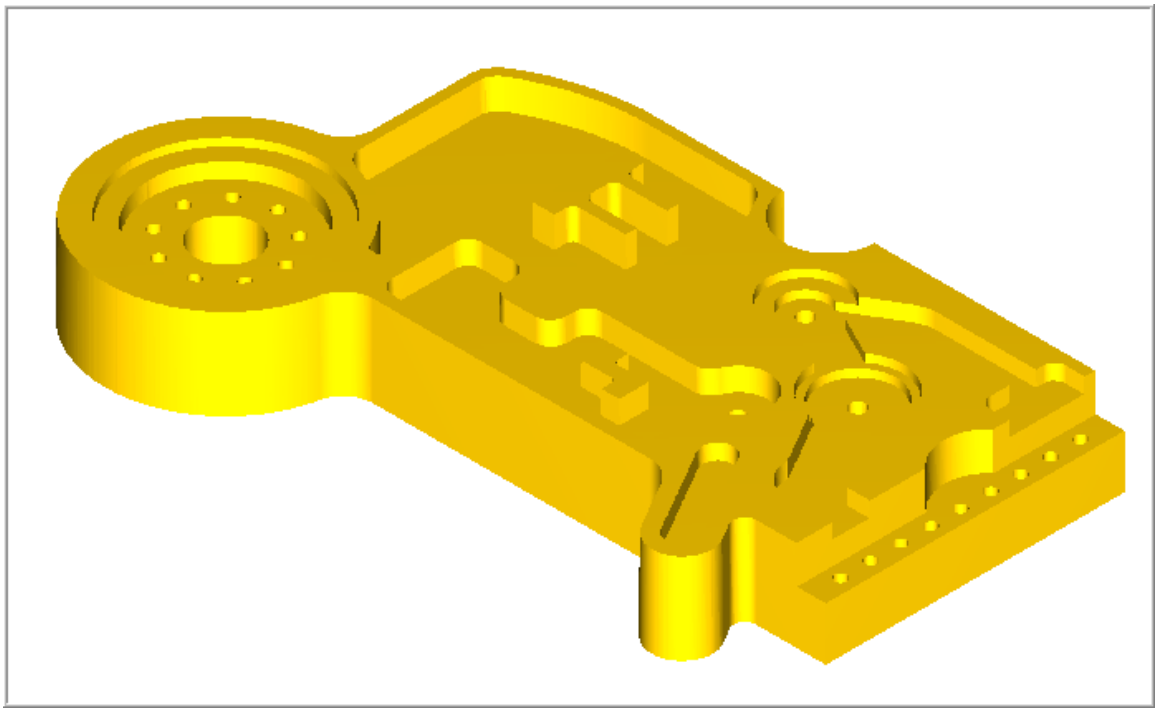
Mastercam 2017

TRAINING GUIDE



MILL-LESSON-FBM-2

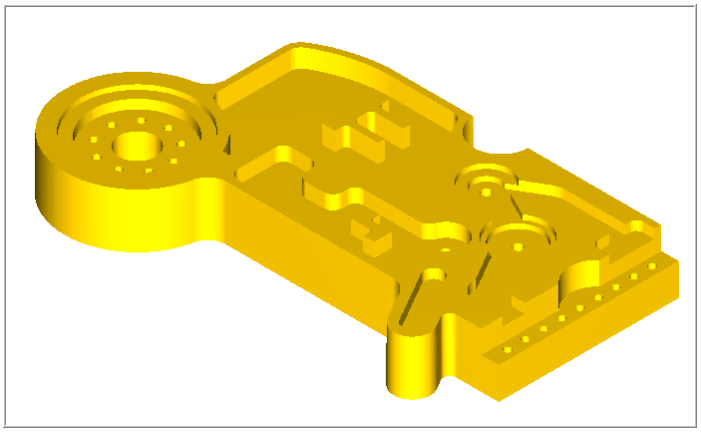
FBM MILL AND FBM DRILL

camInstructor

Objectives

This lesson will use the same Feature Based Machining (FBM) methods used in Mill-Lesson-FBM-1; however, this time applying FBM to much more complex geometry. Using similar configurations techniques as the previous lesson, you will see advantages in programming time using the FBM toolpaths.

➲ **Feature Based Machining (FBM)**
FBM eliminates manual feature identification for programming milling and drilling operations on solid parts. The FBM Mill and FBM Drill operations automatically create the toolpaths needed to machine features that are identified using your specific criteria. FBM's interface is simple and intuitive.



You will generate the toolpaths for Mill-Lesson-FBM-2 to machine the part on a CNC vertical milling machine.

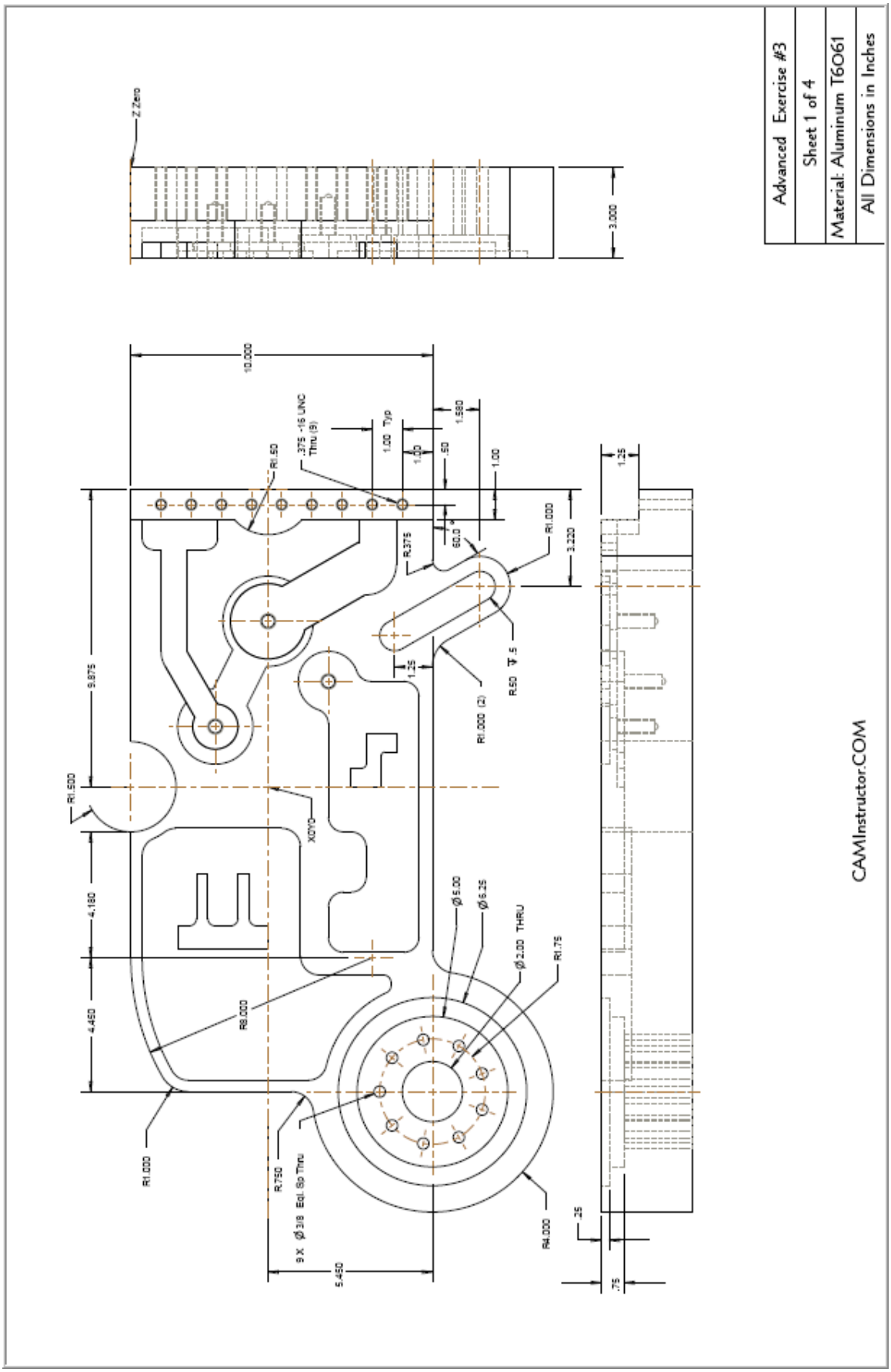
This Lesson covers the following topics:

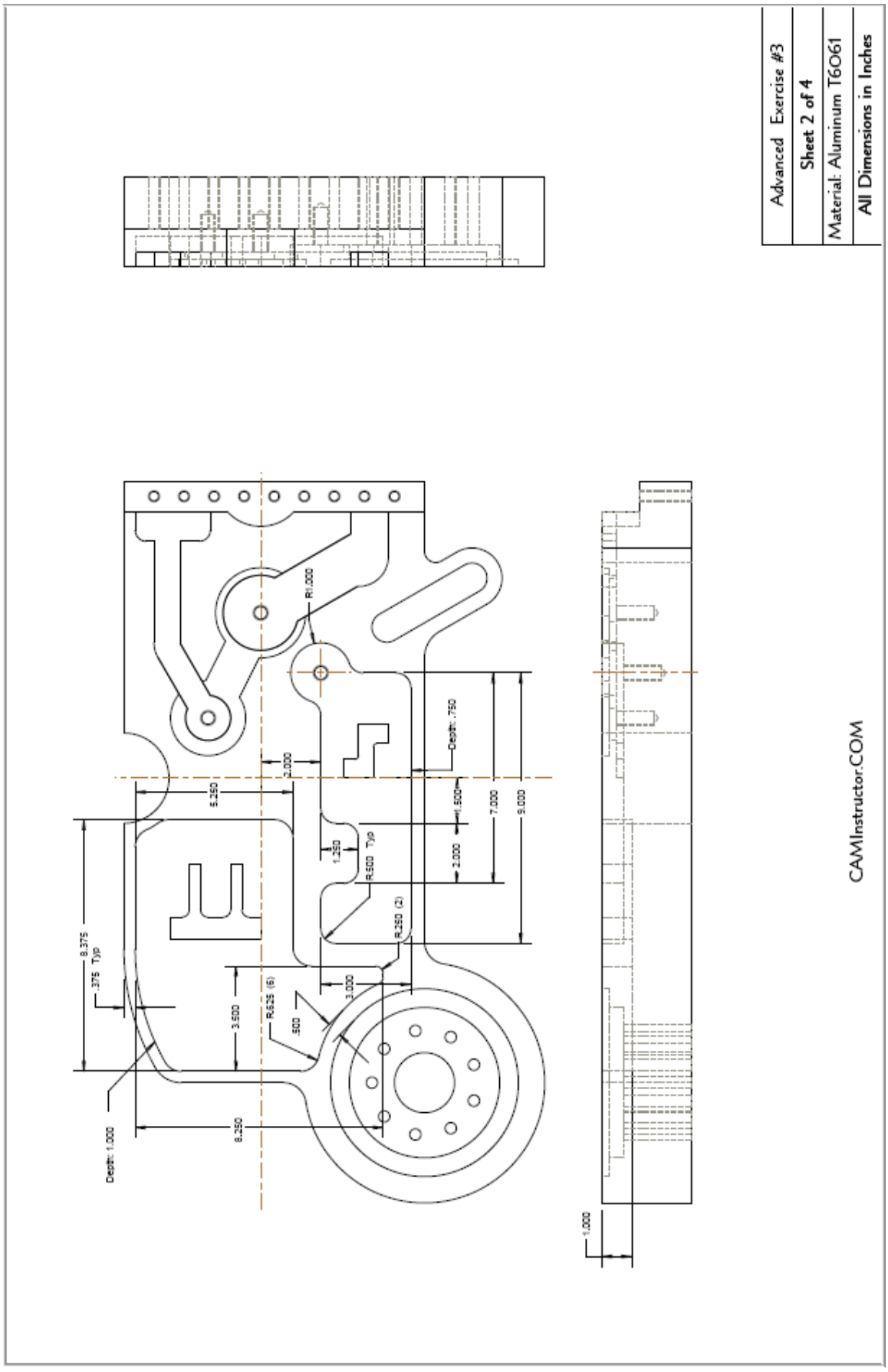
➲ **Establish Stock Setup settings:**
Stock size.
Material for the part.
Feed calculation.

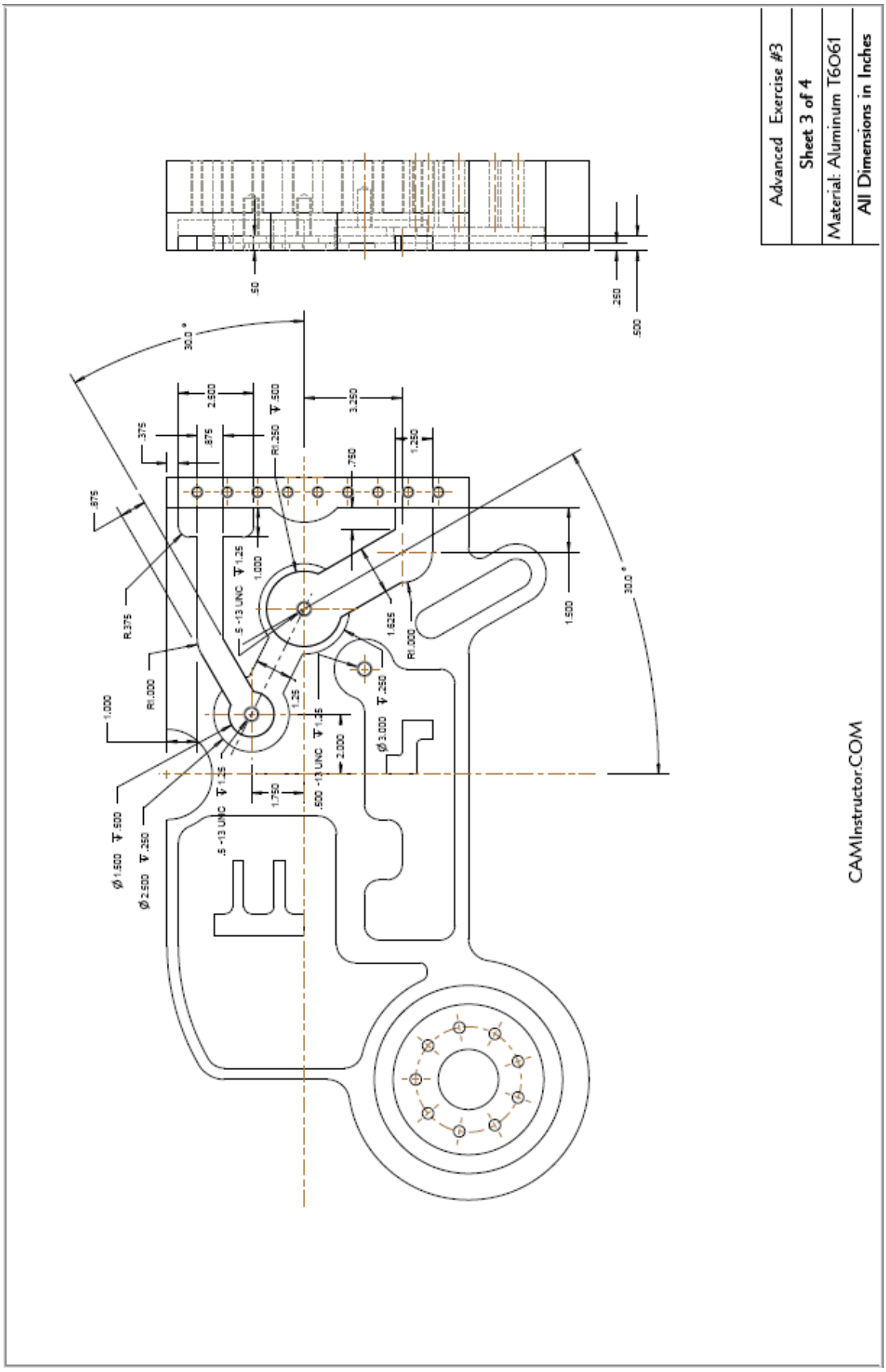
➲ **Generate a 2-dimensional milling toolpath consisting of:**
FBM Mill operation and child toolpath operations.
FBM Drill operation and child toolpath operations.

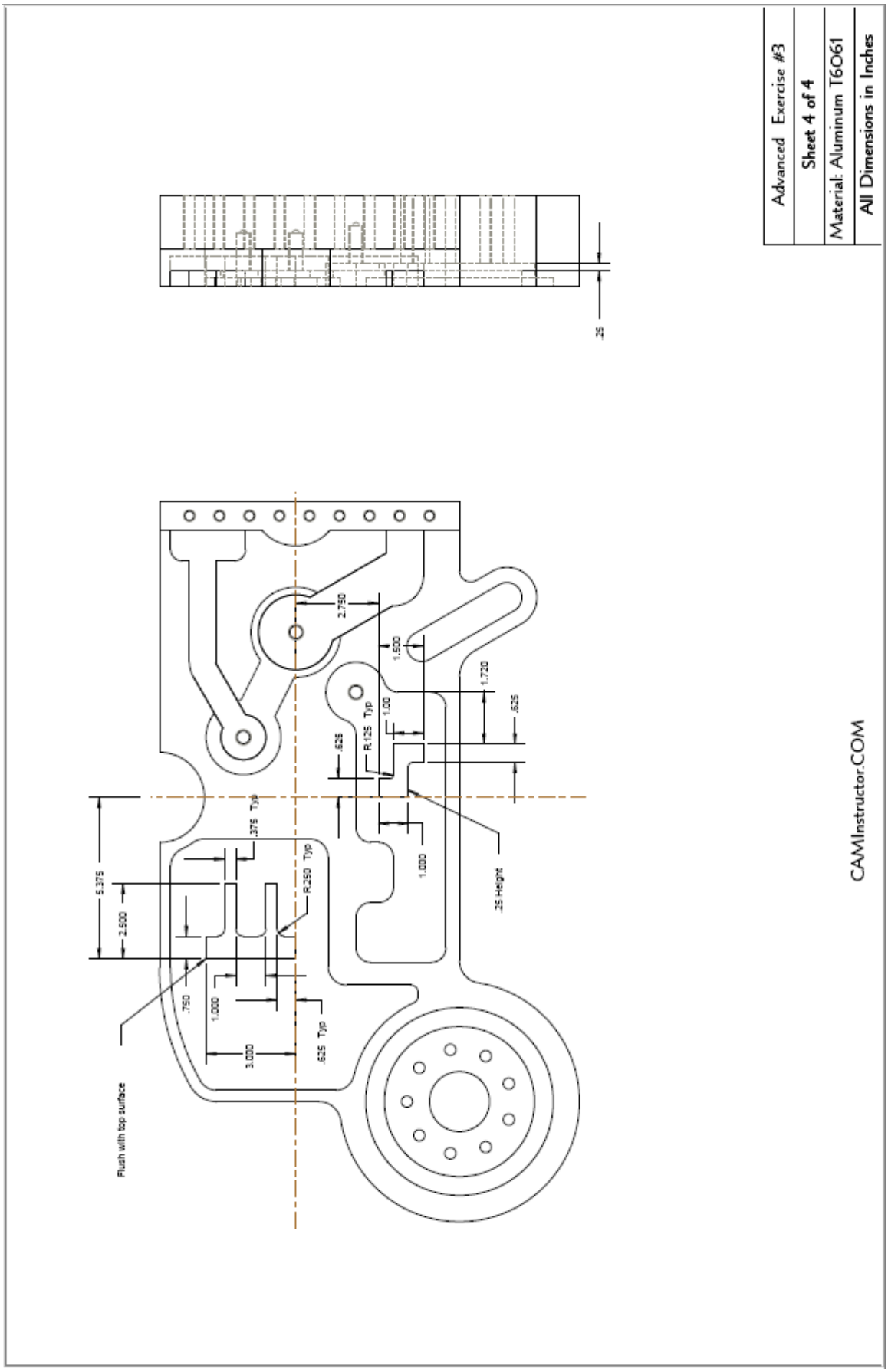
➲ **Inspect the toolpath using Mastercam's Verify by:**
Launching the Verify function to machine the part on the screen.
Saving the FBM Mill results as an in-process STL in Verify.
Applying the FBM Drill results to the in-process stock definition.
Comparing the machined stock to the part geometry.
Generating the NC- code.

MILL-LESSON-FBM-2 DRAWINGS




Advanced Exercise #3
Sheet 2 of 4
Material: Aluminum T6061
All Dimensions in Inches
CAMInstructor.COM
R1.000
Depth: .750
2.000
5.250
1.350
R.500 Typ
.500
2.000
7.000
9.000
R.250 (2)
8.375
.375 Typ
3.500
R.625 (6)
3.000
.500
Depth: 1.000
8.250
1.000

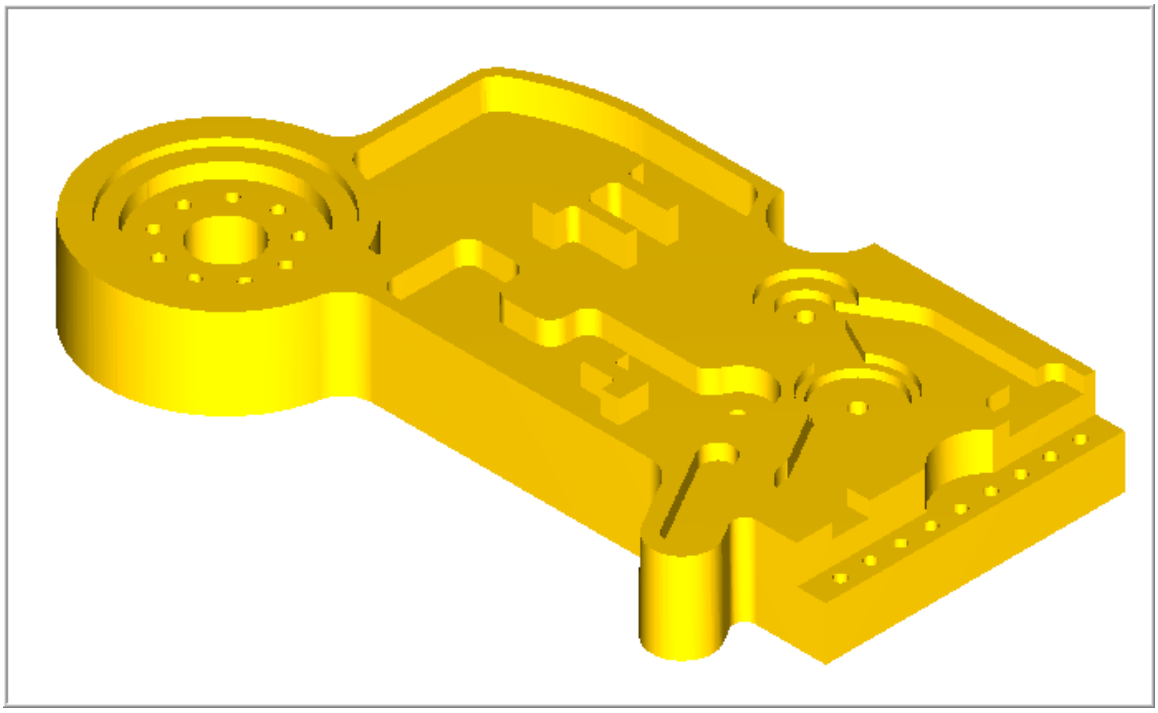

Advanced Exercise #3
Sheet 3 of 4
Material: Aluminum T6061
All Dimensions in Inches
CAMInstructor.COM


Advanced Exercise #3
Sheet 4 of 4
Material: Aluminum T6061
All Dimensions in Inches
CAMInstructor.COM
.25
2.750
1.500
1.720
.625
1.00
R.125 Typ
.625
1.000
.25 Height
.375 Typ
R.250 Typ
5.375
2.500
.750
1.000
3.000
.625 Typ
Flush with top surface

TOOL LIST

- **6.0” face mill**
- **1.0” diameter flat end mill**
- **0.75” diameter flat end mill**
- **0.625” diameter flat end mill**
- **0.5” diameter flat end mill**
- **0.375” diameter flat end mill**
- **0.25” diameter flat end mill**
- **0.125” diameter flat end mill**

- **0.5” spot drill**
- **1/2”-13 UNC tap**
- **0.4219” drill**
- **0.375” drill**
- **3/8”-16 tap**
- **0.3125” drill**



MILL-LESSON-FBM-2 - THE PROCESS

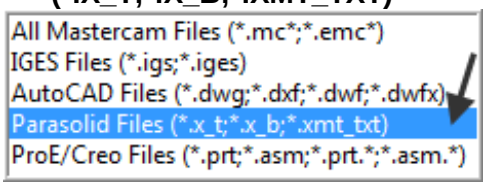
Toolpath Creation

TASK 1: Import part geometry from Parasolid file
TASK 2: Setting the environment
TASK 3: Save part geometry for later inspection
TASK 4: Define the rough stock using stock setup
TASK 5: Run FBM Mill for all milling operations
TASK 6: Verify the toolpath, comparing results to part geometry, and saving out in-process stock
TASK 7: Run FBM Drill for all hole making operations
TASK 8: Verify the toolpath using saved stock
TASK 9: Save the updated Mastercam file
TASK 10: Post and create the CNC code file

TASK 1:
IMPORT PART GEOMETRY FROM PARASOLID

➲ In this lesson you will use an imported solid model.

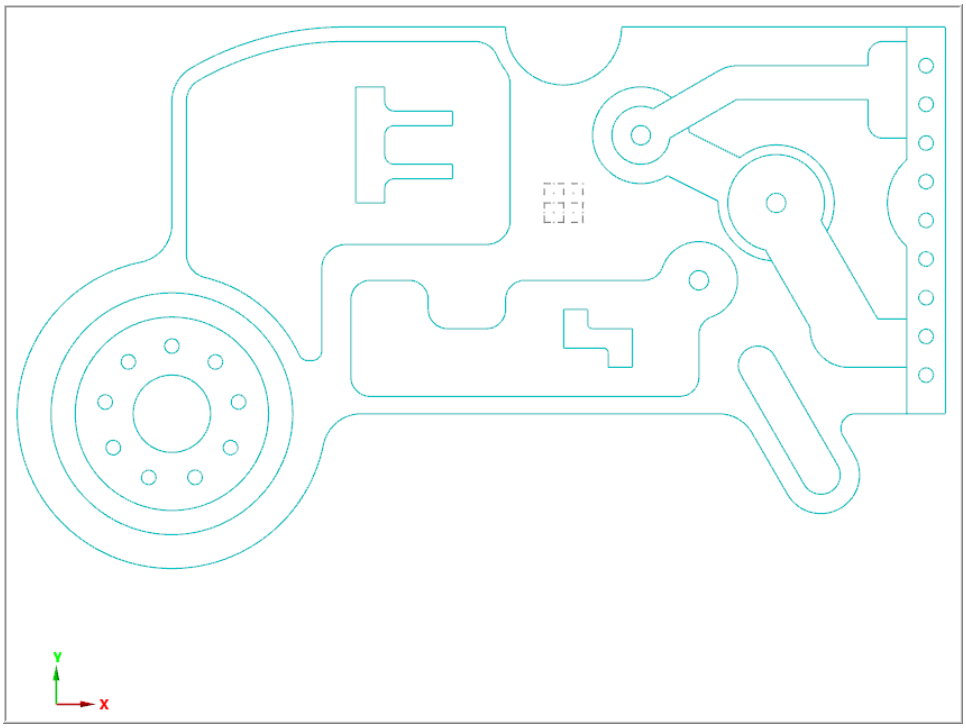
1. Select the **Open** icon from the **Quick Access** toolbar at the top left of the screen.
2. Beside **Files of Type** click on the drop down arrow and select **Parasolid Files (*.X_T;*.X_B;*.XMT_TXT)**



3. On the multimedia DVD that came with this text is **a folder called Mastercam-Files** browse to this location and select **MILL-LESSON-FBM-2.X_T.** If you are an on-line user you will find the file in this lesson.

4. Select the **Open** button to load the file and complete this function.
5. In the **View** tab click on **Outline Shaded** in the **Appearance** section to display the part in shaded mode.
6. For a better view of the part right mouse click in the graphics area and click on **Isometric**.
7. Right mouse click in the graphics area and click on the **Fit** icon  .
8. In the **View** tab click on **Wireframe** in the **Appearance** section.
9. Right mouse click in the graphics area and click on **Top.**
10. Right mouse click in the graphics area and click on the **Fit** icon  .

➲ The screen should look like the one below:



11. Select **File**
12. Select **Save As.**
13. Click on the **Browse** icon.
14. In the File name box, type **Mill-Lesson-FBM-2.**
15. Browse to an appropriate location**.**
16. Select the Save button to save the file and complete this function.

TASK 2:
SETTING THE ENVIRONMENT

➲ Before starting the geometry import you should set up the grid 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. Set the machine type to a Haas Vertical Spindle CNC machine.

TASK 3:
SAVE PART GEOMETRY FOR LATER INSPECTION

1. **NOTE:** Mastercam HLE software will only allow saving files in the .emcx format. Therefore, if you are using the Mastercam HLE software, locate the **Mill-Lesson-FBM-2.STL** file on the DVD accompanying this guide or use the download link in the online instructions. If you are using a full working copy of Mastercam software with the Solids option, continue on to step 2, otherwise move directly to **Task 4**.
2. Select **File and Save as.**
3. Click on the **Browse** icon.
4. **Save as type** click on the drop down arrow and select **StereoLithography Files (*.stl)**

STL files are used with Stereo Lithography systems. An STL file comprises triangular facets of data that represent surface and solid models. Mastercam can read and write both binary and ASCII STL files. STL files are used with

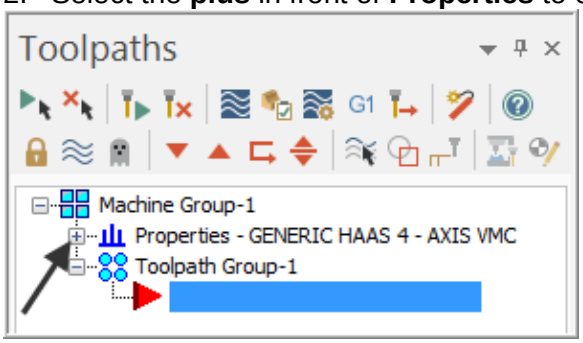
5. In the File name box it should read **Mill-Lesson-FBM-2.**
6. Save to an appropriate location **–** typically in the DATA folder of your Mastercam install folder.
7. Select the **Save** button to save the file and complete this function.

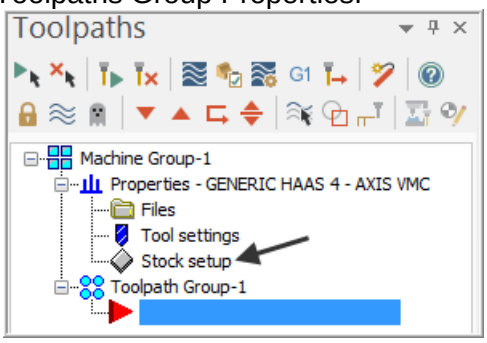
Toolpath Creation

TASK 4:
DEFINE THE ROUGH STOCK USING STOCK SETUP

➲ Defining the Rough Stock using stock setup.

1. Right mouse click in the graphics area and click on the **Fit** icon  .
2. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties.

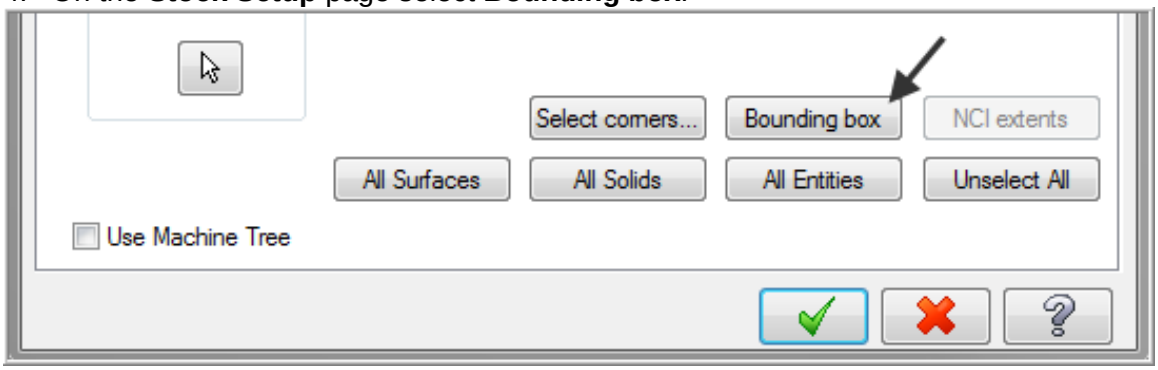




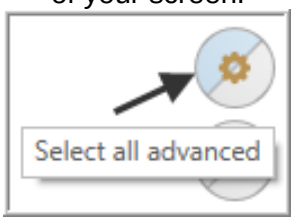
3. Select **Stock setup** in the toolpath manager window as shown above on the right.

➲ The bottom face of the material has been pre-machined and is not part of the toolpath operations for this lesson.
➲ The instructions for this lesson require plus **0.1** stock on thickness, and additional stock in length and width is left to our discretion.
➲ **Z zero** is at the **bottom of the part**.

4. On the **Stock Setup** page select **Bounding box**.



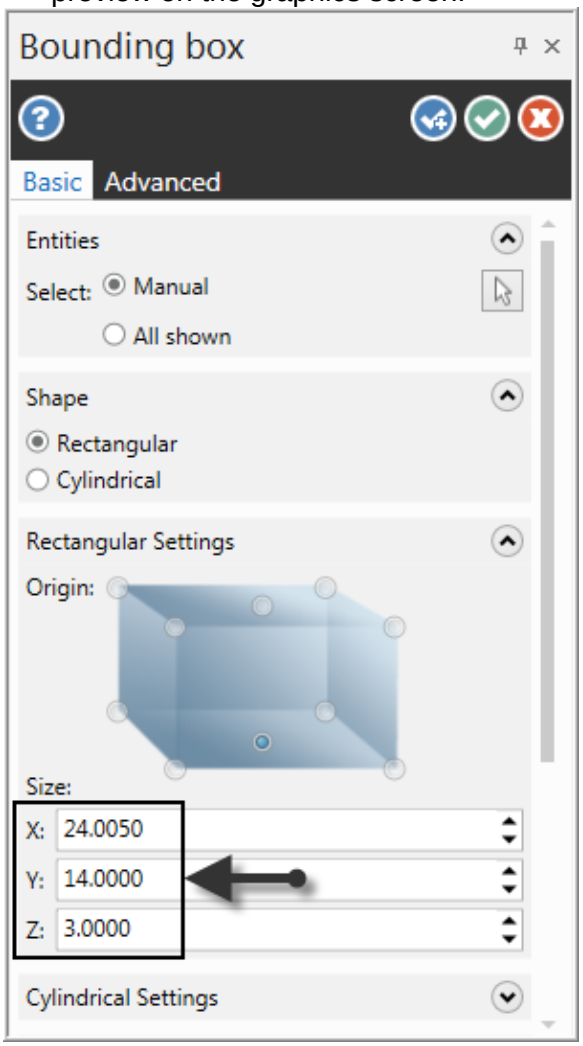
5. You are prompted to **"Select entities**". Select the **Select all advanced** button on the right of your screen.

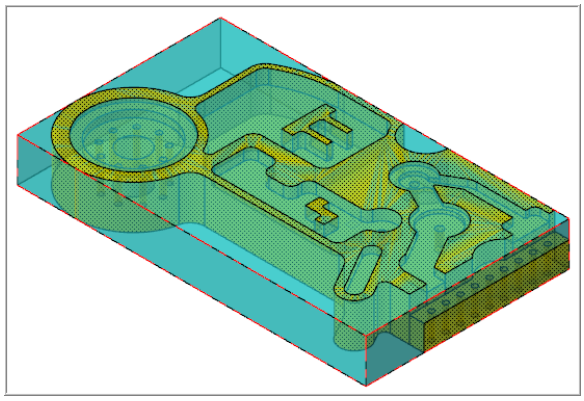


6. Click on **OK**  in the **Select All** window. The model will change colors.

7. Click on **End Selection**  to move onto the next step**.** The Bounding box window appears.

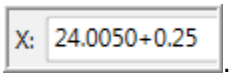
➲ Take note of the X, Y and Z values Mastercam has calculated for the stock size and the preview on the graphics screen.





➲ Now you will **Expand** the **Size** entries by **0.25 in X and Y**, and **0.1 in Z**.

8. In the **Size** section click in the **X and Y values** after the last digit and input **+0.25**.

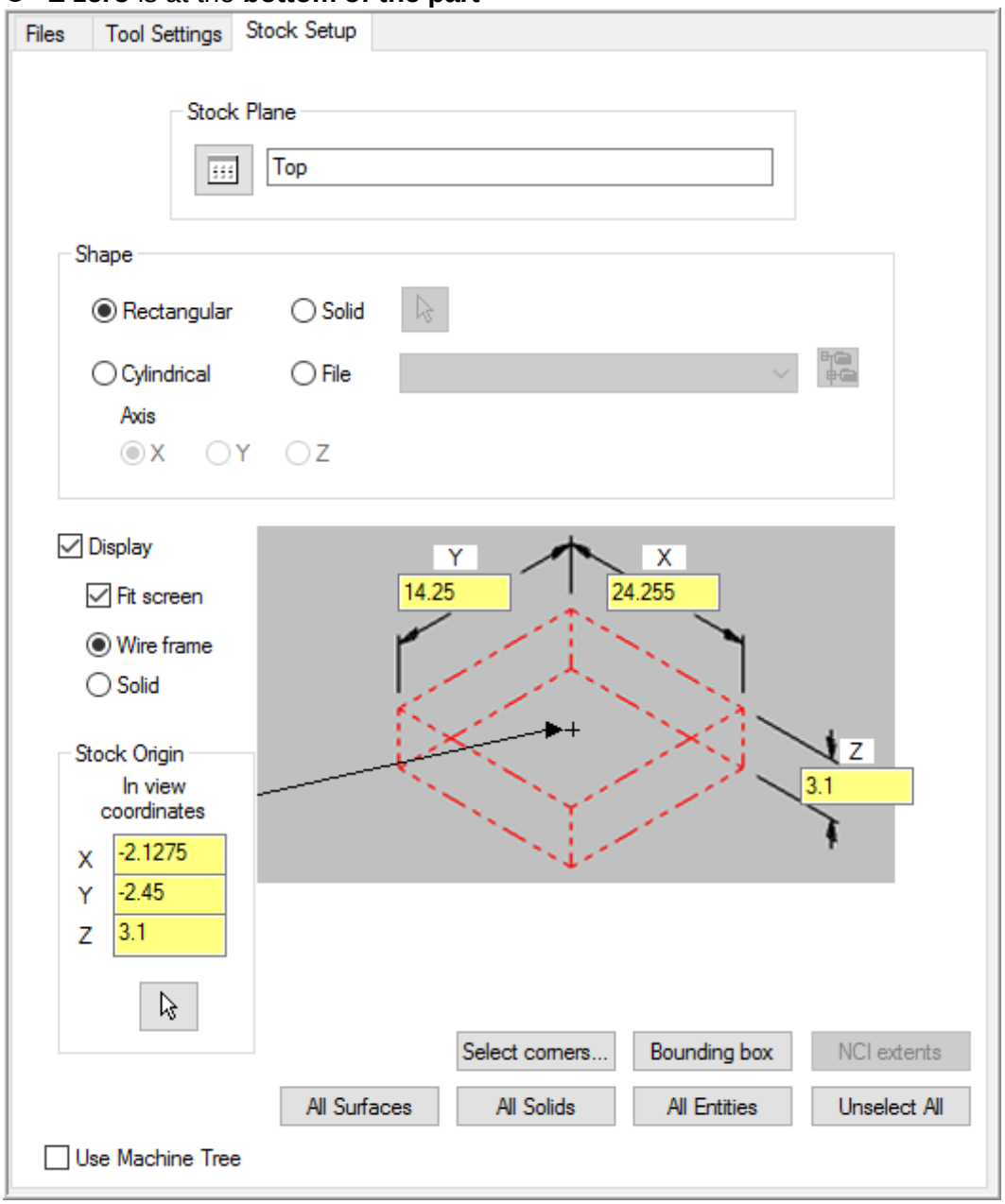


9. Click in the **Z value** after the last digit and input **+0.1**. The results for the Sizes in X, Y and Z should match the sizes shown below.

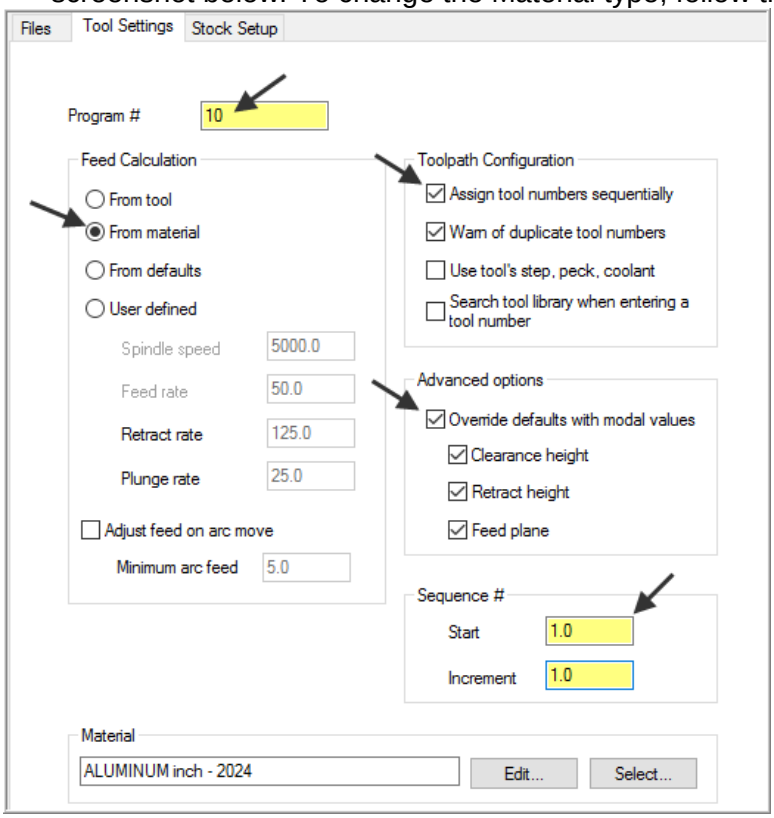
10. Now click on the **OK** icon  in the Bounding box window.

11. The parameters should match the **Stock Setup** screenshot below:

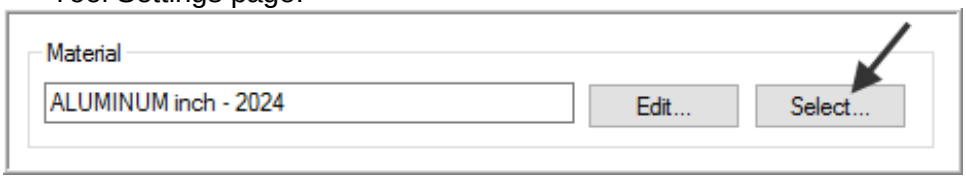
- The bottom face of the material has been pre-machined and is not part of the toolpath operations for this lesson.
- **Z zero** is at the **bottom of the part**



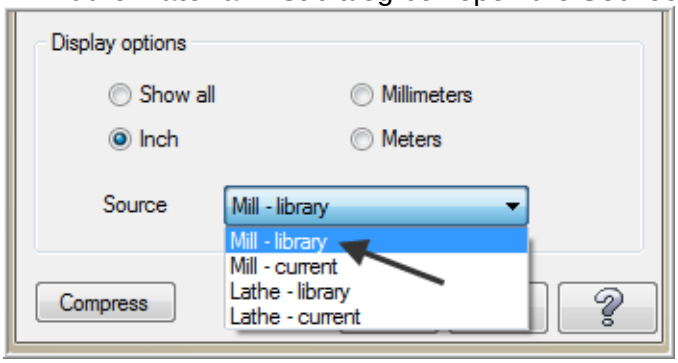
12. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. To change the Material type, follow the next set of instructions.



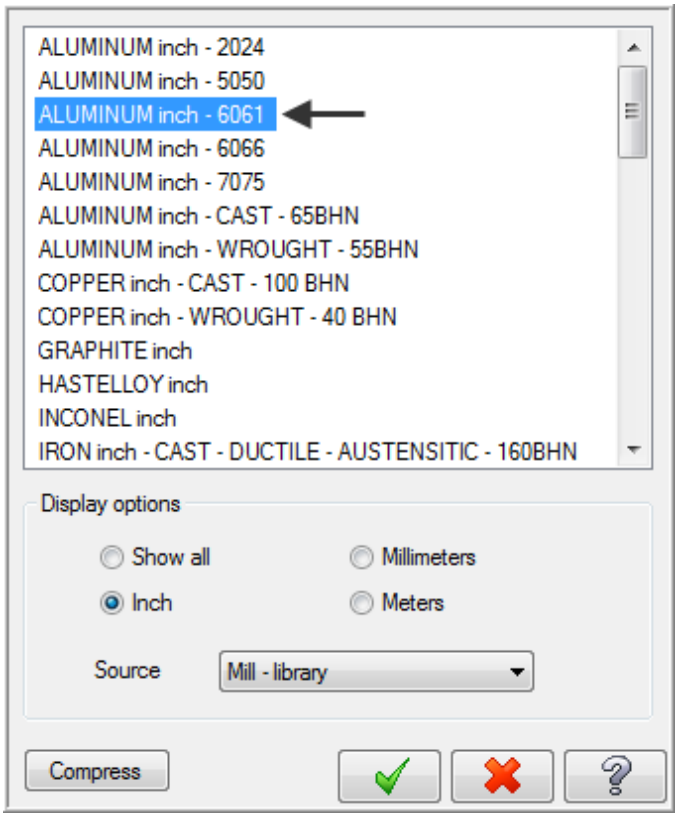
13. To change the **Material type to Aluminum 6061** pick the **Select** button at the bottom of the Tool Settings page.



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



15. From the Default Materials list select **ALUMINUM inch - 6061** and then select  .



With the selection of **Feed Calculation From material** in the **Machine Group Properties Tool Settings** tab, all feeds and speeds are set based on calculated values.

Many users take the time to create a custom tool library with only their tools, with desired feeds and speeds set in the library.

Feeds and speeds can be edited in the resulting FBM child operations if desired.

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

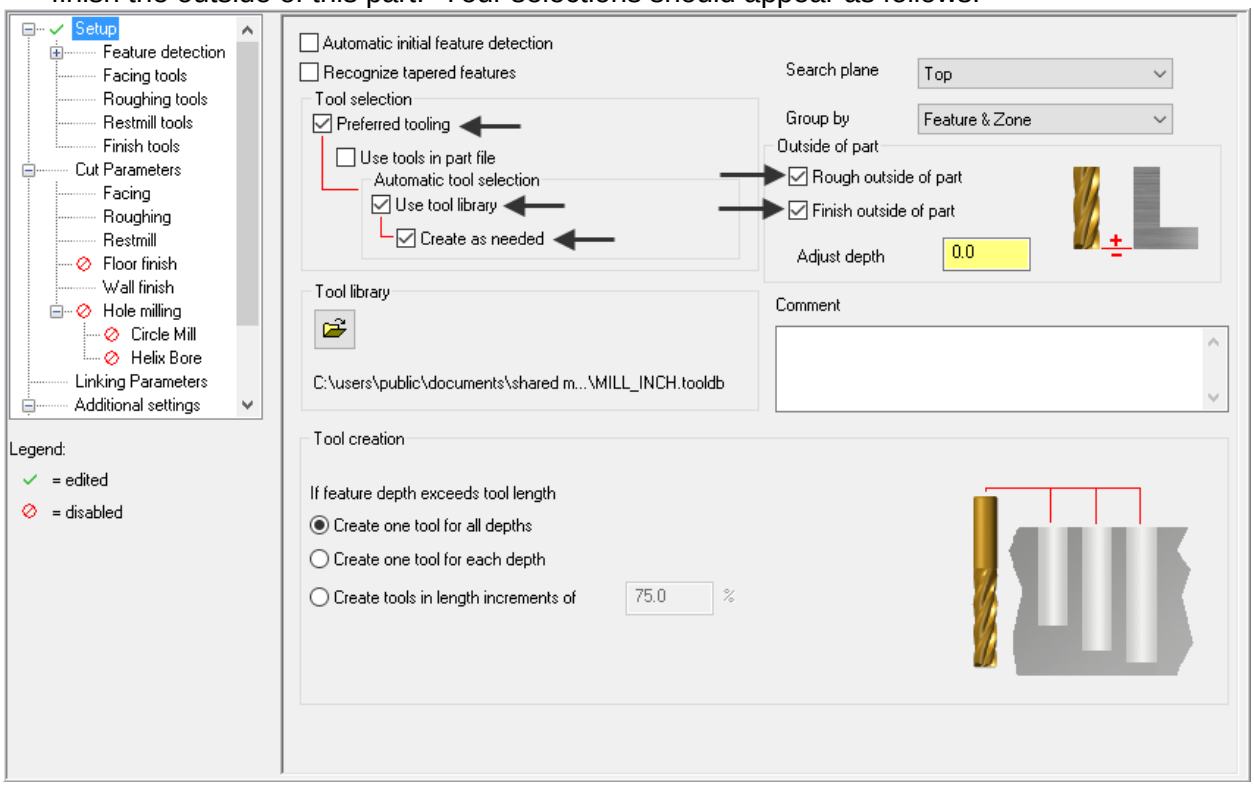
TASK 5:
RUN FBM MILL FOR ALL MILLING OPERATIONS

➲ In this task you configure settings in FBM Mill to set up tools and interrogate the solid to create machining regions and apply toolpath strategies.

FBM Mill pays close attention to flute length during automatic tool selection. If allowing the use of Tools from library, shorter library tools will be selected for shallow pockets. When the pocket depth exceeds the flute length of the library tool, a longer tool with the same diameter will be created to achieve the required depth of cut.

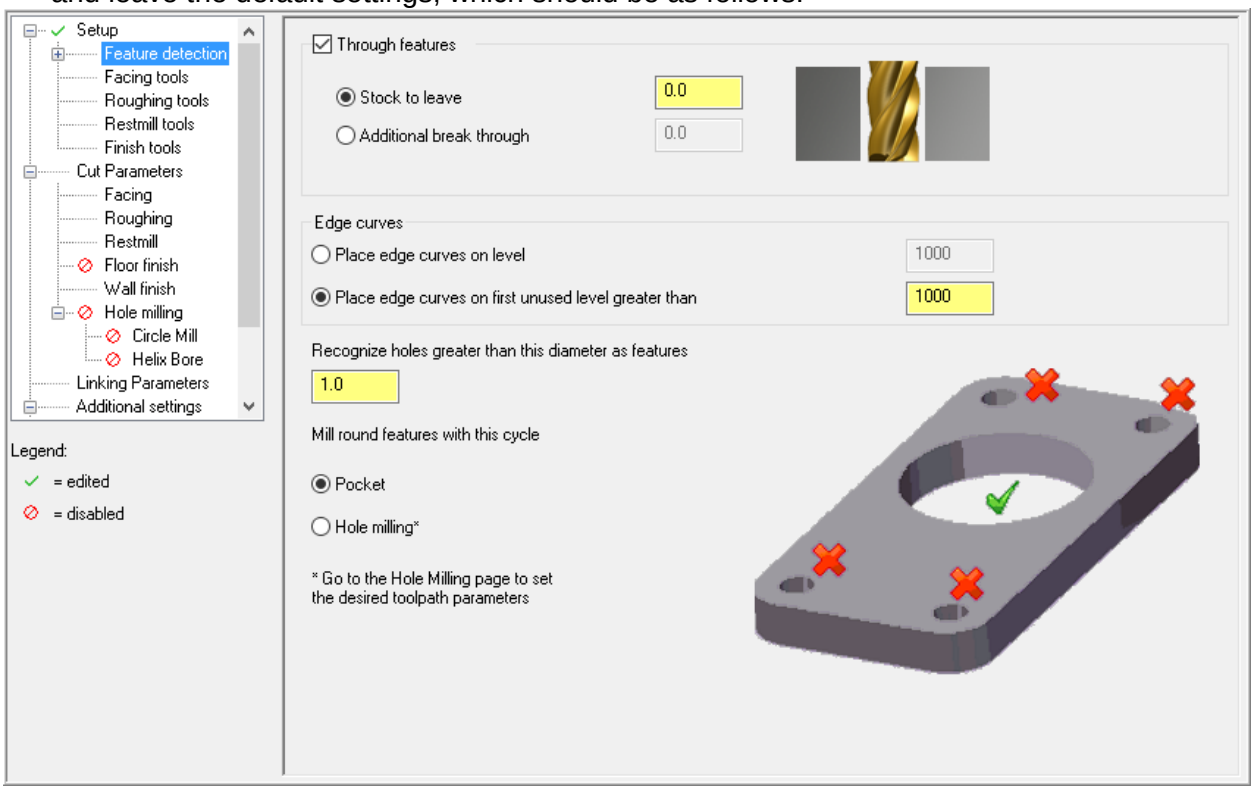
In this exercise we will allow FBM Mill to generate all tools as required to machine the part geometry. In order to avoid multiple same diameter tools, we will avoid the use of library tools.

1. Select the **TOOLPATHS** tab at the top right side of the screen.
2. Select **FBM Mill** in the **2D Milling** toolpath section.
3. On the screen you will now see the **FBM Toolpaths – Mill** dialog box, starting with the **Setup** page. From this page you will select **Create as needed** to have FBM Mill inform us of the tools required to complete machining of the part. You will also select to rough and finish the outside of this part. Your selections should appear as follows:

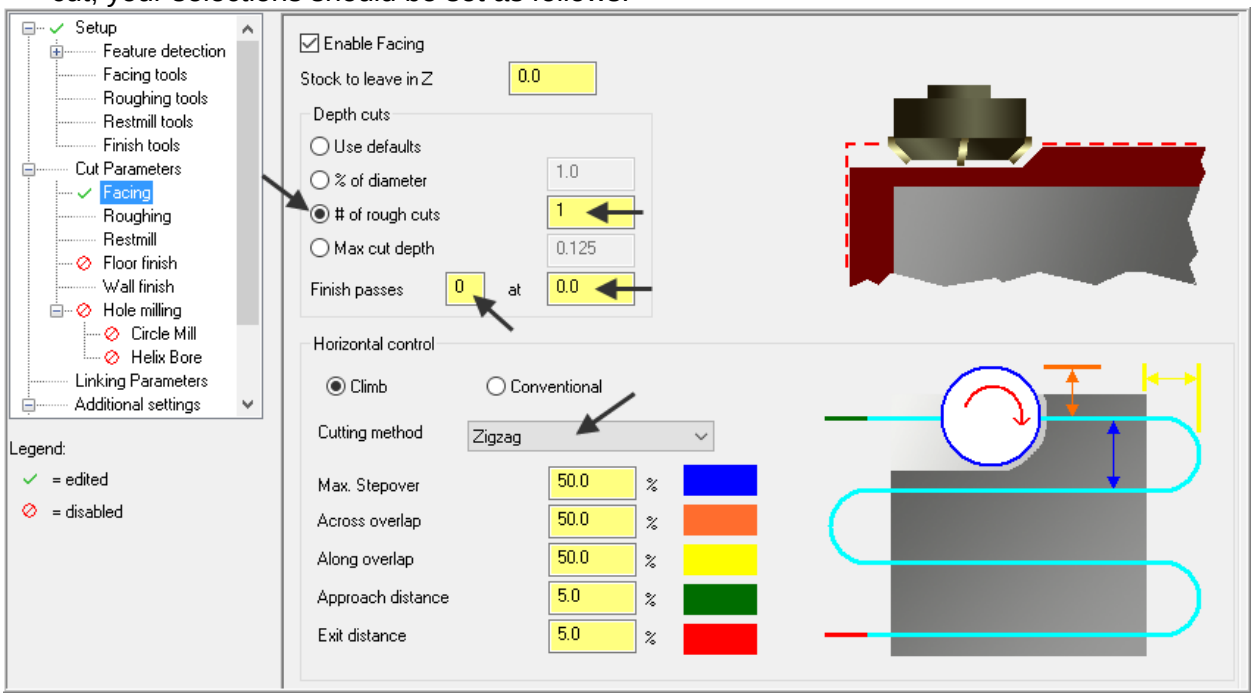


Tool numbering will appear to be non-sequential. Tools are created with the length required for the pocket being processed. If an identical diameter tool is required to be longer for a deeper pocket, the original tool will be replaced with a longer tool, but will have a higher tool number than the original.

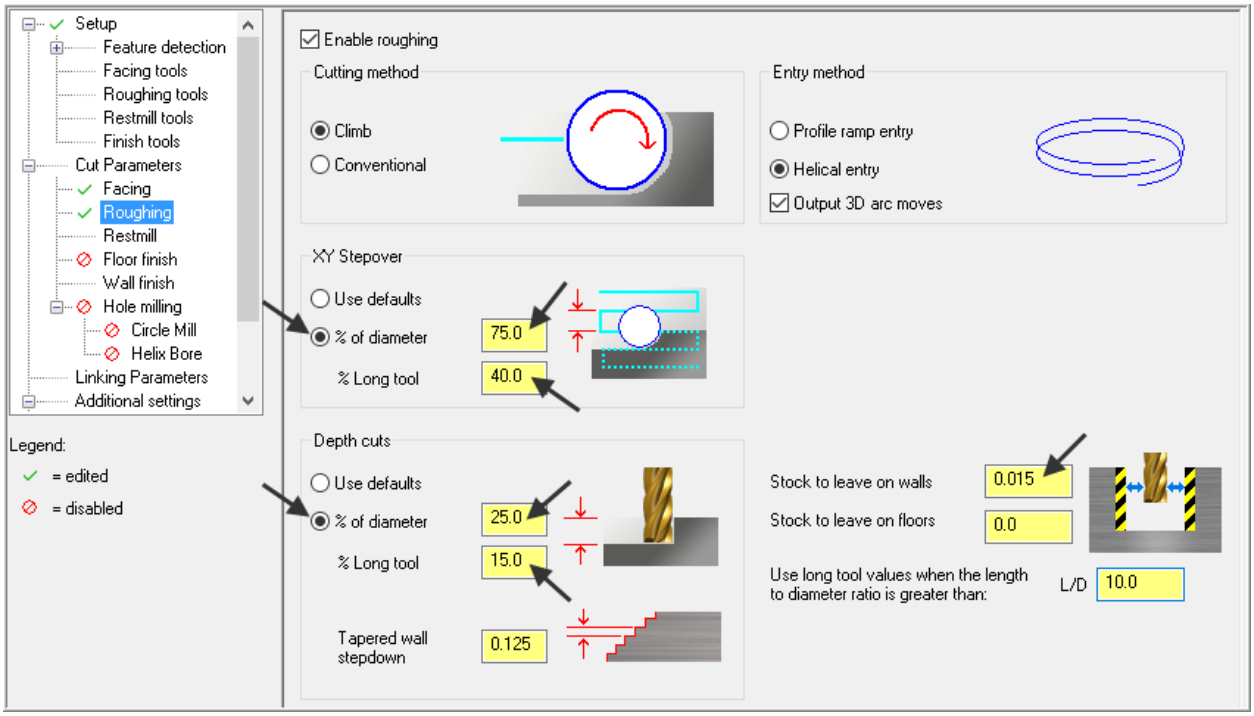
4. Click on **Feature detection** in the Tree Control menu on the left hand side of the dialog box and leave the default settings, which should be as follows:



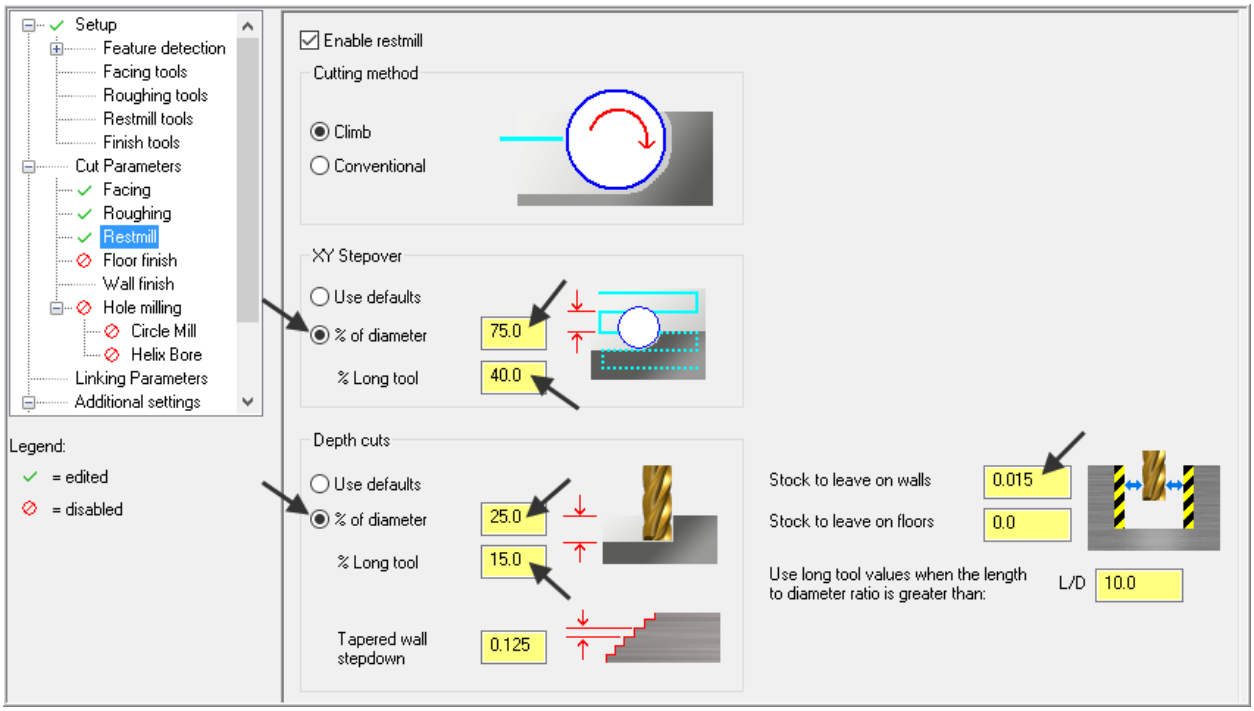
5. You will skip past the **Preferred tool list** for **Facing tools**, **Roughing tools**, **Restmill tools**, and **Finish tools**.
6. Next you will set Facing cut parameters, selecting **Facing** from the Tree Control dialog box, and adjusting settings for **1 rough cut**, and selecting **Zigzag**. To force a single facing depth cut, your selections should be set as follows:



7. Next you will set Roughing cut parameters, selecting **Roughing** from the Tree Control dialog box, and adjusting settings for **Helical entry, 75%** of diameter **XY Stepover**, **25%** of diameter **Depth Cuts**, and **0.015 Stock to leave on walls**. Your selections should be set as follows:



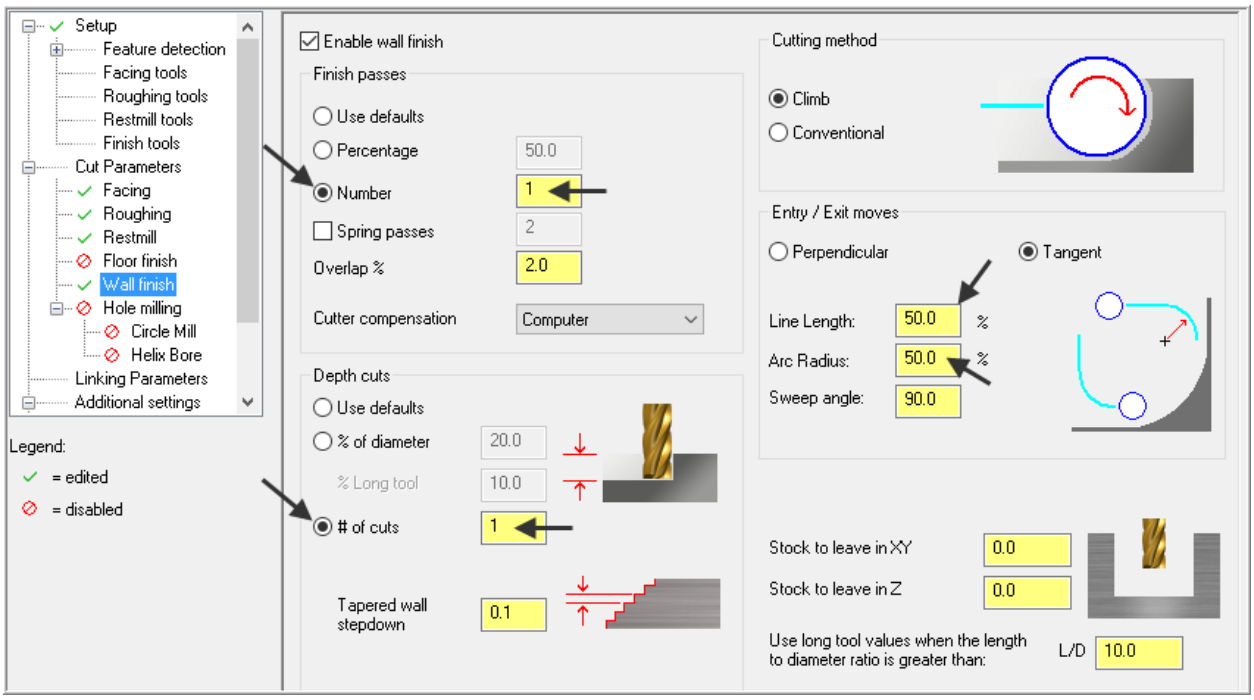
8. Next you will set Restmill cut parameters, selecting **Restmill** from the Tree Control dialog box, and adjusting settings for **75%** of diameter **XY Stepover**, **25%** of diameter **Depth Cuts**, and **0.015 Stock to leave on walls**. Your selections should be set as follows:



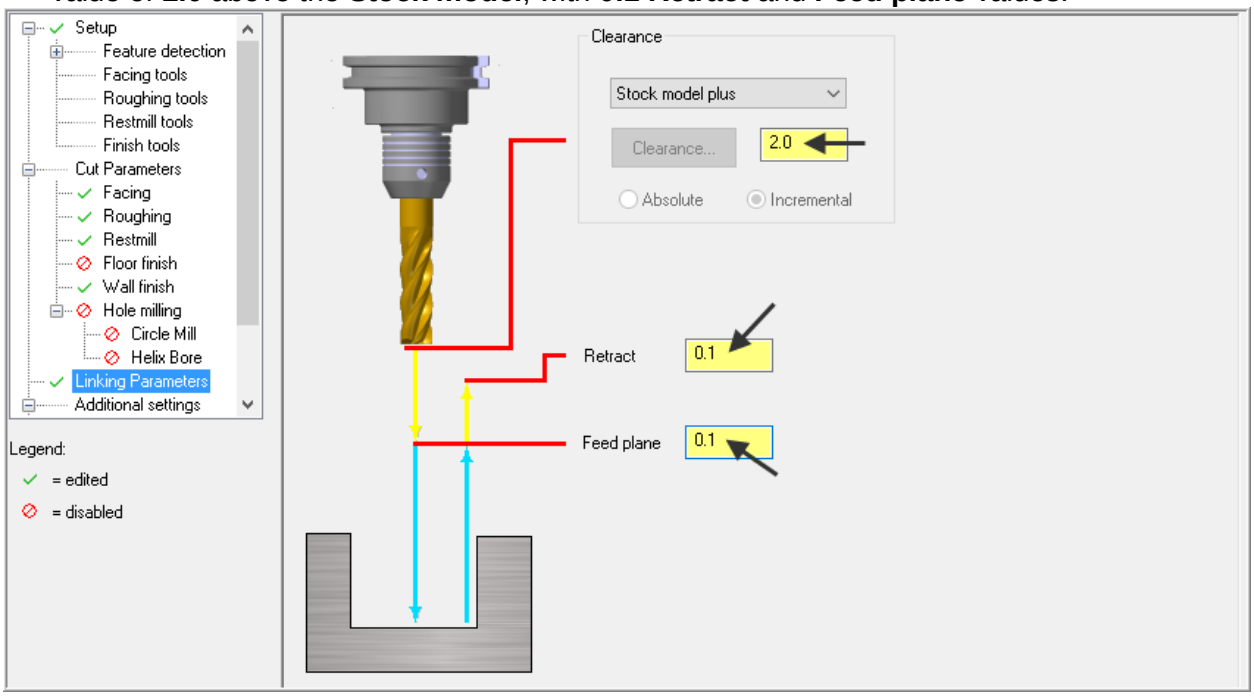
9. Ensure **Floor finish** is disabled 

. If not, select **Floor Finish** from the Tree Control dialog box and uncheck **Enable floor finish** 

in order to disable floor finish.
10. Next you will set Finish cut parameters, selecting **Wall finish** from the Tree Control dialog box, and adjusting settings for **1 Finish Pass**, **1 Depth Cut**, and **Tangent Entry / Exit moves** with lines and arcs that are **50%** of the tool's diameter. Your selections should be set as follows:

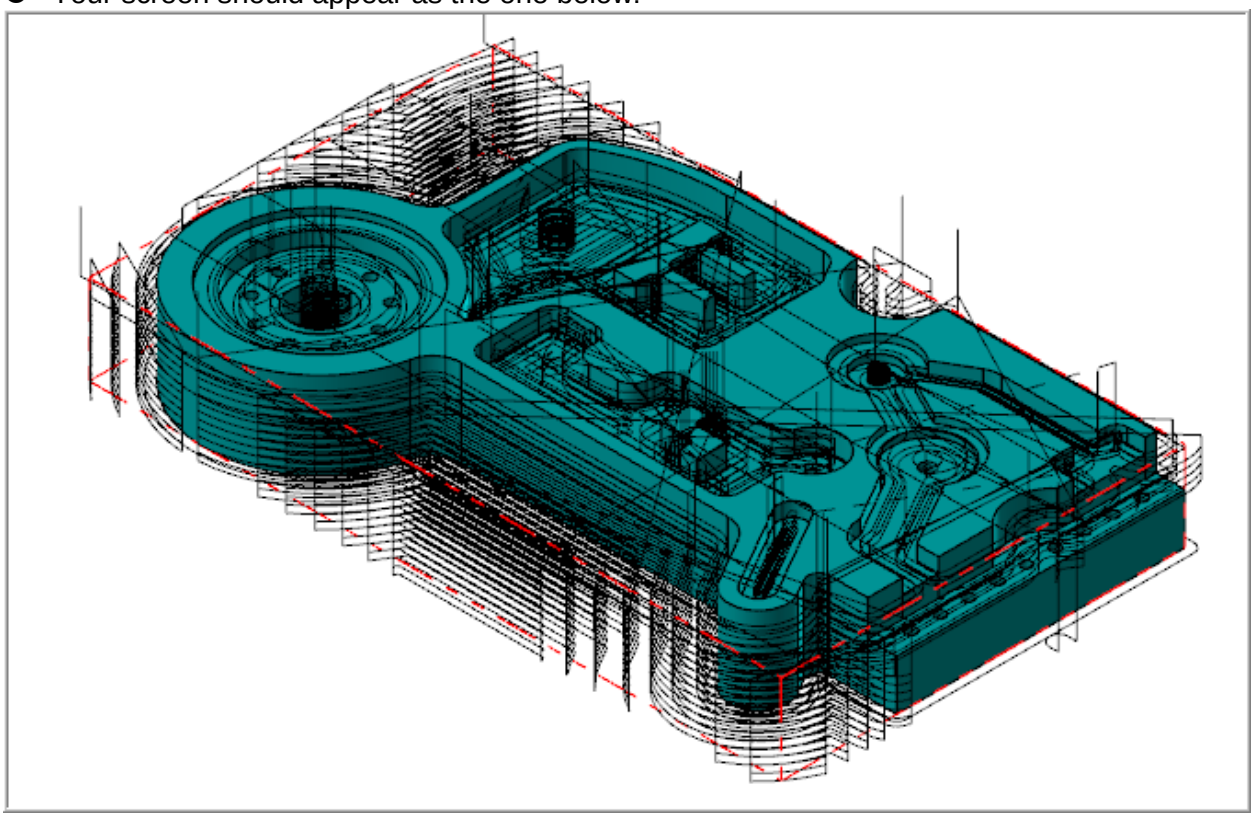


11. Select **Linking Parameters** from the Tree Control dialog box. You will use a **Clearance** value of **2.0** above the **Stock model**, with **0.1 Retract** and **Feed plane** values:



12. Expand **Additional settings** and set **Coolant Flood** to **On.**
13. Select the OK button  to process all milling operations for this part.
14. When prompted to **Enter new NC name** ensure **Mill-Lesson-FBM-2** is displayed and then select the OK button  .
15. Process times can be significant for complex geometry, **so be patient**.
16. In the **View** tab click on **Outline Shaded** in the **Appearance** section to display the part in shaded mode.
17. Right mouse click in the graphics area and click on **Isometric**.
18. Right mouse click in the graphics area and click on the **Fit** icon  .

➲ Your screen should appear as the one below:

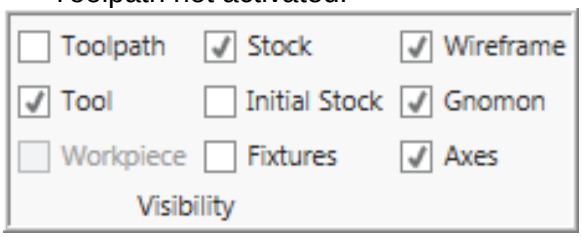


TASK 6:
VERIFY THE TOOLPATH, COMPARING RESULTS TO PART GEOMETRY, AND SAVING OUT IN-PROCESS STOCK

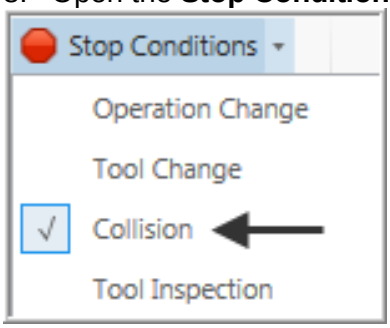
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

1. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon.
2. Select the **Verify selected** operations icon.
3. **Maximize** the Backplot/Verify window if required.
4. Activate the **Both** option in the **Toolpath** section of the **Verify** tab.
5. Now select the **Home** Tab.

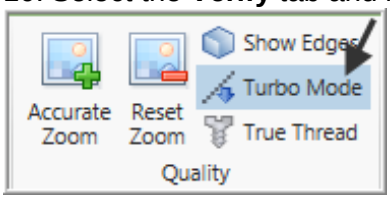
6. Activate the options shown below in the **Visibility** section of the Home tab. **Initial Stock** and Toolpath not activated.



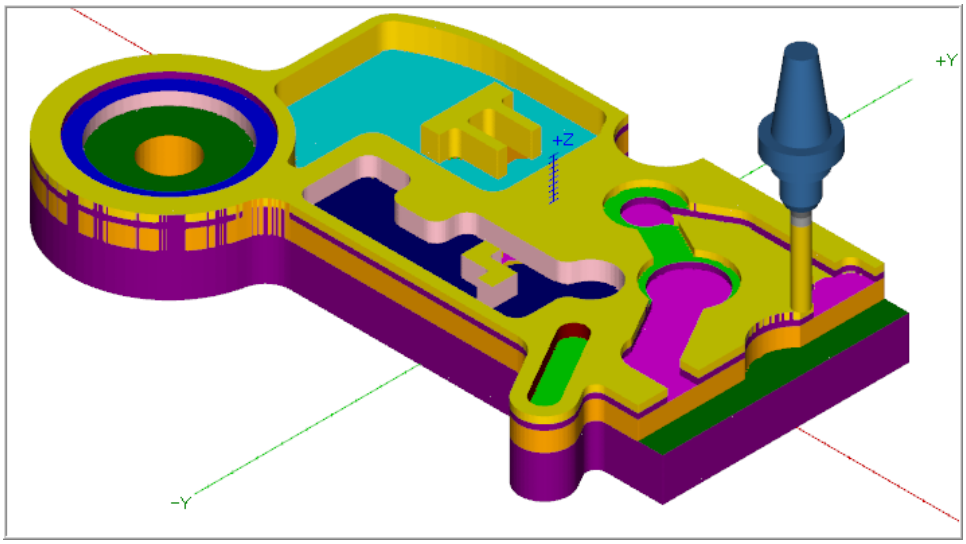
7. Activate the **Color Loop** to change the color of the tools for the verified part.
8. Open the **Stop Conditions** drop down menu and activate stop on Collision.



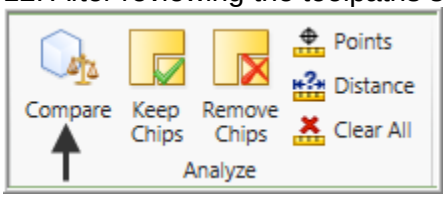
9. At the top of the screen select the **View** tab, the **Isometric** icon and then select **Fit.**
10. Select the **Verify** tab and in the **Quality** section activate **Turbo Mode**.



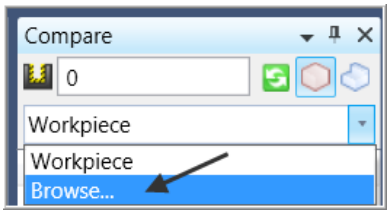
11. Now select the Play Simulation button to review the toolpaths.



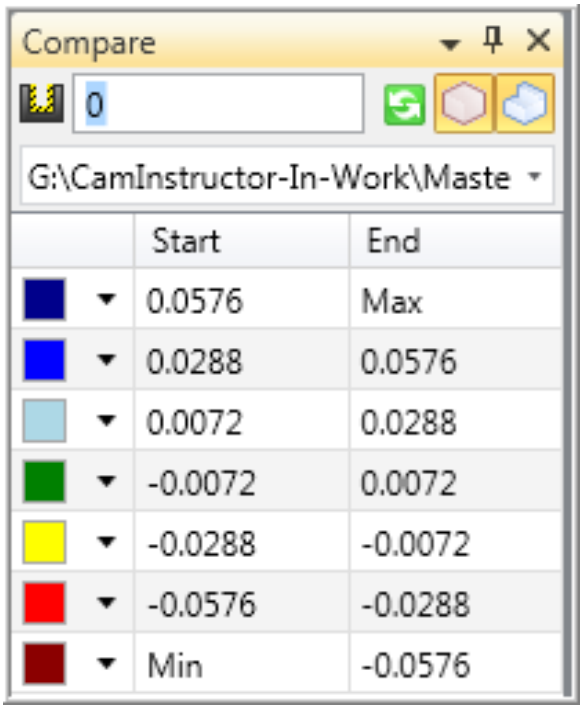
12. After reviewing the toolpaths select the **Compare** icon in the **Analyze** section.



13. The Compare dialog box opens up on the right of the screen. Now open the dropdown menu and select Browse and select the .stl file that you saved in the earlier step. Or if you are using Mastercam HLE use the **Mill-Lesson-FBM-2.STL** file supplied on the multimedia DVD that came with this text in a folder called Mastercam Files.



14. The comparison stock model is now loaded. Hit the green **Refresh** icon. The computed results are shown below:



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

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

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

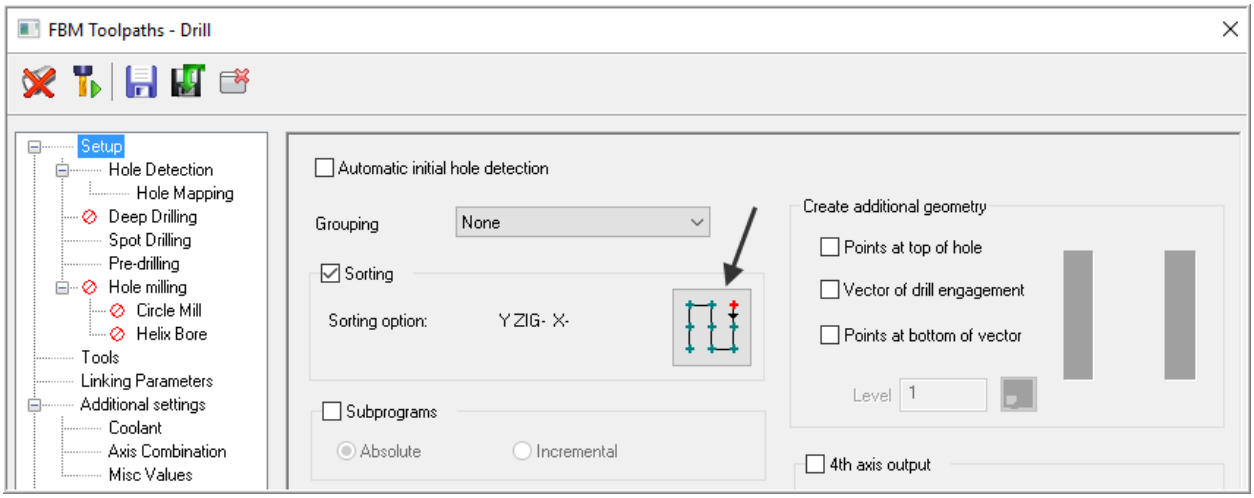
Purple to red shaded areas would indicate part gouges and areas in previous toolpaths that need to be addressed!

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

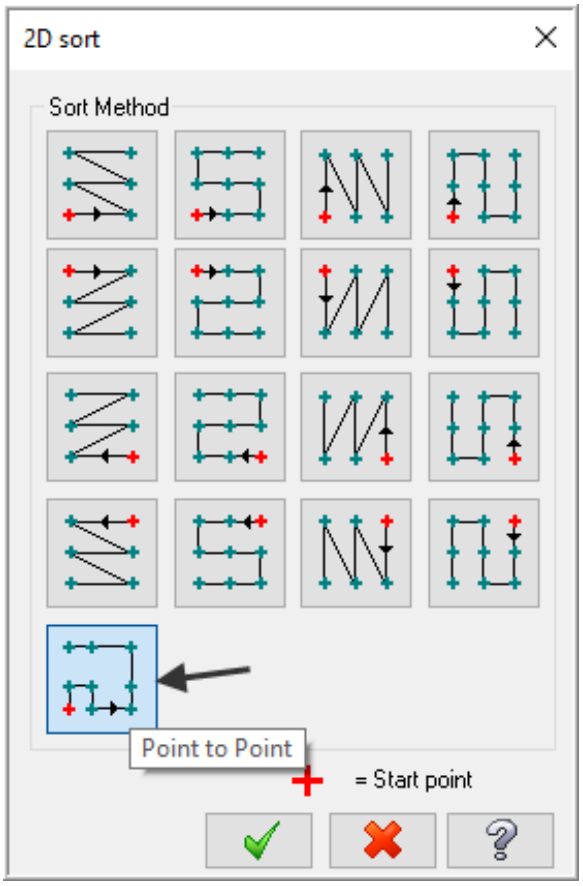
TASK 7:
RUN FBM DRILL FOR ALL HOLE MAKING OPERATIONS

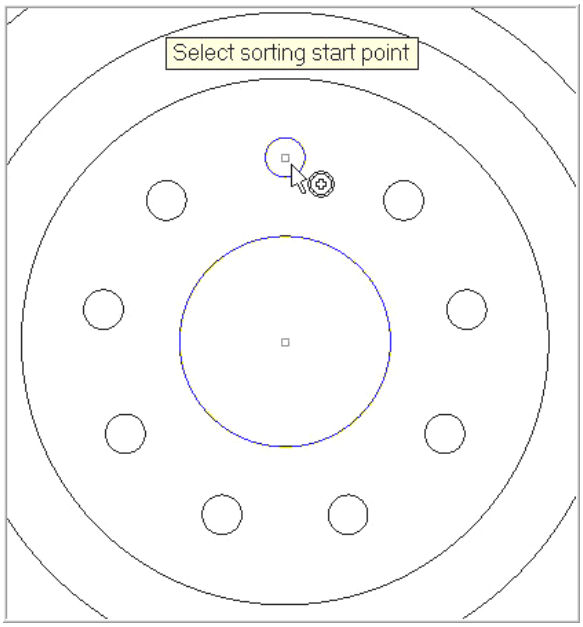
➲ In this task you configure settings in FBM Drill to interrogate the solid, identify hole features, and apply holemaking strategies.

1. In the **View** tab click on **Wireframe** in the **Appearance** section.
2. Right mouse click in the graphics area and click on **Top.**
3. Right mouse click in the graphics area and click on the **Fit** icon  .
4. Select the **TOOLPATHS** tab at the top right side of the screen.
5. Select **FBM Drill** in the **2D Holemaking** toolpath section.
6. On the screen you will now see the **FBM Toolpaths – Drill** dialog box. Select **Setup** in the Tree Control menu and change the **Sorting option** by click on the current Sorting method icon as shown below:



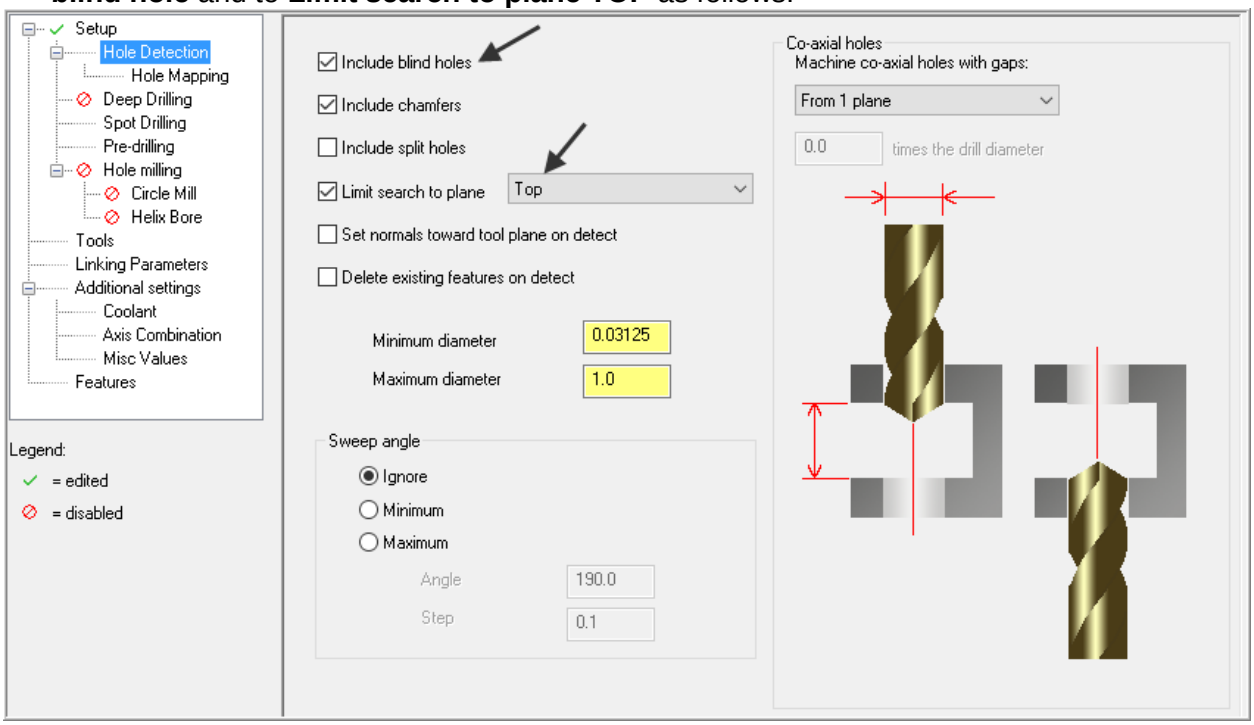
7. Select the **Point to Point 2D Sort Method**.



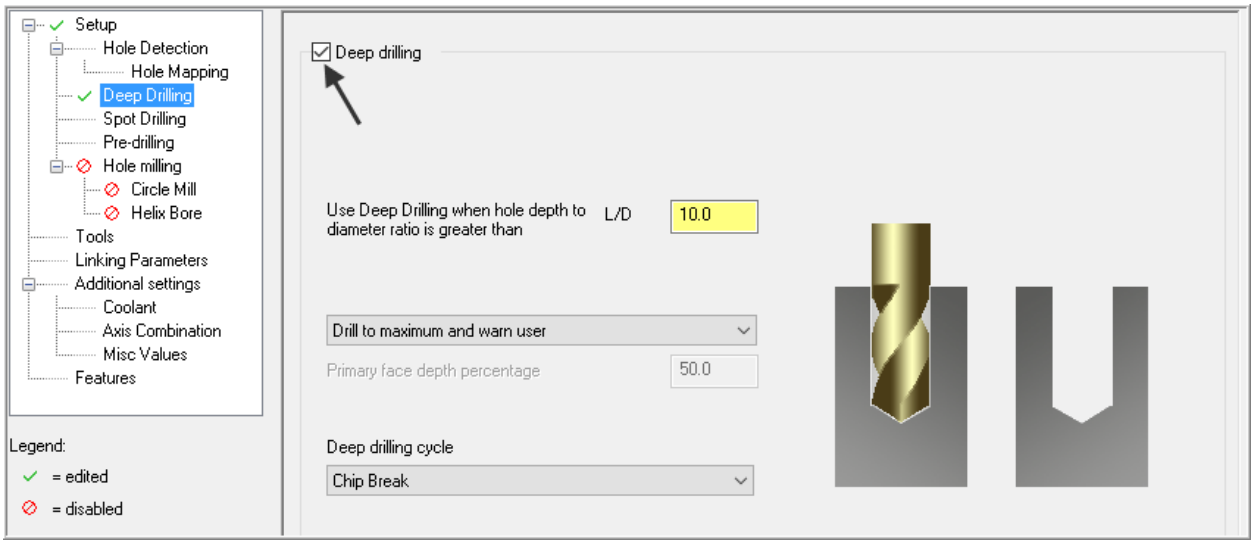


8. Select the OK button  in the **2D sort** page.
9. The prompt now changes to **Select sorting start point**. Move the mouse and position the cursor over the point at the center of circle shown above right; you will notice this icon  appears as you move over the point at the center of this circle. This cross denotes you are snapping to a circle center point. Use the left button of your mouse to pick this point.

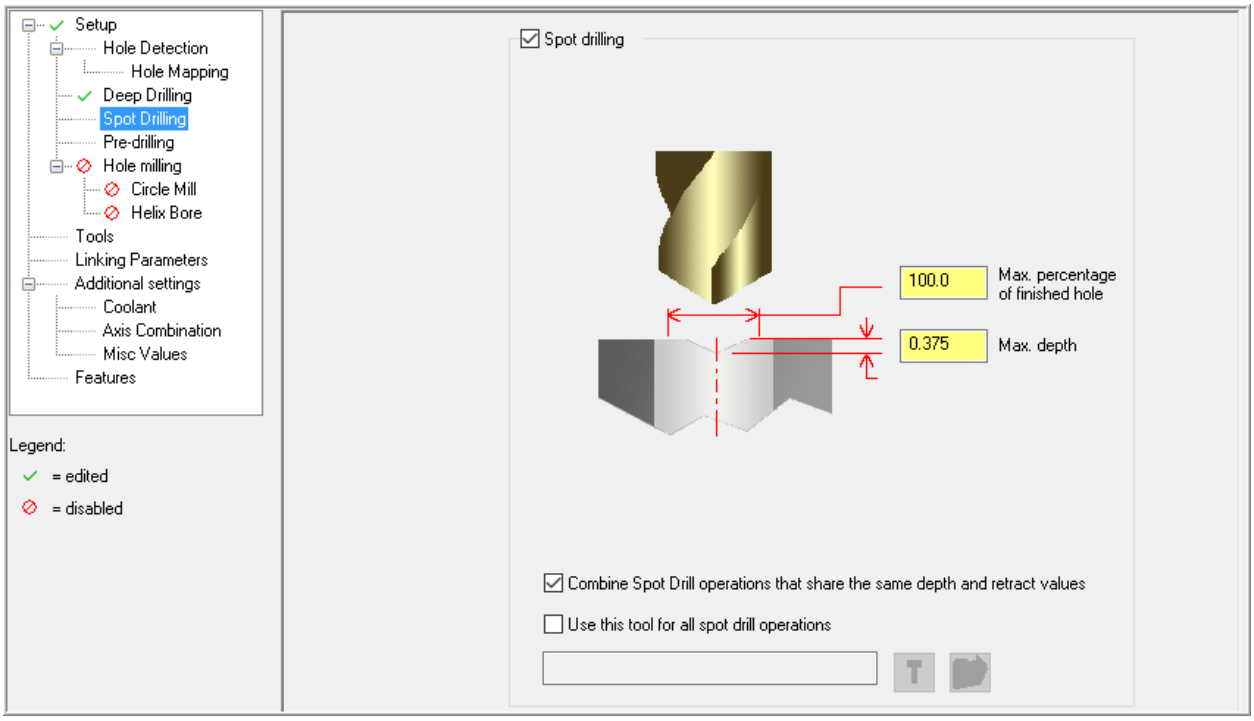
10. Select **Hole Detection** from the Tree Control dialog box and set parameters to **Include blind hole** and to **Limit search to plane TOP** as follows:



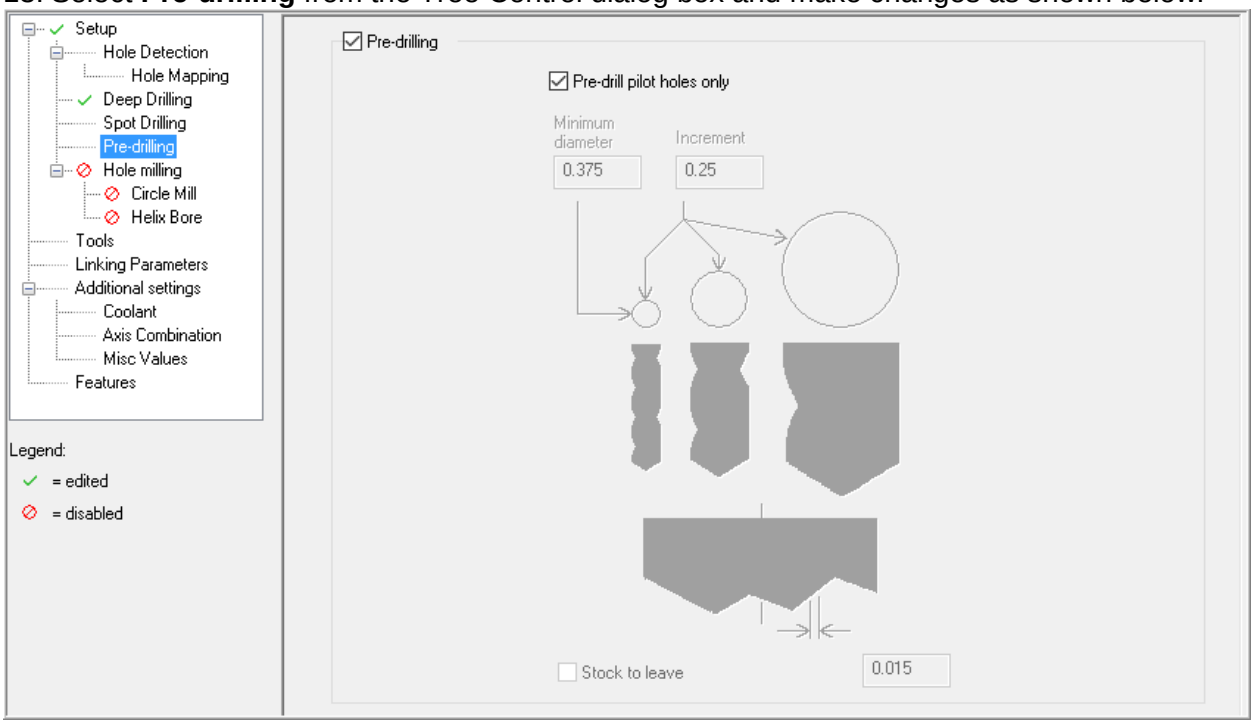
11. Select **Deep Drilling** from the Tree Control dialog box and activate Deep drilling by clicking on the check box as shown below:



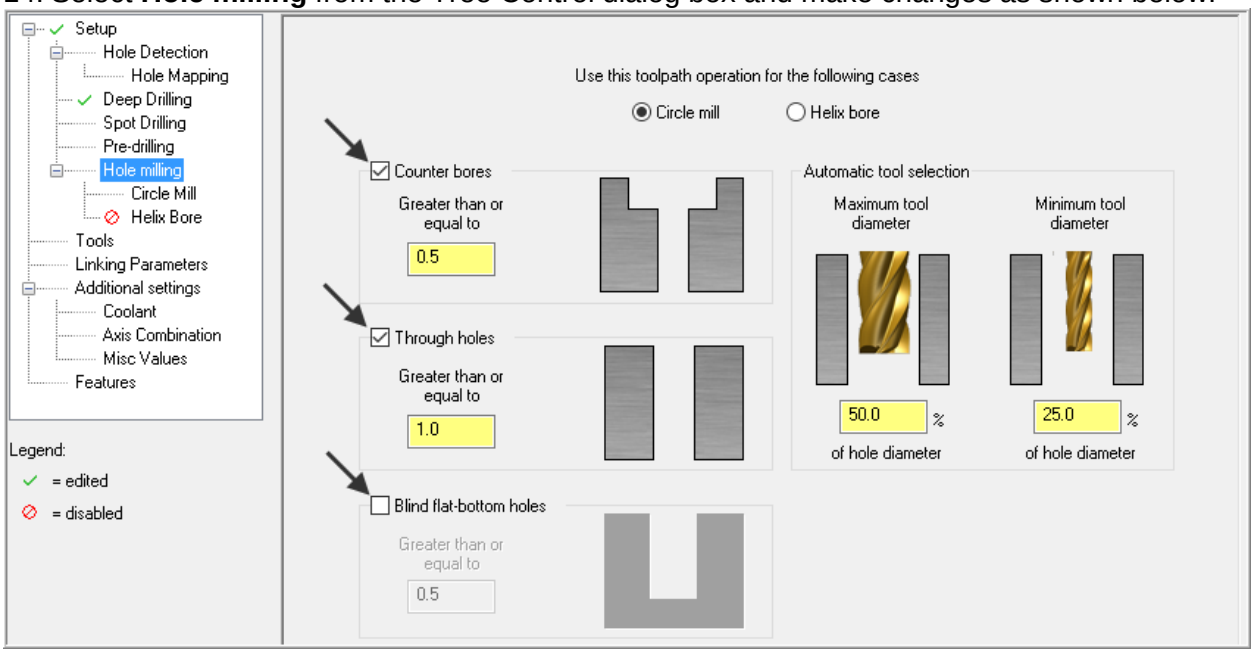
12. Select **Spot Drilling** from the Tree Control dialog box and ensure **Spot drilling** is activated as shown below:



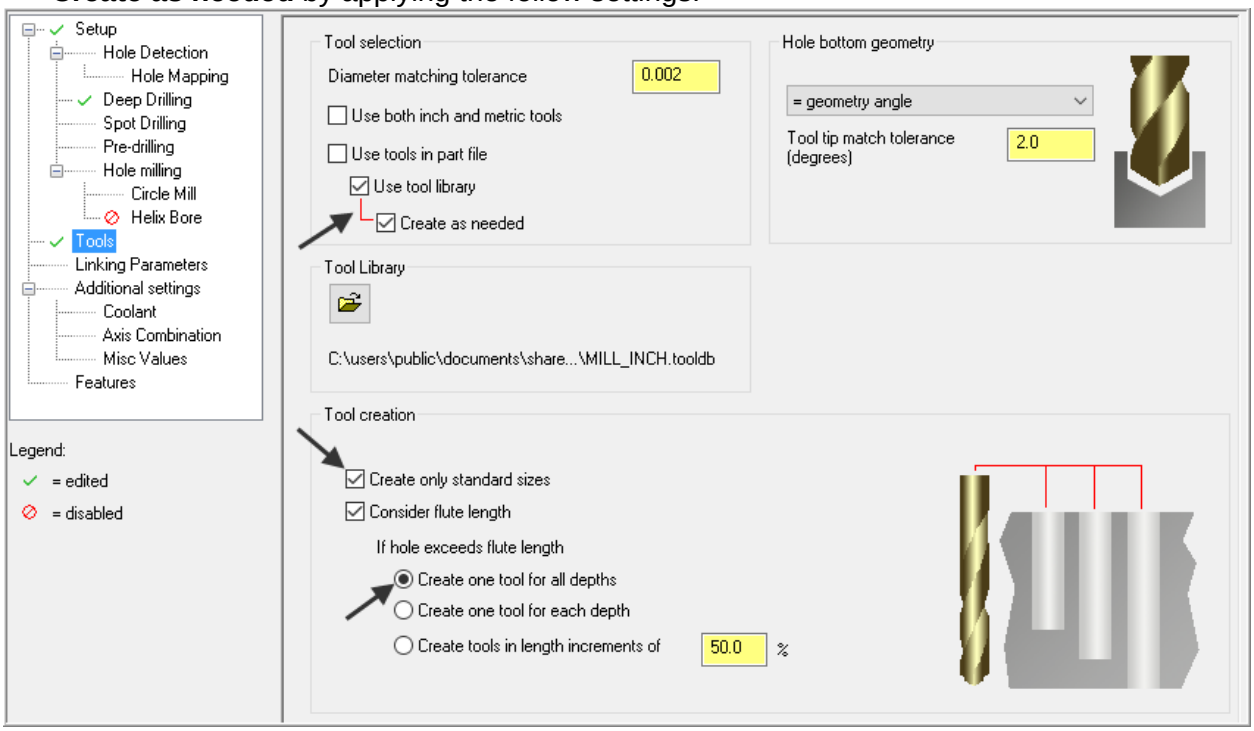
13. Select **Pre-drilling** from the Tree Control dialog box and make changes as shown below:



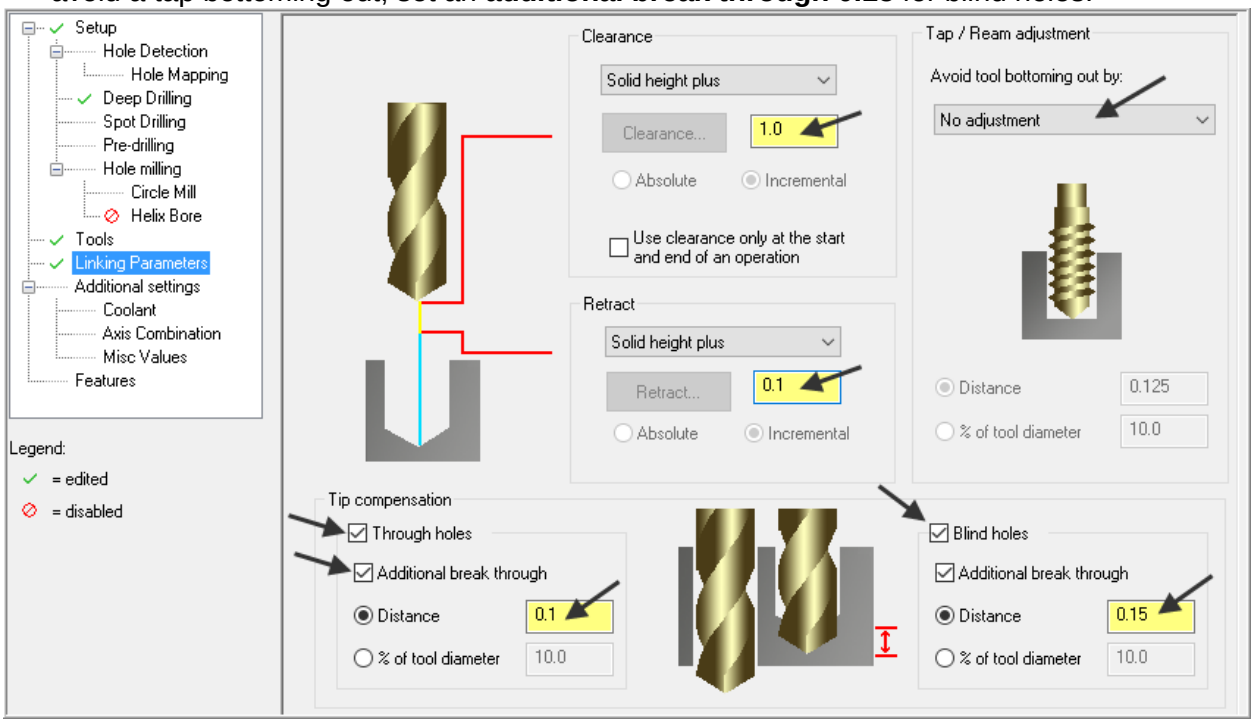
14. Select **Hole milling** from the Tree Control dialog box and make changes as shown below:



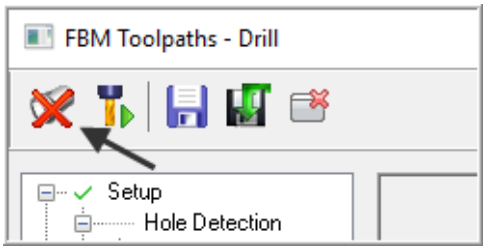
15. Select **Tools** from the Tree Control dialog box. You will allow FBM Drill to automatically **Create as needed** by applying the follow settings:



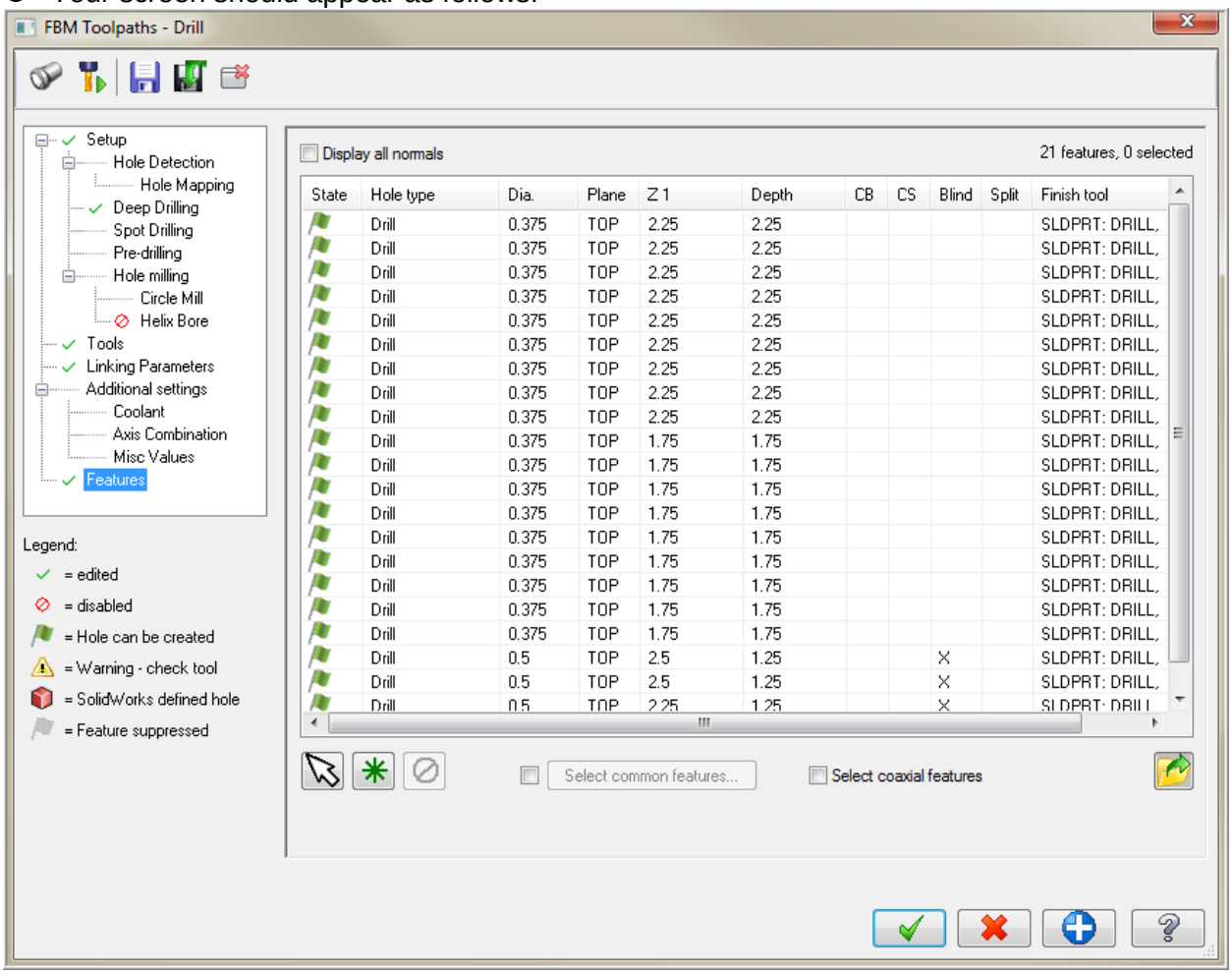
16. Select **Linking Parameters** from the Tree Control dialog box. This dialog box will be used to set our **Clearance** at **Solid height plus 1.0**, **Retract** at **Solid height plus 0.1**, an **Additional break through** of **0.1**, and **No adjustment** for the **Tap / Ream adjustment.** To avoid a tap bottoming out, set an **additional break through 0.15** for blind holes.



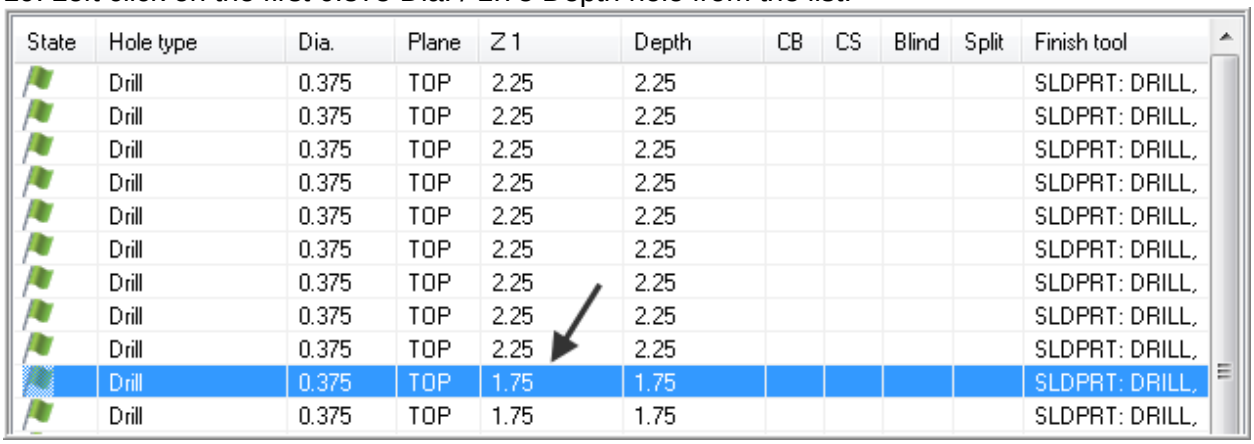
17. Expand **Additional settings** and set **Coolant Flood** to **On.**
18. Click the **Detect...** button at the top of the FBM Toolpaths – Drill dialog box.



➲ Your screen should appear as follows:

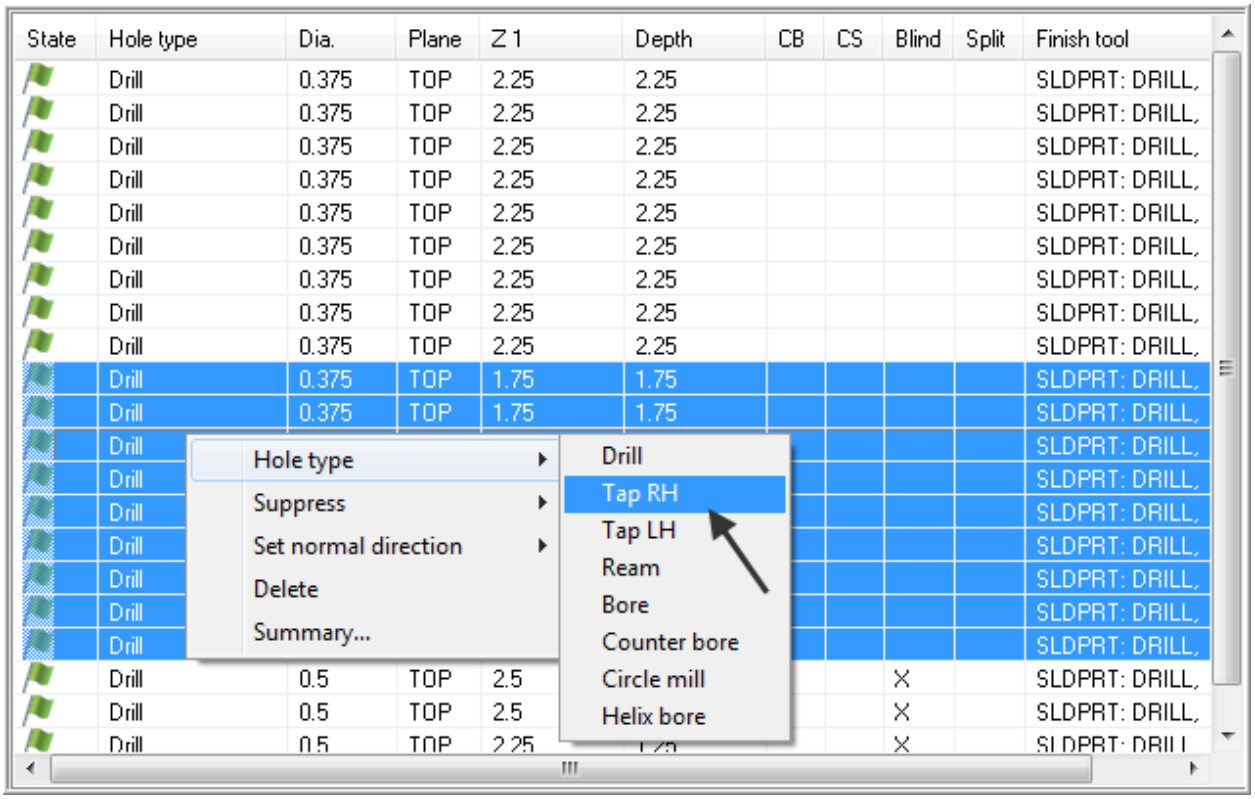


19. Left click on the first 0.375 Dia. / 1.75 Depth hole from the list.

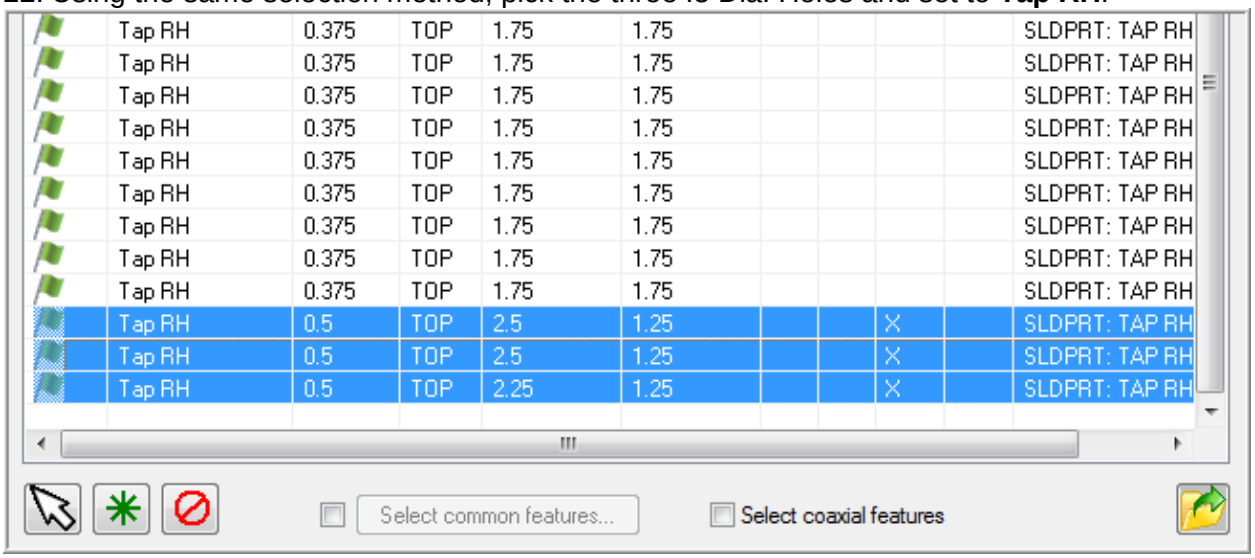


20. Press and hold the **Shift** key on your keyboard. Left click on the last 0.375 Dia. / 1.75 Depth hole from the list. Release the **Shift** key. This will select all rows from the first mouse selection through the second mouse selection.

21. Right click on anywhere in Hole type column of nine selected holes. In the **Hole type** column their value currently reads **Drill.** Hover the cursor over **Hole type** and select **Tap RH** from the list.



22. Using the same selection method, pick the three .5 Dia. Holes and set to **Tap RH**.



Features page
Use this page to manage the list of hole features that the FBM drilling operation detects in the solid model. The holes and machining information in the list are based on the parameters you set in the operation's other pages.

For each hole listed, you can use right-click menu options to change the hole type, tool plane, and finish tool. You can also delete holes from the list (select and press [Delete]), or use the right-click Suppress option to exclude them from the operation.

When you accept the operation, Mastercam creates drill cycles for all holes in the Features list that can be machined.

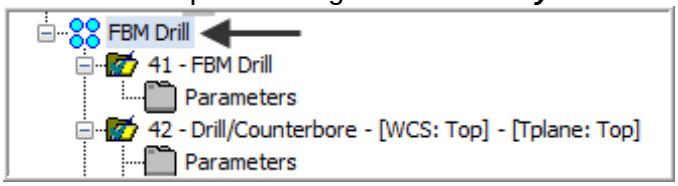
Best practice: Make these modifications only at the final stage of creating the FBM drill operation. Detect rebuilds the Features page list, overwriting any hole type, plane, and finish tool modifications.

23. Select the OK button  to process all holemaking operations for this part.
24. Click on the OK button  if you receive any warnings.

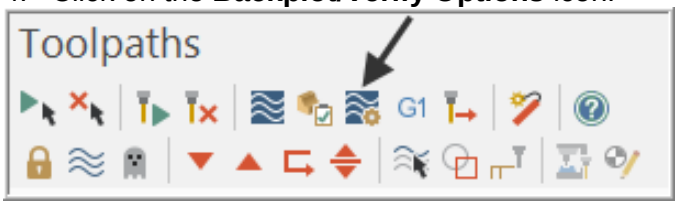
TASK 8:
VERIFY THE TOOLPATH USING SAVED STOCK

- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

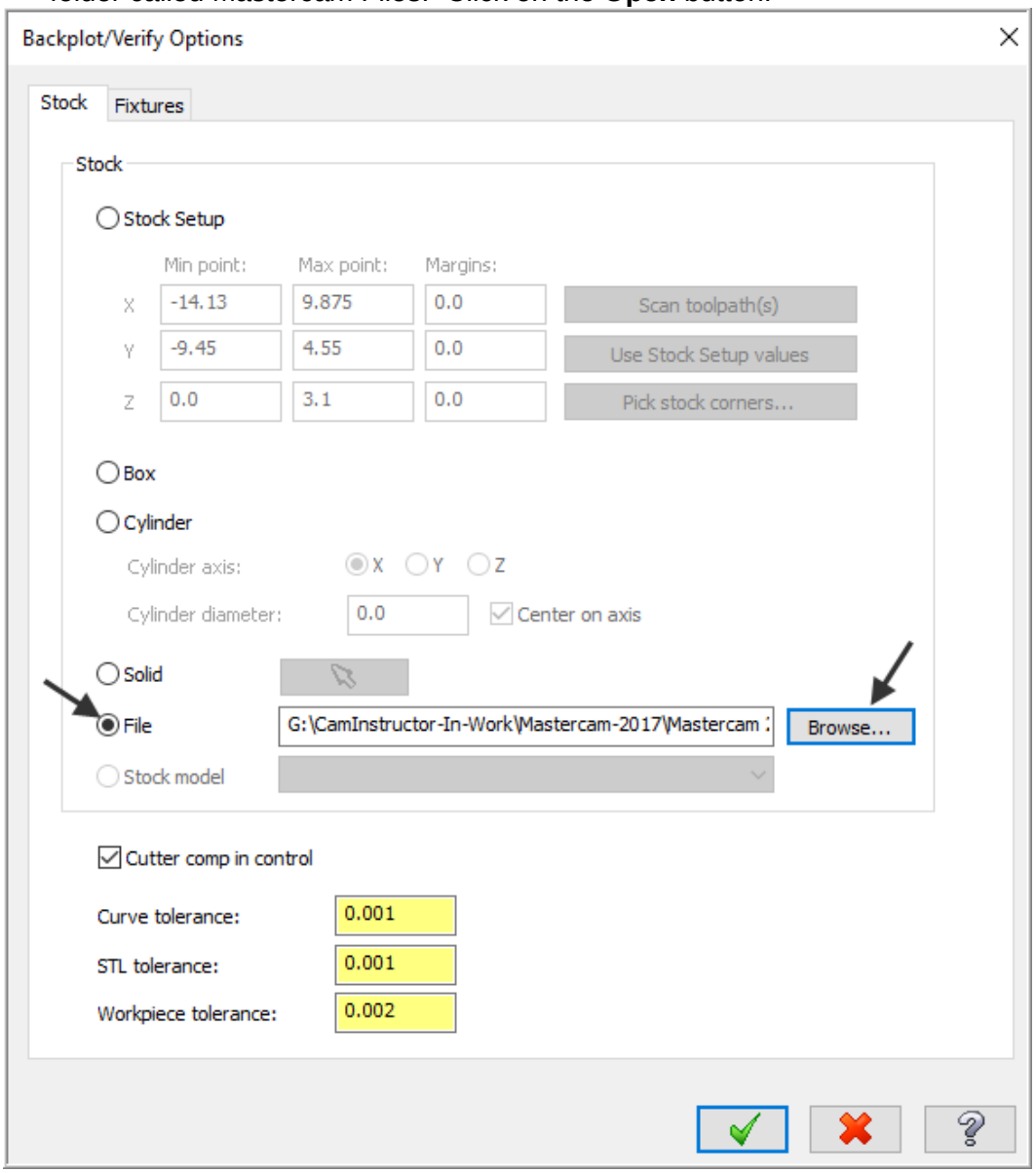
1. Right mouse click in the graphics area and click on **Isometric**.
2. Right mouse click in the graphics area and click on the **Fit** icon  .
3. In the Toolpath Manager left click ***only*** on the **FBM Drill Toolpath Group**.



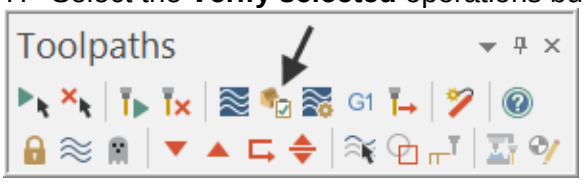
4. Click on the **Backplot/Verify Options** icon.



5. In the **Backplot/Verify Options** change the **Stock** selection to **File**. Browse and select **MILL-LESSON-FBM-2.STL** as your **Stock file**. Or if you are using Mastercam HLE use the **Mill-Lesson-FBM-2.STL** file supplied on the multimedia DVD that came with this text in a folder called Mastercam Files. Click on the **Open** button.



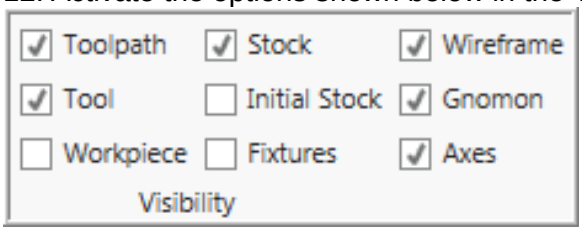
6. Select the OK button  to exit **Backplot/Verify Options**.
7. Select the **Verify selected** operations button shown below:



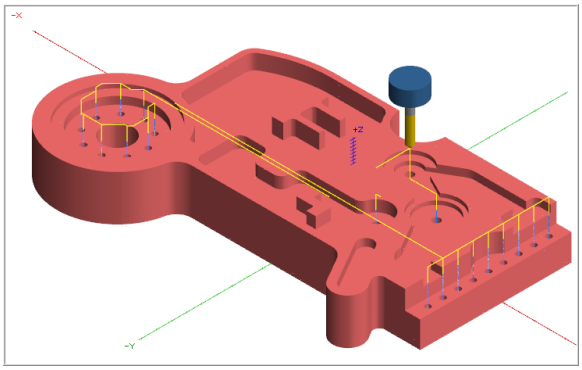
8. **Maximize** the Backplot/Verify window if required.
9. Activate the **Both** option in the **Toolpath** section of the **Verify** tab.
10. At the top of the screen select the **View** tab, the **Isometric** icon and then select **Fit**

11. Now select the **Home** Tab.

12. Activate the options shown below in the **Visibility** section of the Home tab.

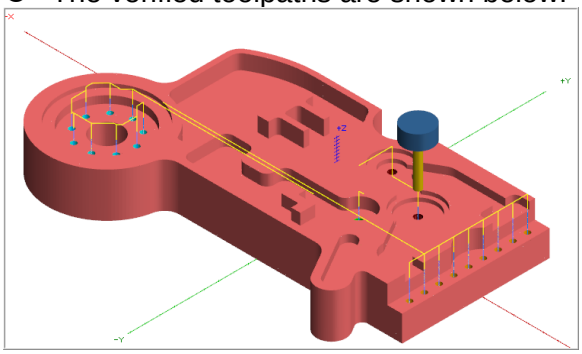


Your screen should show the STL file as below.



13. Activate the **Color Loop** to change the color of the tools for the verified part.
14. Open the **Stop Conditions** drop down menu and activate stop on Collision.
15. Now select the Play Simulation button to review the toolpaths.

➲ The verified toolpaths are shown below.



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

TASK 9:
SAVE THE UPDATED MASTERCAM FILE

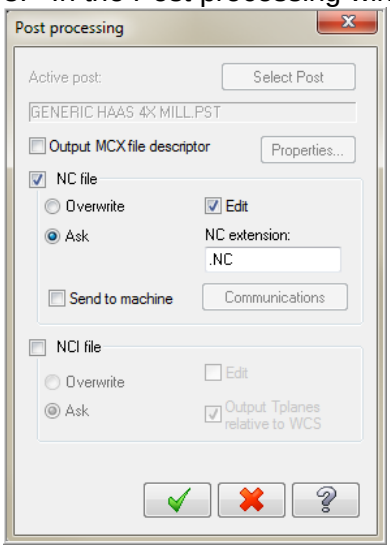
1. Select the **Save** icon from the **Quick Access** toolbar at the top left of the screen.

TASK 10:
POST AND CREATE THE CNC CODE FILE

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

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

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



4. Select the OK button  to continue.
5. Enter the same name as your Mastercam part file name in the NC File name field **Mill-Lesson-FBM-2**.
6. Select the Save button.
7. The CNC code file opens up in the default editor.
8. Select the  in the top right corner to exit the CNC editor.

➲ This completes Mill-Lesson-FBM-2.