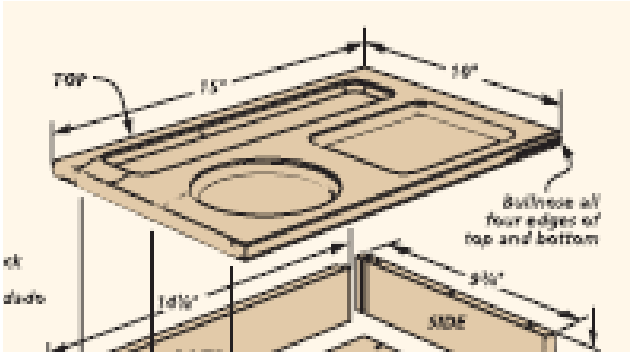
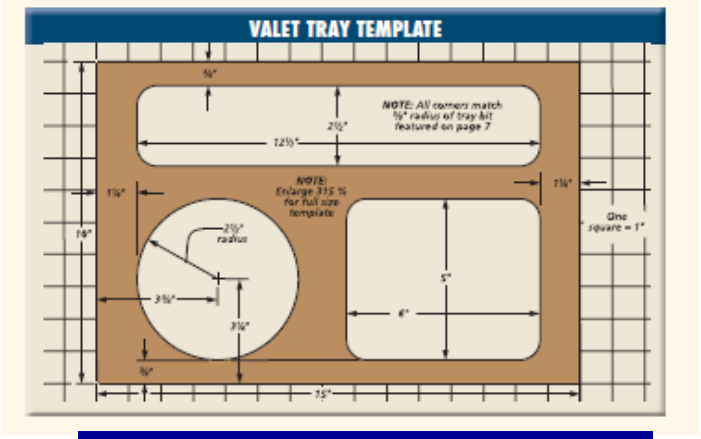
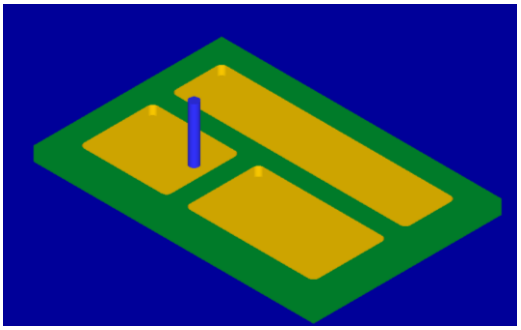
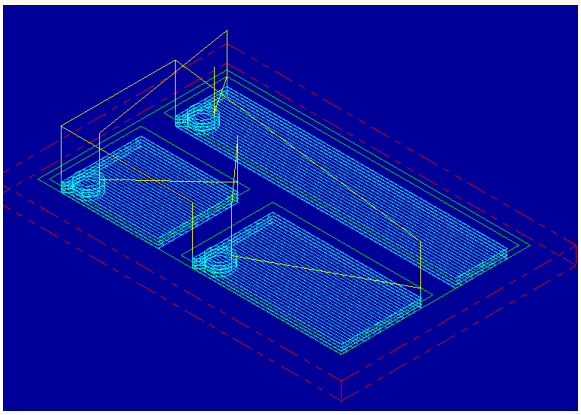
MasterCAM for Dresser Valet



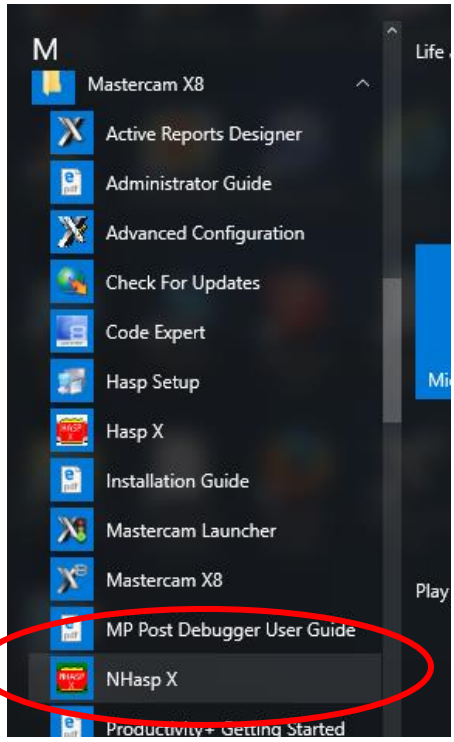




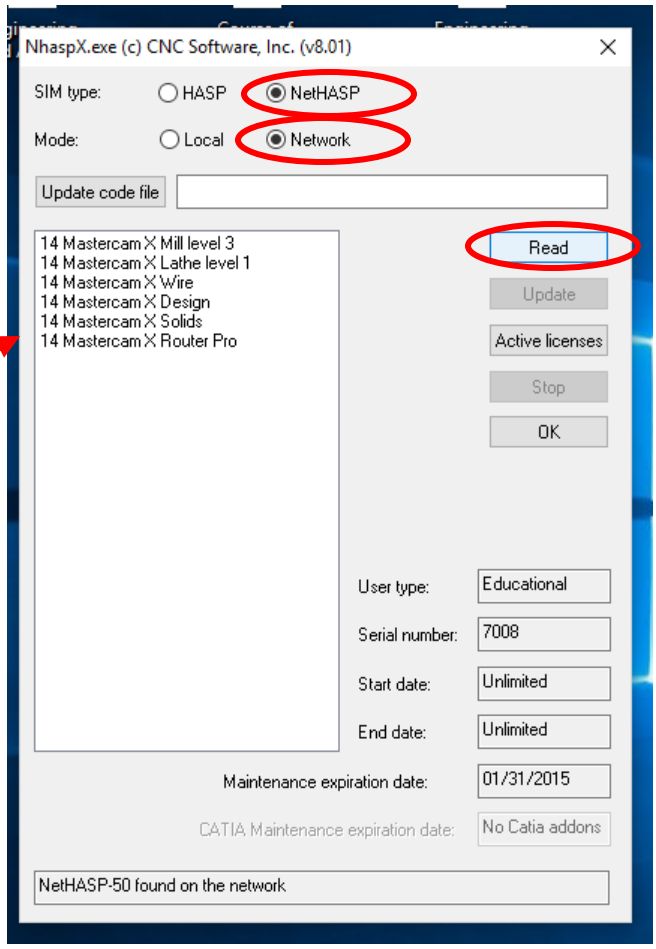


Check to make sure the nethasp is working/turned on to network.

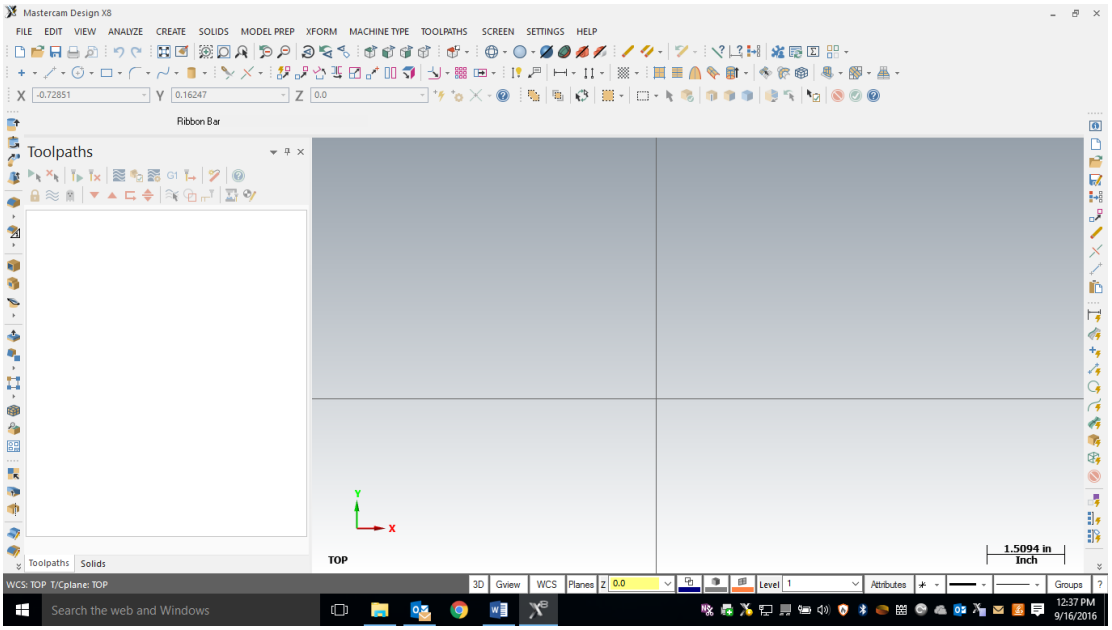
Go to ALL APPs/Mastercam x8/nethasp



After the computer “reads” the nethasp, these programs should show up. If not ask your instructor.

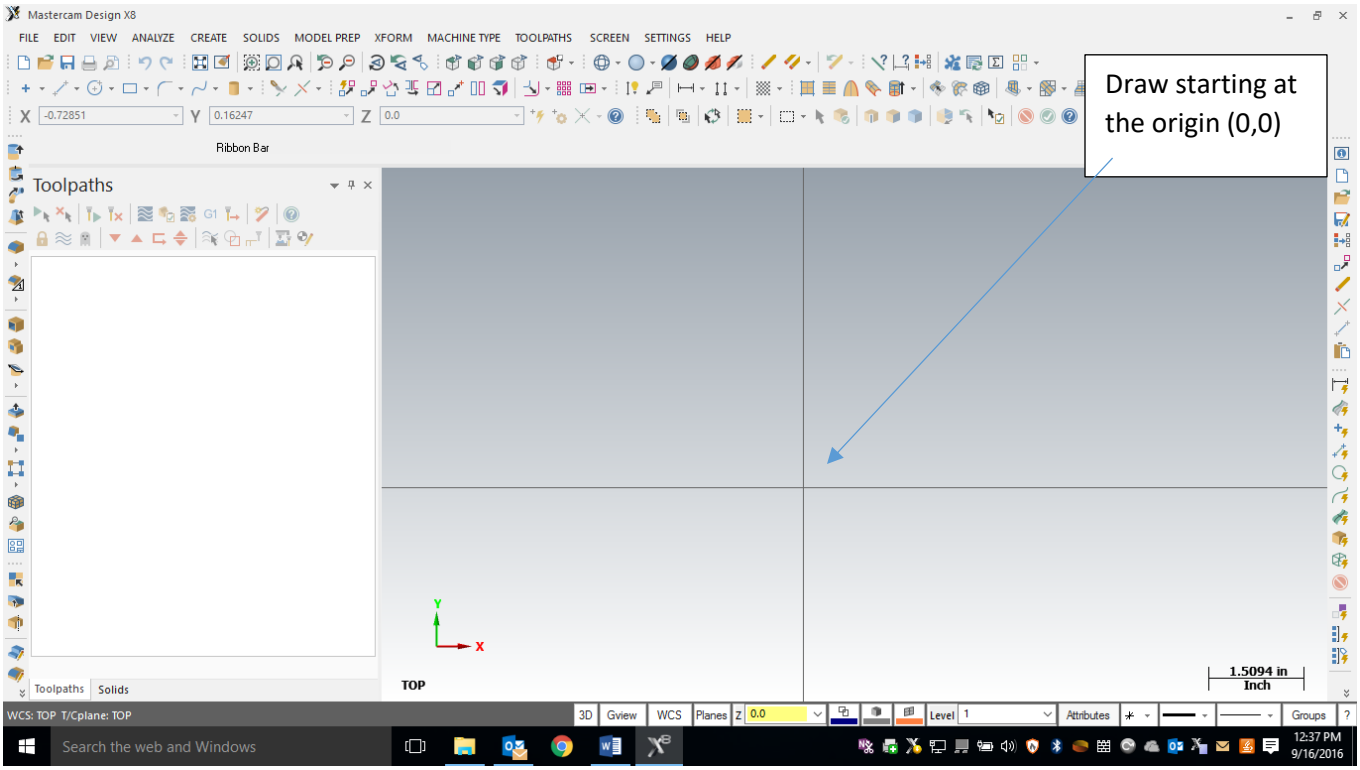


Open the MasterCAM application, it should look something like below.

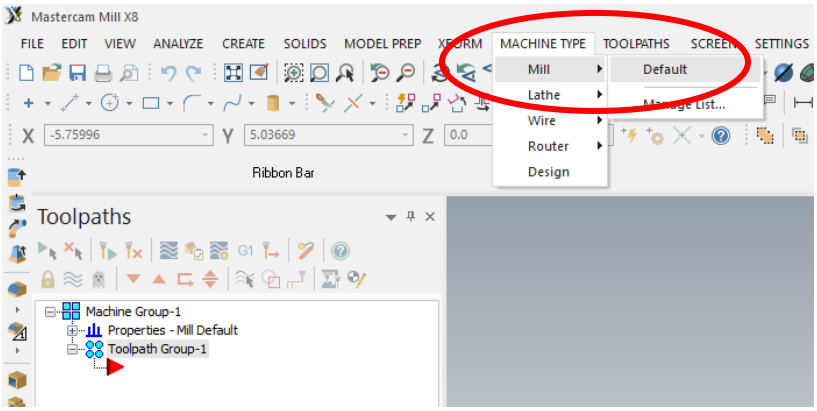


First thing is to figure out what you are making….Using the measurements from your plans or your adjusted measurements from your plans, you will draw your geometry (geometry is a generic term for lines, arcs, etc. in a computer drawing program). Personally I just draw on the piece of wood that is going to be cut and plot the points needed to draw the geometry. This geometry must be drawn in the 1st quadrant of the coordinate system, so positive x and y. The placement of the geometry matters since we will later be cutting out the part using the CNC Router. The CNC Router uses the coordinates from where you draw the geometry.

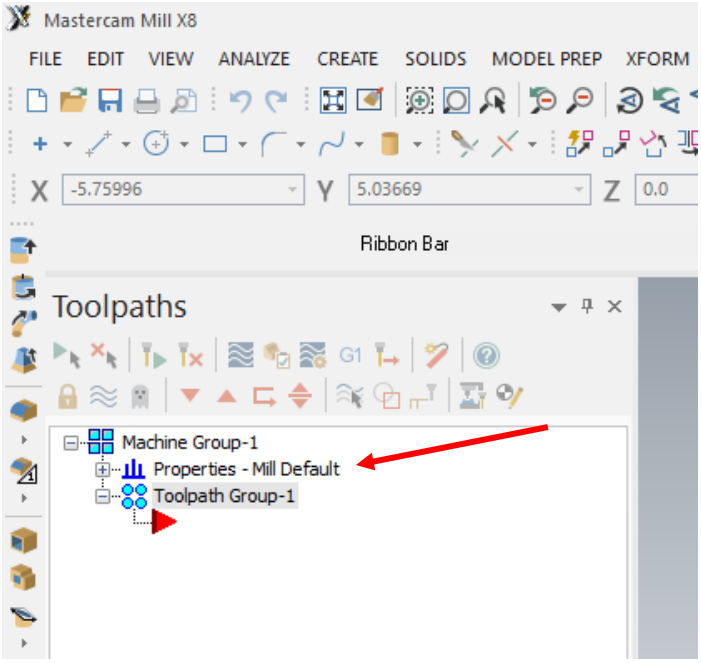
F9 will display the x/y axis such as:



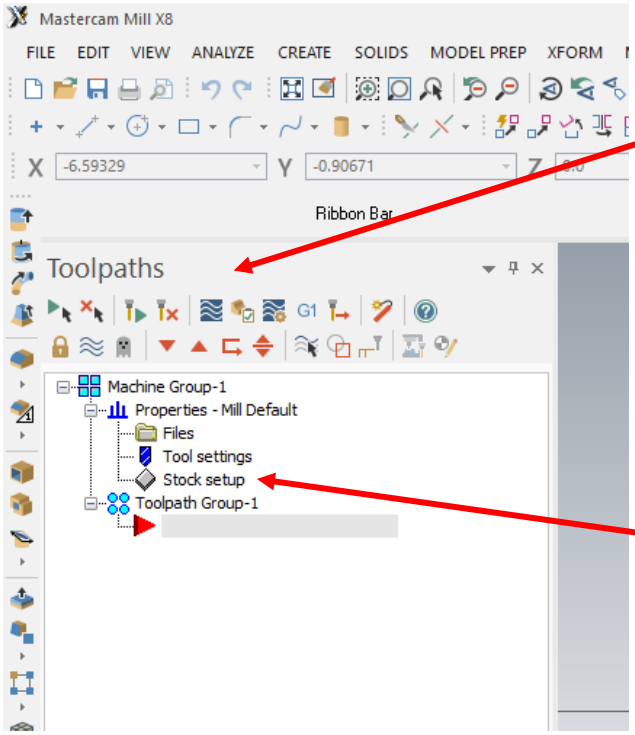
To start a project, we need to set our specific CNC router and set up the stock sizes. MasterCAM can write NC code for different manufacturers of CNC equipment. Our router is a Forest Scientific Velosity 3 axis mill. MasterCAM will write the correct type of code as long as we pick the correct machine definition. This is a critical first step, without the machine definition, the CNC router will crash….litterly the tool bit will dive into the table top. Your instructor will help you load the machine definition when you go to the cnc router, for the time being set the machine definition to default mill. **Goto Machine Type/Mill/Default.**



The result: there should be one machine group that says “Properties Default Mill,” if there is other Machine Groups, right-click and delete them.

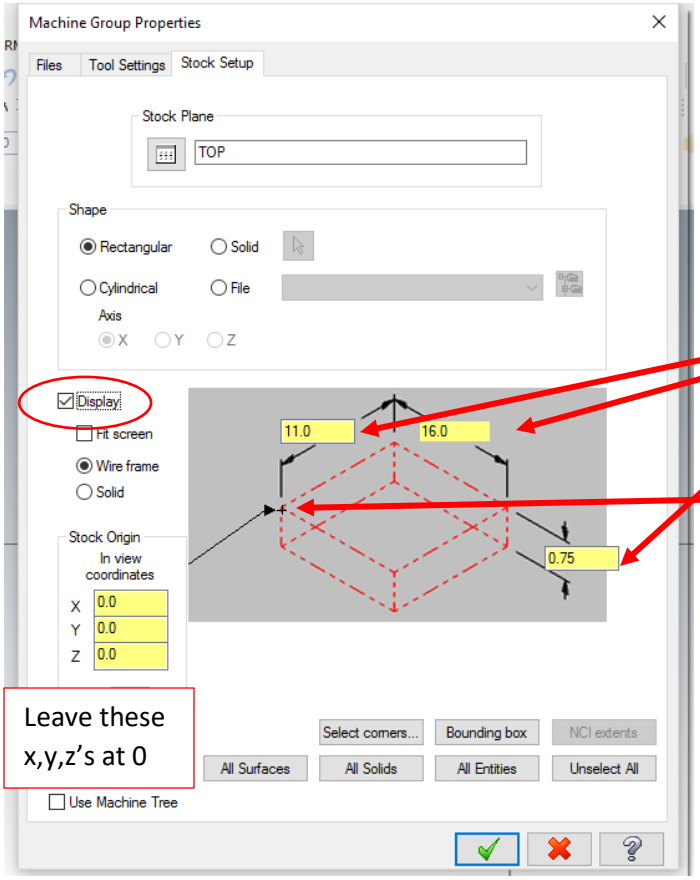


Stock Setup



The Toolpaths Operations Manager is the tool palette that is docked on the left of the screen. This displays all the specific information about the tool paths (what the CNC router will cut).

Expand the properties tab in the operations manager. Then click on stock setup.



Leave these x,y,z's at 0

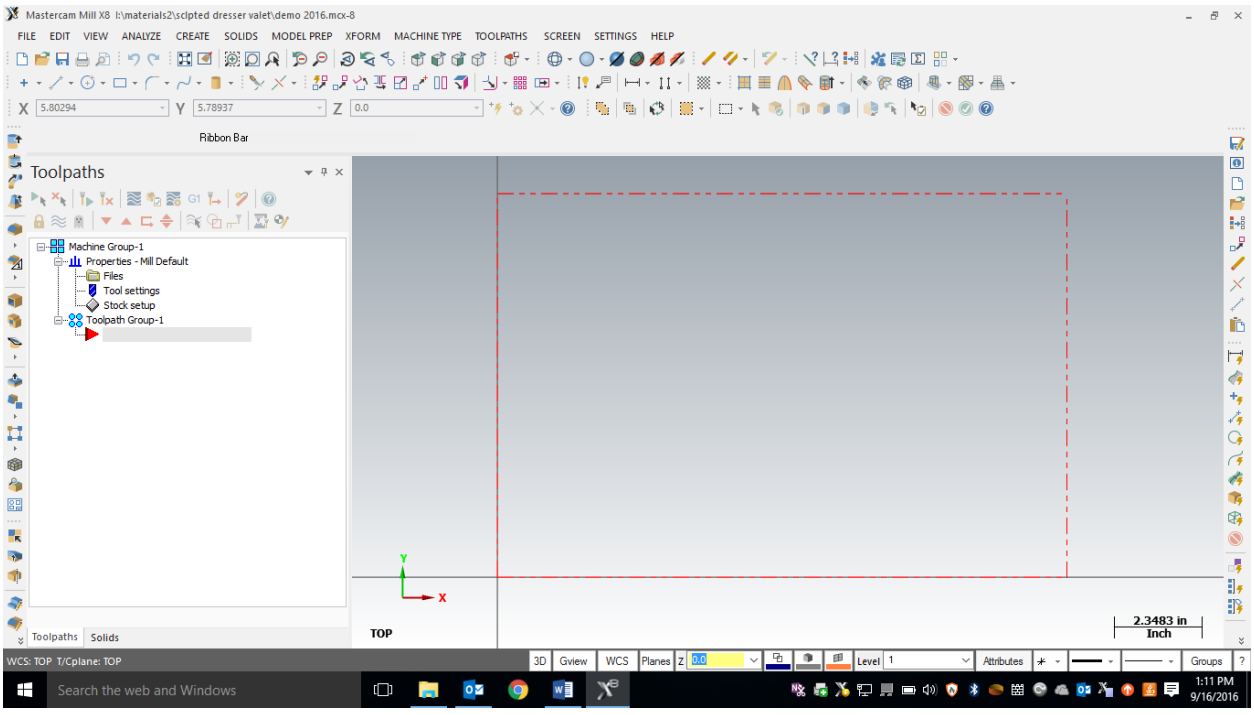
Setup the stock:
Enter the measurements for your piece, I'm putting in 11 (y), 16 (x), .75(z) for my measurements for the project. I'm not including the extra stock for the screws to hold it down to the router.

Set the stock origin by clicking on this corner.

Check "Display"

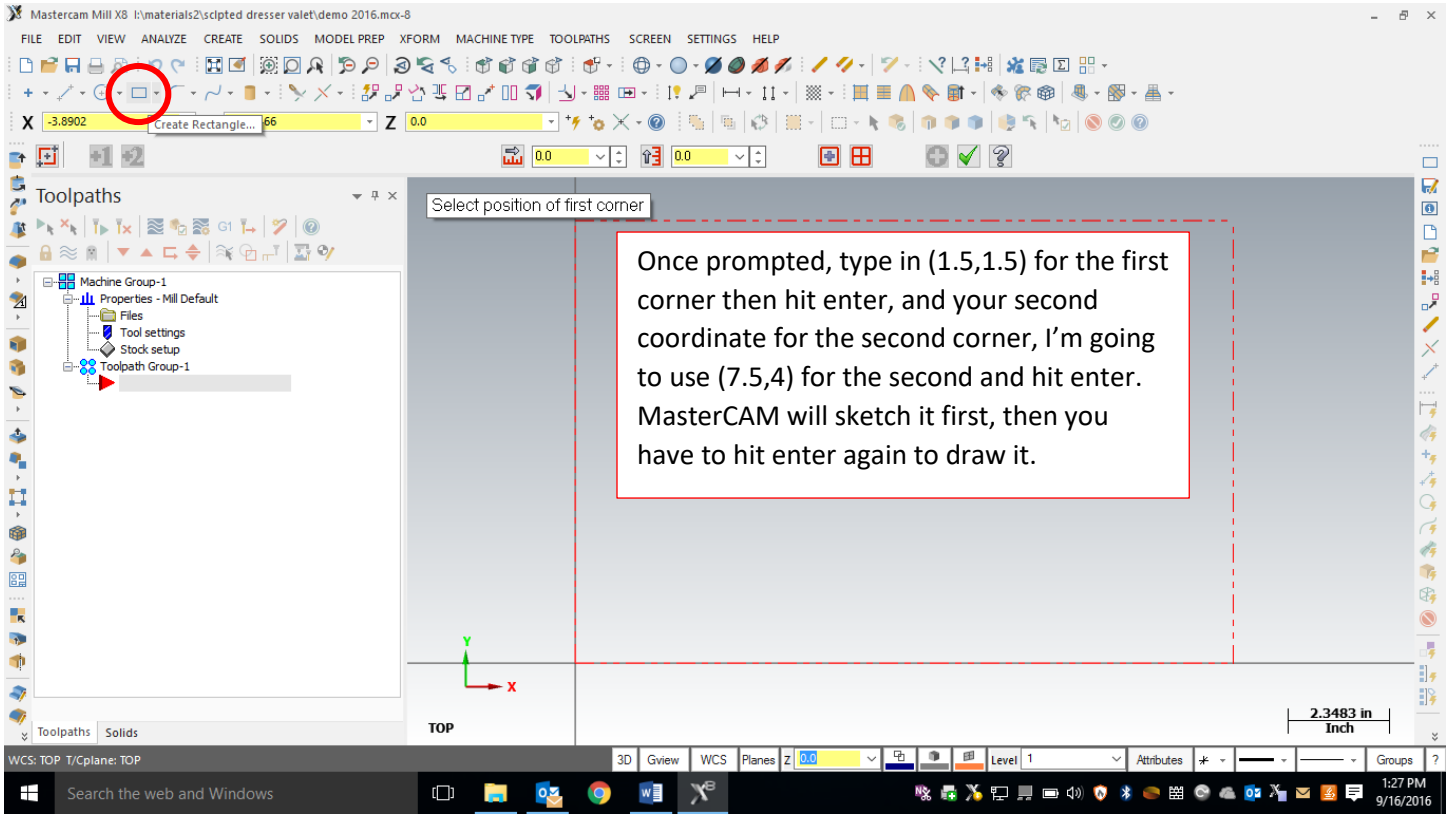
Click the Green Check Mark (OK)

After you click ok in the stock setup, you should see a red dashed rectangle that represents your stock. Zoom in or out so that you see the whole piece, like below.

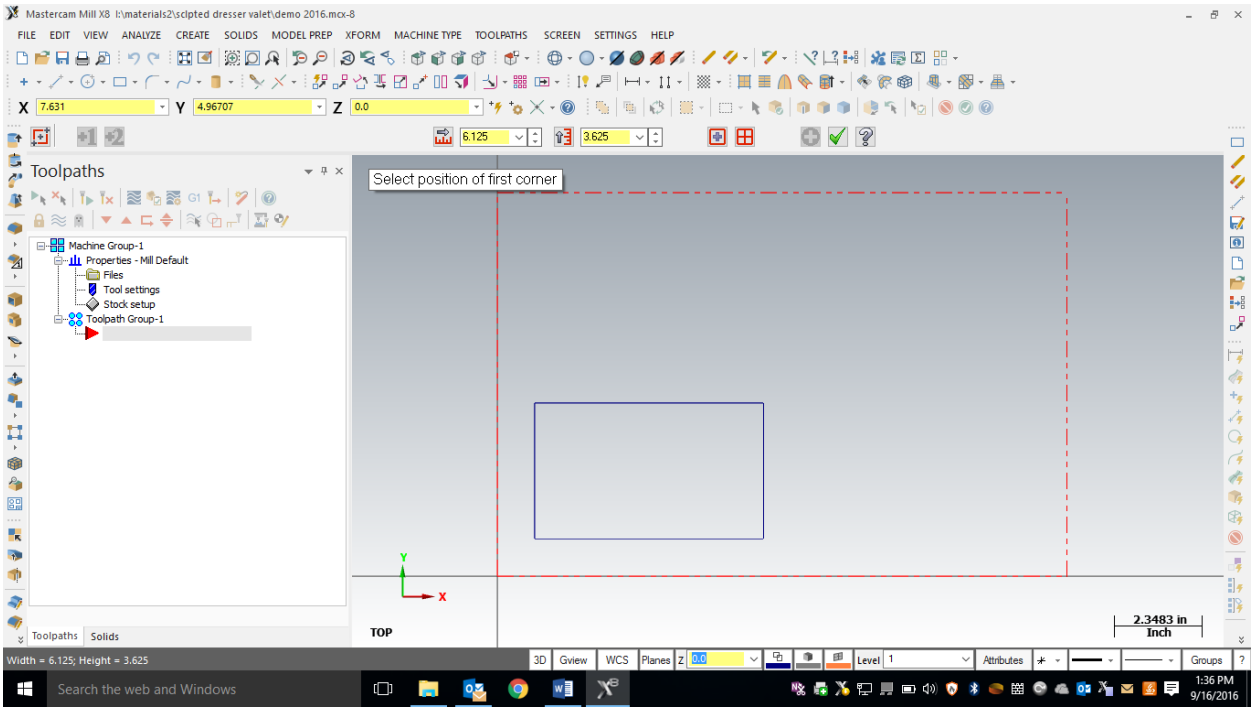


Entering Geometry

It's time to start drawing some geometry, so figure out what pockets you want to cut on your piece, and you will need the corresponding coordinates to draw the geometry. I'm going to leave 1.5" margin around the work piece so that I know that the screws will not intersect a pocket when that project is assembled. Also, I will leave 1" between the pockets so everything looks symmetrical. I'm just going to do 3 rectangles for the shapes of my pockets, but you can do what you like as long as it looks good and is symmetrical/balanced. To draw a rectangle, we need the coordinates of the opposite corners of each rectangle. When you have the coordinates written down then you are ready to go the rectangle tool, then we can just type in the two coordinates for the corners of each rectangle.

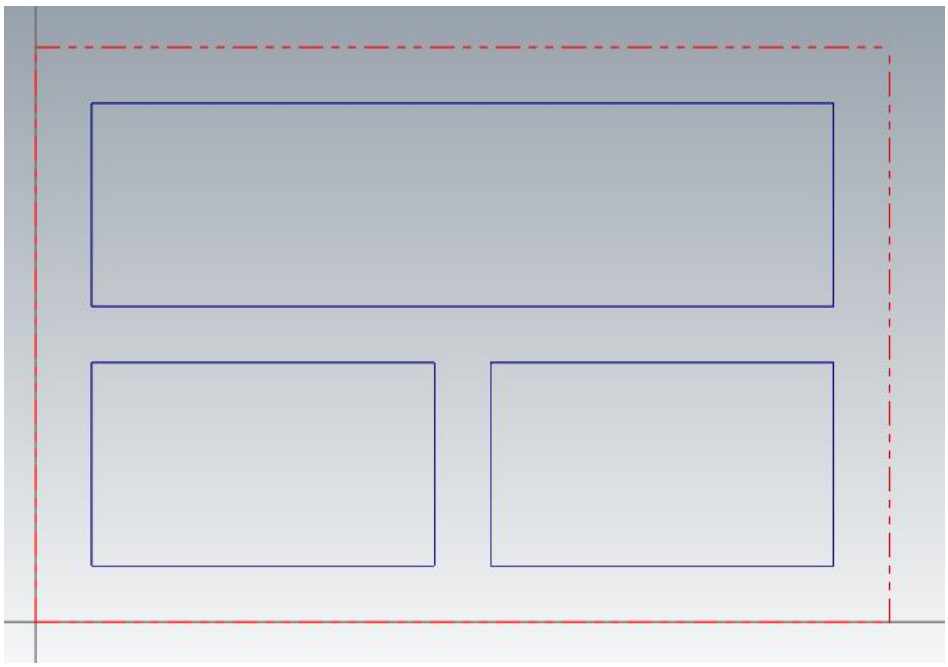


Result:



Now you can keep creating geometry in this method. You might want to draw out you shapes on graph paper or on your board first so you can keep the measurements correct. Here is my example. To complete this, you just need to coordinates of opposite corners of the rectangle. If you want to include a circle or other shape that is fine. They are just drawn from a center point and the radius or diameter.

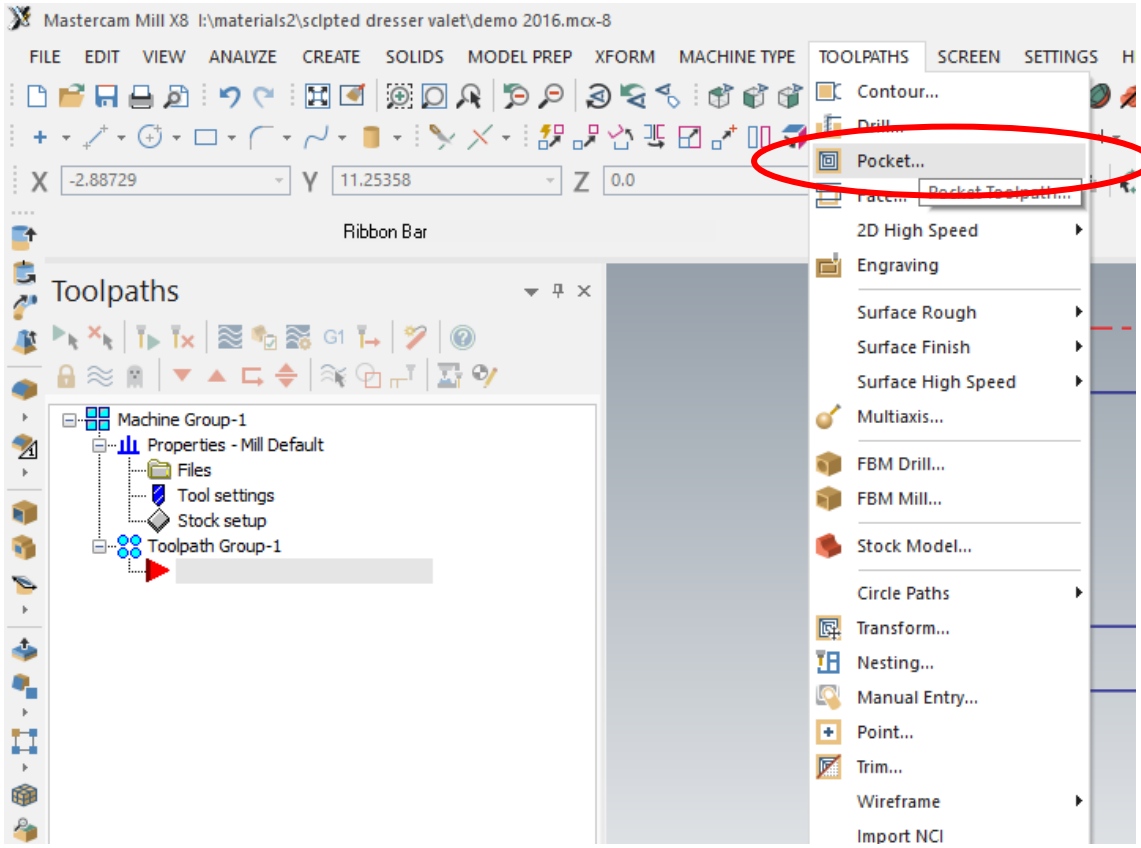
Result:

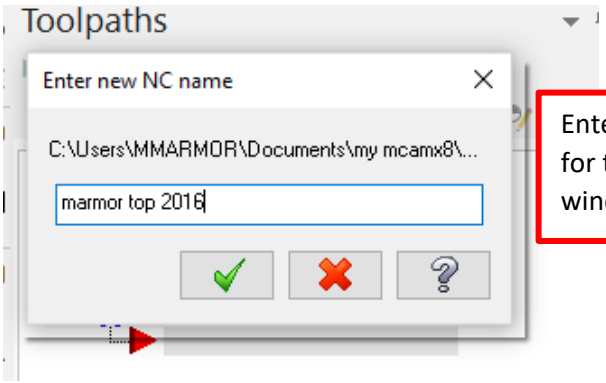


Toolpaths:

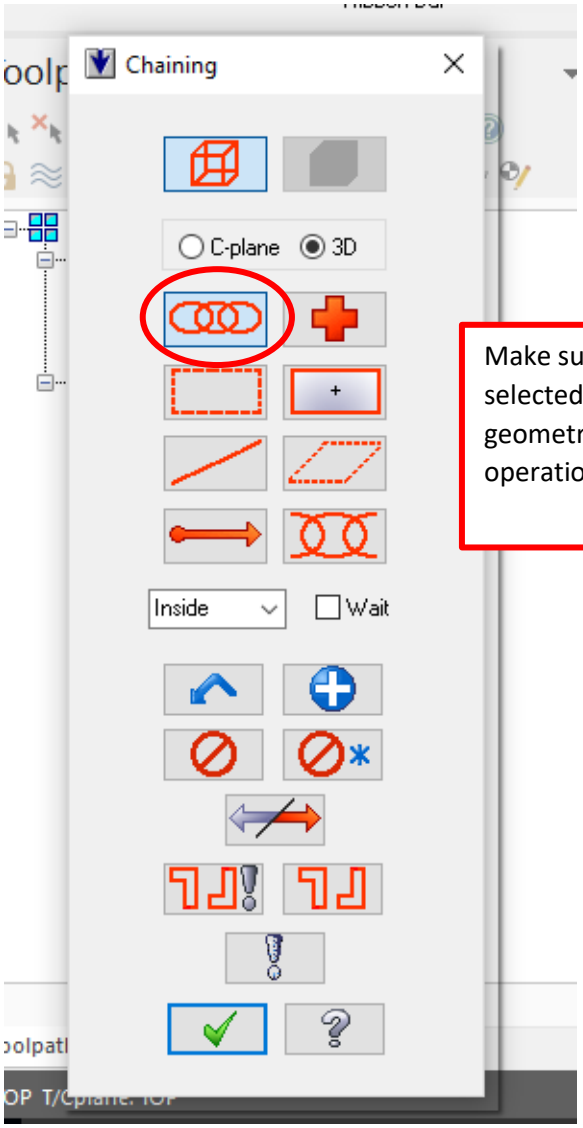
For 2D geometry such as we have, there are 2 main types of tool paths. The first one is a contour, and the second a pocket. In a contour toolpath, the tool bit will follow a path. The path can be one piece of geometry or multiple pieces of geometry linked together end to end. When the geometry is selected you must either pick the single option or the chaining option (multiple objects laid out end to end) before you select the geometry. We are going to complete pocket toolpaths. A pocket toolpath will make a cavity inside the selected geometry. We should be able to use one pocket toolpath and pick all the geometry to complete the cut correctly. When we select the geometry, we will use the chaining option, since the rectangles are really made from four lines each.

To start the toolpaths, go to Toolpaths/Pocket.

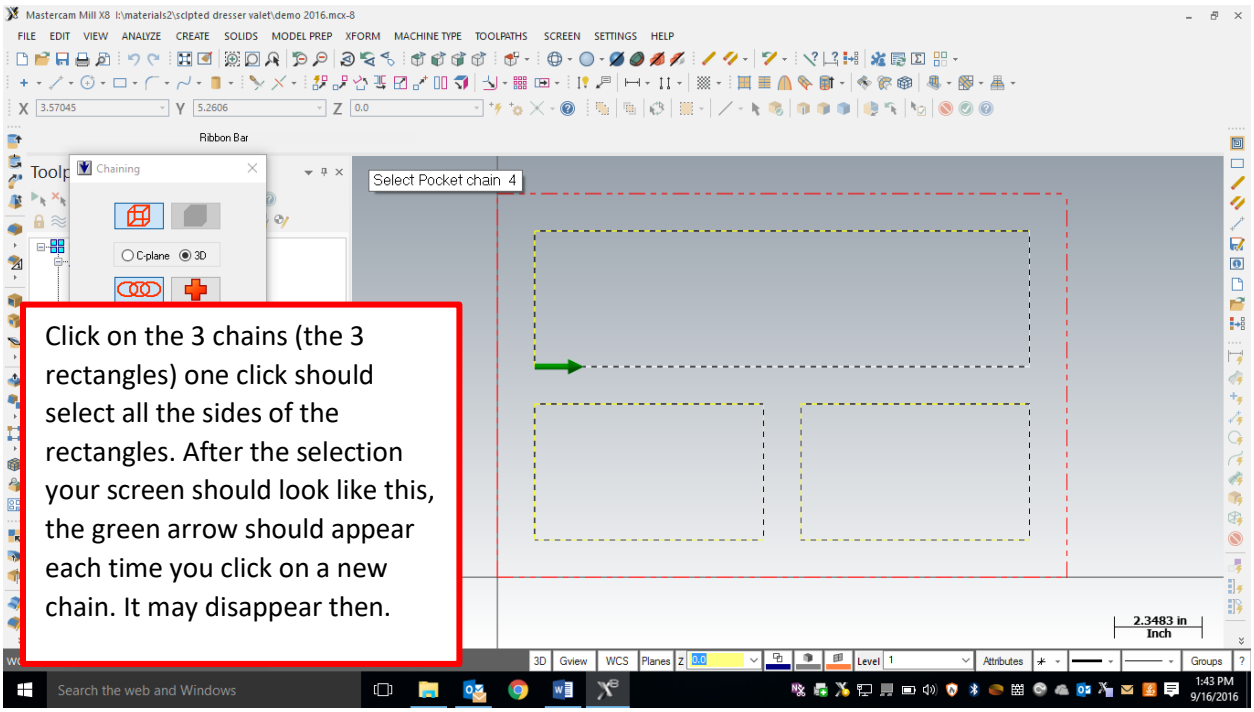


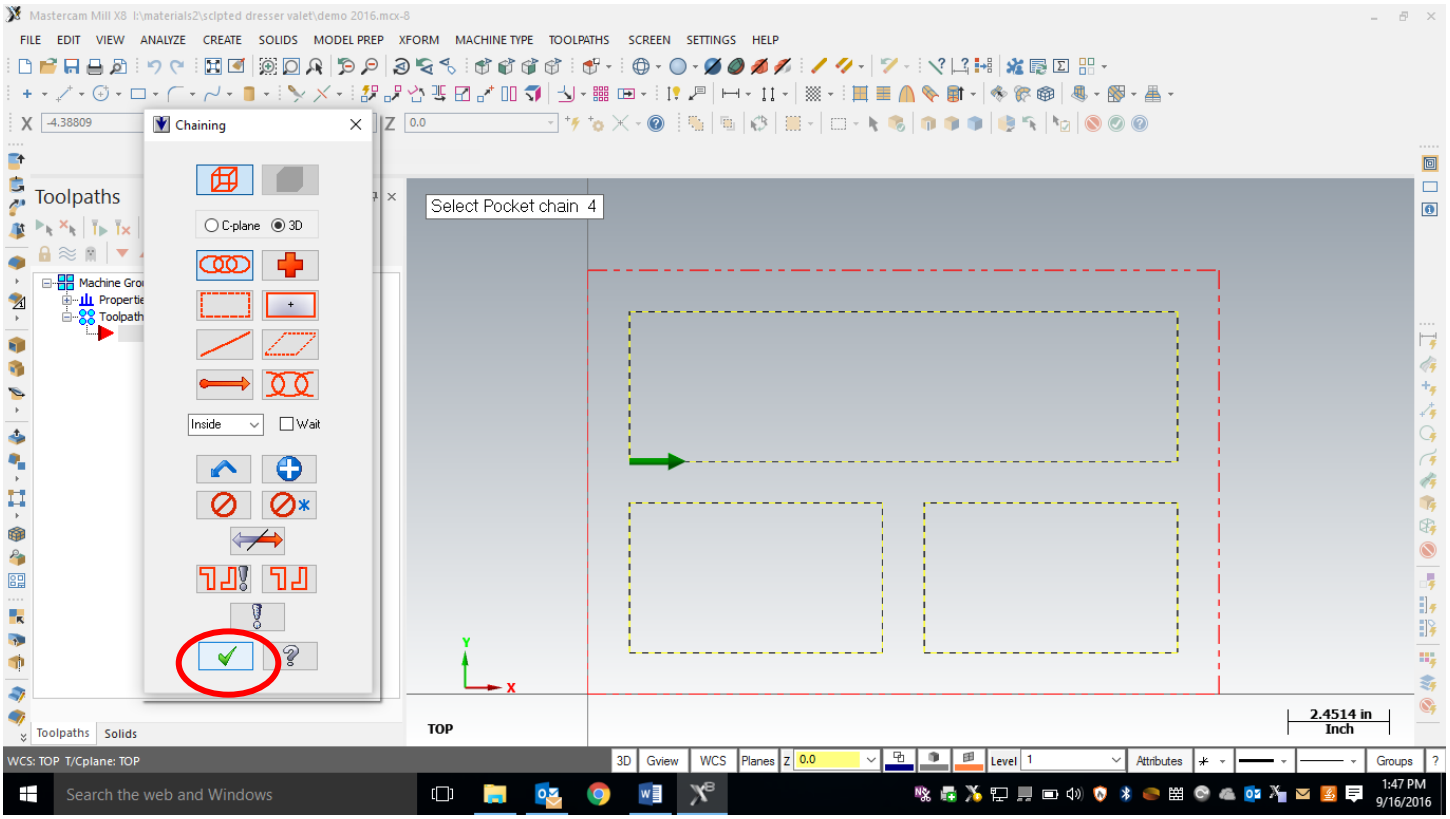


Enter an appropriate name for the toolpath in this window



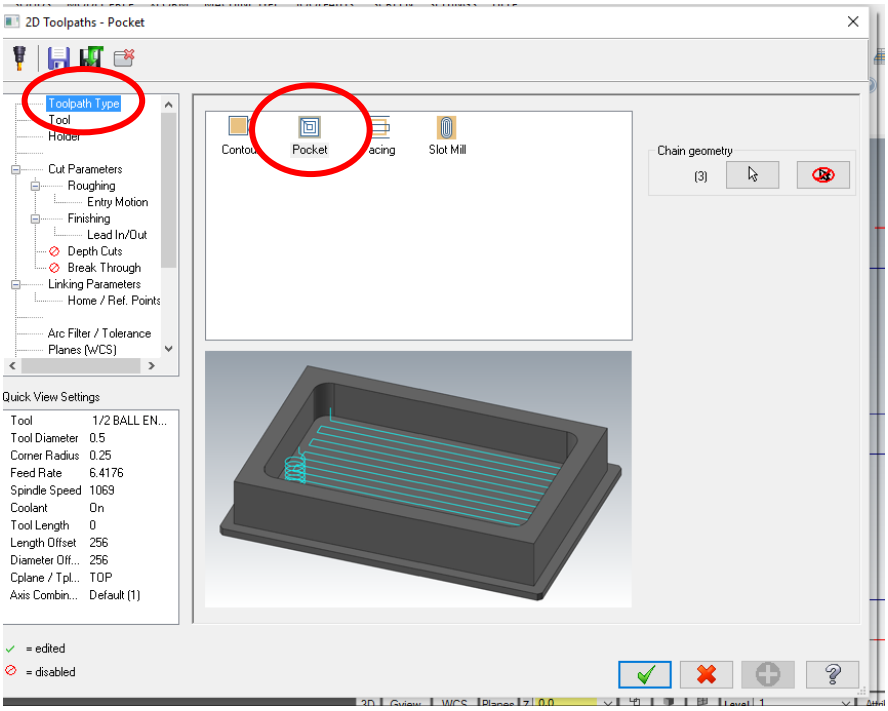
Make sure the chaining option is selected then click on the geometry you want to drive the operation.

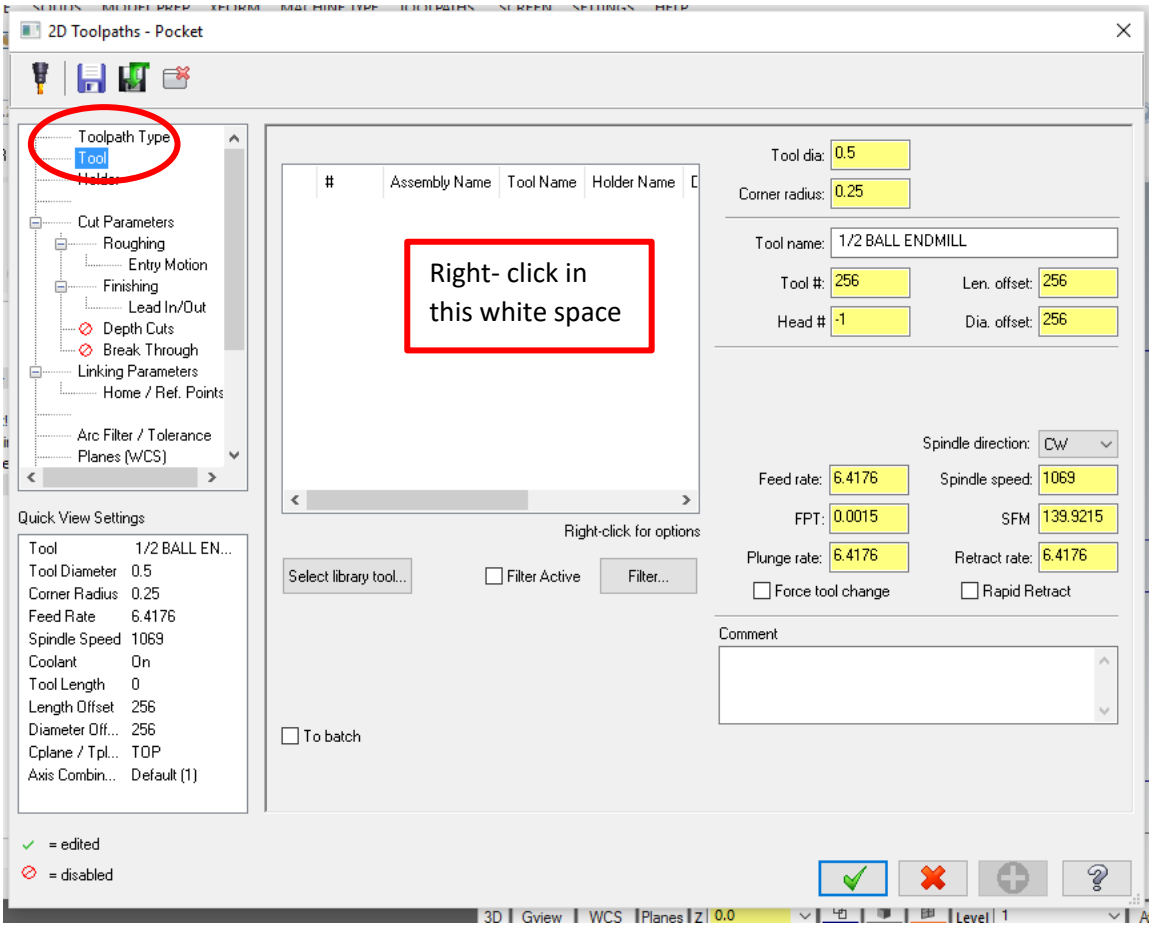


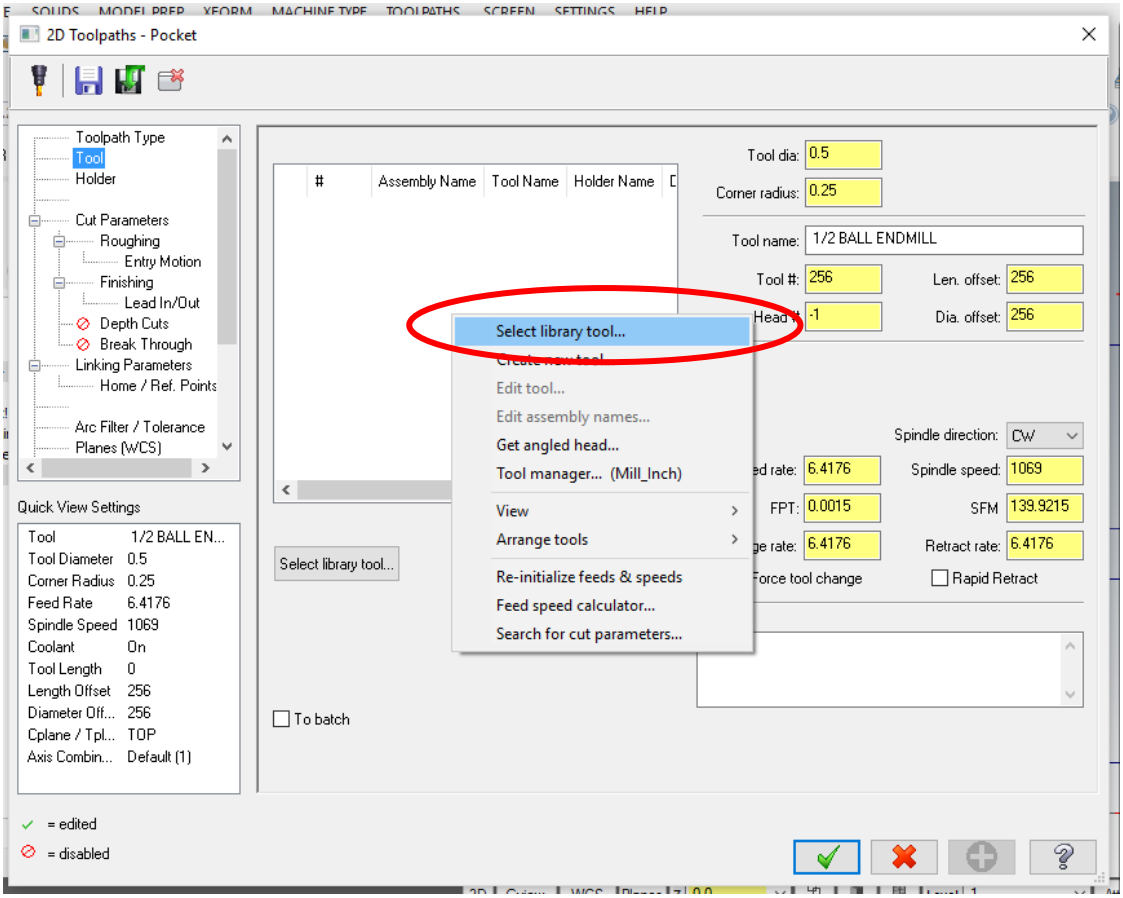


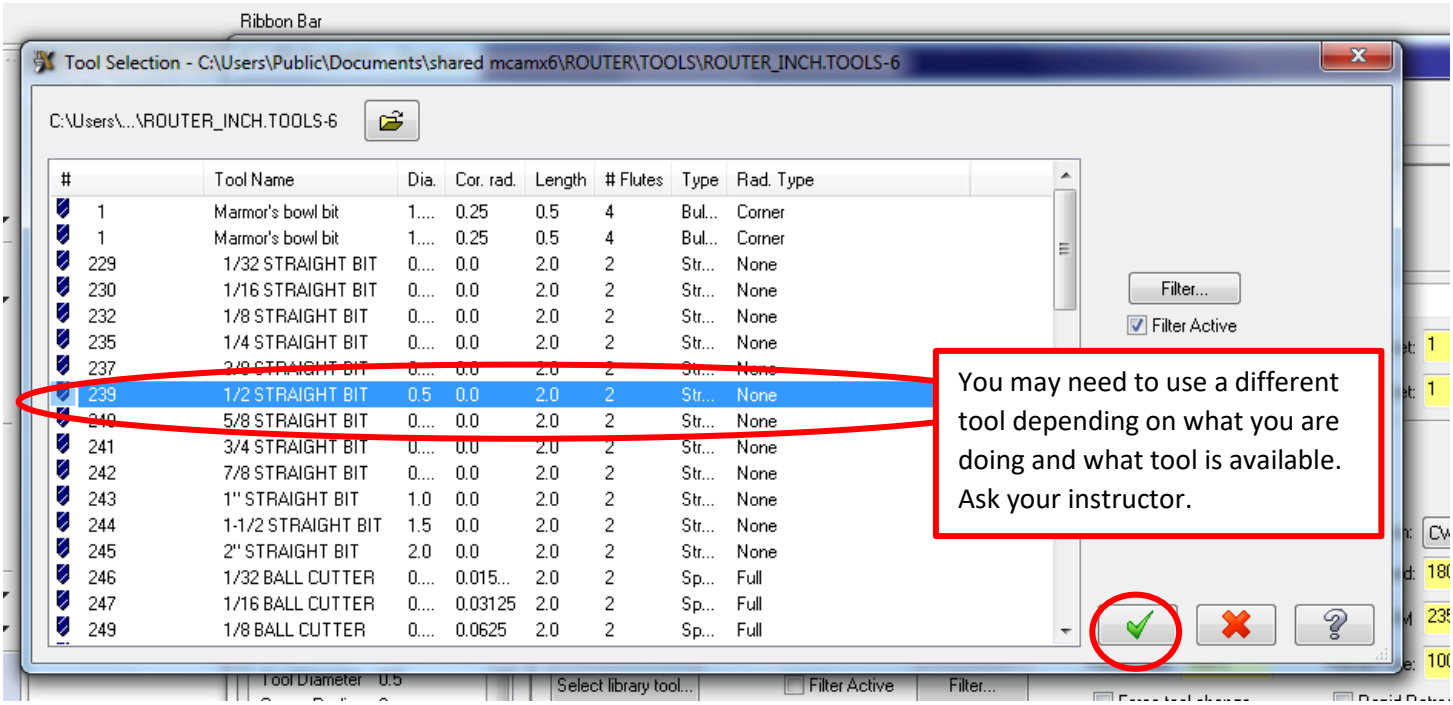
Click ok, to end our selection.

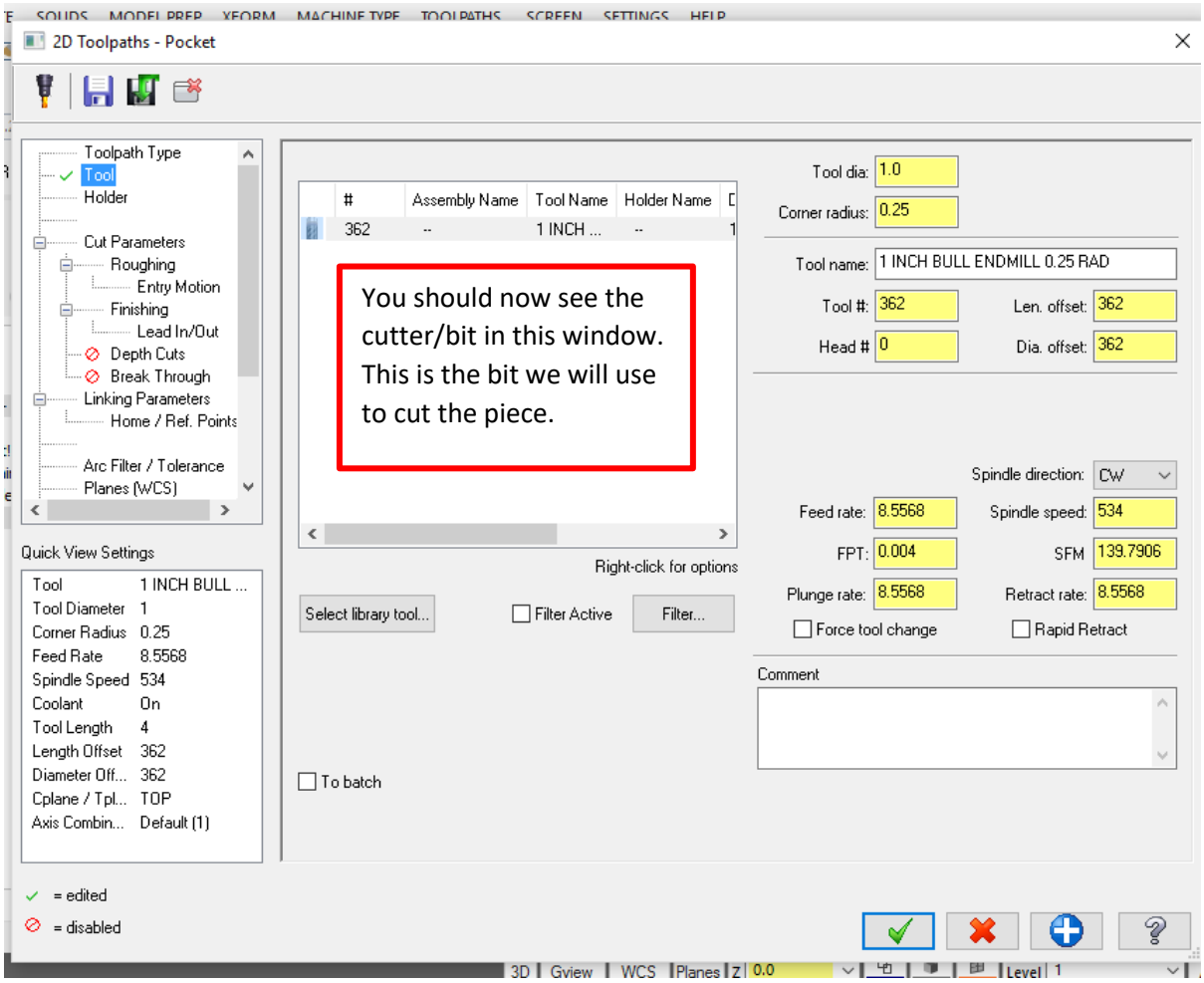
Please enter the following information in the pocket toolpath parameters.

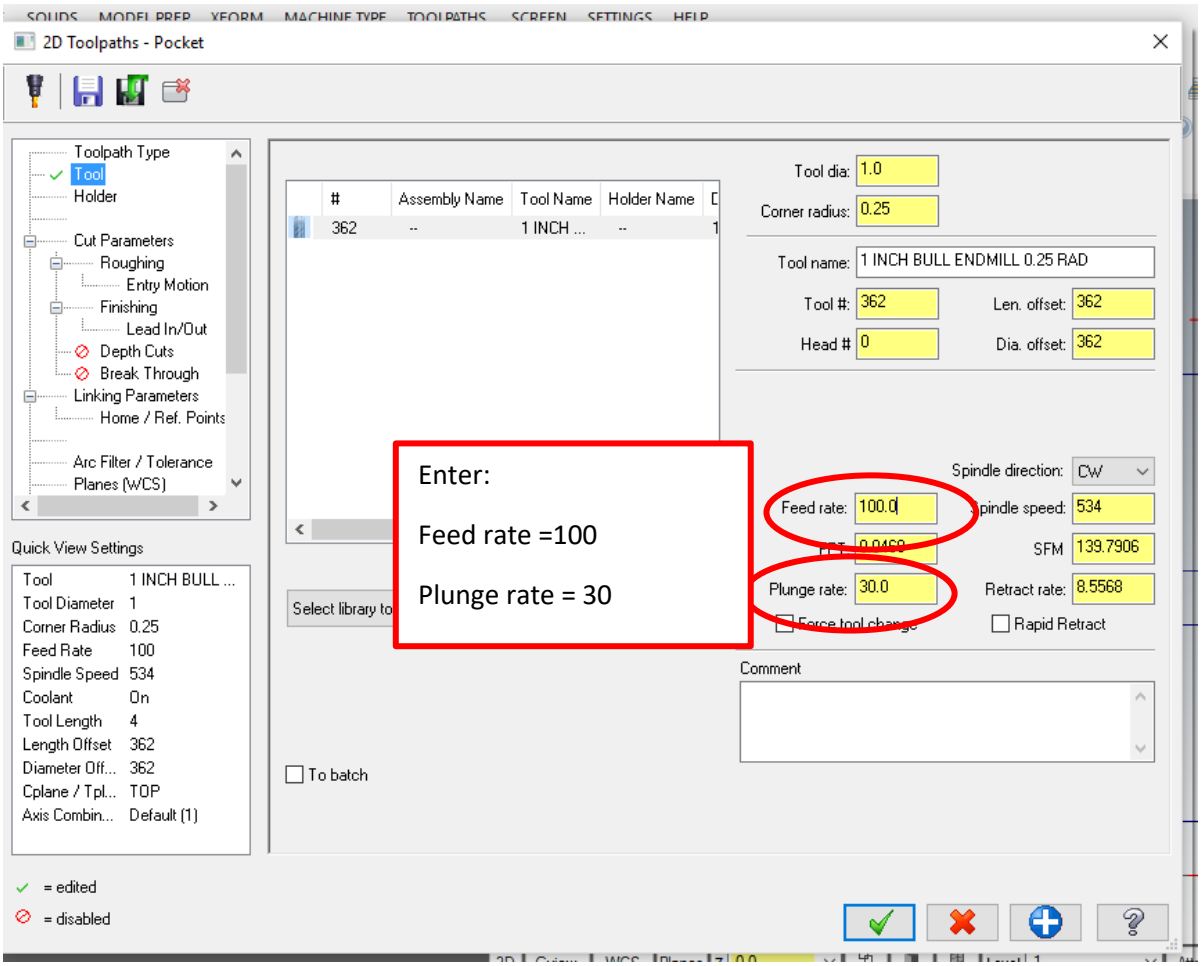


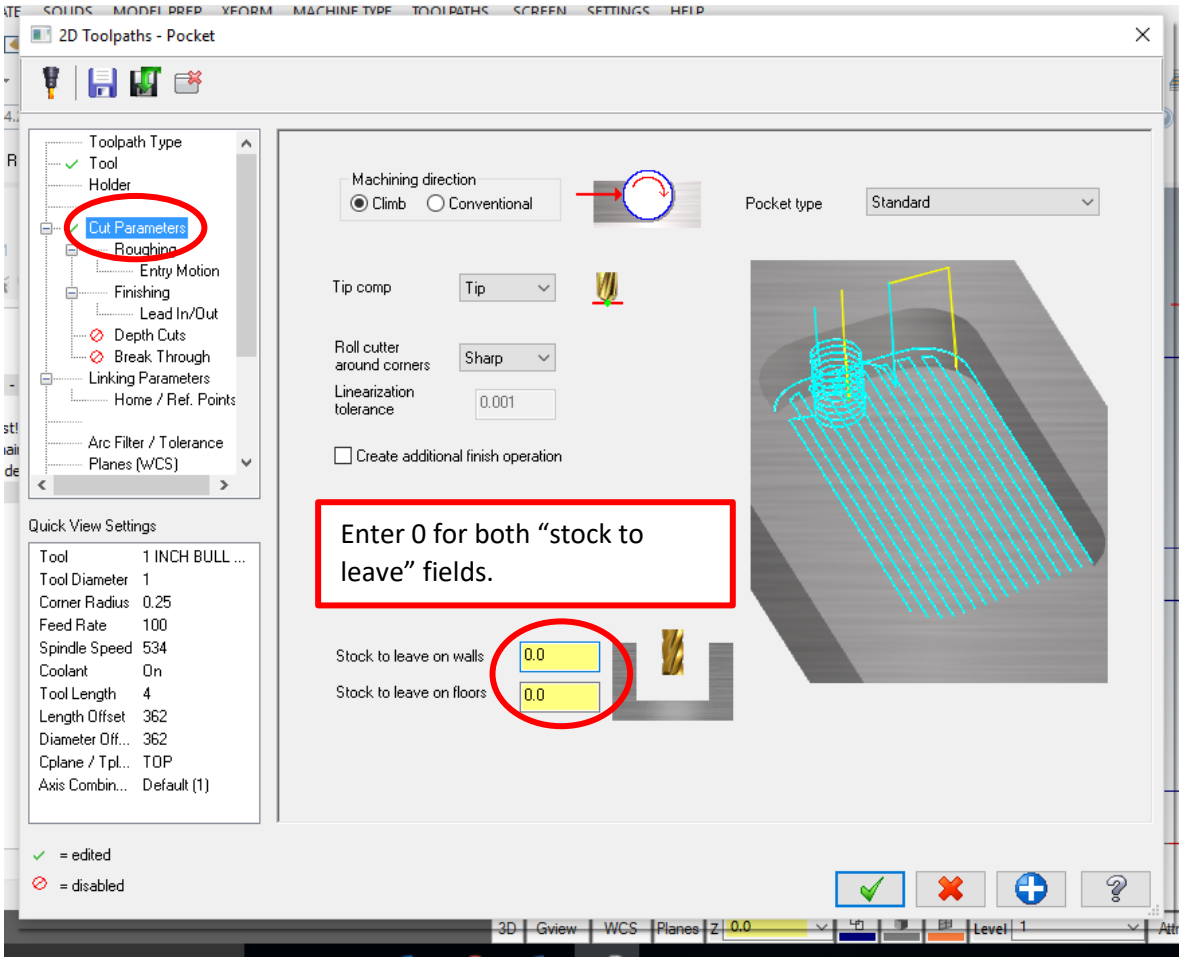


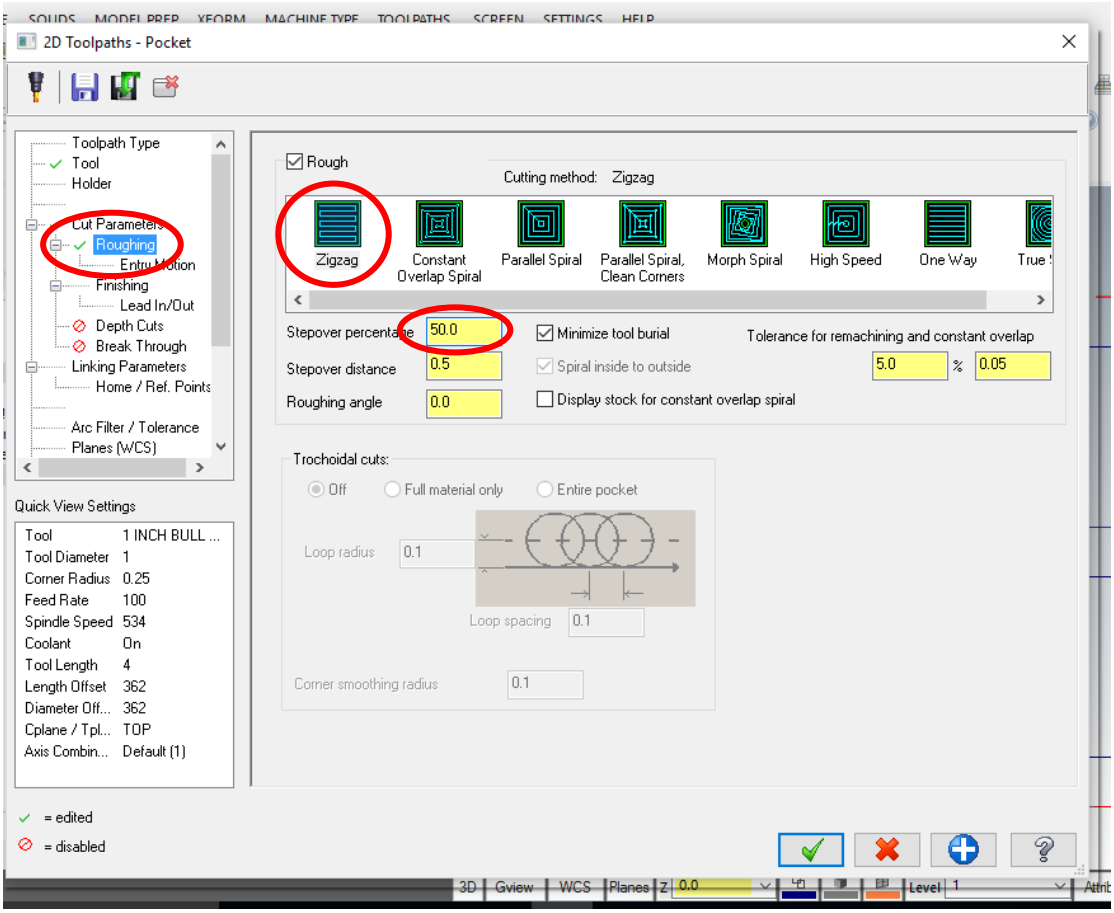

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1/2 BALL EN...
Tool Diameter 0.5
Corner Radius 0.25
Feed Rate 6.4176
Spindle Speed 1069
Coolant On
Tool Length 0
Length Offset 256
Diameter Off... 256
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
Assembly Name Tool Name Holder Name
Select library tool...
To batch
Tool dia: 0.5
Corner radius: 0.25
Tool name: 1/2 BALL ENDMILL
Tool #: 256
Len. offset: 256
Dia. offset: 256
Spindle direction: CW
Spindle speed: 1069
FPT: 0.0015
SFM 139.9215
Retract rate: 6.4176
Rapid Retract
Select library tool...
Edit tool...
Edit assembly names...
Get angled head...
Tool manager... (Mill_Inch)
View
Arrange tools
Re-initialize feeds & speeds
Feed speed calculator...
Search for cut parameters...

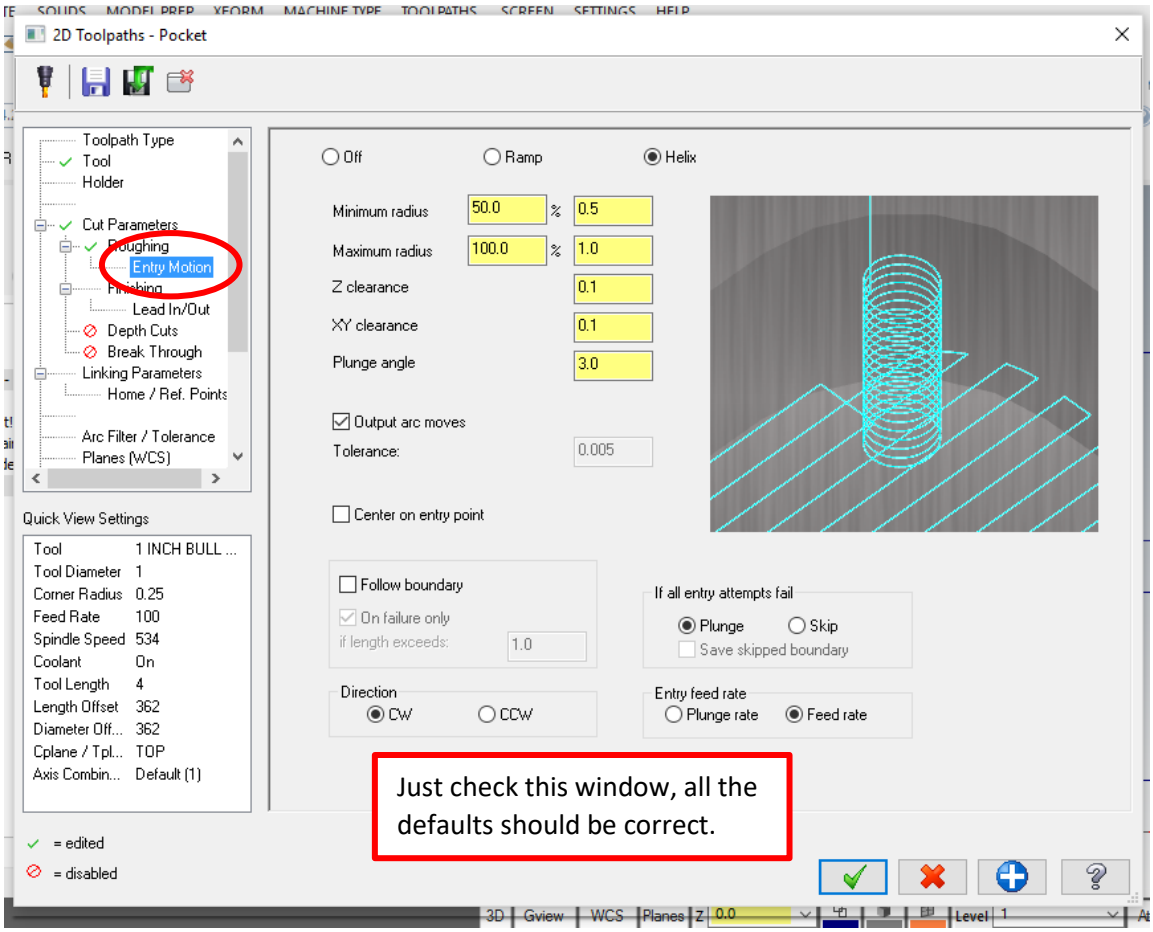

Ribbon Bar
Tool Selection - C:\Users\Public\Documents\shared mcamx6\ROUTER\TOOLS\ROUTER_INCH.TOOLS-6
C:\Users\...\ROUTER_INCH.TOOLS-6
Tool Name Dia. Cor. rad. Length # Flutes Type Rad. Type
1 Marmor's bowl bit 1.... 0.25 0.5 4 Bul... Corner
1 Marmor's bowl bit 1.... 0.25 0.5 4 Bul... Corner
229 1/32 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
230 1/16 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
232 1/8 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
235 1/4 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
239 1/2 STRAIGHT BIT 0.5 0.0 2.0 2 Str... None
249 5/8 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
241 3/4 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
242 7/8 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
243 1" STRAIGHT BIT 1.0 0.0 2.0 2 Str... None
244 1-1/2 STRAIGHT BIT 1.5 0.0 2.0 2 Str... None
245 2" STRAIGHT BIT 2.0 0.0 2.0 2 Str... None
246 1/32 BALL CUTTER 0.... 0.015... 2.0 2 Sp... Full
247 1/16 BALL CUTTER 0.... 0.03125 2.0 2 Sp... Full
249 1/8 BALL CUTTER 0.... 0.0625 2.0 2 Sp... Full
Filter...
Filter Active
You may need to use a different tool depending on what you are doing and what tool is available. Ask your instructor.

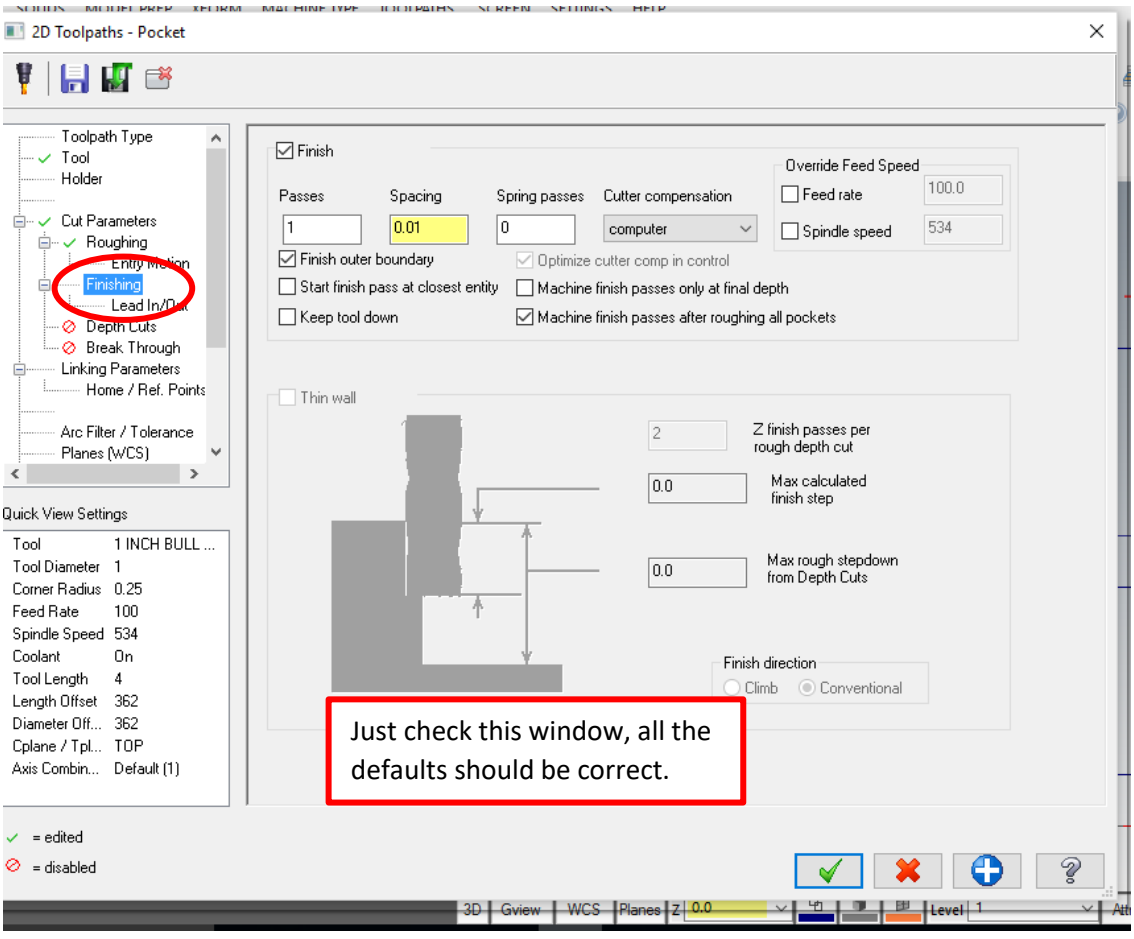

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 8.5568
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
Assembly Name Tool Name Holder Name
362 -- 1 INCH ... --
You should now see the cutter/bit in this window. This is the bit we will use to cut the piece.
Right-click for options
Select library tool...
Filter Active
Filter...
To batch
Tool dia: 1.0
Corner radius: 0.25
Tool name: 1 INCH BULL ENDMILL 0.25 RAD
Tool #: 362
Len. offset: 362
Head #: 0
Dia. offset: 362
Spindle direction: CW
Feed rate: 8.5568
Spindle speed: 534
FPT: 0.004
SFM 139.7906
Plunge rate: 8.5568
Retract rate: 8.5568
Force tool change
Rapid Retract
Comment
= edited
= disabled
3D Gview WCS Planes Z 0.0
Level 1

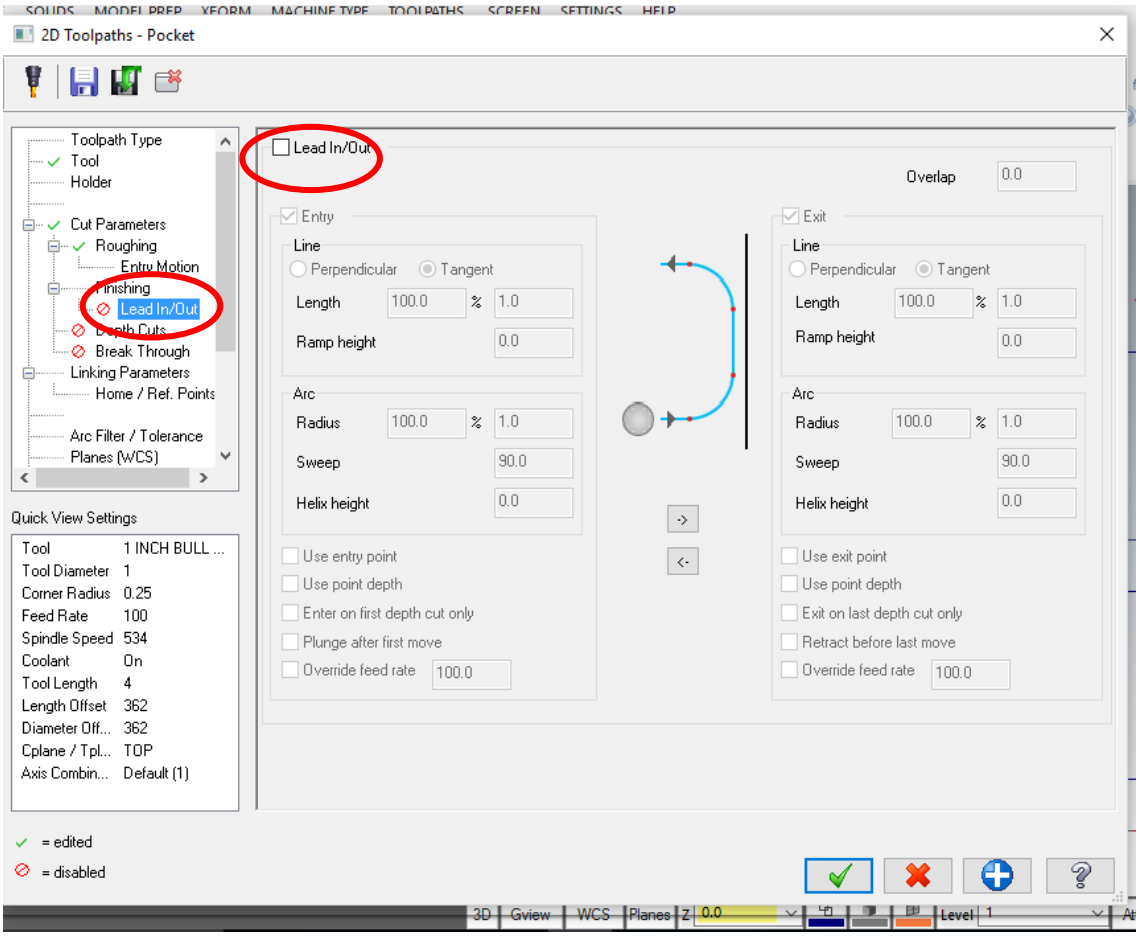

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
Assembly Name Tool Name Holder Name
362 -- 1 INCH ...
Enter:
Feed rate =100
Plunge rate = 30
Select library to
To batch
Tool dia: 1.0
Corner radius: 0.25
Tool name: 1 INCH BULL ENDMILL 0.25 RAD
Tool #: 362
Len. offset: 362
Head #: 0
Dia. offset: 362
Spindle direction: CW
Feed rate: 100.0
Spindle speed: 534
SFM 139.7906
Plunge rate: 30.0
Retract rate: 8.5568
Rapid Retract
Comment
= edited
= disabled

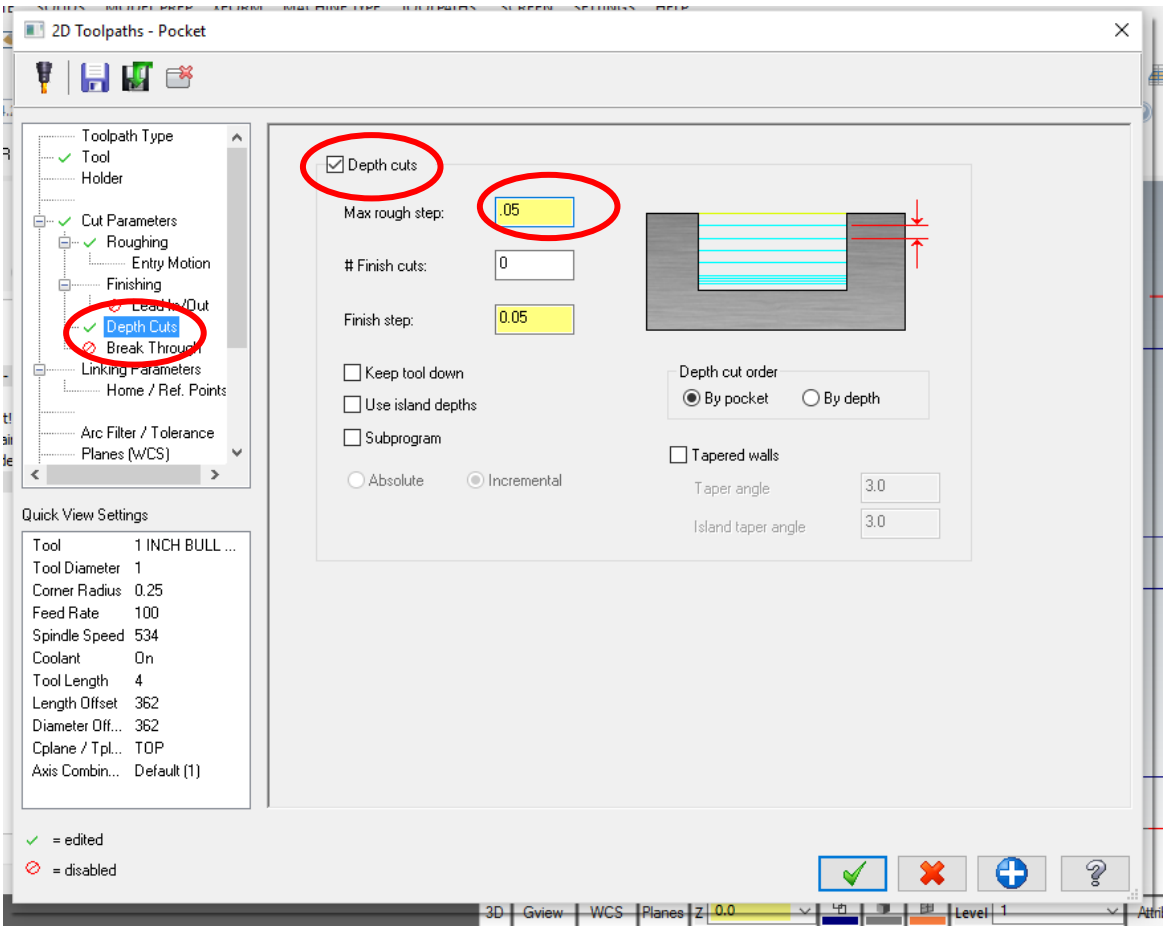

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Machining direction
Climb
Conventional
Pocket type
Standard
Tip comp
Tip
Roll cutter around corners
Sharp
Linearization tolerance
0.001
Create additional finish operation
Enter 0 for both “stock to leave” fields.
Stock to leave on walls
0.0
Stock to leave on floors
0.0
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled

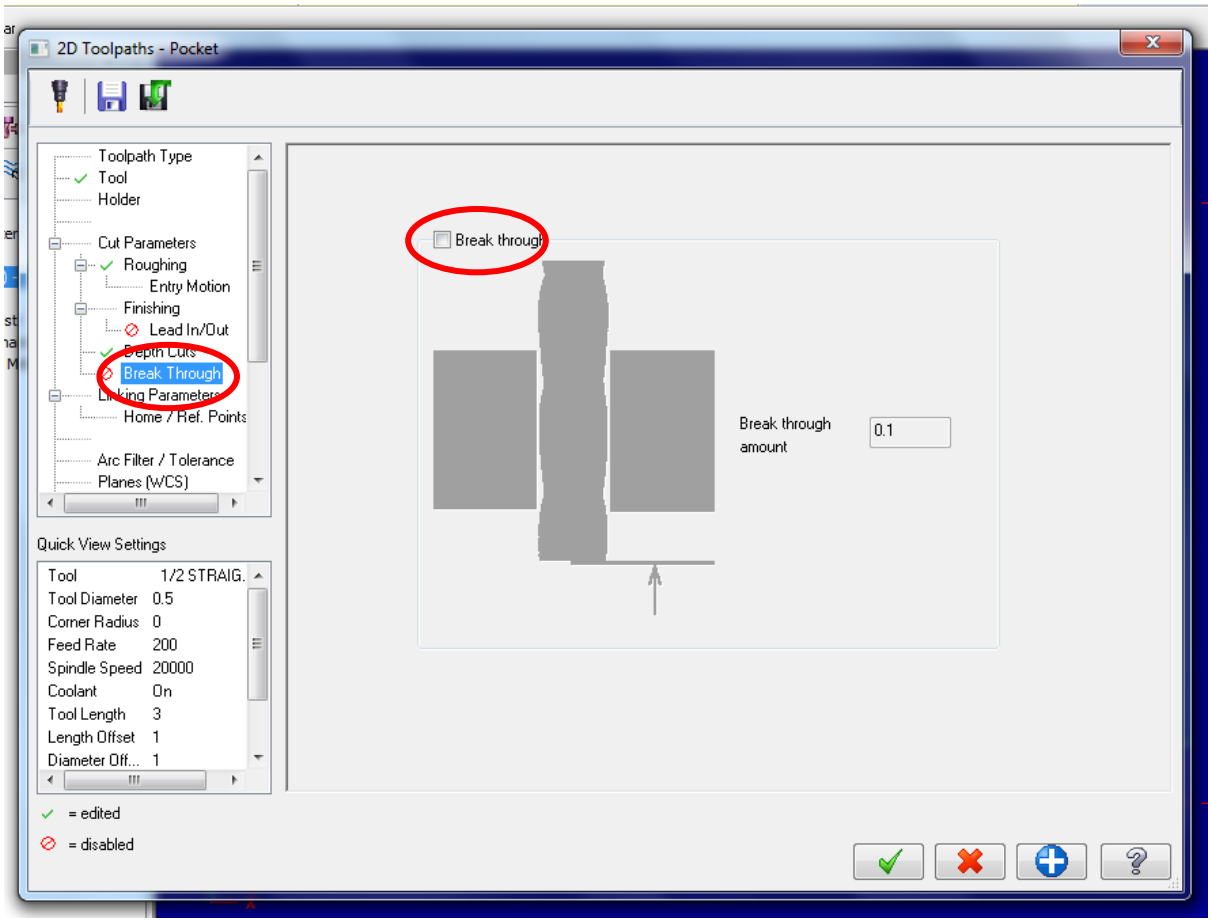

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Rough
Cutting method: Zigzag
Zigzag
Constant Overlap Spiral
Parallel Spiral
Parallel Spiral, Clean Corners
Morph Spiral
High Speed
One Way
Stepover percentage
50.0
Minimize tool burial
Tolerance for remachining and constant overlap
Stepover distance
0.5
Spiral inside to outside
5.0
0.05
Roughing angle
0.0
Display stock for constant overlap spiral
Trochoidal cuts:
Off
Full material only
Entire pocket
Loop radius
0.1
Loop spacing
0.1
Corner smoothing radius
0.1
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled


2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
Off
Ramp
Helix
Minimum radius 50.0 % 0.5
Maximum radius 100.0 % 1.0
Z clearance 0.1
XY clearance 0.1
Plunge angle 3.0
Output arc moves
Tolerance: 0.005
Center on entry point
Follow boundary
On failure only
if length exceeds: 1.0
If all entry attempts fail
Plunge
Skip
Save skipped boundary
Direction
CW
CCW
Entry feed rate
Plunge rate
Feed rate
Just check this window, all the defaults should be correct.
3D
Gview
WCS
Planes
Z 0.0
Level 1

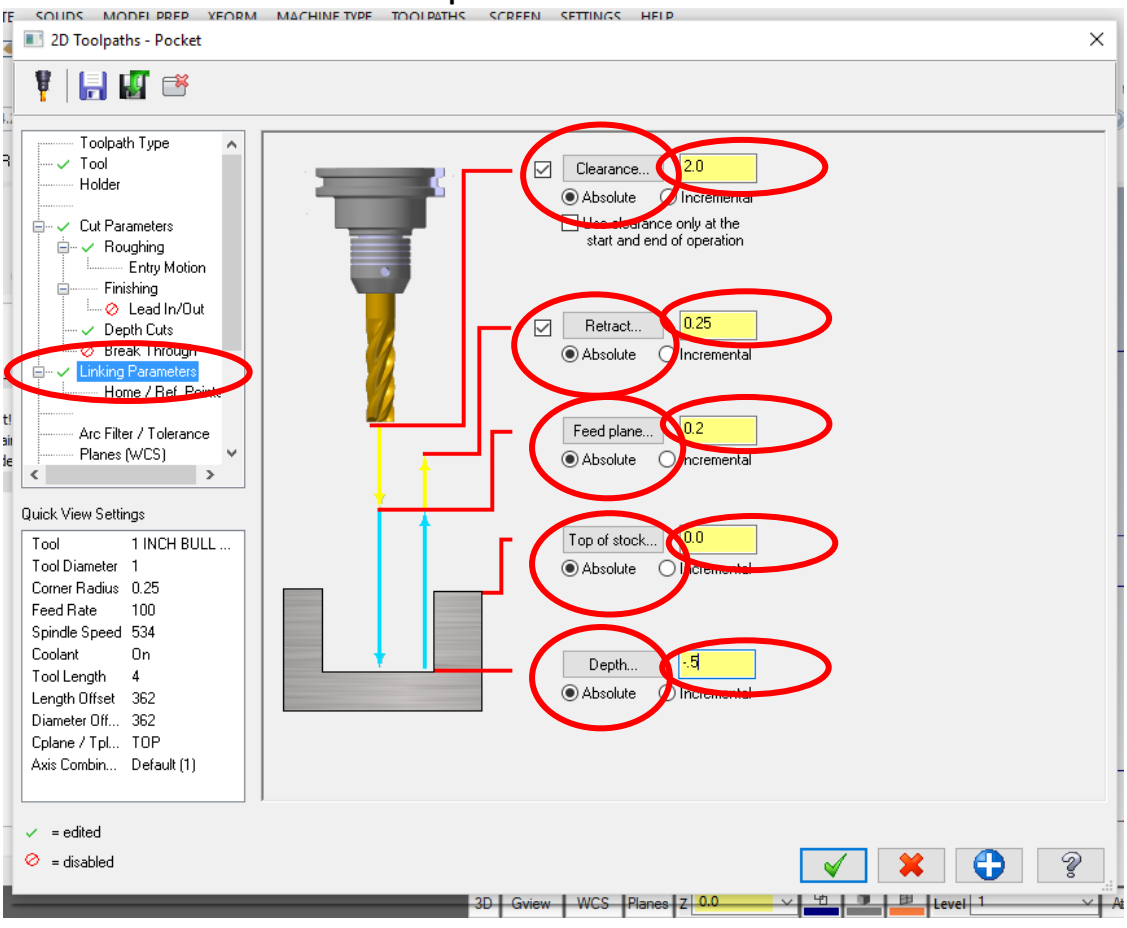

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
Finish
Passes
Spacing
Spring passes
Cutter compensation
1
0.01
0
computer
Override Feed Speed
Feed rate 100.0
Spindle speed 534
Finish outer boundary
Optimize cutter comp in control
Start finish pass at closest entity
Machine finish passes only at final depth
Keep tool down
Machine finish passes after roughing all pockets
Thin wall
2
Z finish passes per rough depth cut
0.0
Max calculated finish step
0.0
Max rough stepdown from Depth Cuts
Finish direction
Climb
Conventional
Just check this window, all the defaults should be correct.
3D
Gview
WCS
Planes
Z 0.0
Level 1

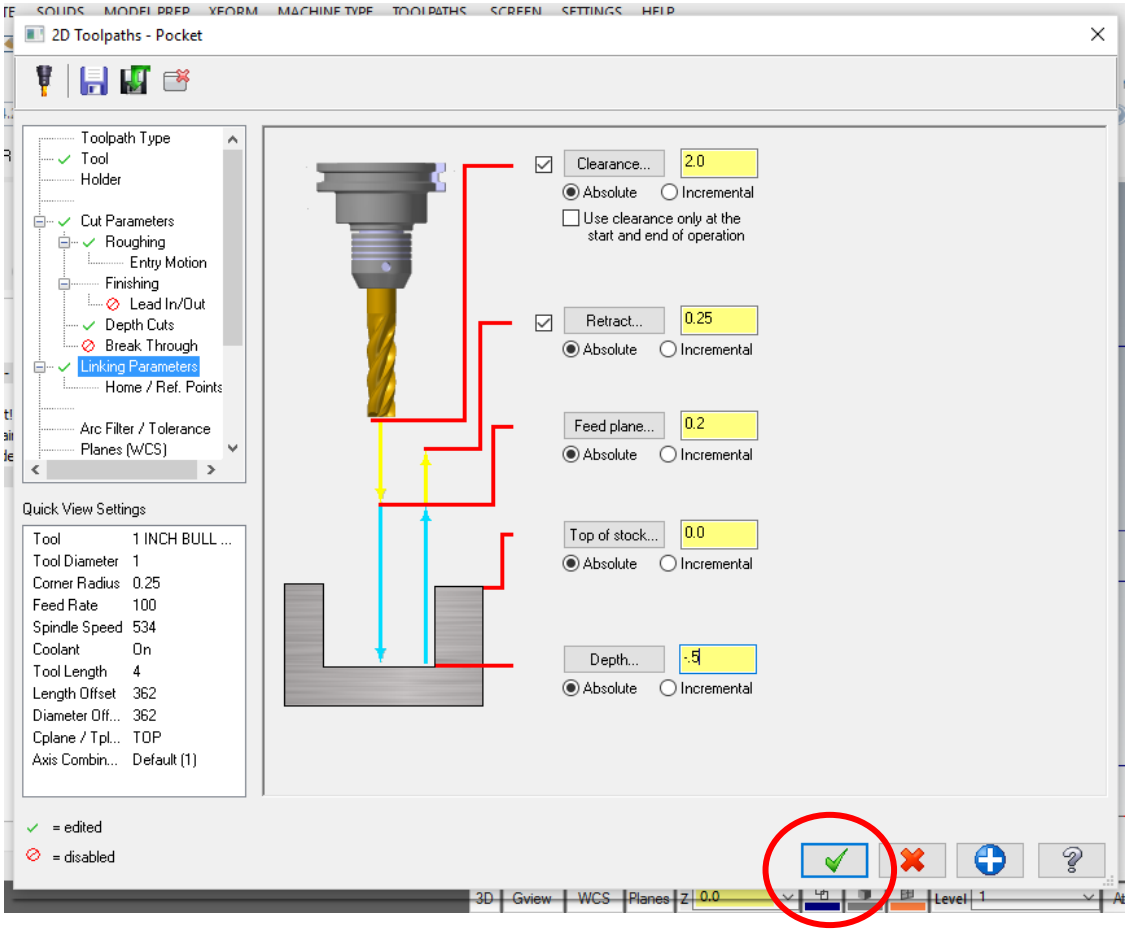

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Lead In/Out
Overlap
0.0
Entry
Line
Perpendicular
Tangent
Length
100.0
%
1.0
Ramp height
0.0
Arc
Radius
100.0
%
1.0
Sweep
90.0
Helix height
0.0
Use entry point
Use point depth
Enter on first depth cut only
Plunge after first move
Override feed rate
100.0
Exit
Line
Perpendicular
Tangent
Length
100.0
%
1.0
Ramp height
0.0
Arc
Radius
100.0
%
1.0
Sweep
90.0
Helix height
0.0
Use exit point
Use point depth
Exit on last depth cut only
Retract before last move
Override feed rate
100.0
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
3D
Gview
WCS
Planes
Z
0.0
Level
1


2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Depth cuts
Max rough step:
.05
Finish cuts:
0
Finish step:
0.05
Keep tool down
Use island depths
Subprogram
Absolute
Incremental
Depth cut order
By pocket
By depth
Tapered walls
Taper angle
3.0
Island taper angle
3.0
Quick View Settings
Tool 1 INCH BULL ...
Tool Diameter 1
Corner Radius 0.25
Feed Rate 100
Spindle Speed 534
Coolant On
Tool Length 4
Length Offset 362
Diameter Off... 362
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
3D
Gview
WCS
Planes
Z
0.0
Level
1

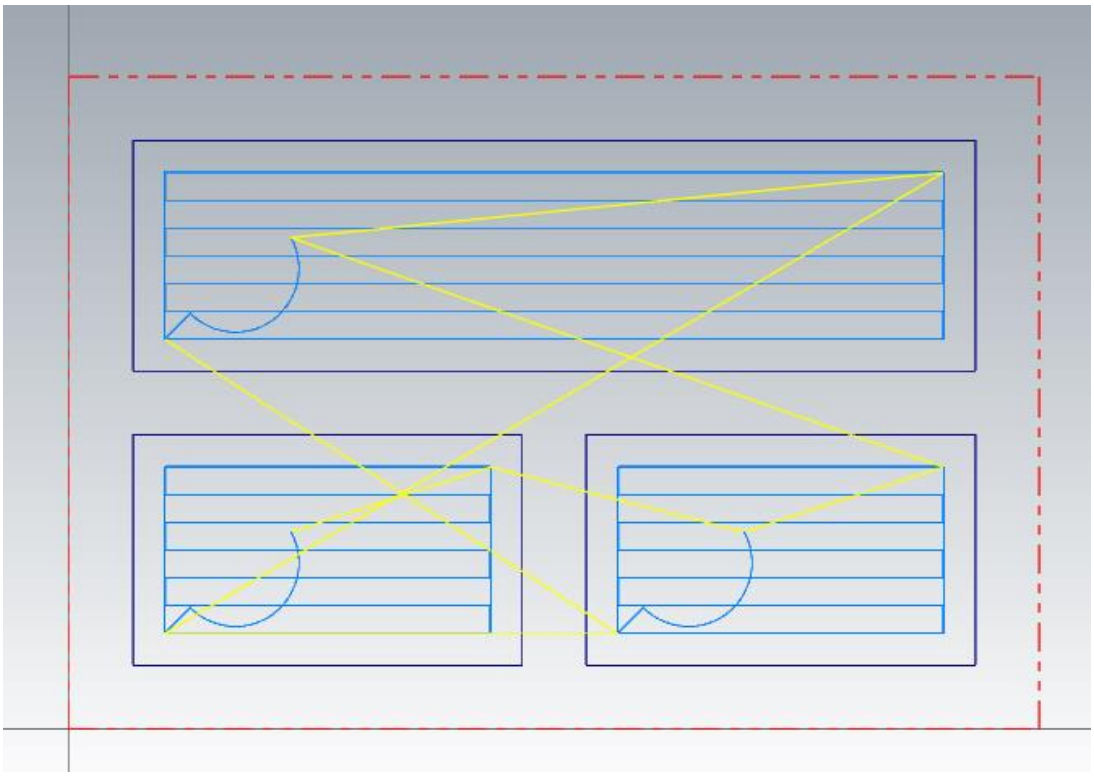


To set the depth of the cut, please enter the below values. In the Linking Parameters tab. Notice that all the values are **“Absolute” and the depth is a -.5.**

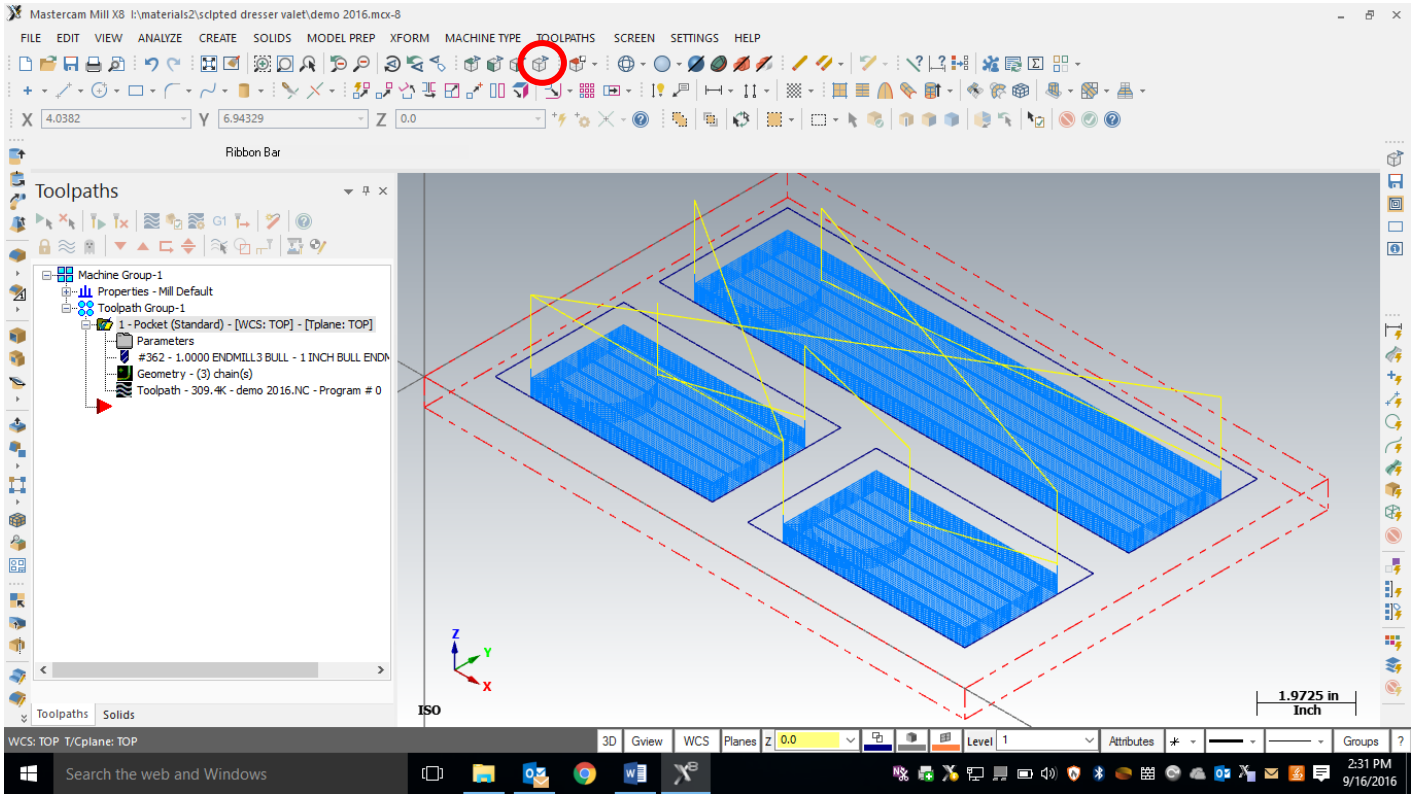


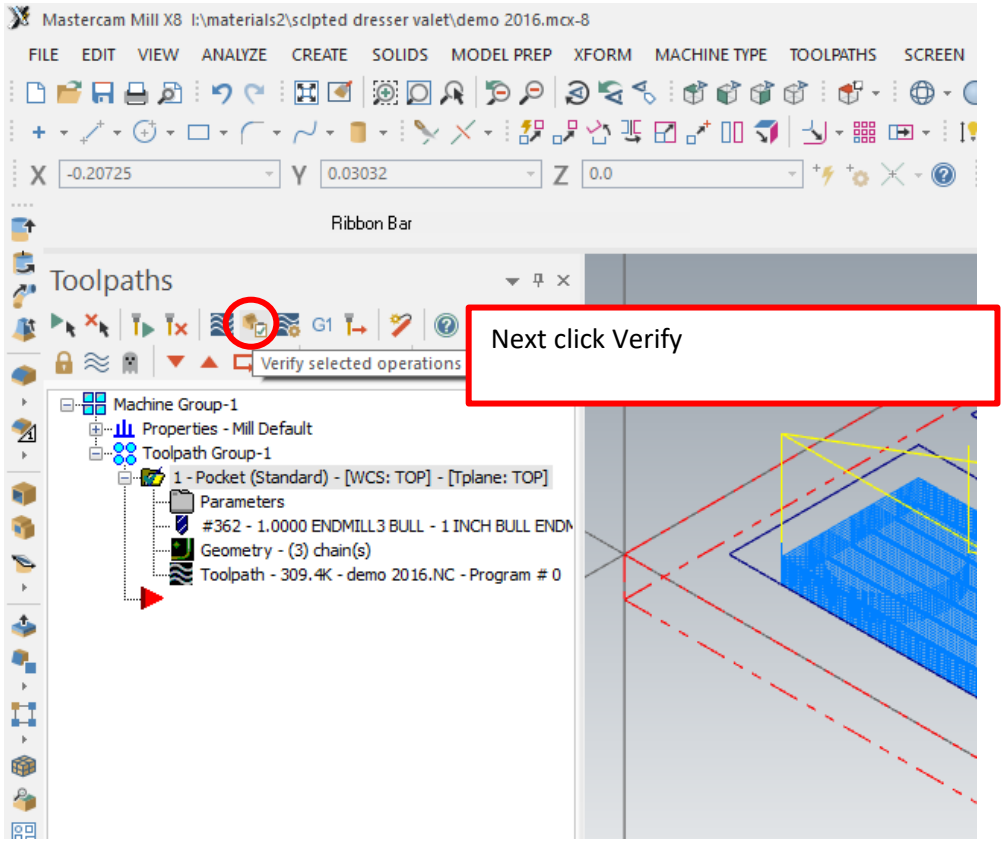


After you hit OK, you should see masterCAM draw the toolpaths. The blue lines represent the center of the cutter when it is cutting material, and the yellow lines represent the center of the cutter when it moves between geometry.



To Verify your toolpath, go to an isometric view, zoom in/out, and pan the drawing so you see something such as below:





This will open up in a new window titled “MasterCAM Simulator”, and you can push the play button at the bottom of the window. It should see your part cut out virtually. Please show your instructor.