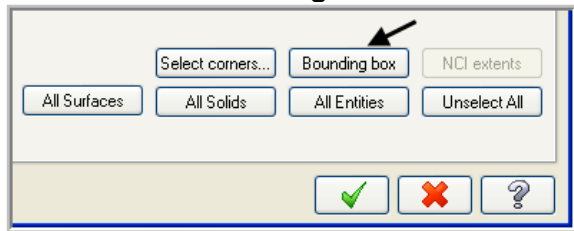
5. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will **Show/hide** Operations Manager pane.



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



7. Select the **Bounding box** button.

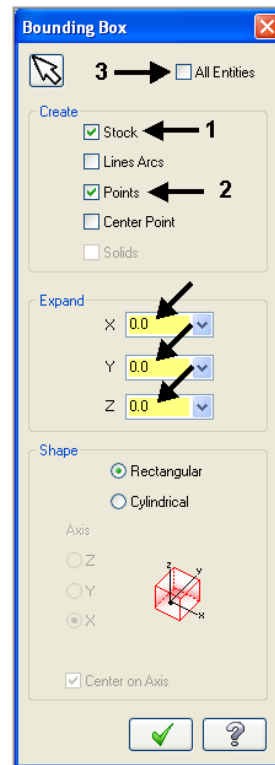


8. In the **Bounding Box** dialog window first activate **Create Stock - #1**

9. **Second** activate **Points - #2**.

10. The **Expand** values are all set at Zero.

11. **Third** uncheck **All Entities - #3** as shown on the right.



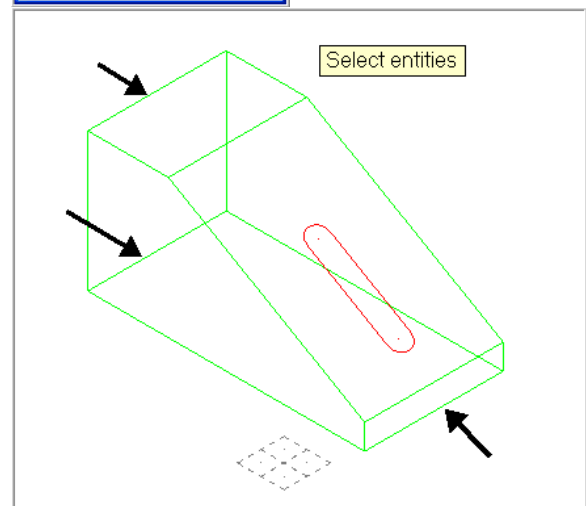
12. You are now prompted to **Select Entities**.

Select the three lines as shown on the right; you can select the lines in any order. The stock size needs to be the same size as the part, picking these three lines defines the X, Y and Z values of the stock.

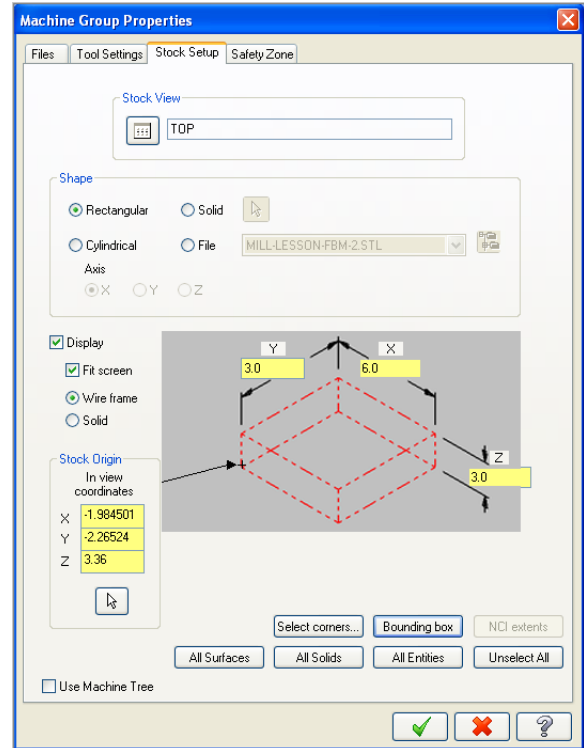
13. Click on the **End Selection** icon.




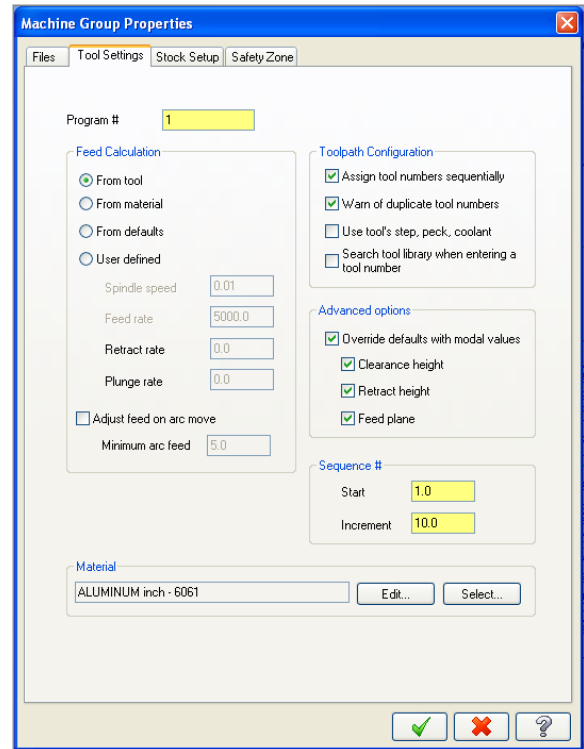
14. Select the OK button  to complete this Bounding Box function.




15. Your stock sizes should now be **X6.0, Y3.0, and Z3.0** as shown on the right. Complete the remaining parameters to match the **Stock Setup** screenshot on the right.



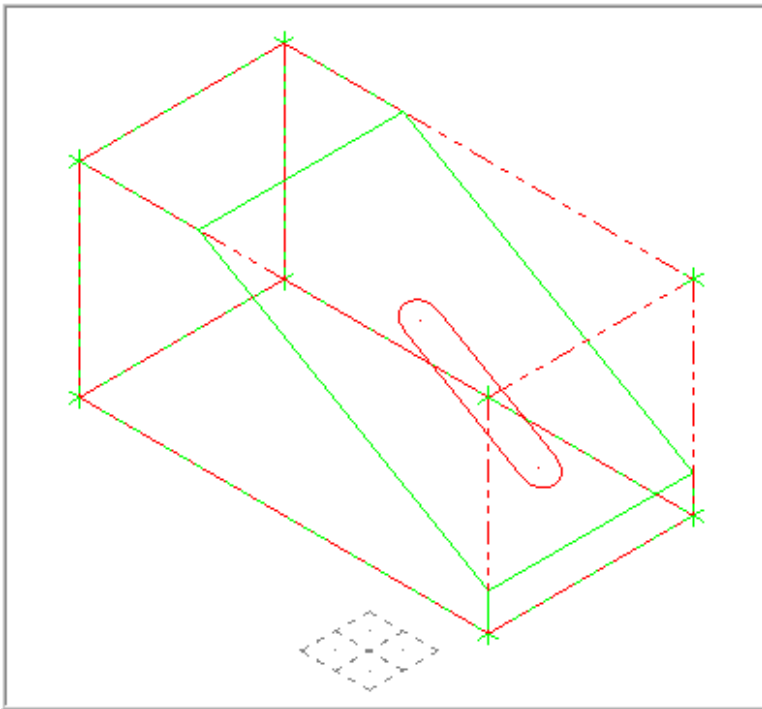
16. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. To change the Material type follow the next set of instructions.
17. To change the Material type to Aluminium 6061 pick the Select button at the bottom of the Tool Settings page.
18. At the **Material List** dialog box open the **Source** drop down list and select **Mill – library**.
19. From the Default Materials list select **ALUMINIUM inch -6061** and then select .



20. Select the OK button  again to complete this Stock Setup function.

Mastercam Training Guide

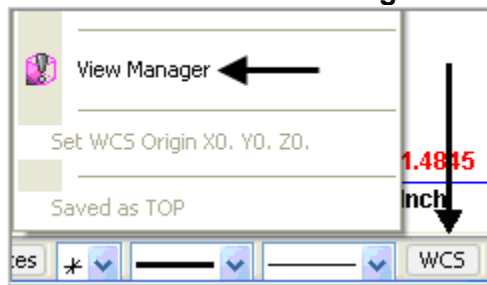
- Your part should look similar to the screen shot below. **Note the points** on the corners of the stock. These points will be used later on in this lesson when you use the Trim toolpath function.



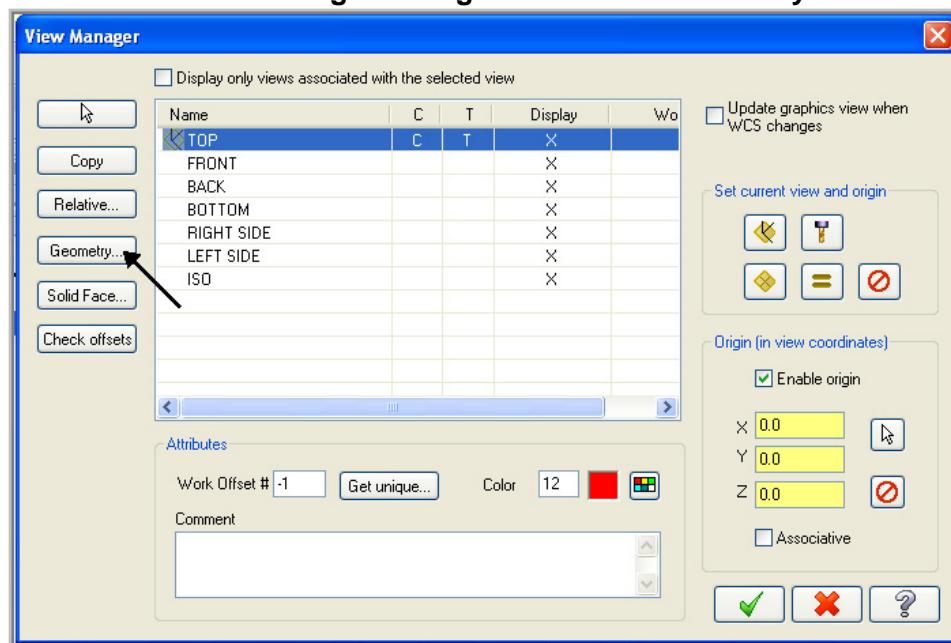
TASK 5: USE VIEW MANAGER TO SET UP NEW CPLANES AND TPLANES

- **Cplane** is an abbreviation for construction plane; the plane where geometry is created.
- Typically, this is used in connection with drafting when you draw entities, select a Cplane to orient the geometry in space. When you create a toolpath, the construction plane is the plane in which the tool is compensated. For almost all toolpaths, the construction plane should be the same as the tool plane (which is the plane normal to Z, or to the vertical tool axis). However, it is possible to have them differ for advanced contour applications or Swiss machining.
- **Tplane** is an abbreviation for tool plane; a 2D plane that represents the CNC machine's XY axis and origin.
- 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. Changing the Tplane typically produces a rotary motion code when you post your toolpath. The exact effect depends on your machine tool and post processor; for example, rotating the tool plane 30 degrees about the X-axis might tilt the tool axis on a 5-axis machining center, or could rotate a fixture on a table, depending on how your post is written.

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

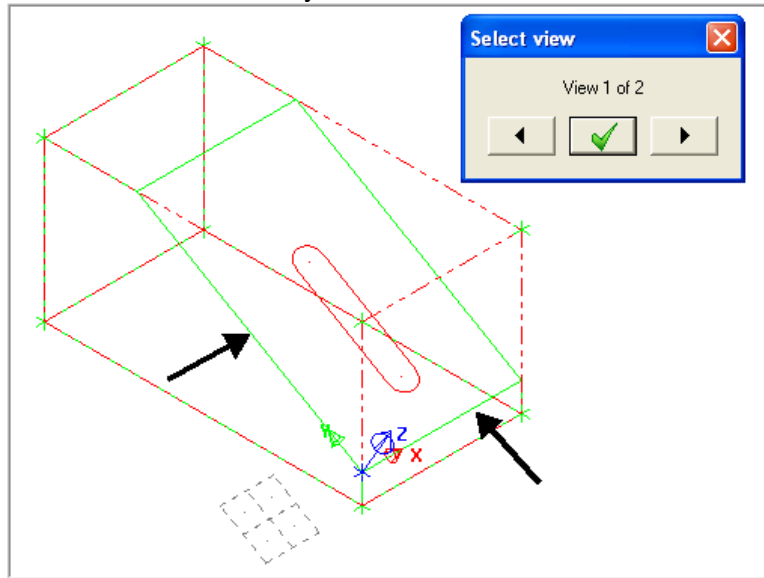


2. From the **View Manager Dialog** box select the **Geometry** button as shown below.

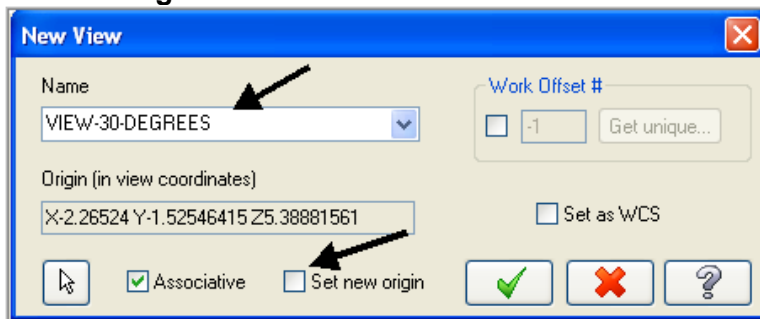


Mastercam Training Guide

- You will be returned to the graphics screen with a prompt to: **Select construction plane by geometry. Cplane by geometry.** Select the **two lines shown below**. Make sure the **XYZ tripod** is the same as the example below right with the **Z axis** arrow pointing up from angled face. If required use the **arrow buttons** in the Select view screen to cycle through to the correct coordinate system.

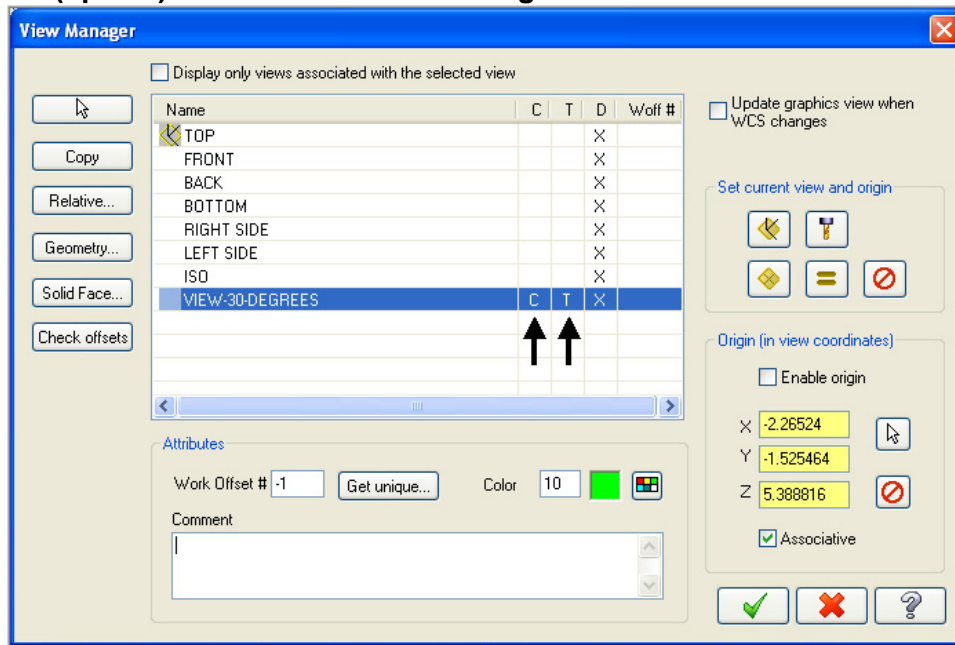


- Select the green check mark for **OK**.
- In the **New View** dialog box enter **VIEW-30-DEGREES** in the name section. Un-check **Set new origin**.

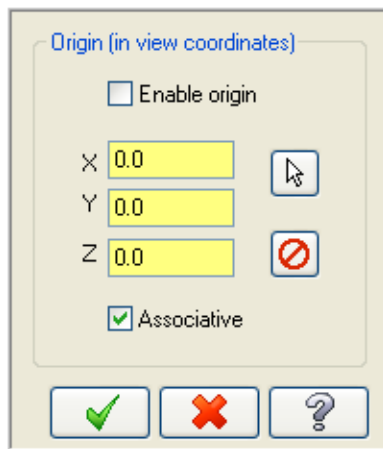


- Click on the OK icon  to complete this feature.

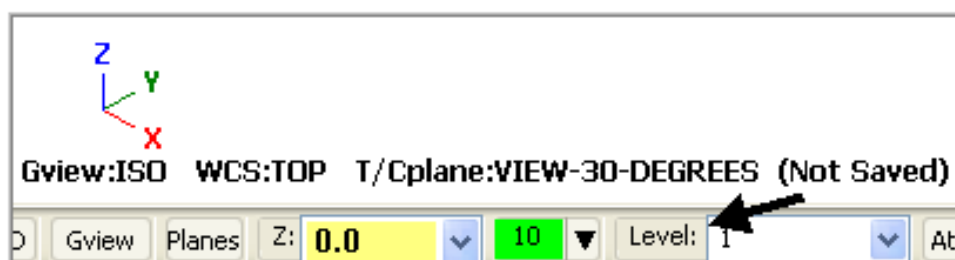
7. Now you are back in the View Manager dialog box pick in the **C (Cplane)** column and **T (Tplane)** column of the **View-30-Degrees** row as shown below.



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

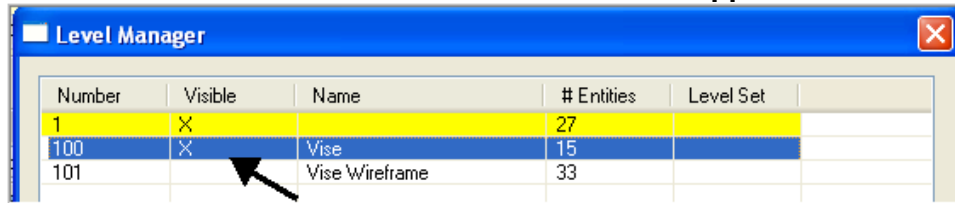


9. Click on the **OK** icon to complete this feature.
 10. Select on your keyboard **ALT and F9**. This displays on the screen the orientation of the **Cplane**, **Tplane** and the **WCS**.
 11. Select on your keyboard **ALT and F9** to turn off the display.
 12. Now you will make the **Level visible** that contains the geometry for the vise. From the **Status** bar at the bottom of the screen select **Level**.



Mastercam Training Guide

13. The **Level Manager** dialog window will now appear. In the **Visible** column **click on the visible column for Level Number 100** so the **X** appears to make this level visible.

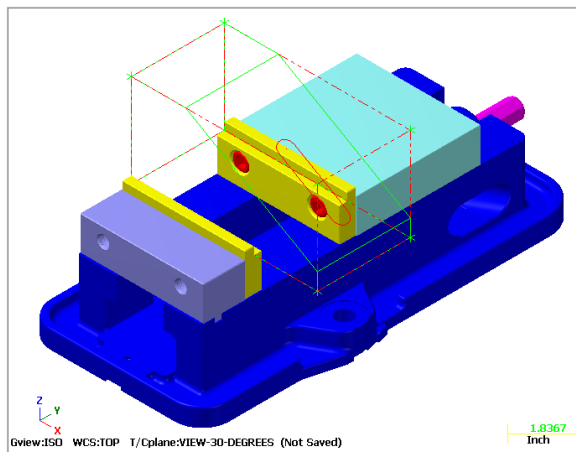


14. Click on the OK button when done .

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



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



☞ Notice at the bottom of the graphics screen the status of the **Tool and Construction** plane, it is set to **VIEW-30-DEGREES (Not Saved)**. Before you start to work on the toolpath for the pocket you need to understand what happens when you start to display different views of the part, such as an Isometric or Front view and how it affects the Cplane and Tplane.

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

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

☞ Notice at the bottom of the graphics screen the status of the Tool and Construction plane, it is now set to **TOP** as shown below.

☞ If you had **dynamically** moved the view around the **T/Cplane** would have stayed at **VIEW-30-DEGREES**

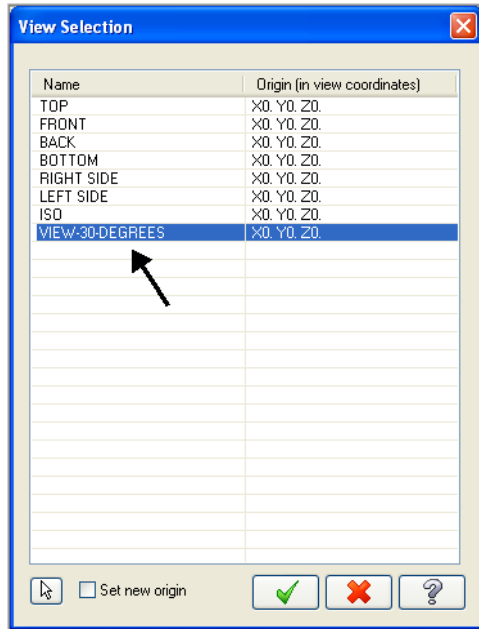
☞ Before you start to work on the toolpaths you need to change the **T/Cplane** to the **VIEW-30-DEGREES** created earlier.



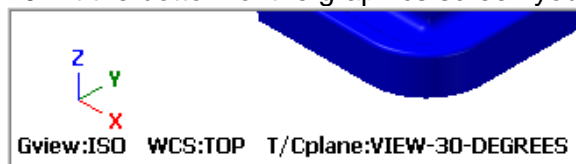
18. Select **Planes** from the **Status bar** and then select **Named Views...**



19. From the **View Selection** dialog window click on **VIEW-30-DEGREES** and then select the **OK** button to exit



20. At the bottom of the graphics screen you will now see **T/Cplane:VIEW-30-DEGREES**.



Please Note:

- Before you start to work on any of the toolpaths in this lesson look at the bottom of the graphic screen and ensure the status of the **T/Cplane** is set to **VIEW-30-DEGREES** as the picture above shows.
- If it is not set to **VIEW-30-DEGREES** use the steps just prescribed, for example:
 1. On the Status bar select **Planes**
 2. **Named Views...**
 3. From the **View Selection** dialog window click on **VIEW-30-DEGREES**

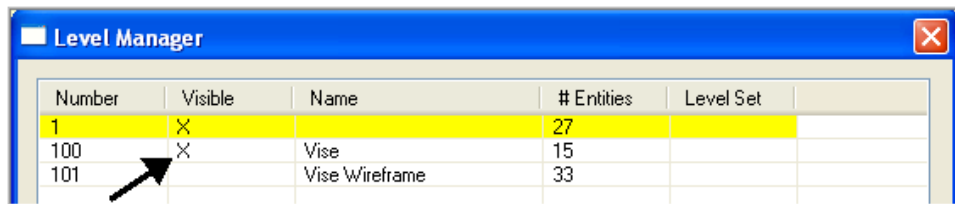
TASK 6: FACE MILL 30 DEGREE ANGLE

- In this task you will use a 2.0 diameter face mill to face the 30 degree angle. The tool axis will be perpendicular to this 30 degree machined face.
- Before you create the toolpath the Level that contain the vice will be turned off, it will make it a lot easier to select the entities in this next set of instructions.

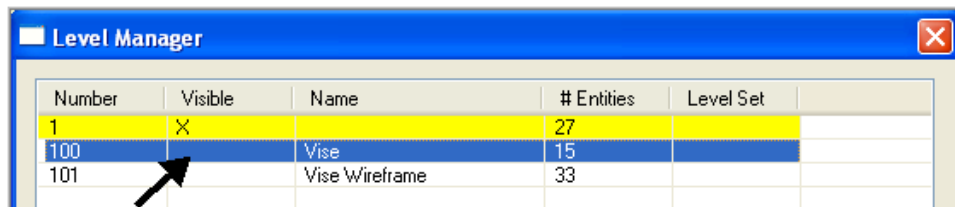
1. From the **Status** bar at the bottom of the screen select **Level**.



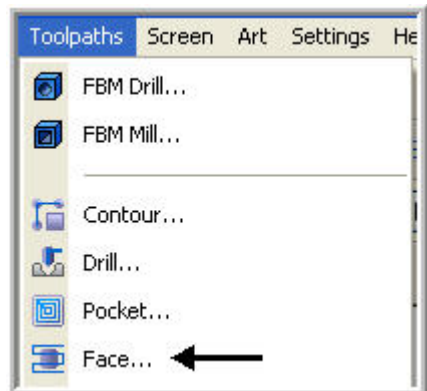
2. The **Level Manager** dialog window will now appear. In the **Visible** column **click on the X for Level Number 100**.



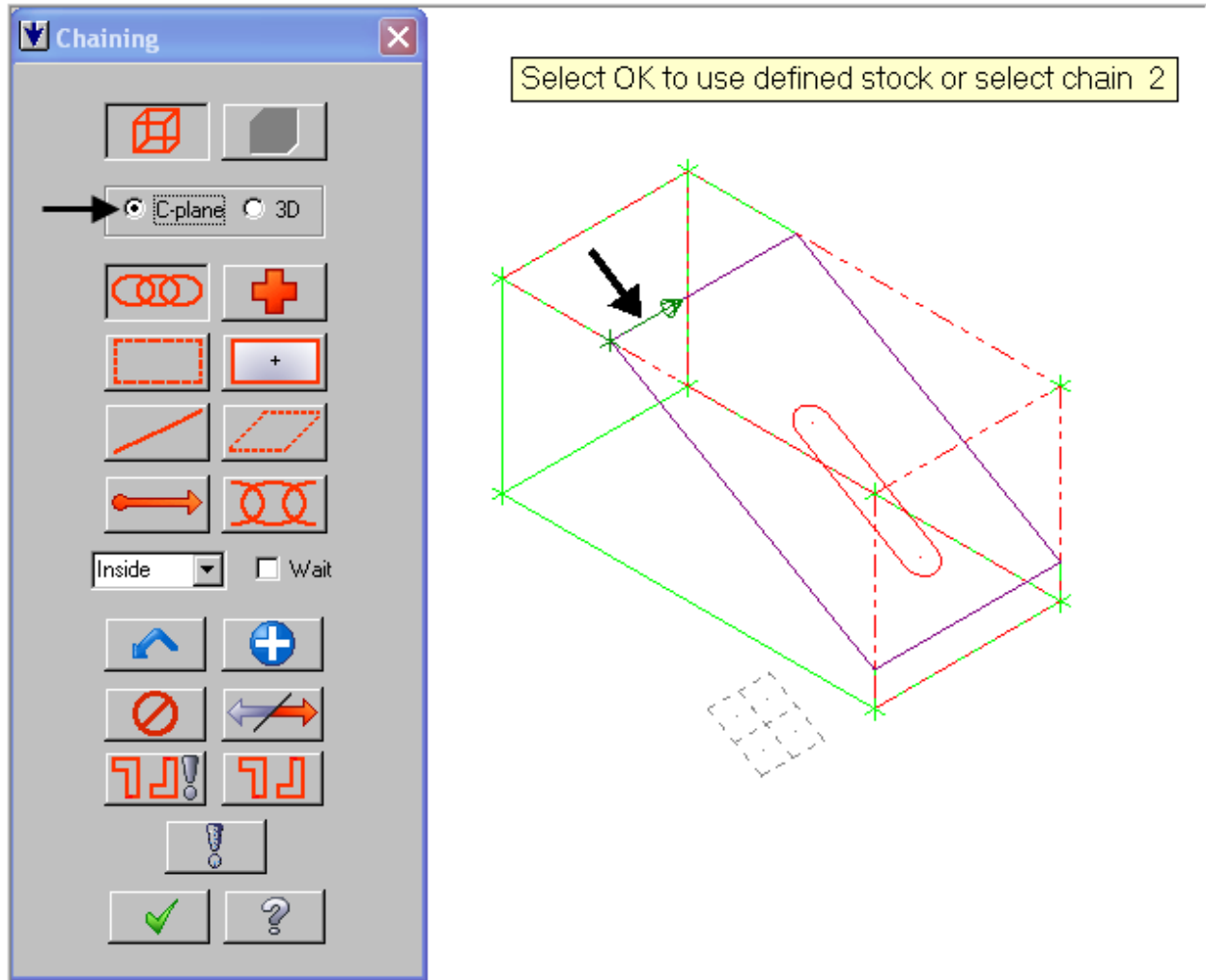
3. Ensure the Level Manager dialog window appears as below, with the **X for Level Number 100 removed**. This makes the entities on that level **invisible**. Click on the OK button when done .



4. From the menu bar select **Toolpaths>Face...**

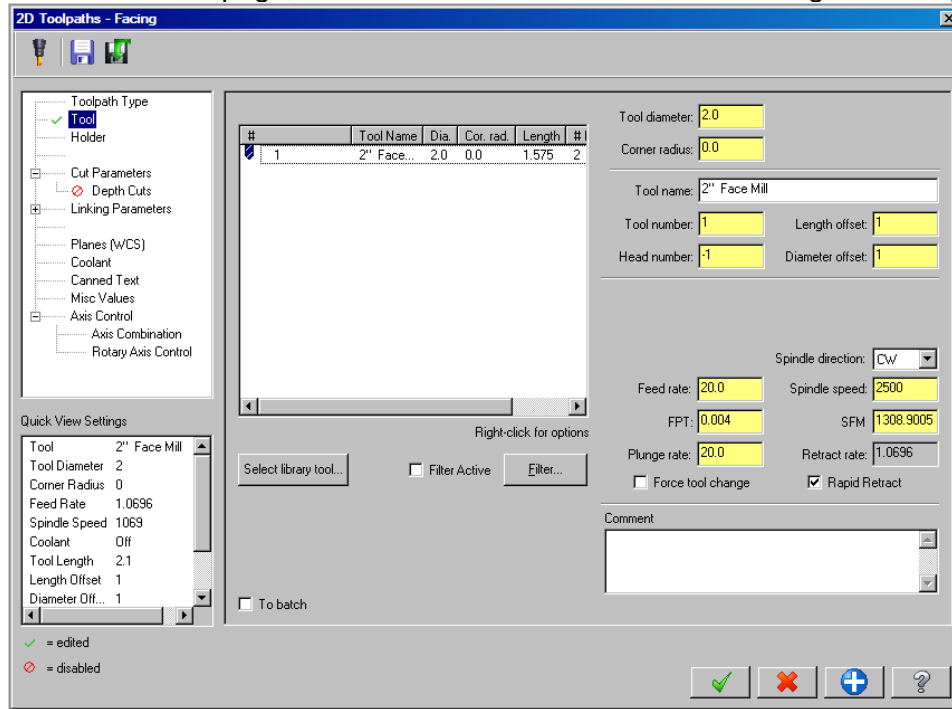


5. If prompted to **Enter new NC name** ensure **Multi-Axis-Lesson-1** is displayed and then select the OK button . If not prompted then continue to the next step.
6. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Select OK to use defined stock or select chain 1.**
7. Activate the **C-plane** radio button.
8. Select the line as shown below to chain the area being faced.

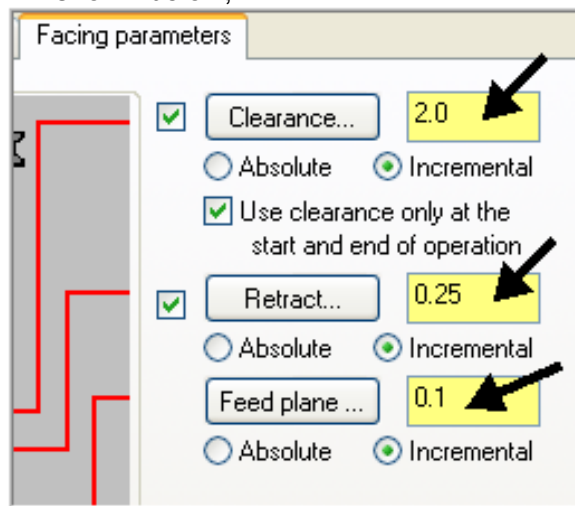


9. Select the OK button to exit the chaining dialog box.

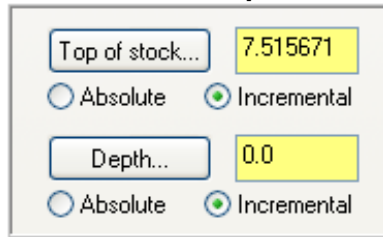
10. On the **Tool** page, select the **2" Face Mill** and make changes to this page as shown below:



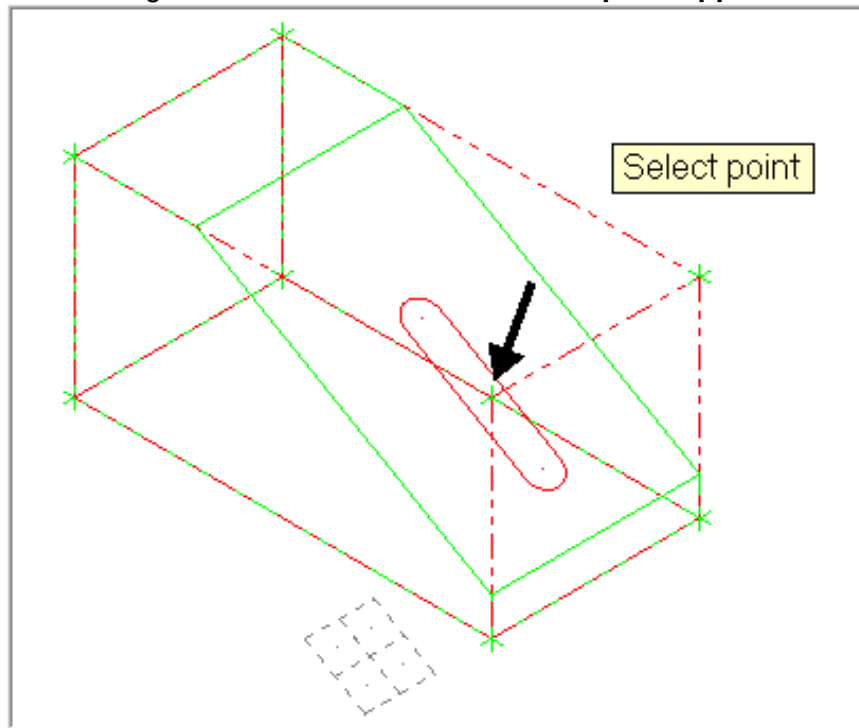
11. Select the **Linking parameters** page and first **activate Clearance** then change the **Clearance, Retract and Feed plane** to **Incremental**. Now set the remaining values as shown below;



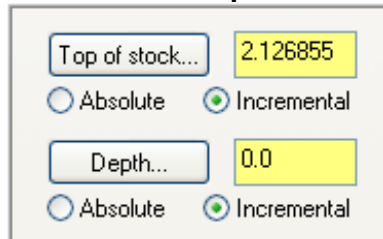
12. Still on the **Facing parameters** page change the **Depth** and **Top of Stock** to **Incremental** and set the **Depth** value to **0**.



13. Now click on the **Top of stock** button. When prompted to **Select point** select the point on the top of the stock as shown below. This is one of the points you created earlier using **Bonding Box**. **Ensure the visual cue for point appears** before you select the point.



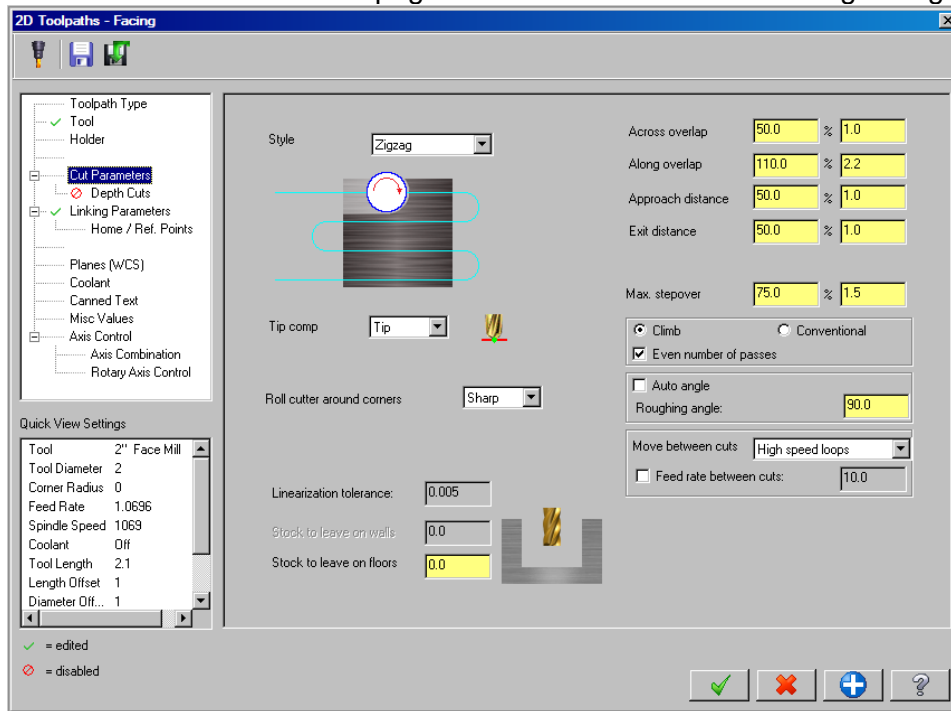
14. Notice the **Top of Stock** has now changed to 2.126855 after picking the point.



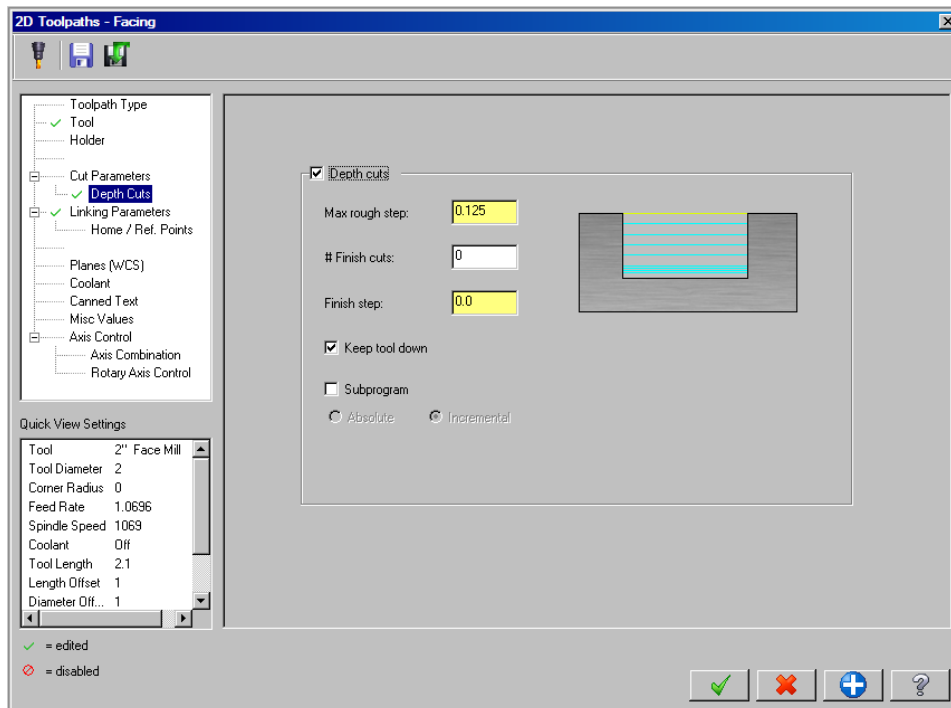
Top of stock

- ☞ Sets the height of the material in the Z axis.
- ☞ Many other toolpath parameters are measured from this position in incremental mode, such as **Clearance**, **Retract**, and **Feed planes**.
- ☞ Choose the **Top of stock** button and select a point on the geometry or enter a value.
- ☞ Incremental values are measured from the chained geometry.

15. On the **Cut Parameters** page review and make the remaining changes as shown below:



16. Select the **Depth Cuts** page and select the **Depth Cuts** button and make changes as shown below:



17. Select the **OK** button  to exit **Depth cuts**.

18. Select the **OK** button  to exit **Face parameters**.


TASK 7:

USE TRIM TOOLPATH TO EDIT THE FACING TOOLPATH

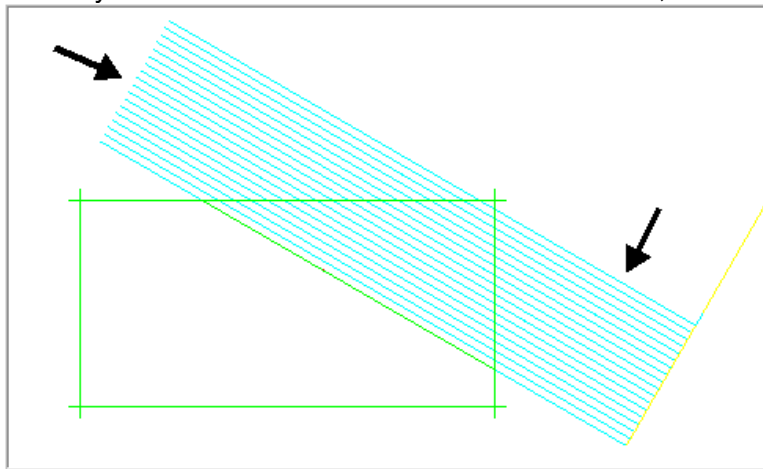
- ☞ In this task you will trim the facing toolpath to remove the fresh air cutting. This Trim function allows you to create a Trim operation, which trims an existing toolpath to one or more closed boundaries that you select.
- ☞ To accomplish this we need to create a trim containment area. One of the points you created earlier in Stock setup will now be used to create a couple of lines for the boundary for this trim operation.

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



2. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen  and zoom as required to display the toolpath as shown below.

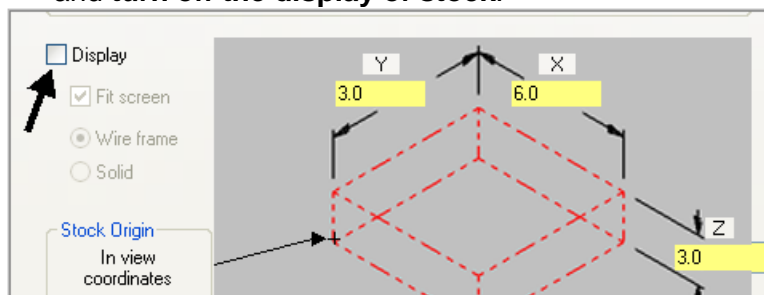
- ☞ As you can see there are lot of fresh air moves, as shown below:



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

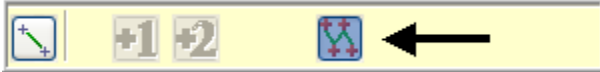


4. Select **Alt-T** on your keyboard to hide the display of toolpaths.
5. To make it easier to select the entities in this next set of instructions move into **Stock Setup** and **turn off the display of stock**.

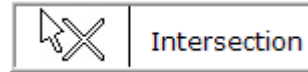
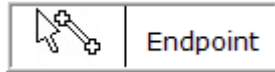


Mastercam Training Guide

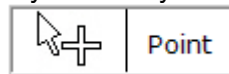
6. Select from the pull down menu: **Create>Line>Endpoint...**
7. Activate **Multi-Line** on the line ribbon bar.



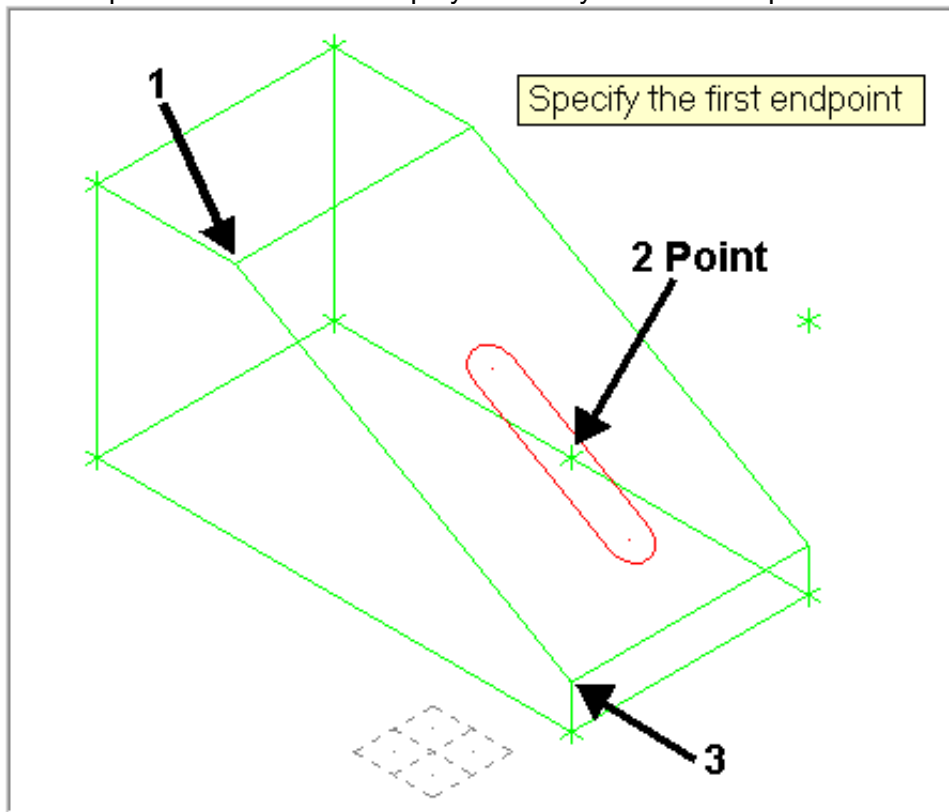
8. When prompted to **Specify the first point** pick at **position 1**. This point is the intersection or endpoint of the lines shown below at **position 1**. Ensure the visual cue for end point or intersection displays before you select the position. **Zoom in as required.**




9. When prompted to **Specify the second point** pick at **position 2**. This point is the **Point** that was created earlier in Stock setup. You may wish to dynamically rotate the display by holding the middle mouse button down and rotating the display to better view the point. Ensure the visual cue for Point displays before you select the position.



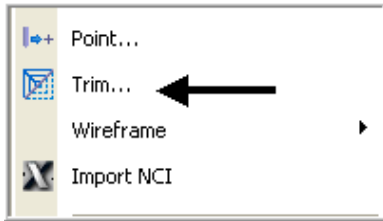
10. When prompted to **Specify the second point** again pick at **position 3**. This point is the intersection or endpoint of the lines shown below at **position 3**. Ensure the visual cue for end point or intersection displays before you select the position.



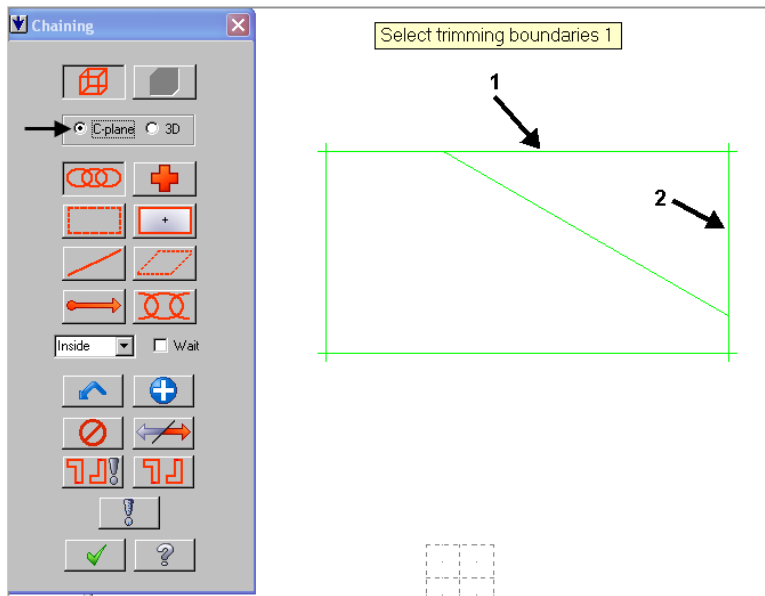
11. Click on the **OK** icon  to complete this feature.
12. Set the graphics view to the **Front View** by using the toolbar icon at the top of the screen.



13. From the menu bar select **Toolpaths>Trim...**

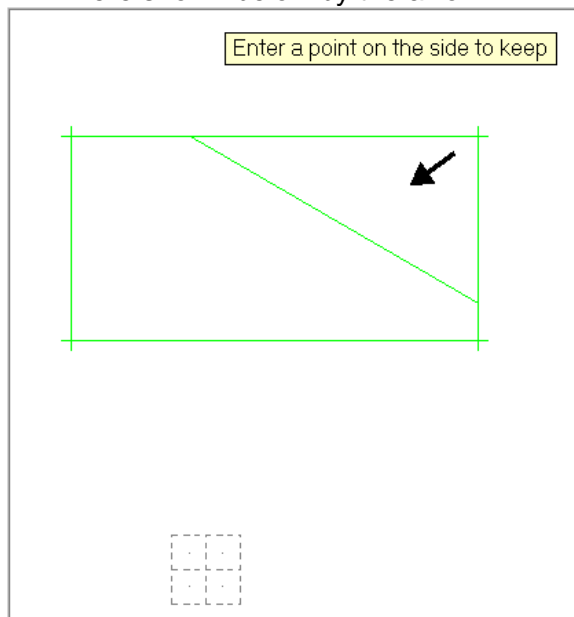


14. In the Chaining dialog window **activate Cplane** and ensure **Chain** is set. When prompted to **Select trimming boundaries 1** – select the line created earlier at **Position 1**. This will chain the two lines at **Position 1 and 2** created earlier.

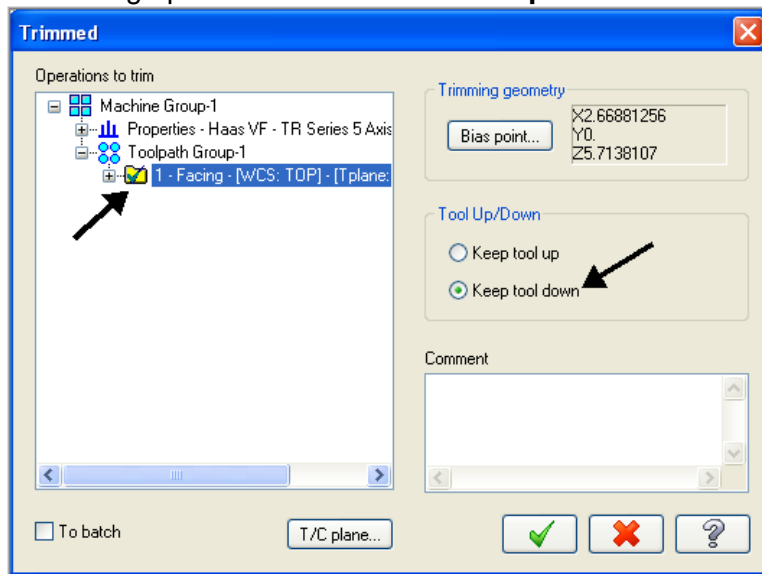



15. Select the OK button  to exit Chaining.

16. When next prompted to **Enter a point on the side to keep** pick a point approximately where shown below by the arrow.

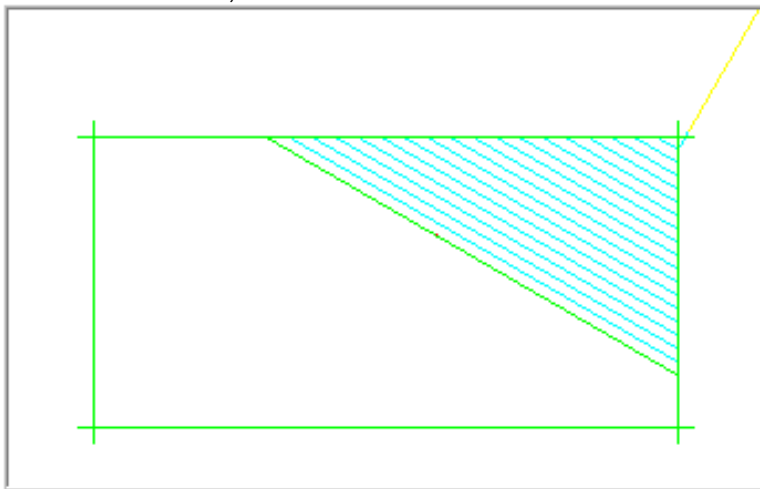


17. After picking this point the **Trimmed** dialog window appears. Select the **Facing** operation from the list of **Operations to trim** on the left. Ensure you have a check mark on this Facing operation. Next activate **Keep tool down**.



18. Select the OK button  to exit the Trimmed dialog window.

19. Select **Alt-T** on your keyboard to **display toolpaths**. The Trimmed facing toolpath should look as below, with the fresh air moves removed

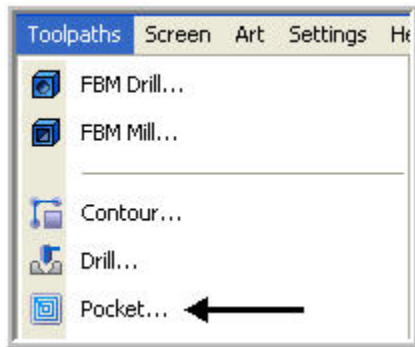


Mastercam Training Guide

6. At the bottom of the graphics screen you will now see **T/Cplane:VIEW-30-DEGREES**.



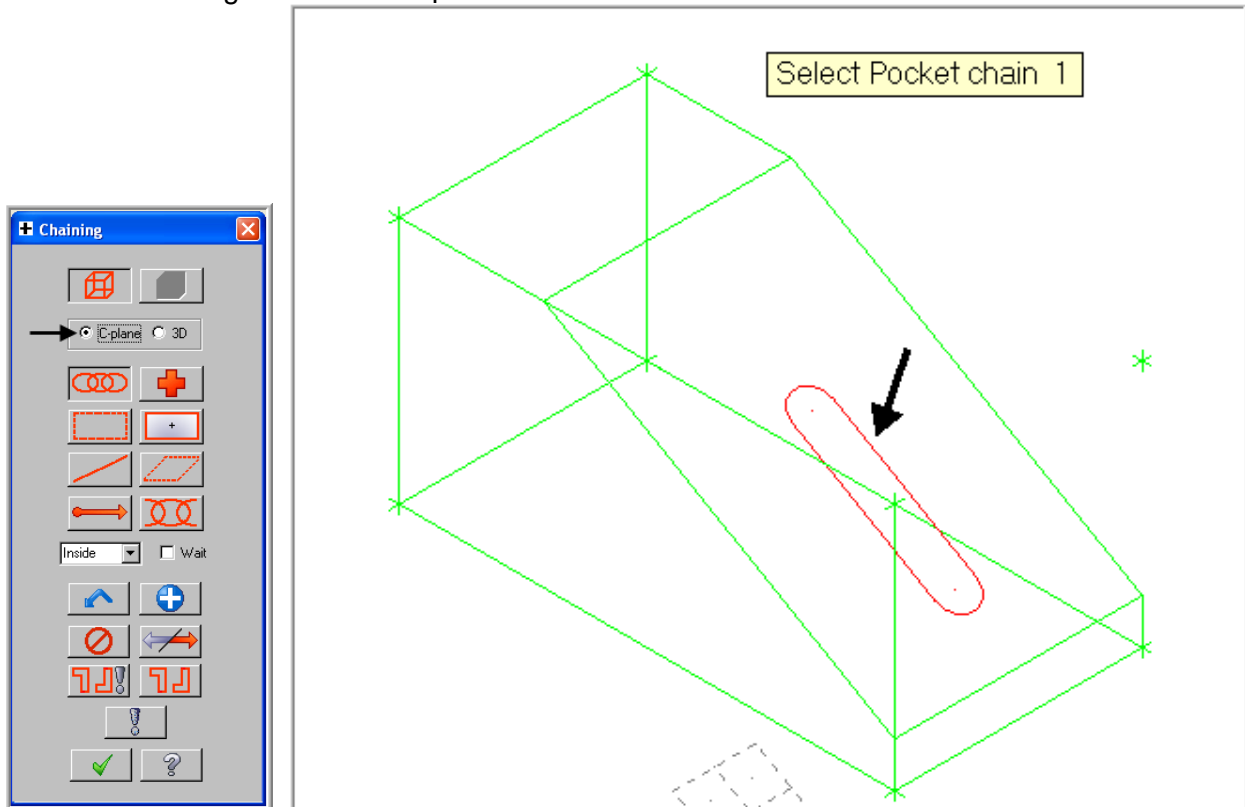
7. From the menu bar select **Toolpaths>Pocket...**



8. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Select Pocket chain 1**.


9. Activate the **C-plane** radio button in the **Chaining dialog box**.

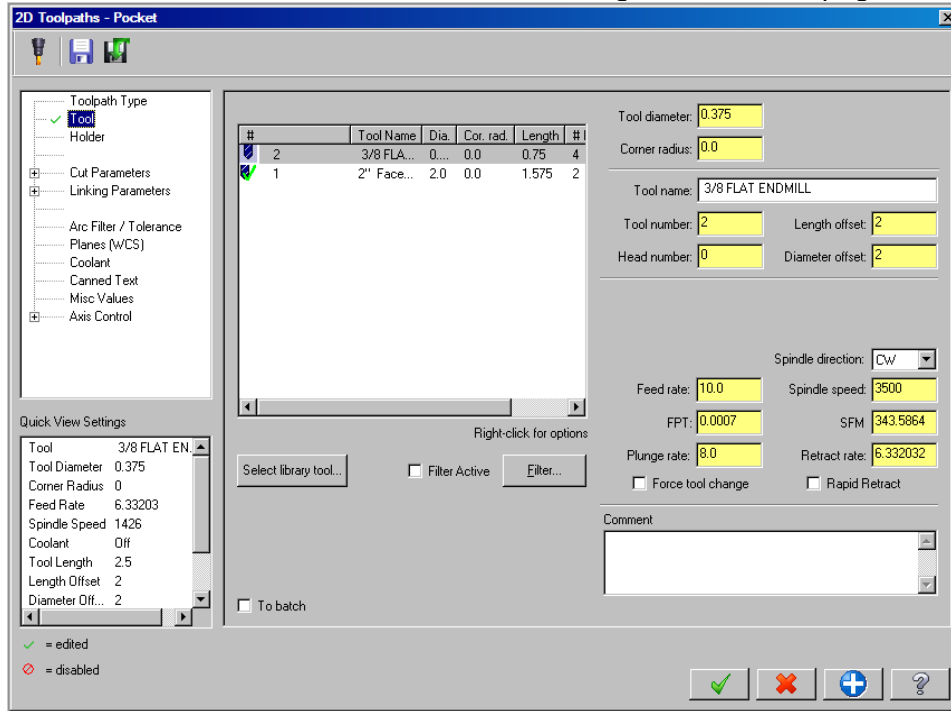
10. Select the angled line for the pocket as shown below to chain the slot.



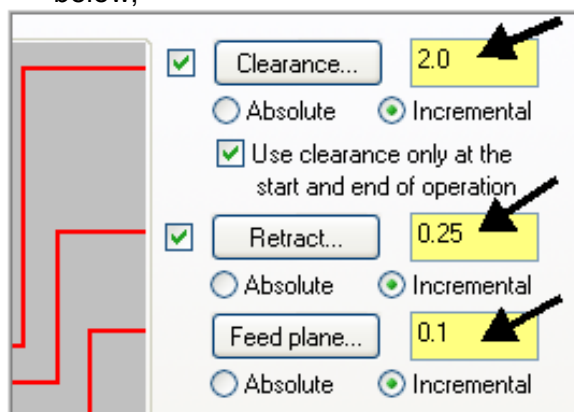
11. After selecting the line the **arrows should be pointing upwards**, if not see below.
12. If the arrow is not pointing upwards select the **arrow** from the Chaining dialog box shown below to reverse the direction.



13. After the pocket has been successfully chained select the **OK** button  at the bottom of the Chaining dialog box.
14. Select **3/8 FLAT ENDMILL** and make changes to the **Tool** page as shown below:



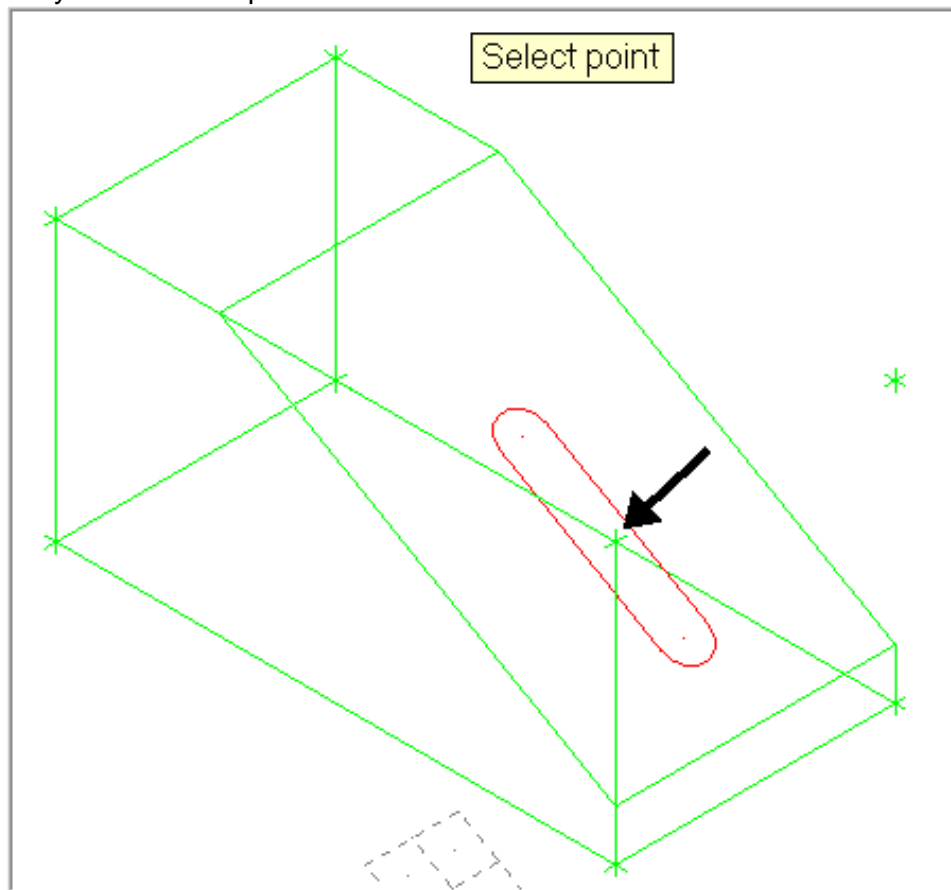
15. Select the **Linking Parameters** page and if required **activate Clearance** then change the **Clearance, Retract and Feed plane** to **Incremental** and set the remaining values as shown below;



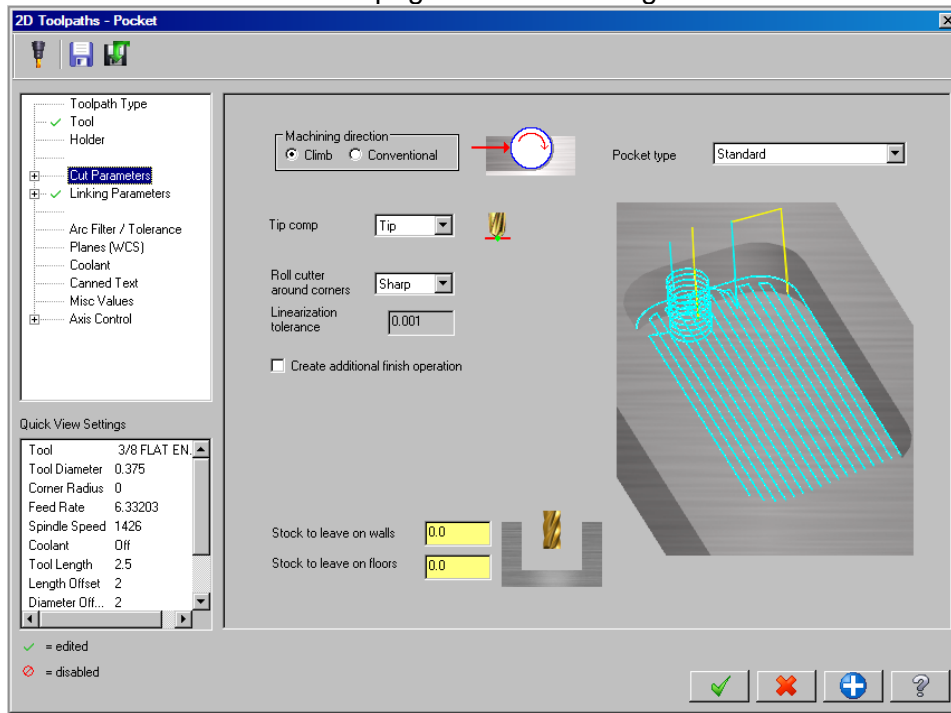
16. Still on the **Linking Parameters** page change the **Depth and Top of Stock** to **Incremental** and set the **Depth** to **-0.125**.

Top of stock...	2.126855
<input type="radio"/> Absolute	<input checked="" type="radio"/> Incremental
Depth...	-0.125
<input type="radio"/> Absolute	<input checked="" type="radio"/> Incremental

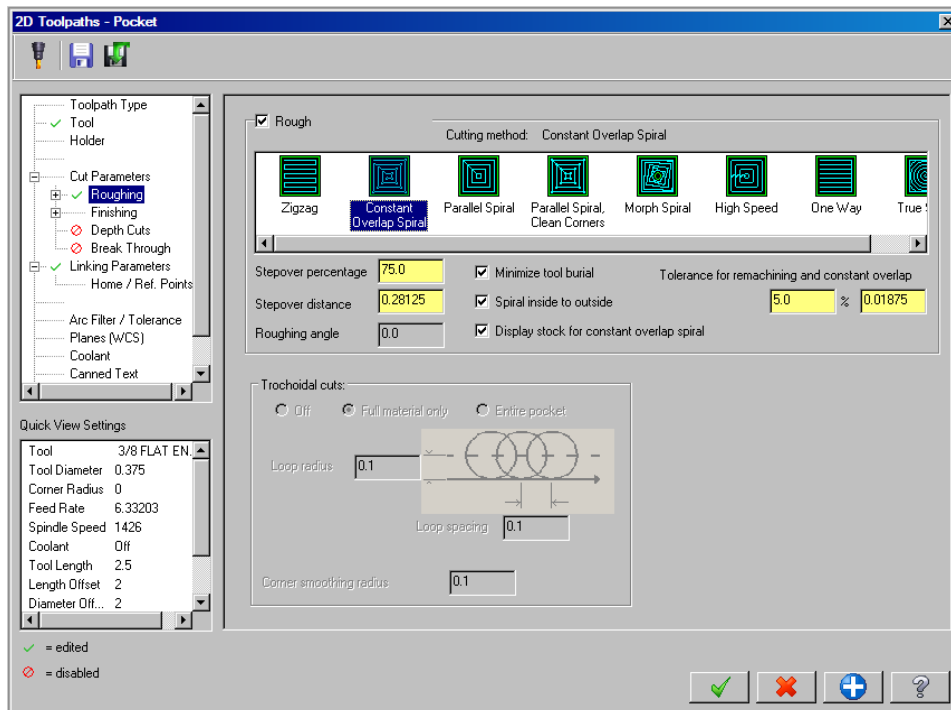
17. Now click on the **Top of stock** button. When prompted to **Select point** select the point on the top of the stock as shown below. This is one of the points you created earlier using Bonding Box. **Ensure the visual cue for end point, intersection or point appears** before you select this point.



18. On the **Cut Parameters** page make the changes as shown below:

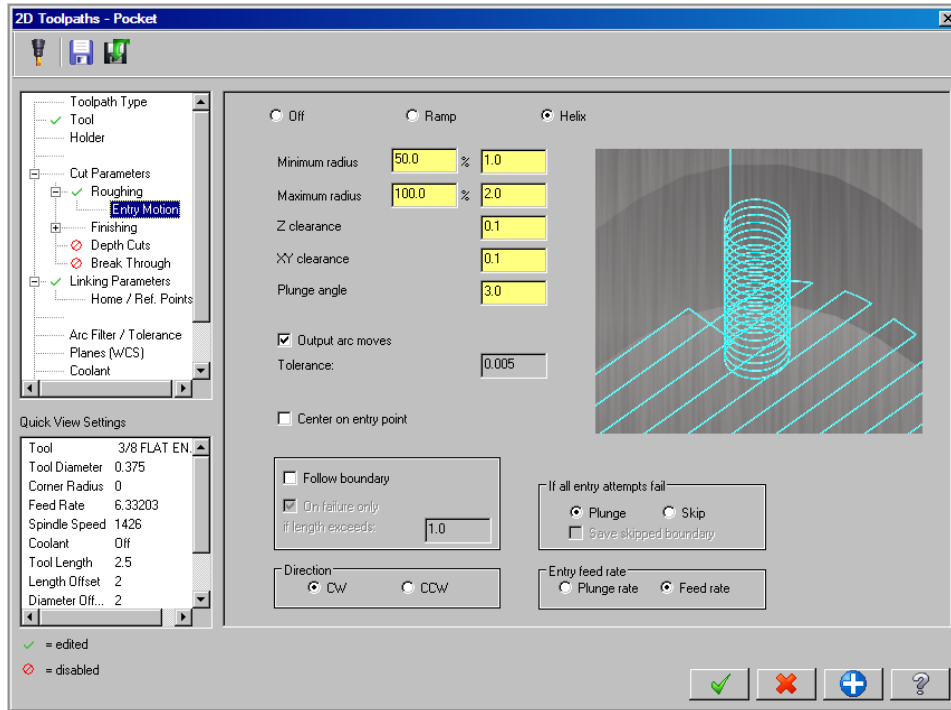


19. Expand the **Cut Parameters** menu and select the **Roughing** page. Make the changes as shown below:

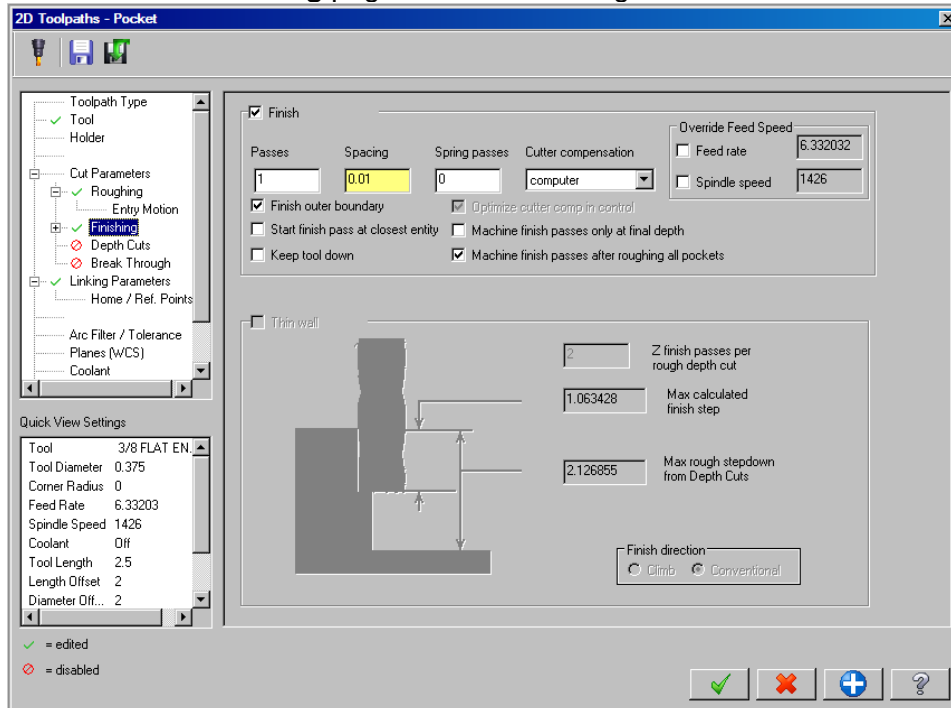


Mastercam Training Guide

20. Expand the **Roughing** menu and select the **Entry Motion** page. Make the changes as shown below:



21. Select the **Finishing** page. Make the changes as shown below:



22. Select the **OK** button  to exit Pocket parameters.

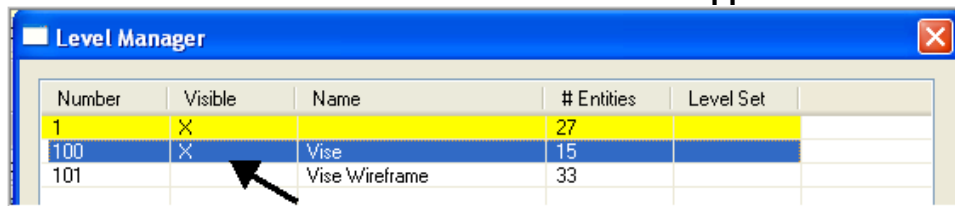
TASK 9: BACKPLOT THE TOOLPATH

- ☞ In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.
- ☞ Backplot will enable you to review the cutting motions and identify any problem areas when cutting the part.

1. Now you will make the **Level visible** that contains the geometry for the vise. From the **Status** bar at the bottom of the screen select **Level**.



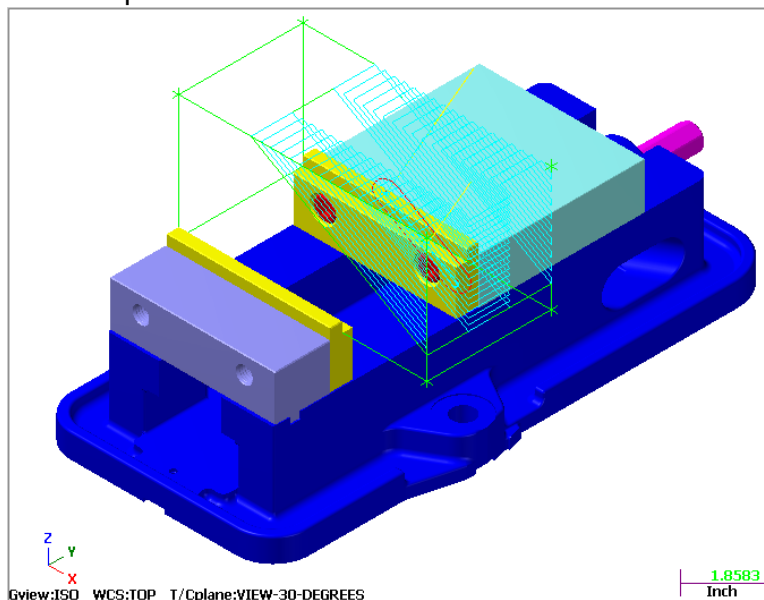
2. The **Level Manager** dialog window will now appear. In the **Visible** column click on the **visible** column for Level Number 100 so the X appears to make this level visible.



3. Click on the OK button when done .
4. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .
5. Select **Alt-T** on your keyboard to **display toolpaths**.
6. Activate a shaded view by selecting the icon at the top of the screen.

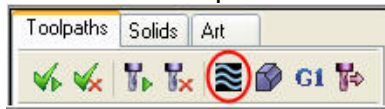


- ☞ Your part should look similar to the screen shot below.

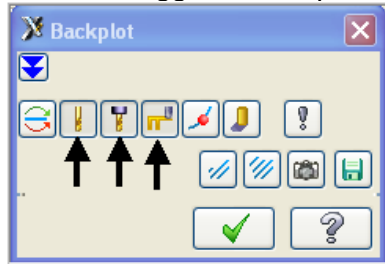


7. To pick all the operations to backplot pick the **Select All** icon  circled below:

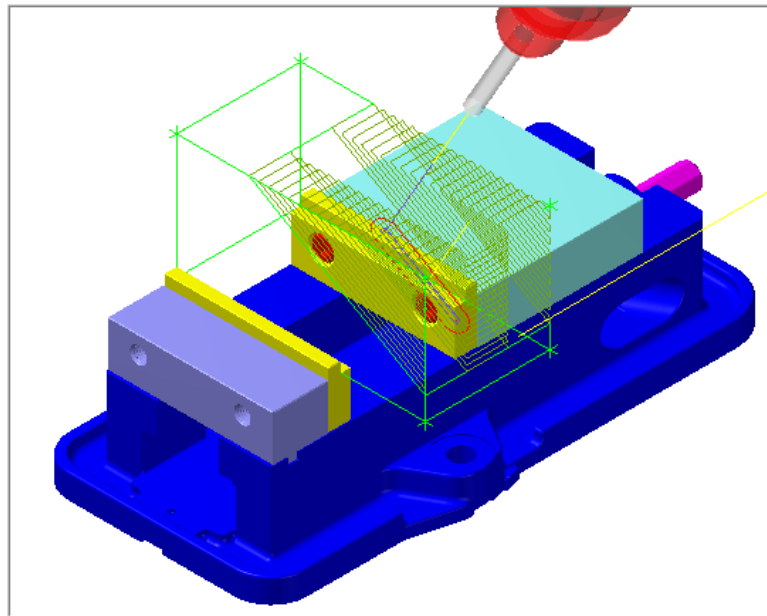
8. The next step is to select the **Backplot selected operations** icon shown below:



9. Before you Backplot the toolpath ensure the three icons shown below are activated. The options will **Display Tool**, **Display Holder** and **Display rapid moves**. These buttons act like a toggle switch, pressed in activates the function.




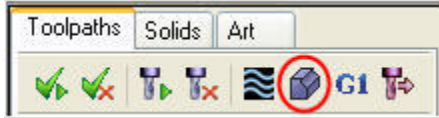
10. Set the **run speed** on the Backplot VCR midway along the speed bar as shown by the arrow below and then select the **play** button.



11. After reviewing the backplot of the two toolpaths select the **OK** button  to exit Backplot.

TASK 10: VERIFY THE TOOLPATHS

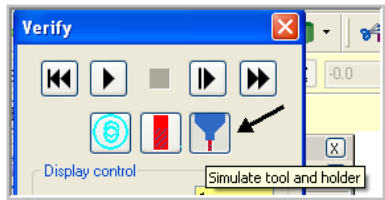
1. In the Toolpath Manager pick all the operations to verify by picking the Select All icon .
2. Select the Verify selected operations button circled below:



3. Adjust the Verify speed to midway along the speed control bar.



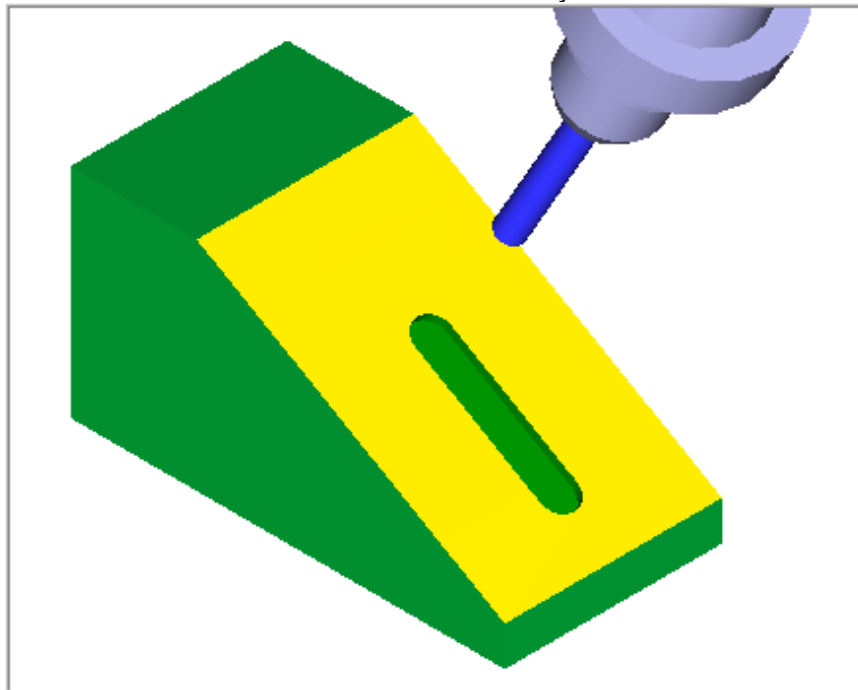
4. Activate the Simulate tool and holder option as shown below.




5. Select the play button to verify the toolpath.

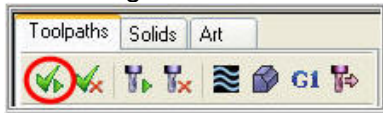


6. Select the OK button  to exit Verify.

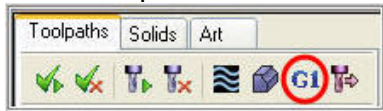


TASK 11: POST AND CREATE THE CNC CODE FILE

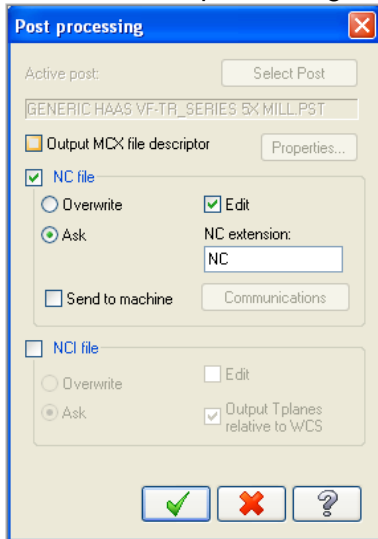
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.



2. Select the **Post selected operations** button from the Toolpath manager.
3. **Please Note:** If you cannot see **G1** click on the right pane of the Toolpath manger window and expand the window to the right.

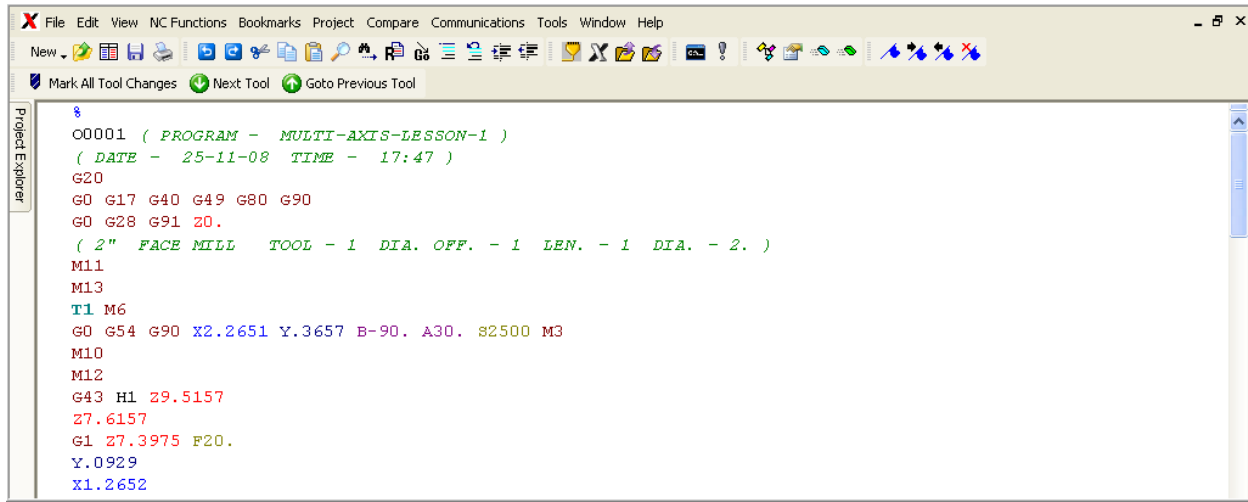


4. In the Post processing window, make the necessary changes as shown below:




5. Select the OK button  to continue.

6. Enter the same name as your Mastercam part file name in the NC File name field **MULTI-AXIS-LESSON-1**.
7. Select the **Save** button.

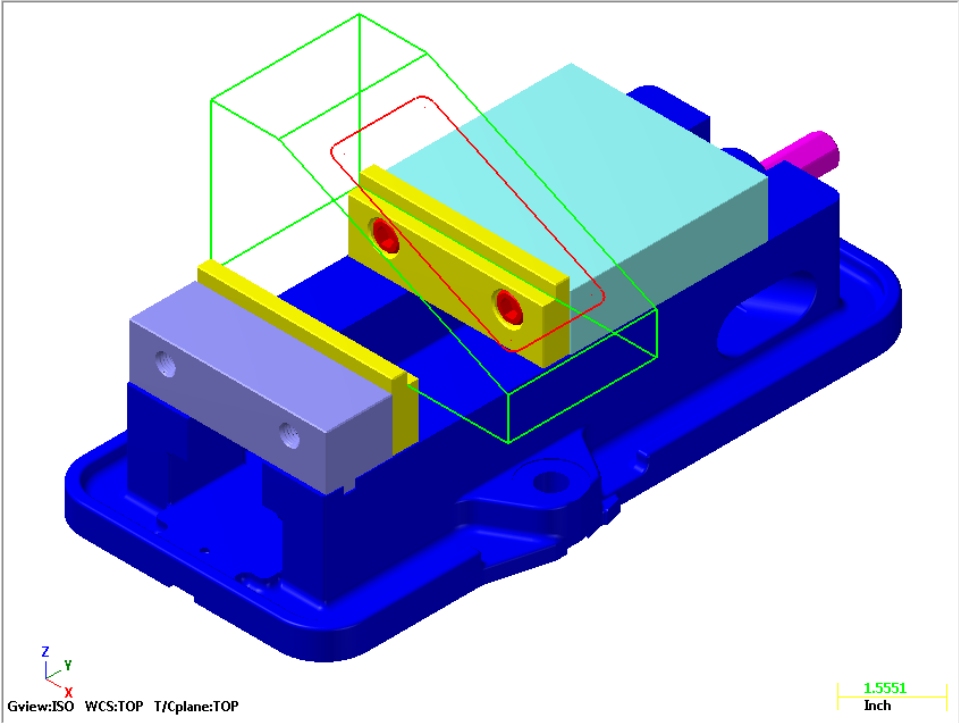


The screenshot shows the Mastercam CNC editor window. The menu bar includes File, Edit, View, NC Functions, Bookmarks, Project, Compare, Communications, Tools, Window, and Help. The toolbar contains various icons for file operations and tool management. Below the toolbar, there are buttons for 'Mark All Tool Changes', 'Next Tool', and 'Goto Previous Tool'. The main text area displays the following G-code program:

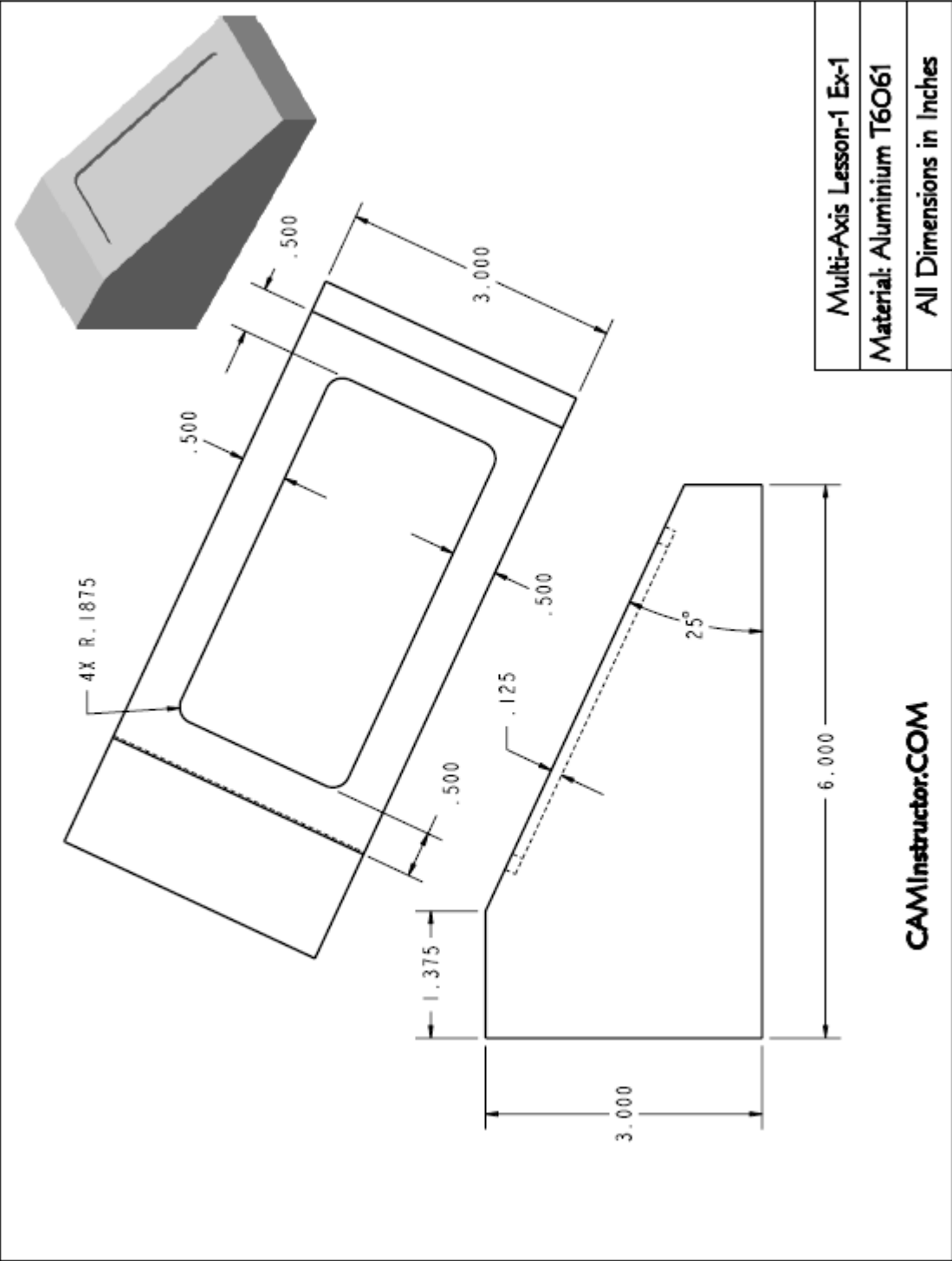
```
O0001 ( PROGRAM - MULTI-AXIS-LESSON-1 )
( DATE - 25-11-08 TIME - 17:47 )
G20
G0 G17 G40 G49 G80 G90
G0 G28 G91 Z0.
( 2" FACE MILL TOOL - 1 DIA. OFF. - 1 LEN. - 1 DIA. - 2. )
M11
M13
T1 M6
G0 G54 G90 X2.2651 Y.3657 B-90. A30. S2500 M3
M10
M12
G43 H1 Z9.5157
Z7.6157
G1 Z7.3975 F20.
Y.0929
X1.2652
```

8. Select the  in the top right corner to exit the CNC editor.
9. This completes **Multi-Axis-Lesson-1**.

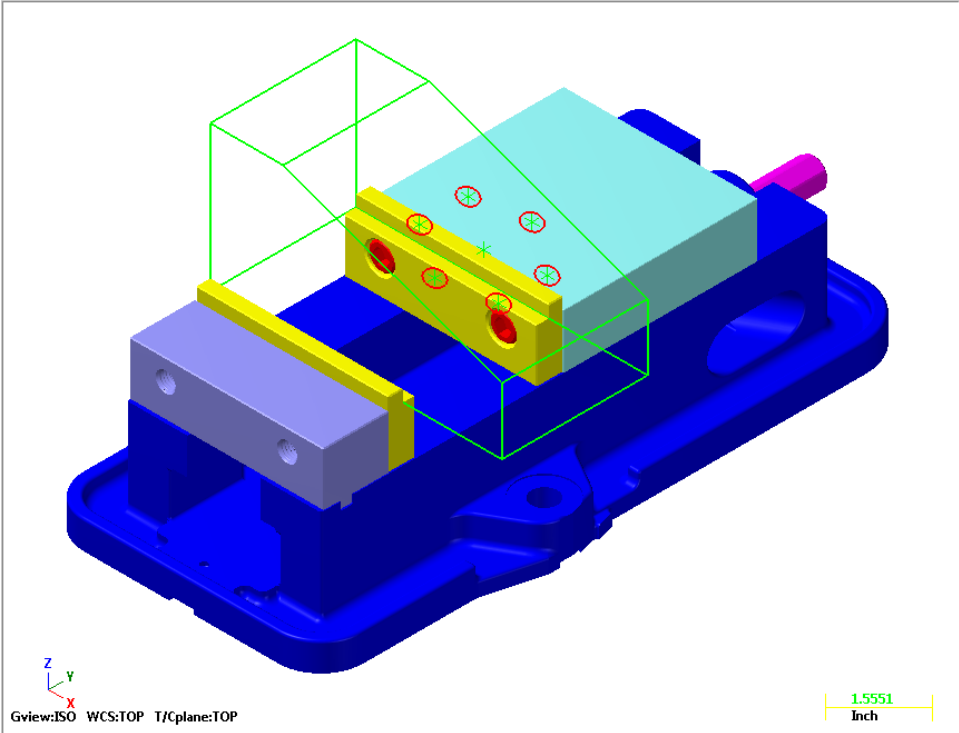
MULTI-AXIS-LESSON-1 EXERCISE #1
MULTI-AXIS-LESSON-1-EX-1.MCX



MULTI-AXIS-LESSON-1 EXERCISE #1



MULTI-AXIS-LESSON-1 EXERCISE #2
MULTI-AXIS-LESSON-1-EX-2.MCX



MULTI-AXIS-LESSON-1 EXERCISE #2

