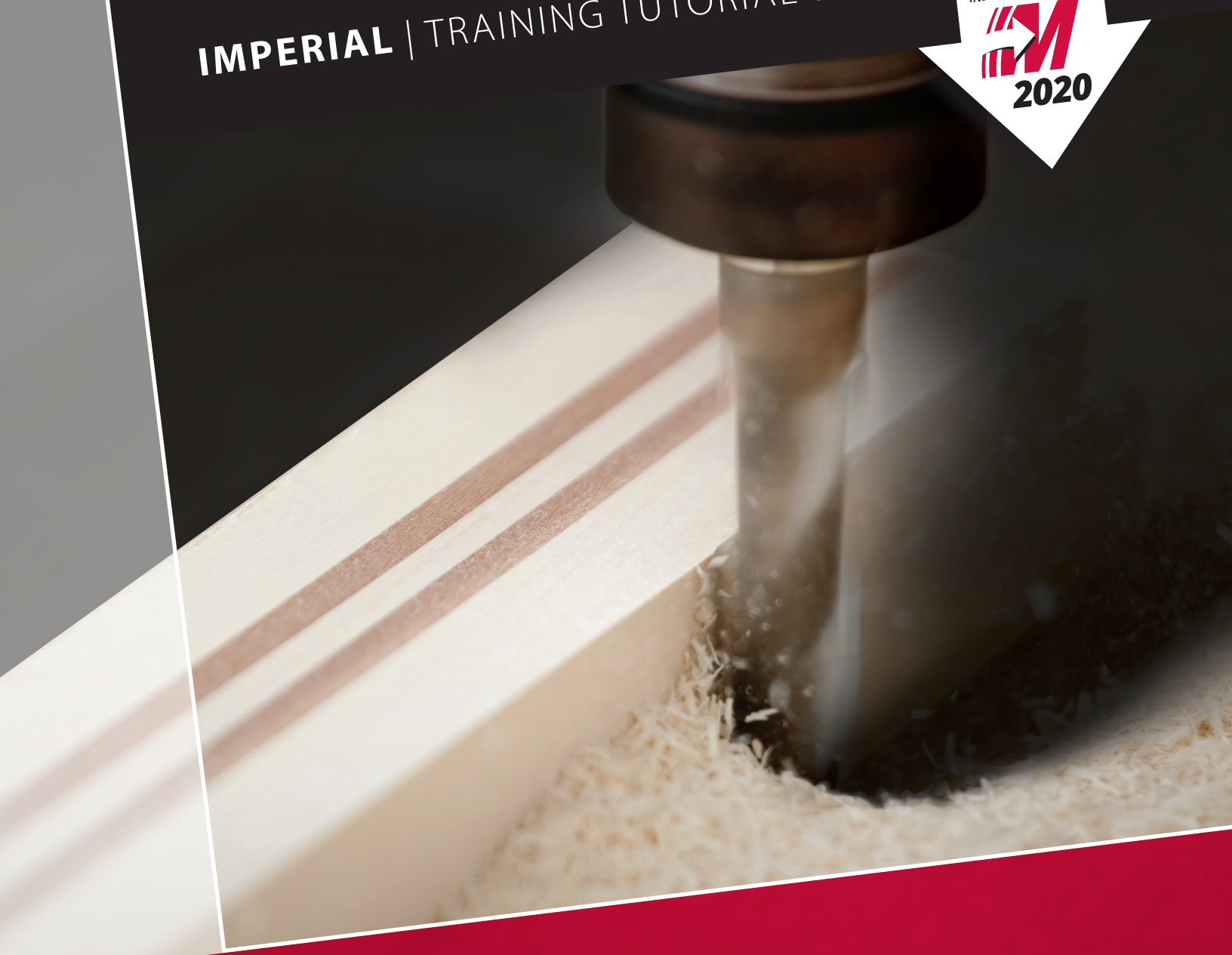
Mastercam® 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE DOWNLOAD
INSTRUCTIONS INSIDE
2020



ROUTER

Mastercam 2020

ROUTER TRAINING TUTORIAL

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2020 Router Training Tutorial

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-839-8

Date: August 26, 2019

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributer, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

Trademarks

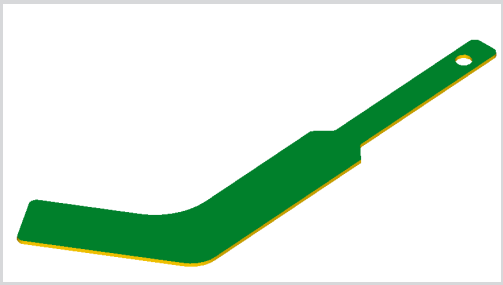
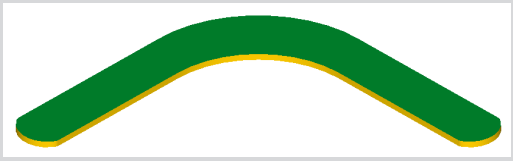
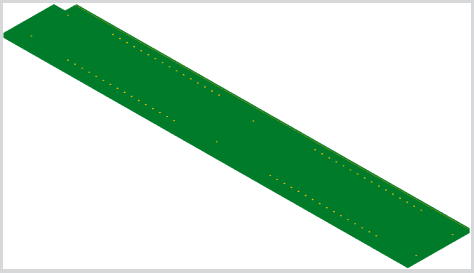
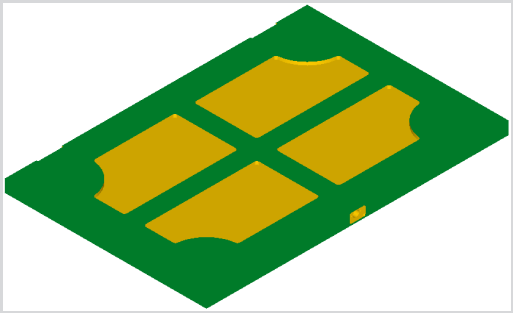
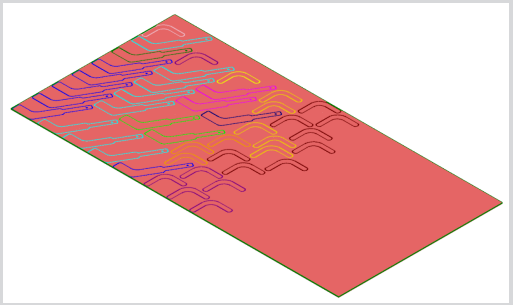
Mastercam is a registered trademark of CNC Software, Inc.

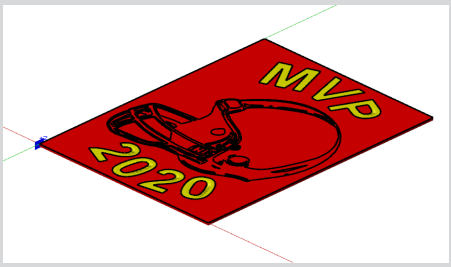
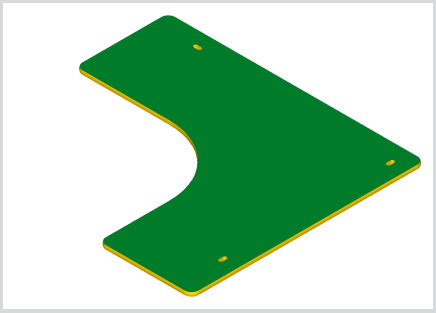
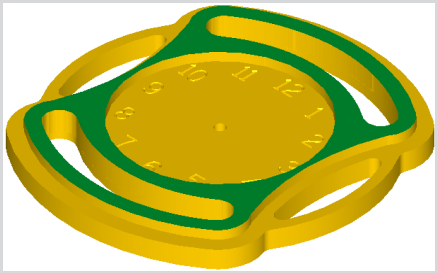
Microsoft, the Microsoft logo, are registered trademarks of Microsoft Corporation;

Windows 10 is a registered trademarks of Microsoft Corporation.

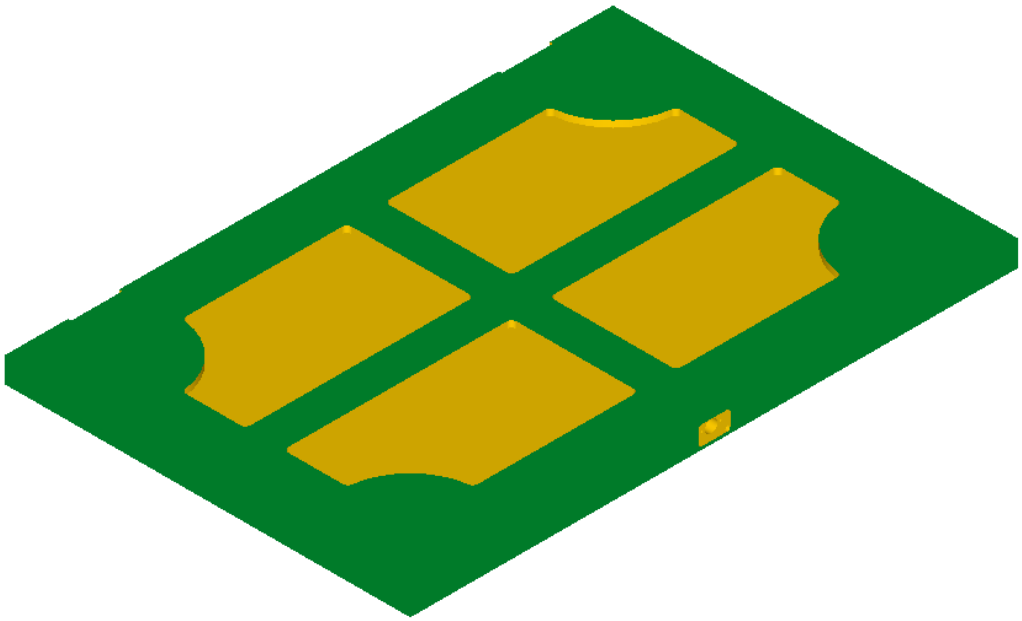
Table Of Contents

Router Projects

Tutorial	Geometry Functions	Toolpath Creation
#1 	Create Polar Line. Create Perpindicular Line. Create Parallel Lines. Trim Entities. Create Fillets. Xform Mirror. Xform Translate. Create Circle Center Point. Create Rectangle.	Create a Drilling Toolpath. Create a Contour Toolpath.
#2 	Create Lines. Create Fillets. Create Arcs.	Create a Contour Toolpath.
#3 	Create Rectangle. Create Parallel Lines. Create Circle Center Point. Xform Rectangular Array. Trim Geometry. Delete Construction Lines.	Create a Drilling Toolpath. Create a Block Drilling Toolpath. Create a Contour Toolpath.
#4 	Create Door. Xform Translate. Change Graphic View and Construction Plane. Create Parallel Lines. Create Rectangles. Delete Construction Lines. Create Circle Center Point. Create Rectangular Shapes.	Create a Pocket Toolpath. Create a Engraving Toolpath. Create Toolpaths on Left and Right Plane. Create Drilling Toolpaths. Create a Circle Mill Toolpath.
#5 	Open Tutorial #1. Merge Tutorial #2. Xform Geometry Nesting. Option #2 (True Shape Nesting). Option #3 (Rectangular Nesting The Toolpaths).	Create Drill Toolpath. Create a Contour Toolpath.

Tutorial	Geometry Functions	Toolpath Creation
#6 	Download the File. Use RAST2VEC.DLL to Open The File. Create a Point. Create Letters.	Create a Contour Toolpath. Create an Engraving Toolpath.
#7 	Create Rectangle. Create Line Parallel. Create Fillets. Create Rectangular Shapes. Create a Custom Tool. Trim Entities. Create Arc Endpoints. Delete Construction Lines.	Create a Slot Mill Toolpath. Create Contour Toolpath Using The Custom Tool Created.
#8 	Download the File.	Create a Circle Mill Toolpath. Create a Drilling Toolpath. Create a Engraving Toolpath. Create a Contour (Ramp) Toolpath. Create a Pocket Toolpath.

Tutorial 4



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

From Drawing to CAD Model:

- The student should examine the drawing on the following page to understand what part is being created in the tutorial.
- From the drawing we can decide how to go about creating the geometry in Mastercam.

Create the CAD Model used to generate Toolpaths from:

- The student will create the wireframe needed to create the surfaces.
- Create a door using the Door command.
- Create side holes.

Create the necessary Toolpaths to machine the part:

- The student will set up the stock size and the clamping method used.
- Machine the pockets using a straight bit.
- Machine the pocket walls using an engraver tool to create a chamfer.
- Drill the two side holes.
- Use an aggregate to mill the door jam.

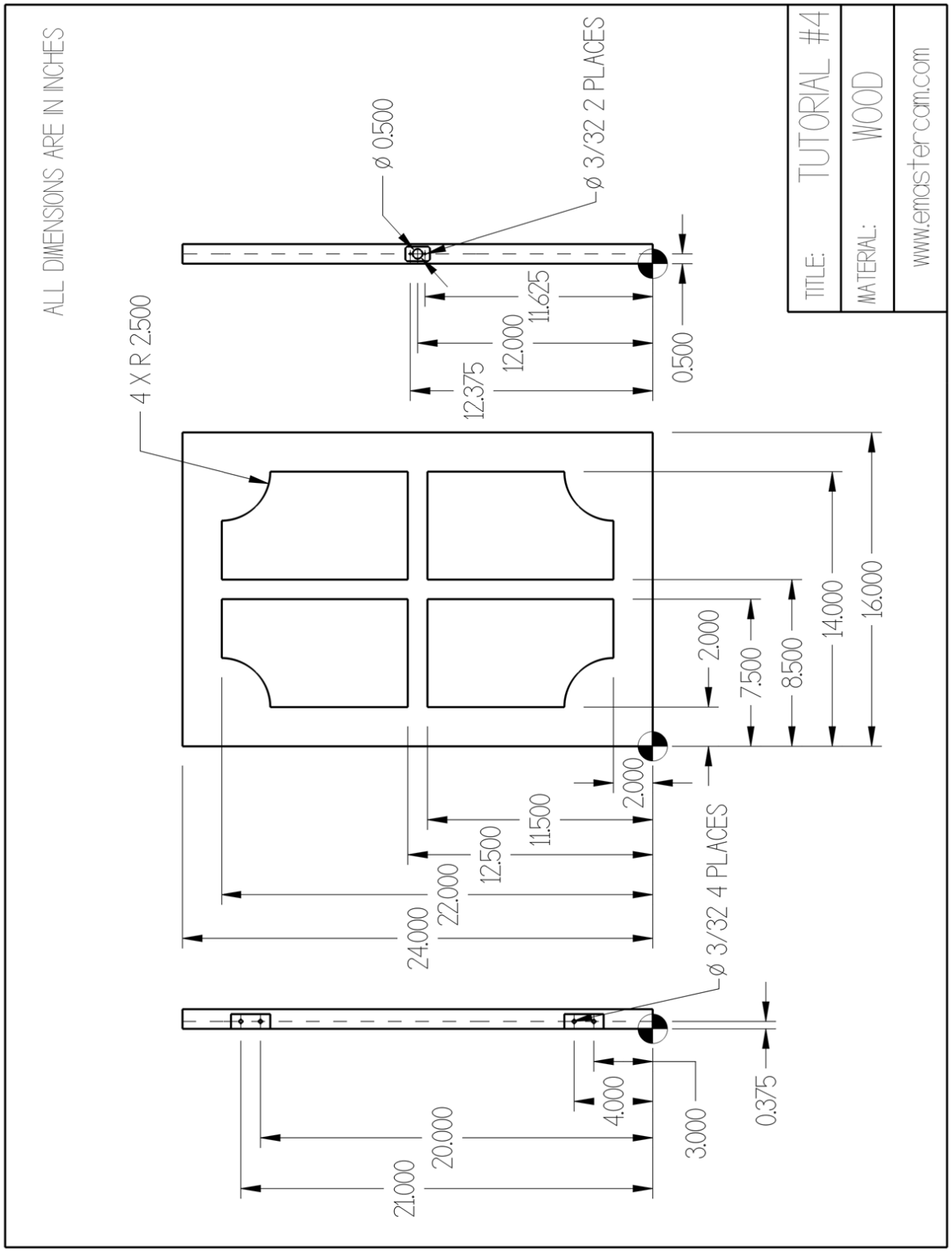
Backplot and Verify the file:

- Backplot will be used to simulate a step by step process of the tool's movements.
- Verify will be used to watch a tool machine the part out of a solid model.

Post Process the file to generate the G-code:

- The student will then post process the file to obtain an NC file containing the necessary code for the machine.

This tutorial takes approximately one hour to complete.


ALL DIMENSIONS ARE IN INCHES
TITLE: TUTORIAL #4
MATERIAL: WOOD
www.emastercam.com
ø 0.500
ø 3/32 2 PLACES
4 X R 2.500
11.625
12.000
12.375
0.500
2.000
7.500
8.500
14.000
16.000
2.000
11.500
12.500
22.000
24.000
ø 3/32 4 PLACES
0.375
3.000
4.000
20.000
21.000

GEOMETRY CREATION

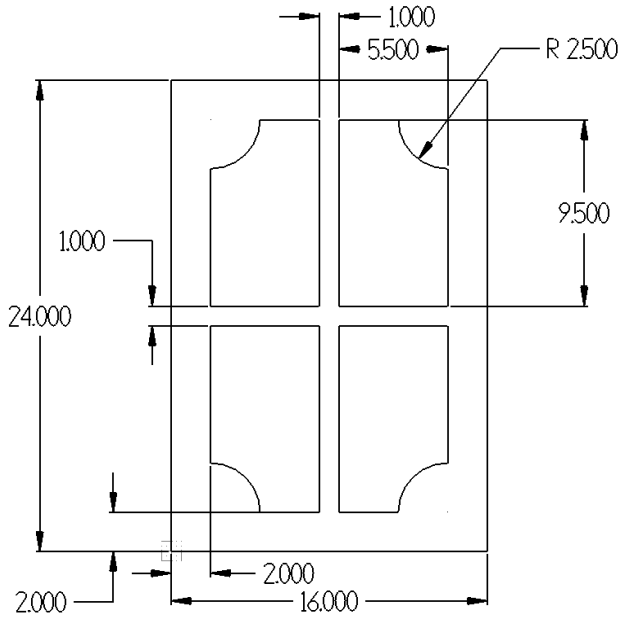
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the Getting Started section to set up the graphics user interface.

STEP 2: CREATE A DOOR

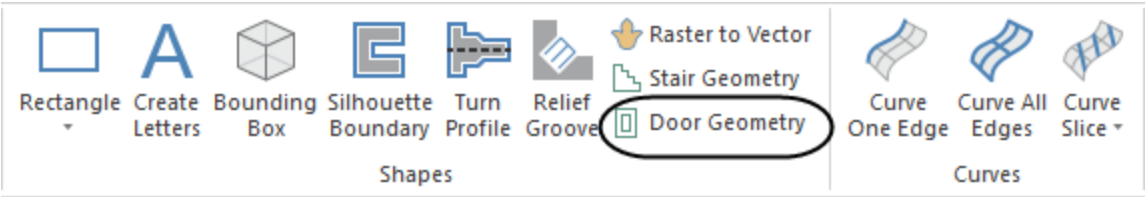
In this step you will learn how to create a door using the **Create Door Geometry** command. This command automates the process so you are required to input only a few values to create a door.

Step Preview:

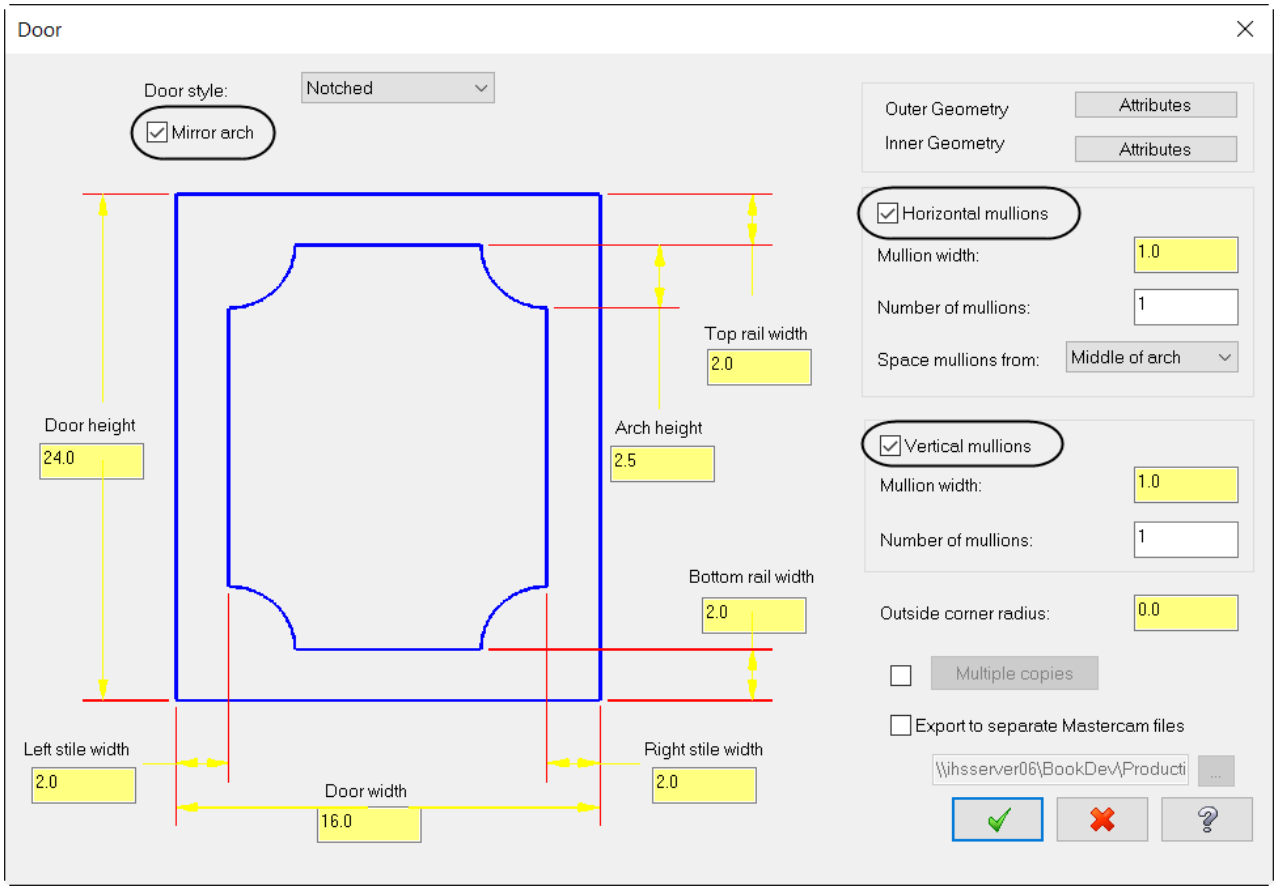


Wireframe

- From the **Router** group, select **Create Door Geometry** as shown.

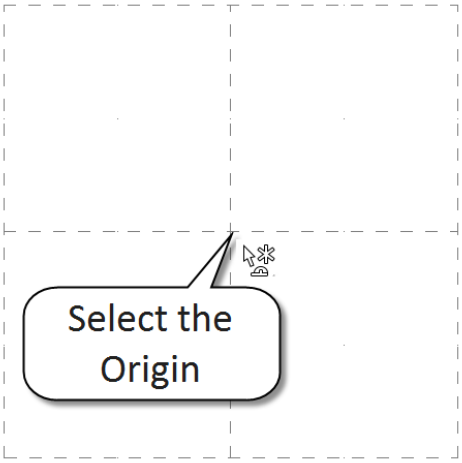


♦ Select the Door style **Notched** and modify the door size and inside shape dimensions as shown.

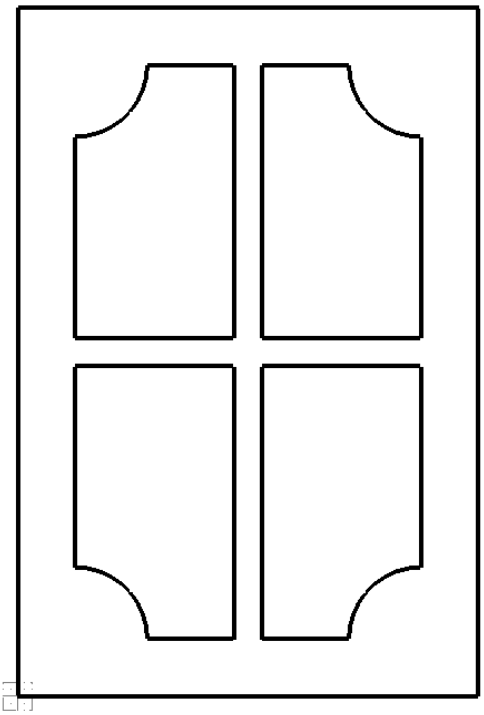


♦ Select the **OK** button once the parameters have been changed to create the door. 

- To place the lower left corner of the door select the **Origin** as shown.



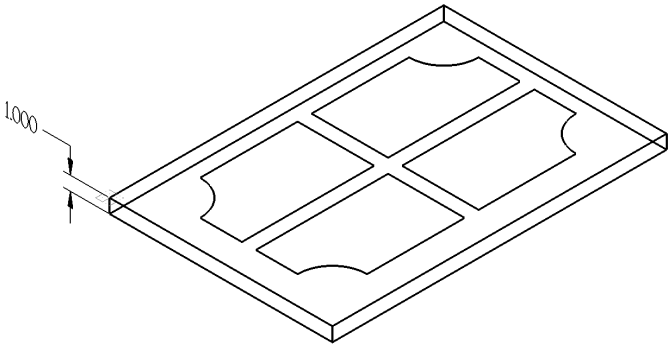
- Press **Alt + F1** to view the geometry.
- Press **Esc** to finish the command.
- The geometry should look as shown.



STEP 3: TRANSLATE THE GEOMETRY

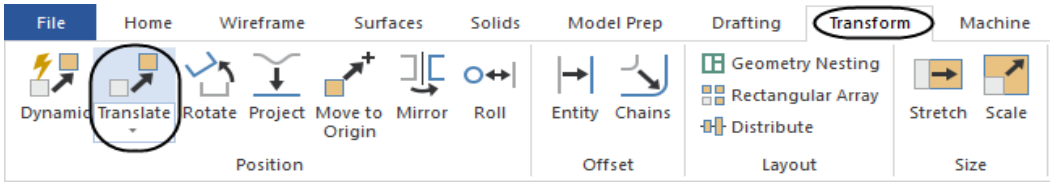
In this step you will translate the geometry to give the door a 3D appearance. This will help us when we create the holes for the hinges and striker holes.

Step Preview:



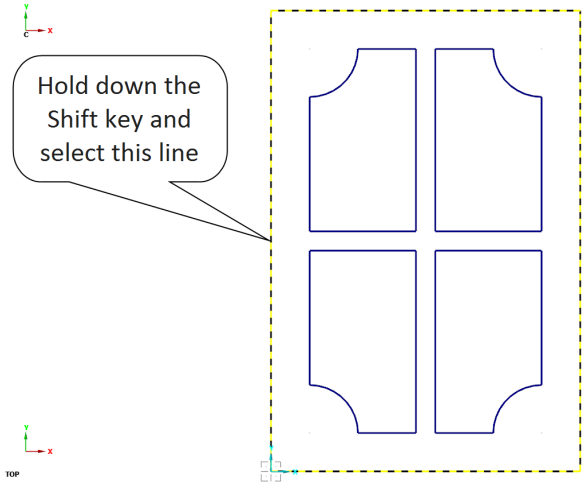
Transform

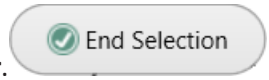
- From the **Position** group, select the **Translate** icon as shown.



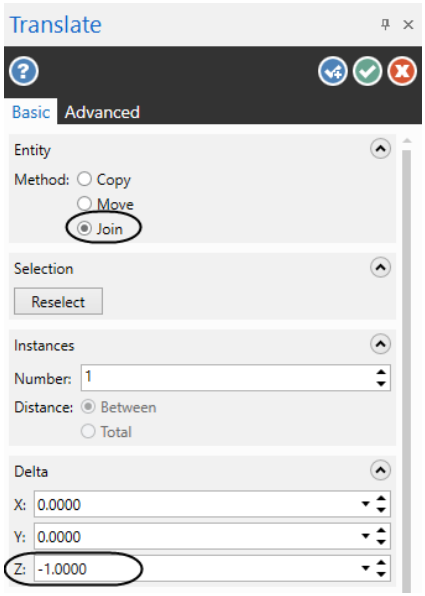
- Hold down the **Shift** key and select the line as shown.

Note: *Holding the **Shift** key allows you to select multiple entities that are touching each other creating a chain.*



- Once the entities are selected click on the **End Selection** button or press **Enter**. 

- In the **Translate** dialog box enable **Join** and enter a **Delta Z** value of **-1.0** as shown.

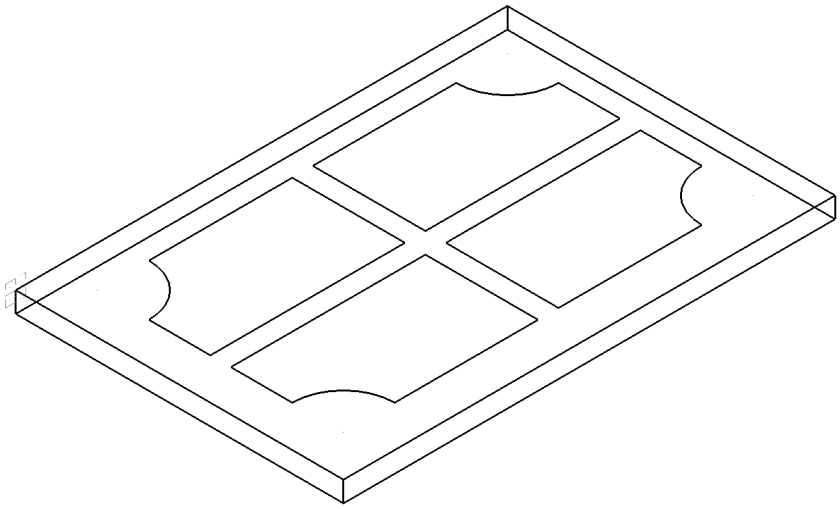


- Select the **OK** button to exit the **Translate** panel. 
- Right mouse click on the graphics window and select the **Clear Colors** button to reset the color to the system color.

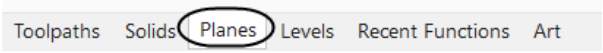
STEP 4: CHANGE THE GRAPHIC VIEW AND CONSTRUCTION PLANE

You will change the Graphics view to Isometric to view the translation and change the Construction plane to the Right plane so we can create geometry on the right side of the door.

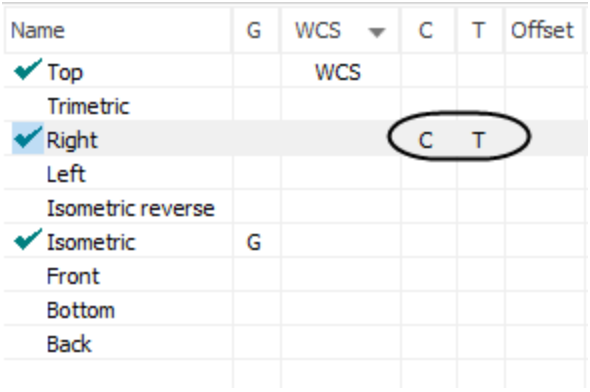
Step Preview:



- Select the **Planes** tab to open **Planes Manager** as shown.



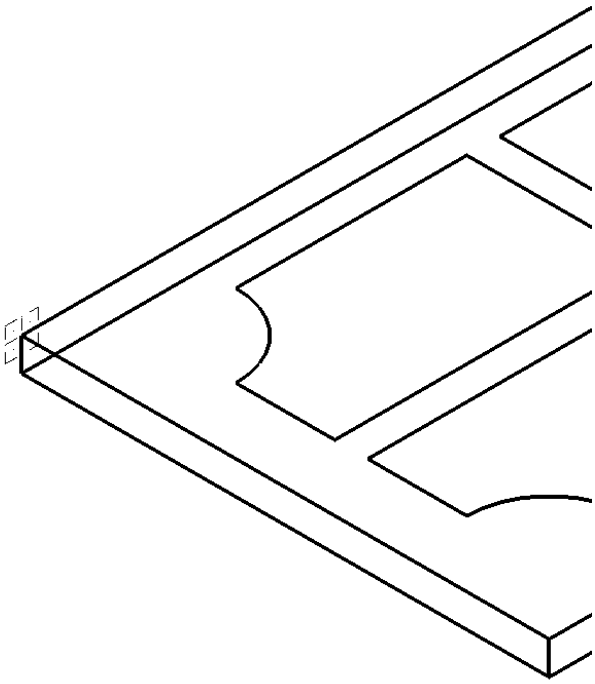
- Set the **Construction plane** and the **Toolpath plane** to the **Right** as shown.



Note: *If you select the Isometric Graphics view at anytime while being in a different plane, the planes will be reset to Top.*

- Press **Alt + F1** to fit the geometry to the screen.

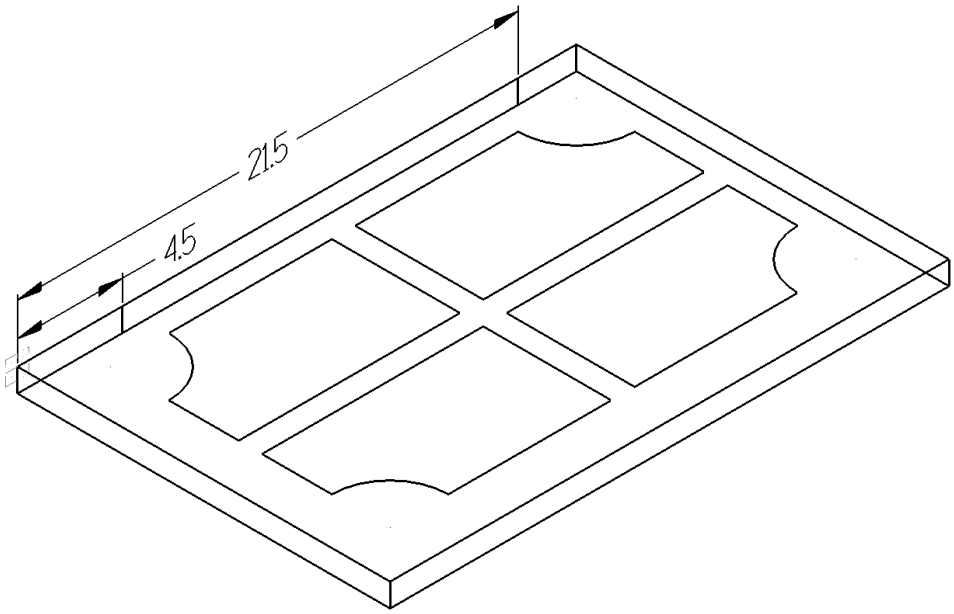
♦ The grid should look as shown.



STEP 5: CREATE PARALLEL LINES

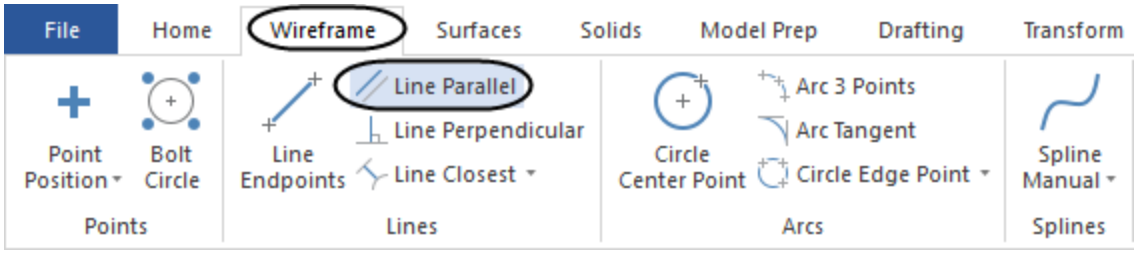
You will create parallel lines in the right plane, knowing the offset direction and distance. These lines will be used to help us establish the side pocket locations.

Step Preview:

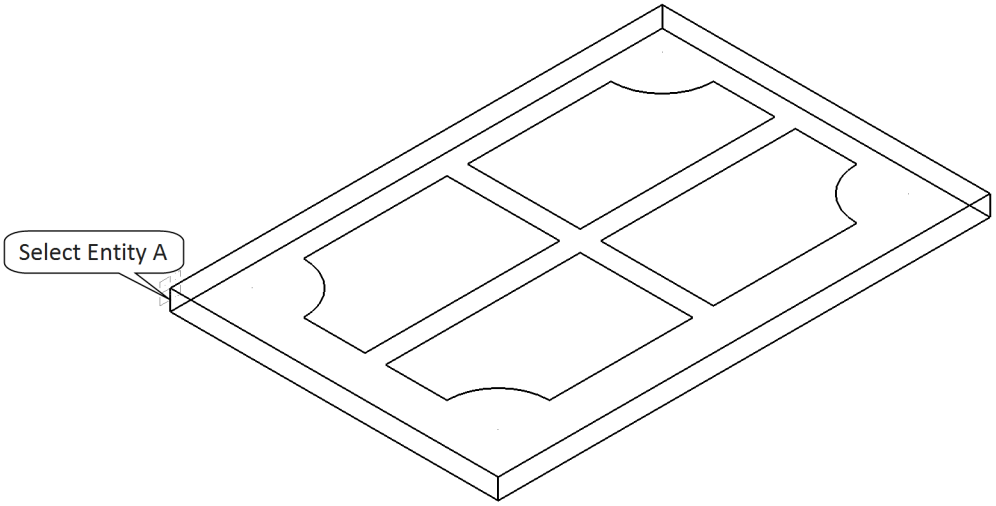


Wireframe

- From the **Lines** group, select **Line Parallel** as shown.



- [Select a line]: Select **Entity A** as shown.



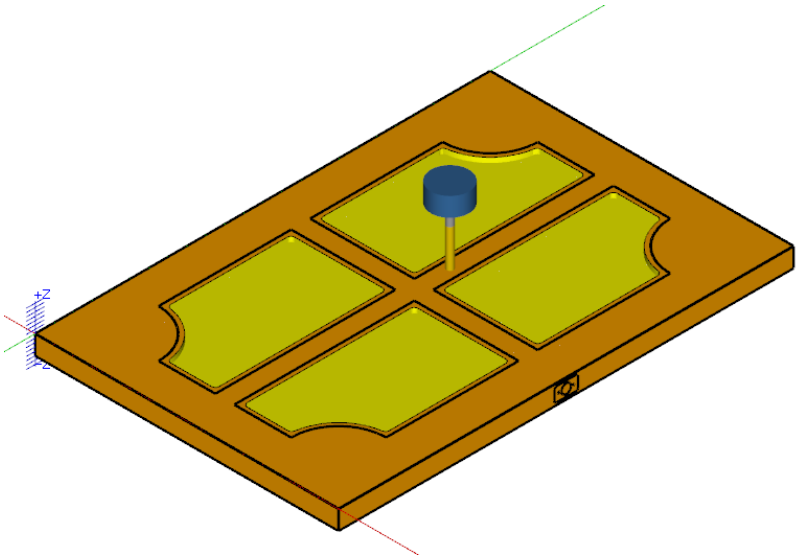
- [Select the point to place the parallel line through]: Select a point to the right of the selected line.

Note: *The color of the line is cyan which means the entity is "live" and you can still change the entities parameters.*

STEP 13: POCKET THE INSIDE PROFILES

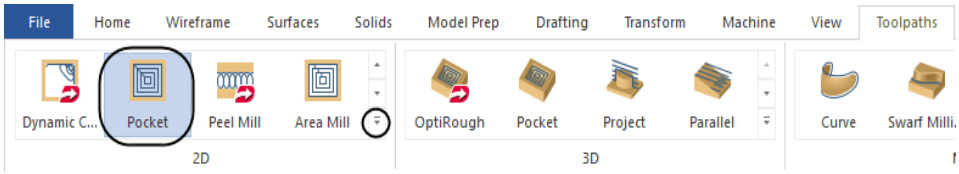
Pocket toolpath is used to clean out material from an enclosed boundary. You will create a pocket toolpath to cut the inside profiles. This will give the door a 3D appearance.

Step Preview:

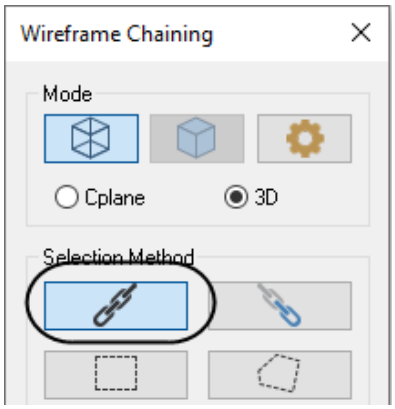


Toolpaths

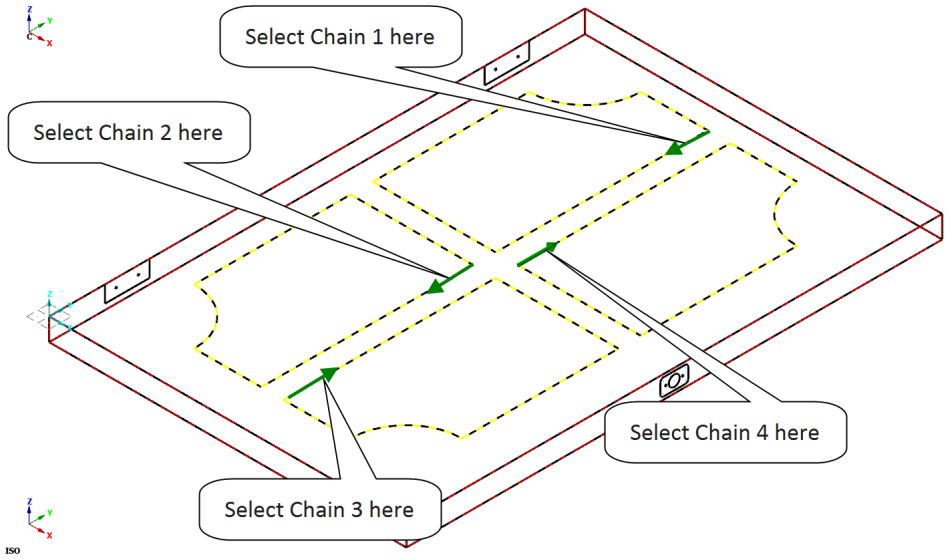
- From the **2D** group, select **Pocket** as shown.



- Make sure that the **Chain** button is selected in the **Chaining** dialog box.



- Select the four chains in a CW direction and in the order shown.



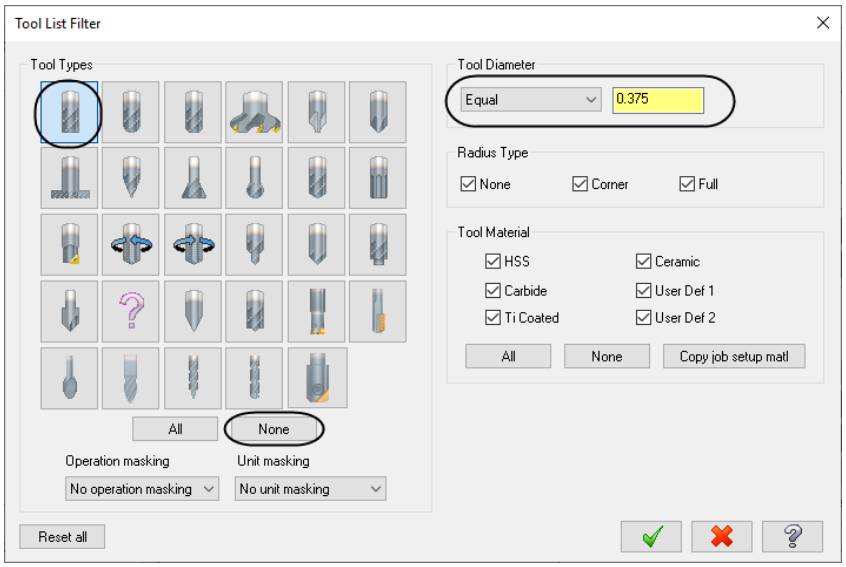
Note: *The arrows shown disappear as you select the next chain. They are showing you the chaining direction.*

- Select the **OK** button to exit the **Chaining** dialog box.
- On the **Toolpath Type** page make sure that **Pocket** is selected.

13.1 Select a 3/8" Straight Bit from the library and set the tool parameters

- Select **Tool** from the **Tree view area**.
- Click on the **Select library tool** button. Select library tool...
- Then select the **Filter** button. Filter...

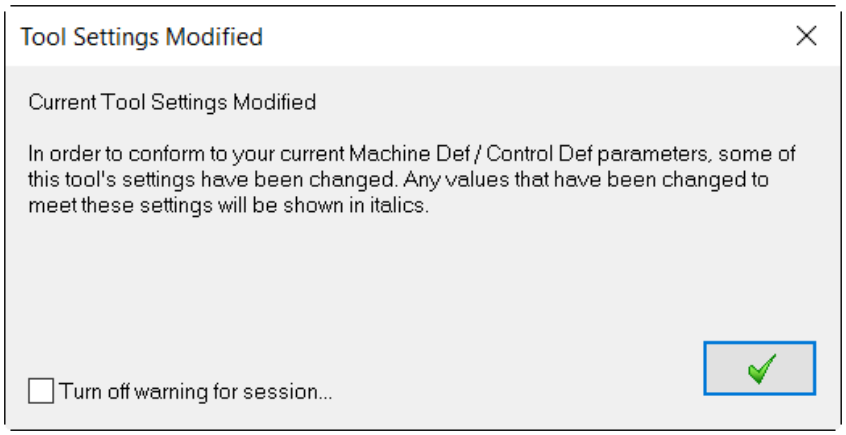
- Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon as shown.
- Under **Tool Diameter**, select **Equal** and enter the value **0.375**.



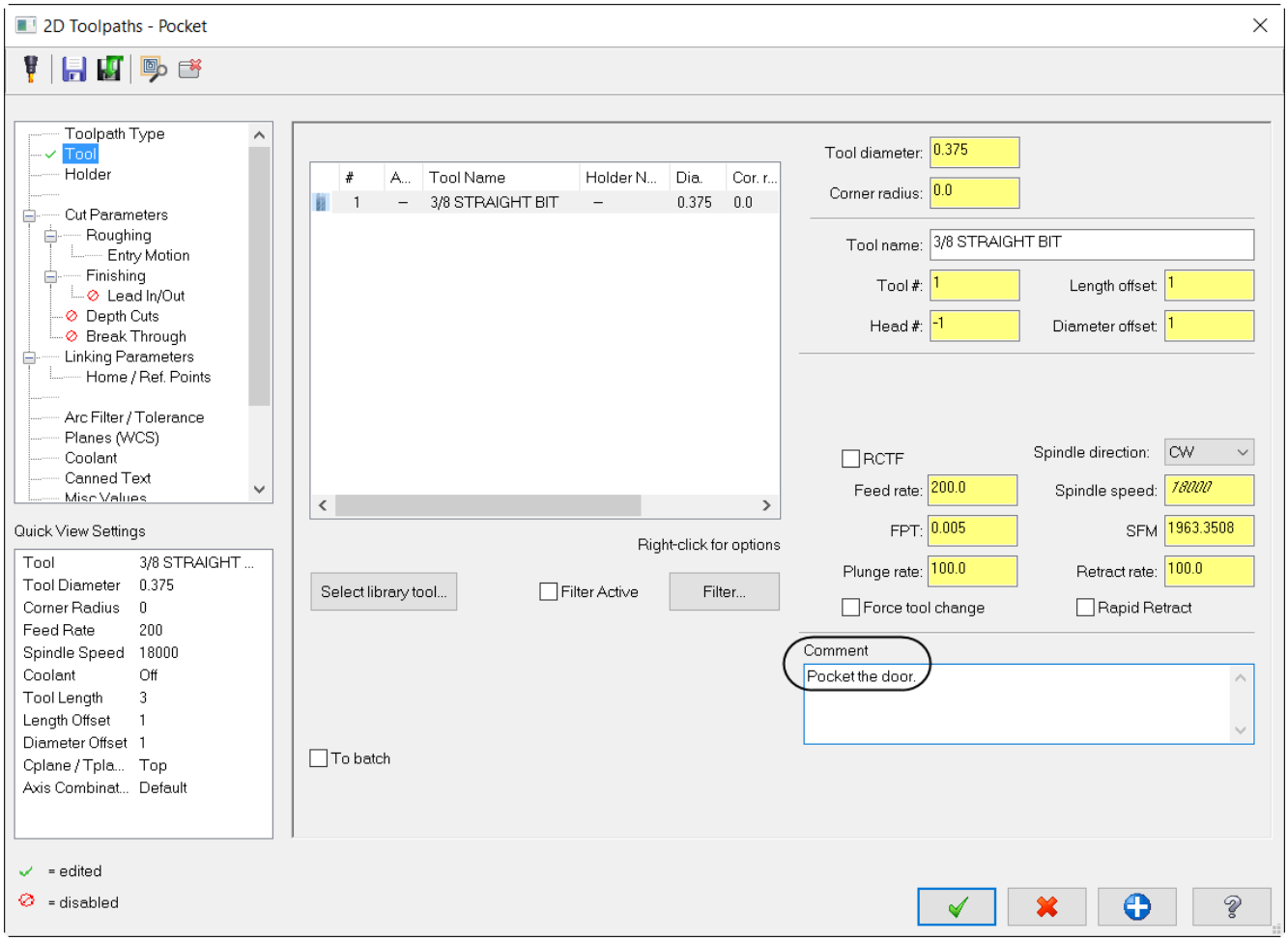
- Select the **OK** button to exit the **Tool List Filter**. 
- In the **Tool Selection** dialog box you should only see a **3/8" Straight Bit**.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
146	–	3/8 STRAIGHT BIT	–	0.375	0.0	2.0	2	Strai...	None

- Select the **3/8" Straight Bit** in the **Tool Selection** page and then select the **OK** button to exit. 
- Select the **OK** button to accept the **Tool Settings Modified** dialog box. 

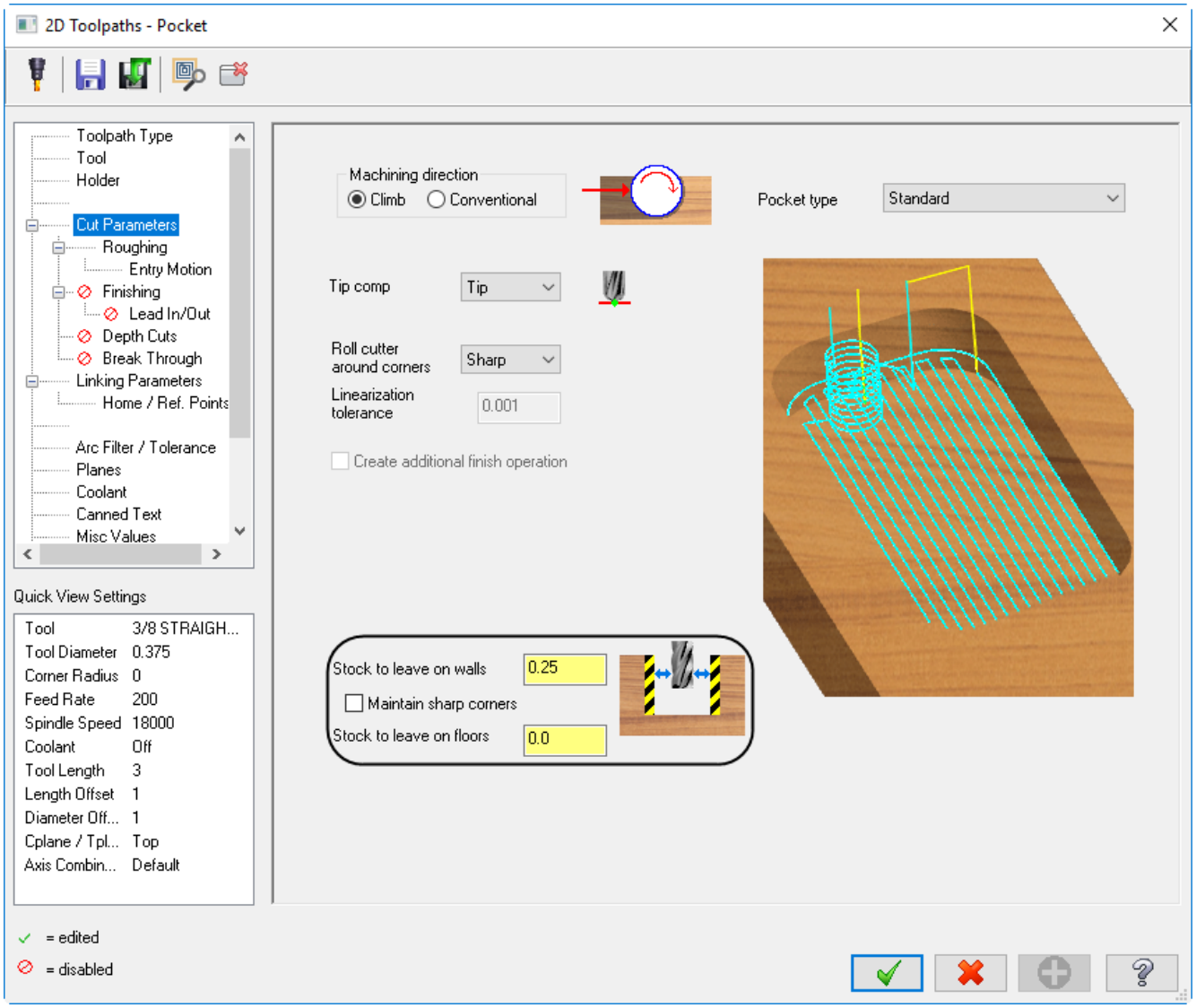


♦ Make all the necessary changes as shown.



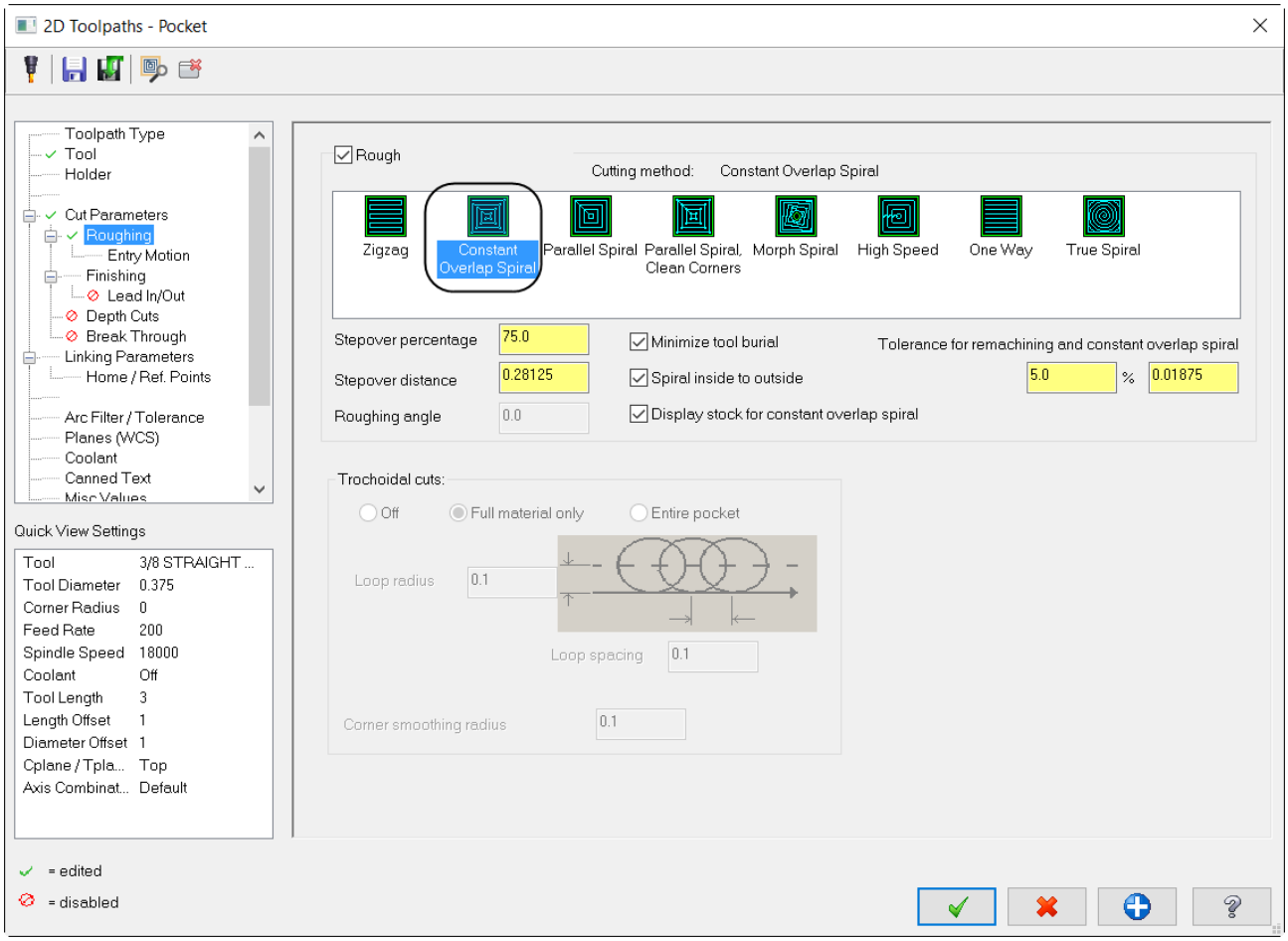
13.2 Set the Cut Parameters

- In the **Tree view area**, select **Cut Parameters**.
- Set the **Stock to leave on walls** to **0.25** and ensure the rest of the parameters appear as shown.



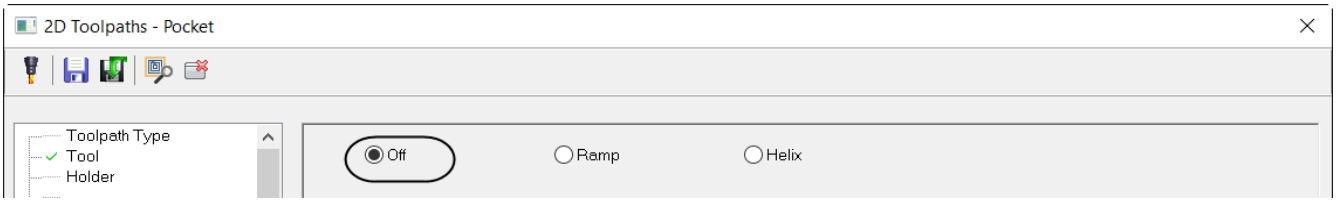
13.3 Set the Roughing Parameters

- Select **Roughing** in the **Tree view area**.
- Change the **Cutting method** to **Constant Overlap Spiral** as shown.



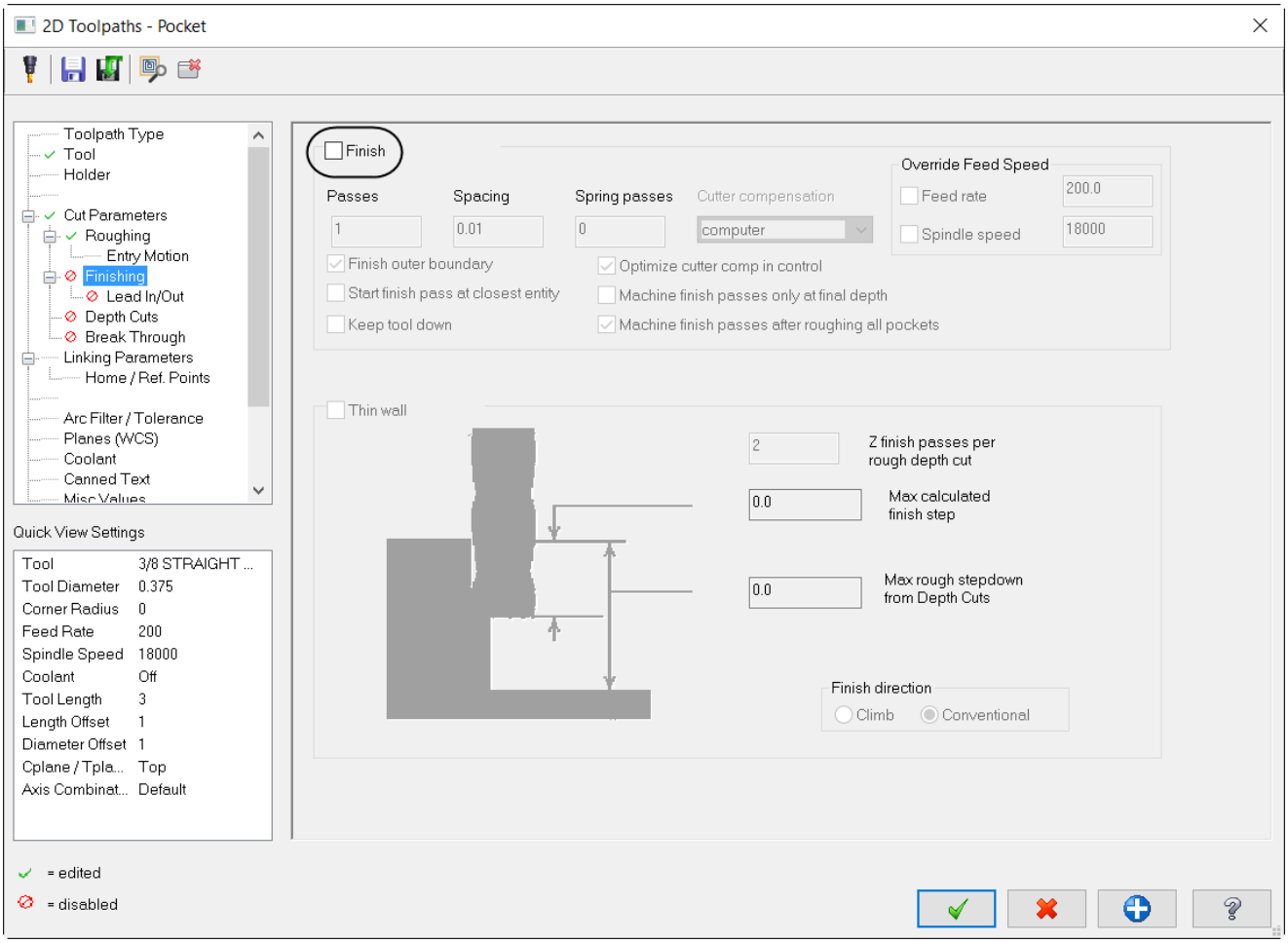
13.4 Set the Entry Motion

- Select **Entry Motion** from the **Tree view area**.
- Set the **Entry Motion** to **Off** as shown.



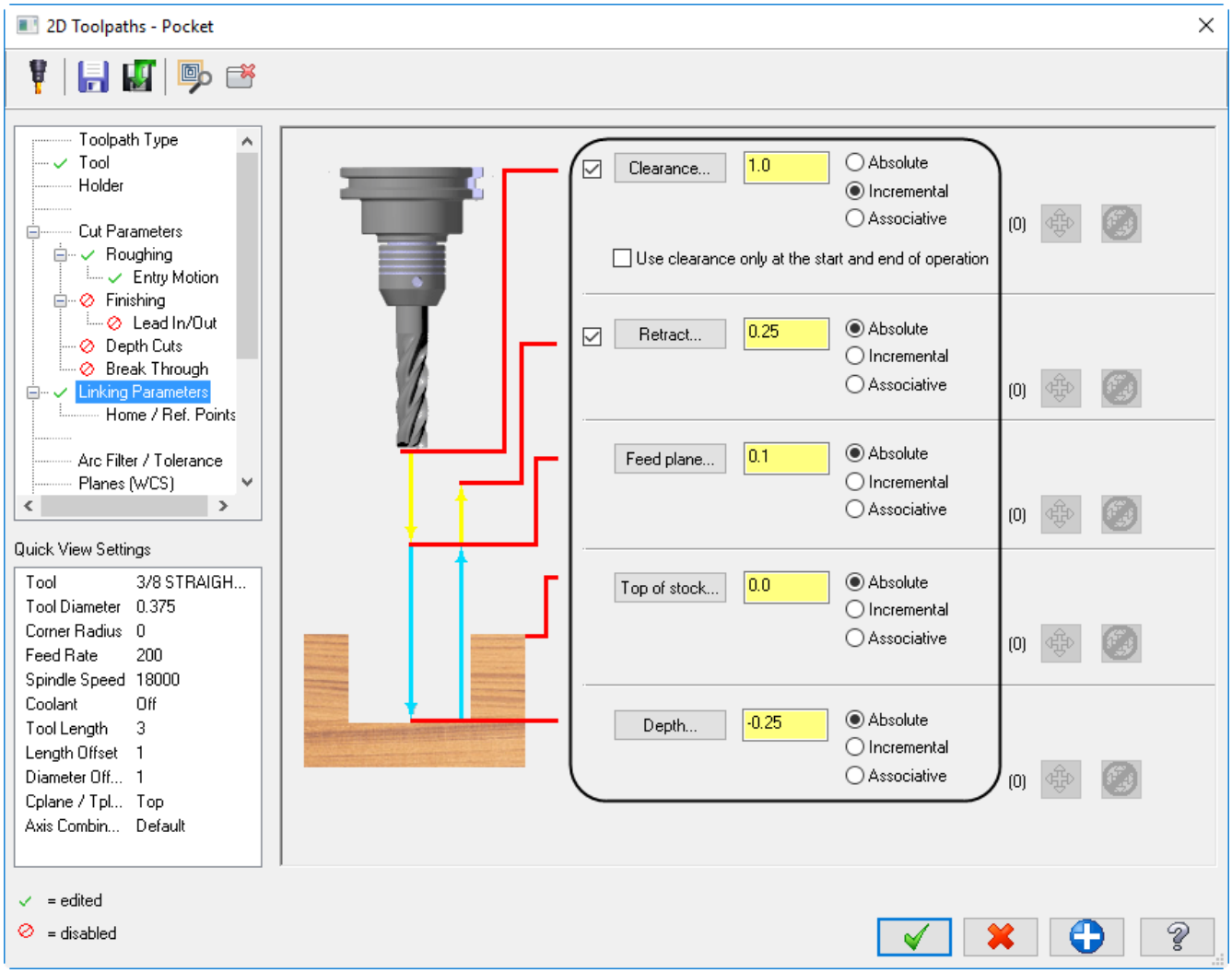
13.5 Set the Finishing Parameters

- Select **Finishing** and disable **Finish** as shown.



13.6 Set the Linking Parameters

- Select **Linking Parameters** from the **Tree view area**.
- Set the **Linking Parameters** as shown.

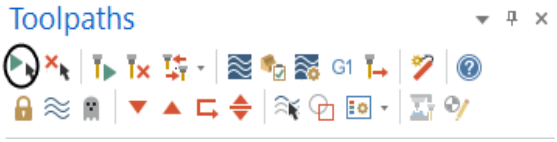


- Select the **OK** button to generate the toolpath. 

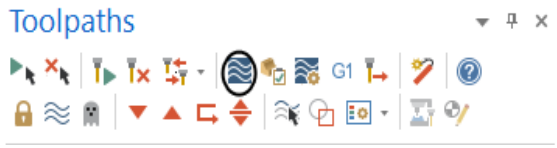
STEP 14: BACKPLOT THE TOOLPATHS

Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

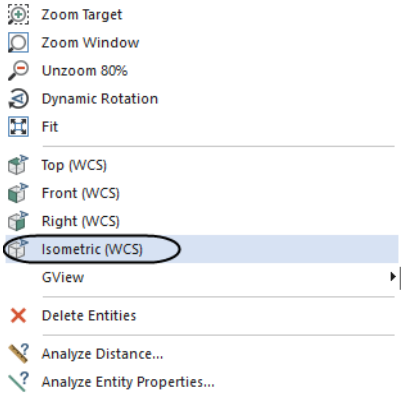
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon). If the operation is not selected choose the **Select all operations** icon.



- Select the **Backplot selected operations** button.

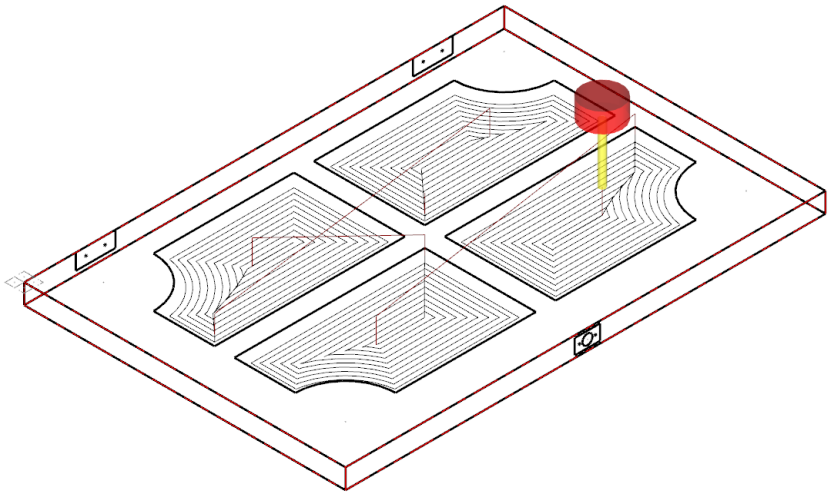


- Right mouse click in the graphics window and select **Isometric**.



- To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select **Fit**.
- You can step through the **Backplot** by using the **Step forward**  or **Step back**  buttons.
- You can adjust the speed of the backplot. 
- Select the **Play** button to run **Backplot**. 

♦ The toolpath should look as shown.

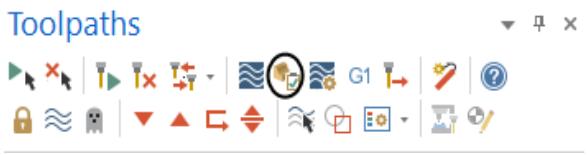


♦ Select the **OK** button to exit **Backplot**. 

STEP 15: SIMULATE THE TOOLPATH IN VERIFY

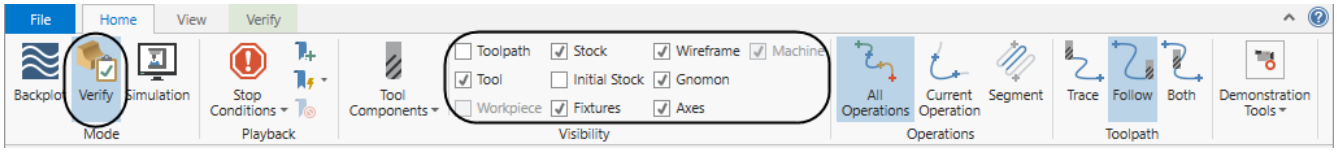
Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

♦ From the **Toolpaths Manager**, select **Verify selected operations** icon as shown.



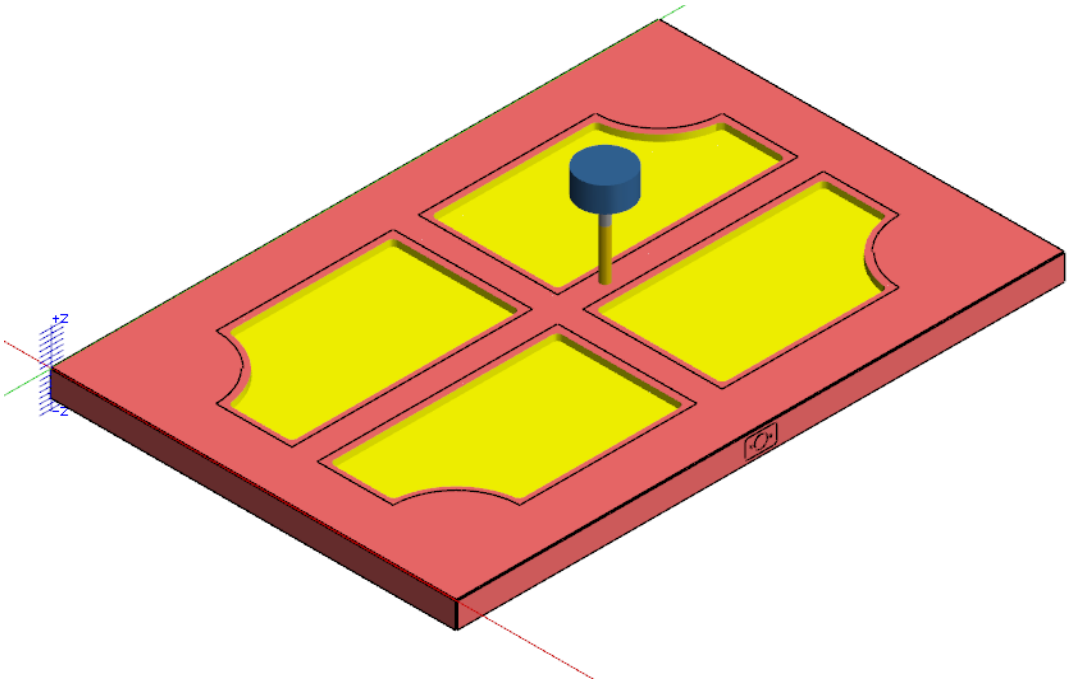
Note: *Mastercam launches a new window that allows you to check the part using* ***Verify****.*

♦ Change the settings for **Visibility** as shown.



♦ Select the **Play (R)** button to run **Verify**. 

♦ The part should look as shown.



Note: *To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction.*

*To **Zoom In** or **Out** hold down the mouse wheel and scroll up or down as needed.*

♦ To go back to Mastercam window, minimize **Mastercam Simulator** window as shown.