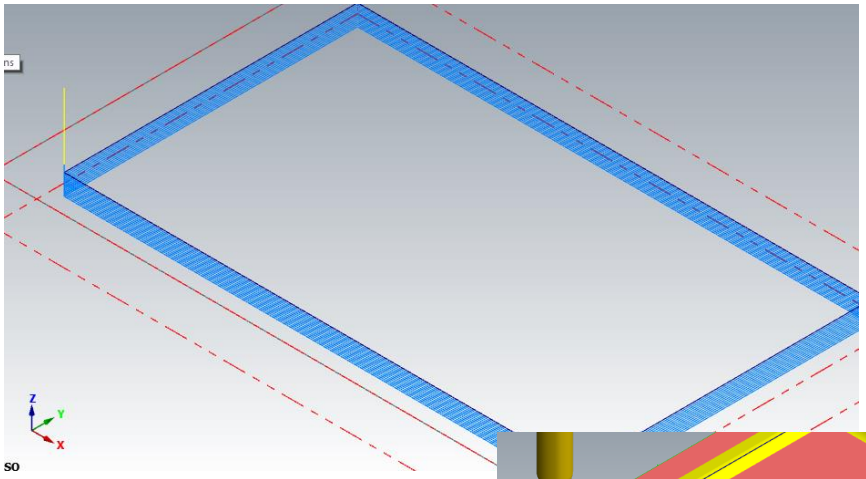
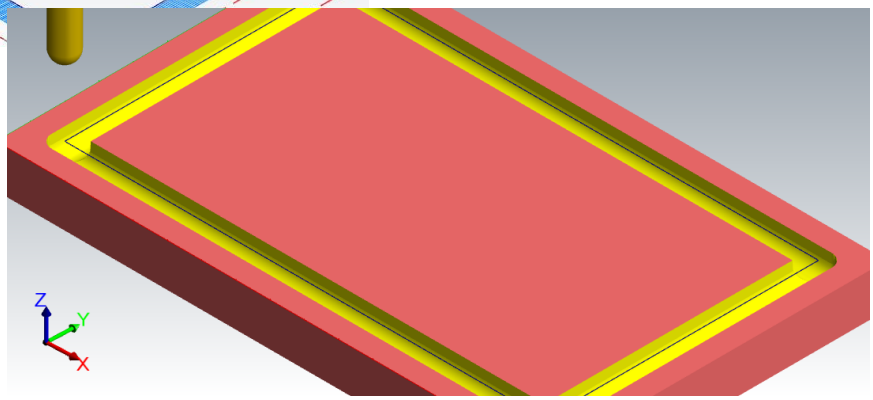
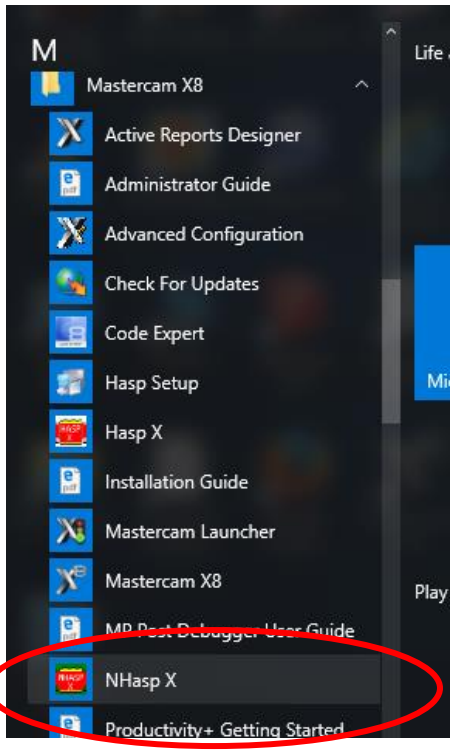
10” x 16” Cutting Board - Juice Groove in MasterCAM



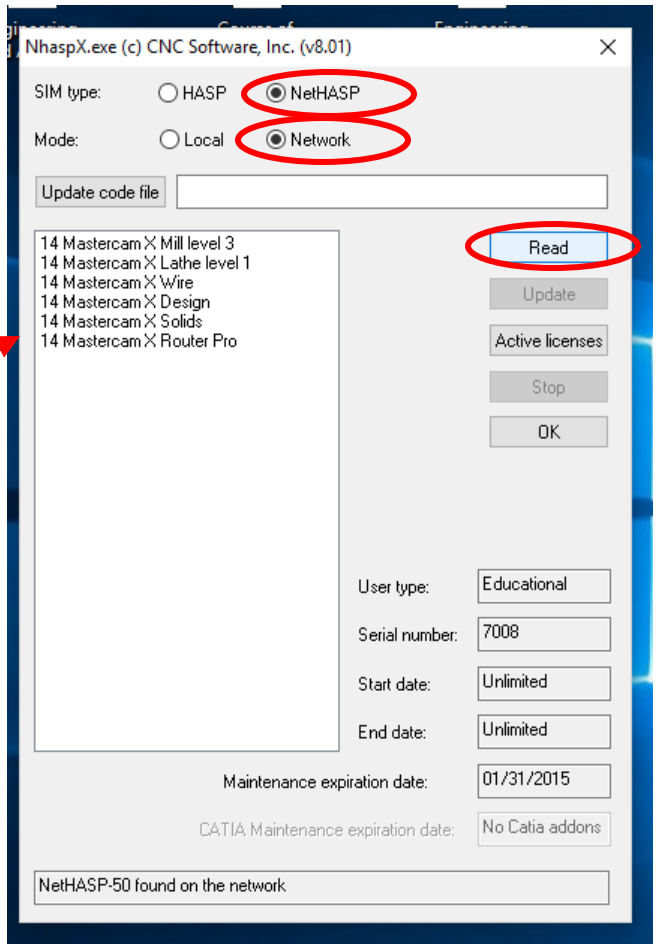


Check to make sure the nethasp is working/turned on to network.

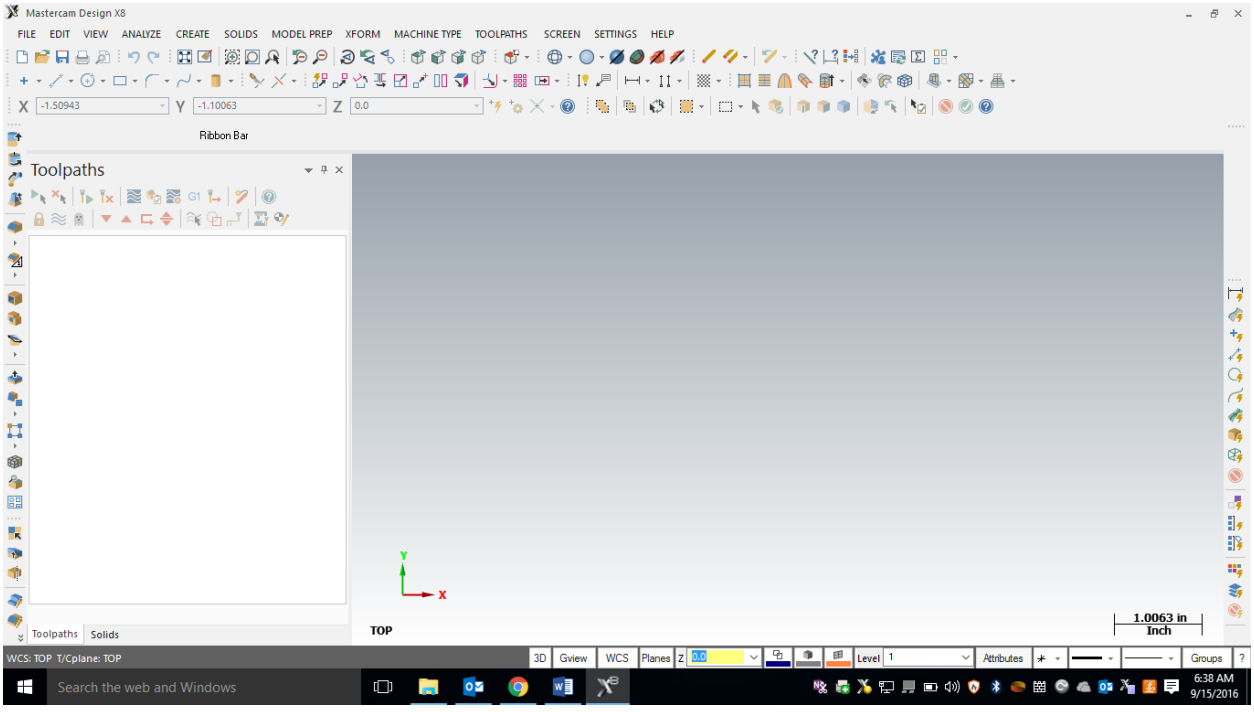
Go to ALL APPs/Mastercam x8/nethasp



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

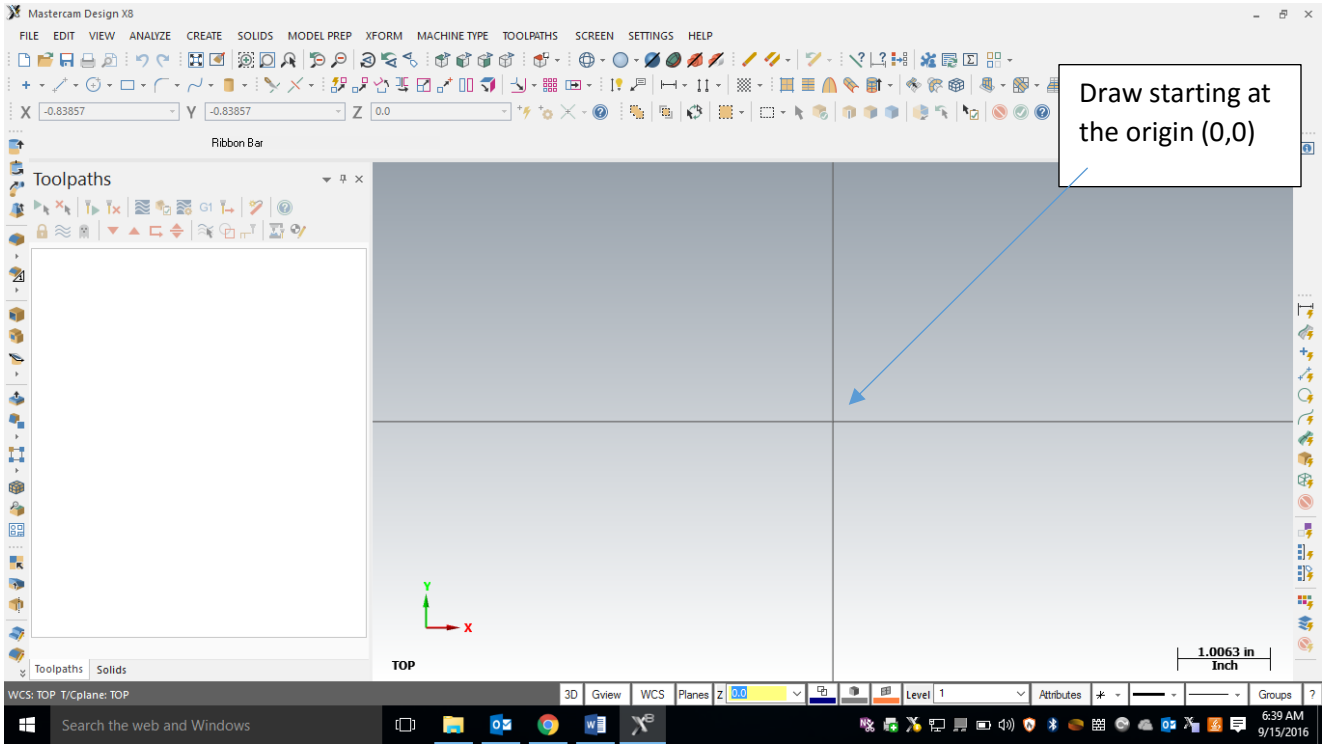


After you check that, please open the MasterCAM application, it should look something like below.

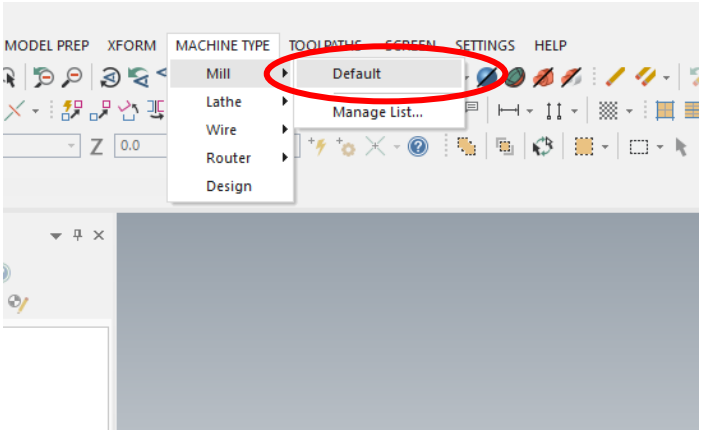


First thing is to figure out what you are making….Using the measurements from your plans, you will draw your geometry (geometry is a generic term for lines, arcs, etc. in a computer drawing program). This geometry must be drawn in the 1st quadrant of the coordinate system, so positive x and y. The placement 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.

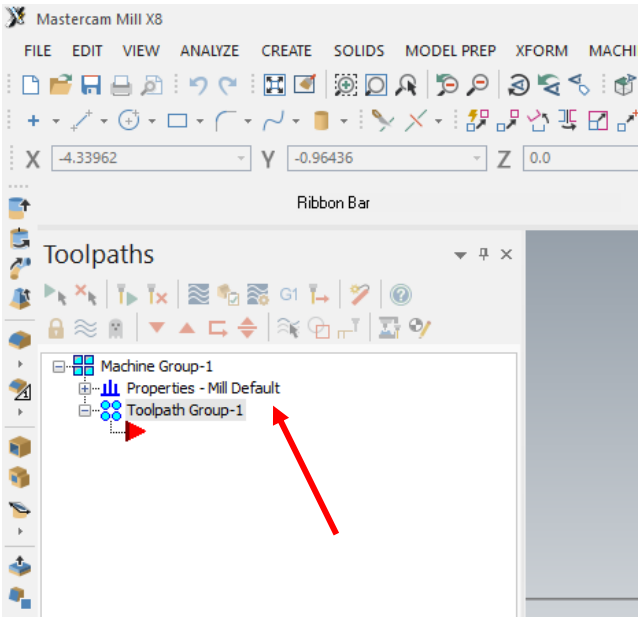
F9 will display the x/y axis such as:



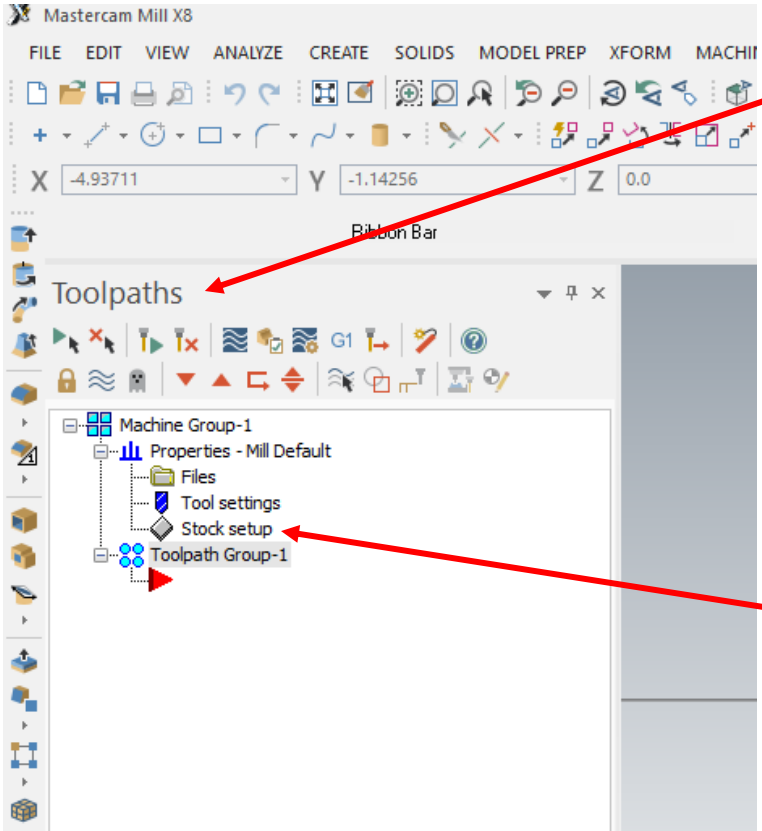
To start a project, we need to set our specific CNC router and set up the stock sizes. MasterCAM can write NC code for different manufacturers of CNC equipment. Our router is called a Forest Scientific Velocity 3 axis mill. MasterCAM will write the correct type of code as long as we pick the correct machine definition. Currently the only computer with this machine definition is the one hooked to the CNC router, so please just pick the default, then your instructor will change it at the CNC machine. This is a critical first step, without a machine definition, the CNC router will crash….litterly the tool bit will dive into the table top. **Goto Machine Type/Mill/Default.**



The result: there should be one machine group (“Machine Group -1”) that says “Properties – Mill Default”, if there is other Machine Groups, right-click and delete them.

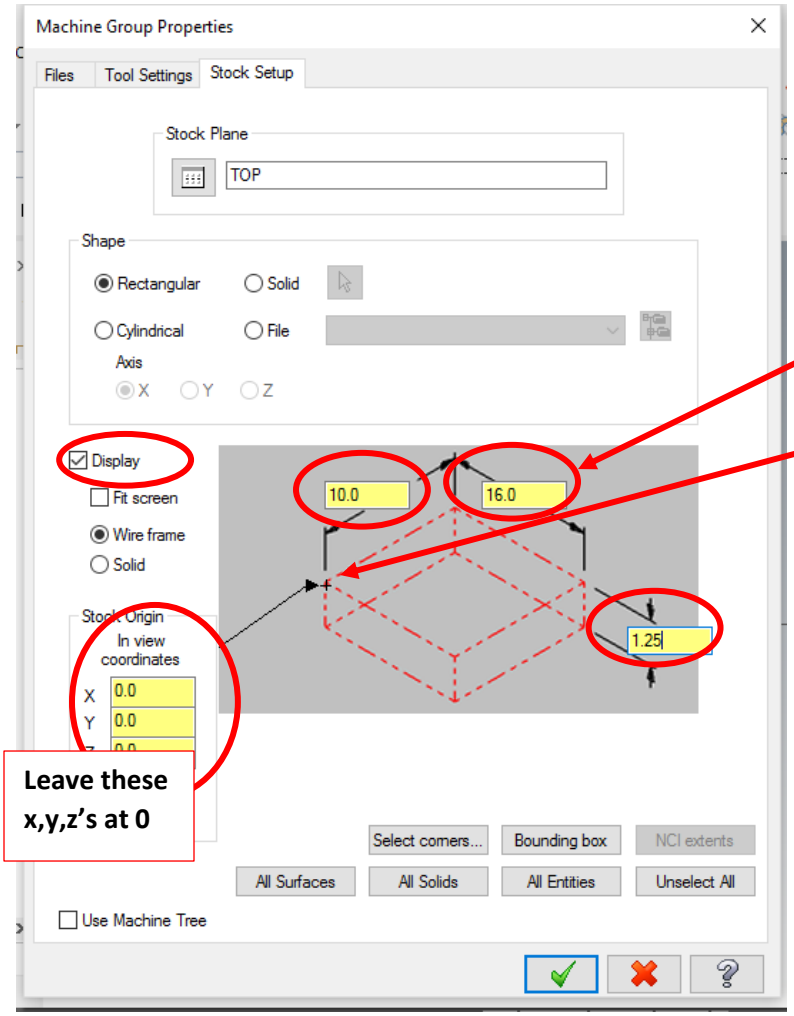


Stock Setup



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

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

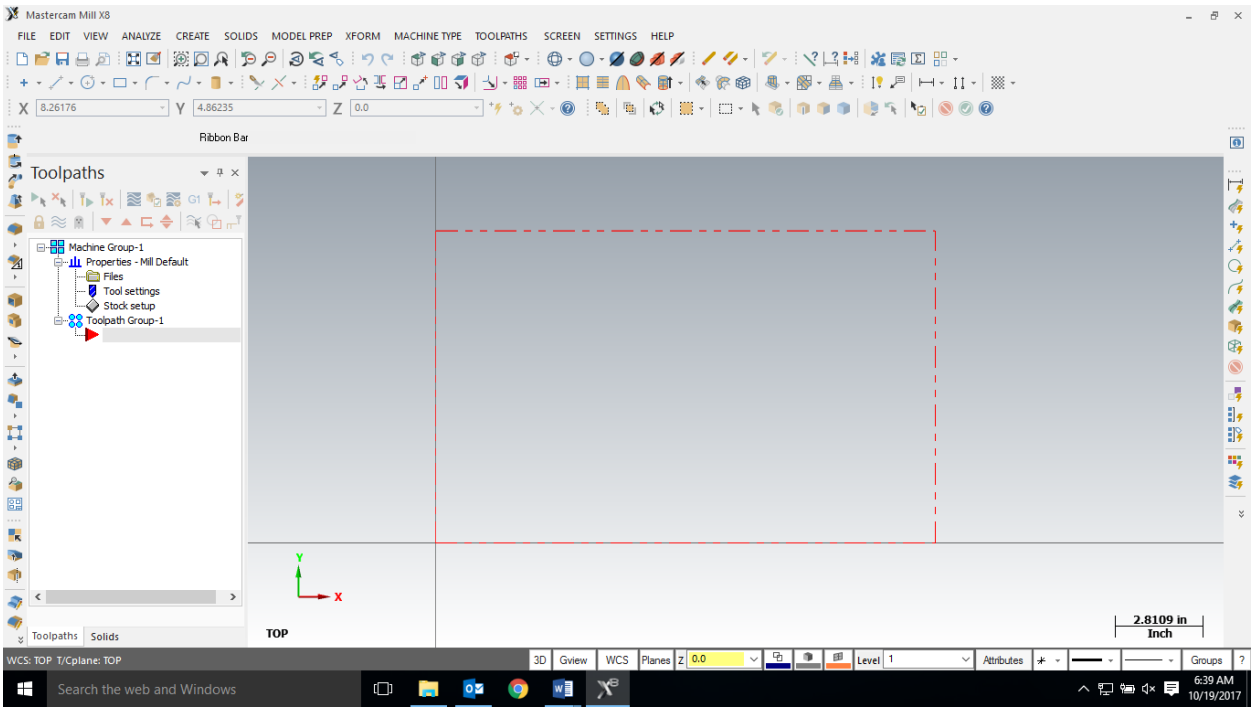


Setup the stock:
Enter the measurements
16 for y
10 for x
1.25 for z
Set the stock origin by clicking on this corner.

Check "Display"

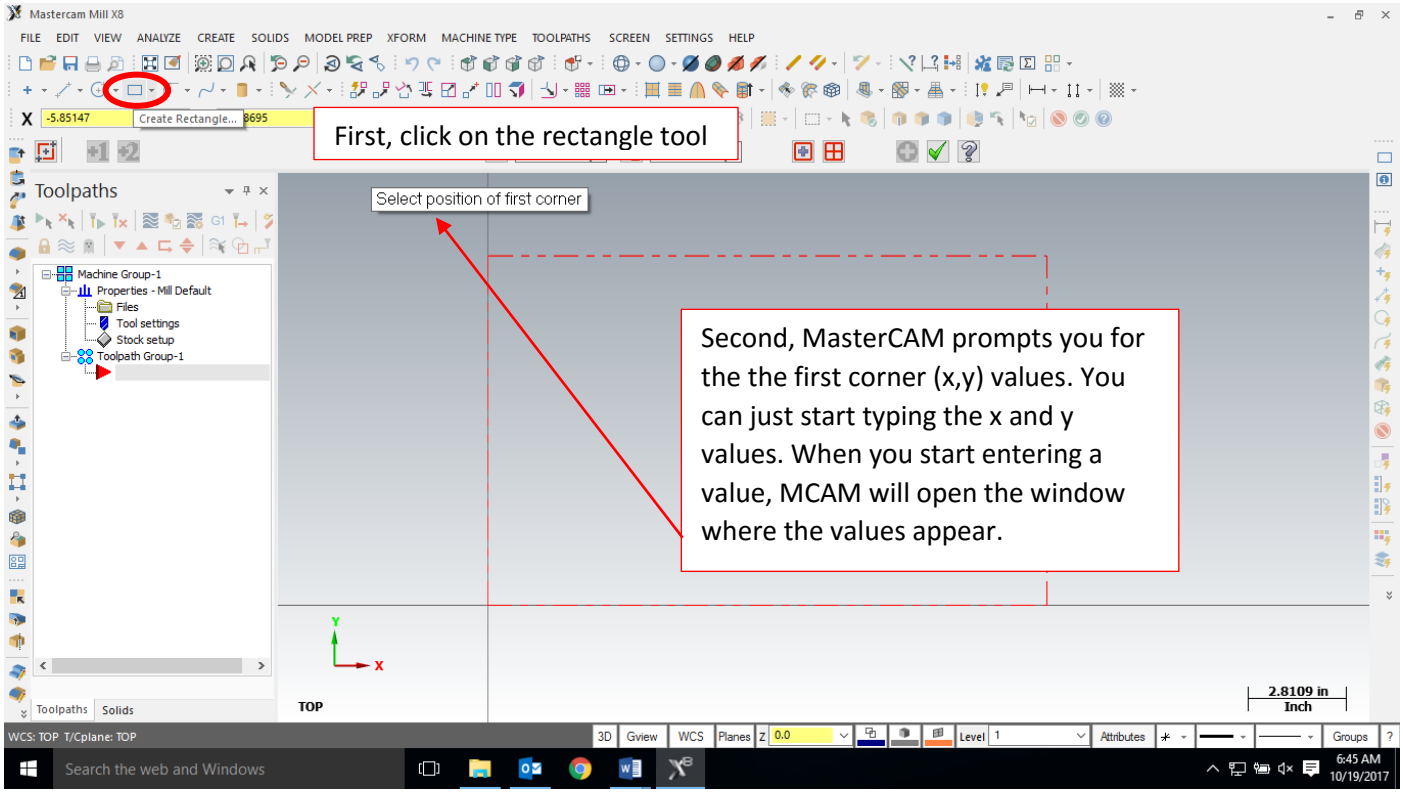
Click the Green Check Mark (OK)

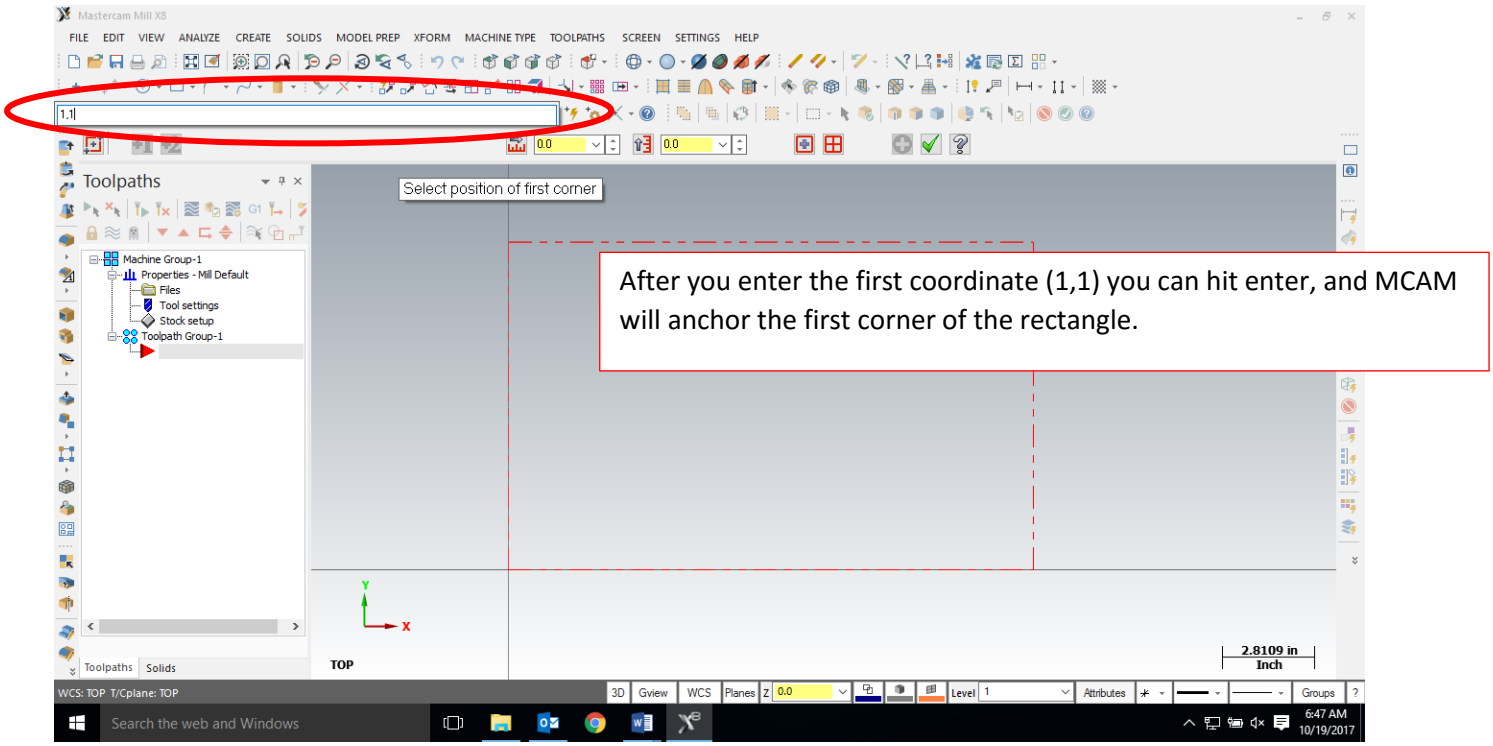
After you click ok in the stock setup, you should see a red dashed red rectangle that represents your stock. Zoom in or out so that you see the whole piece. Zoom with the scroll wheel on the mouse, and use the arrow keys to move left/right/up/down.



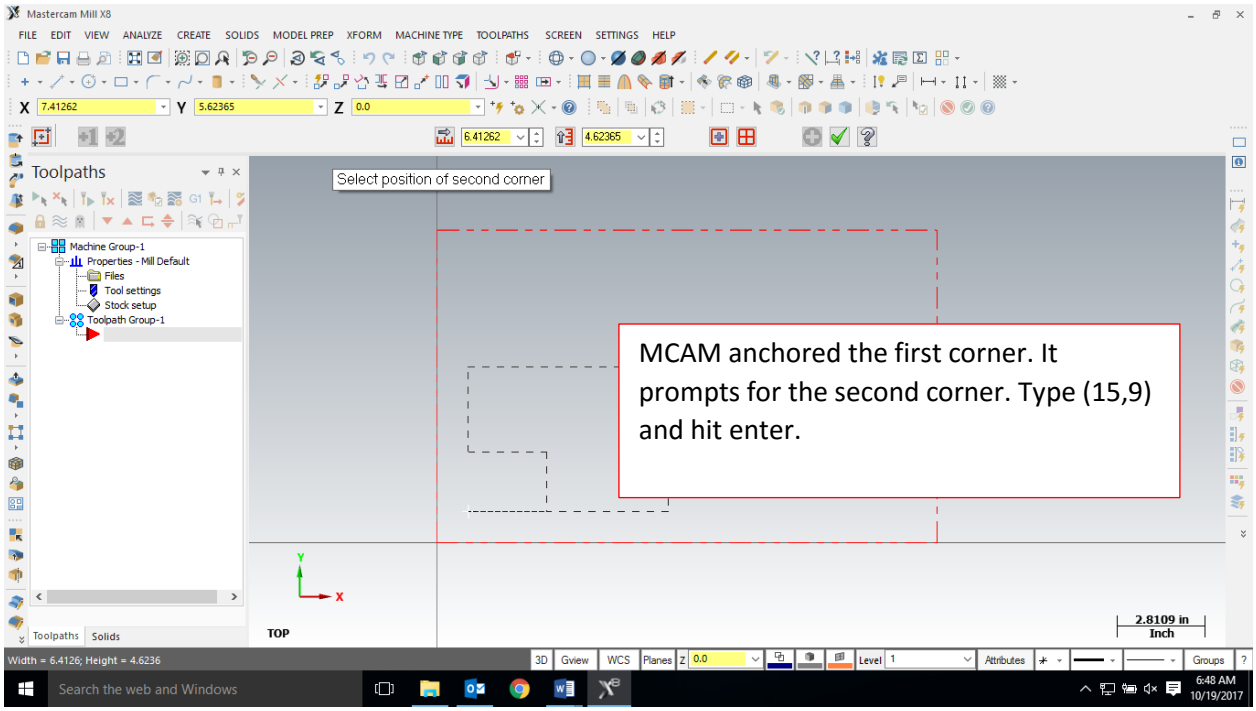
Entering Geometry

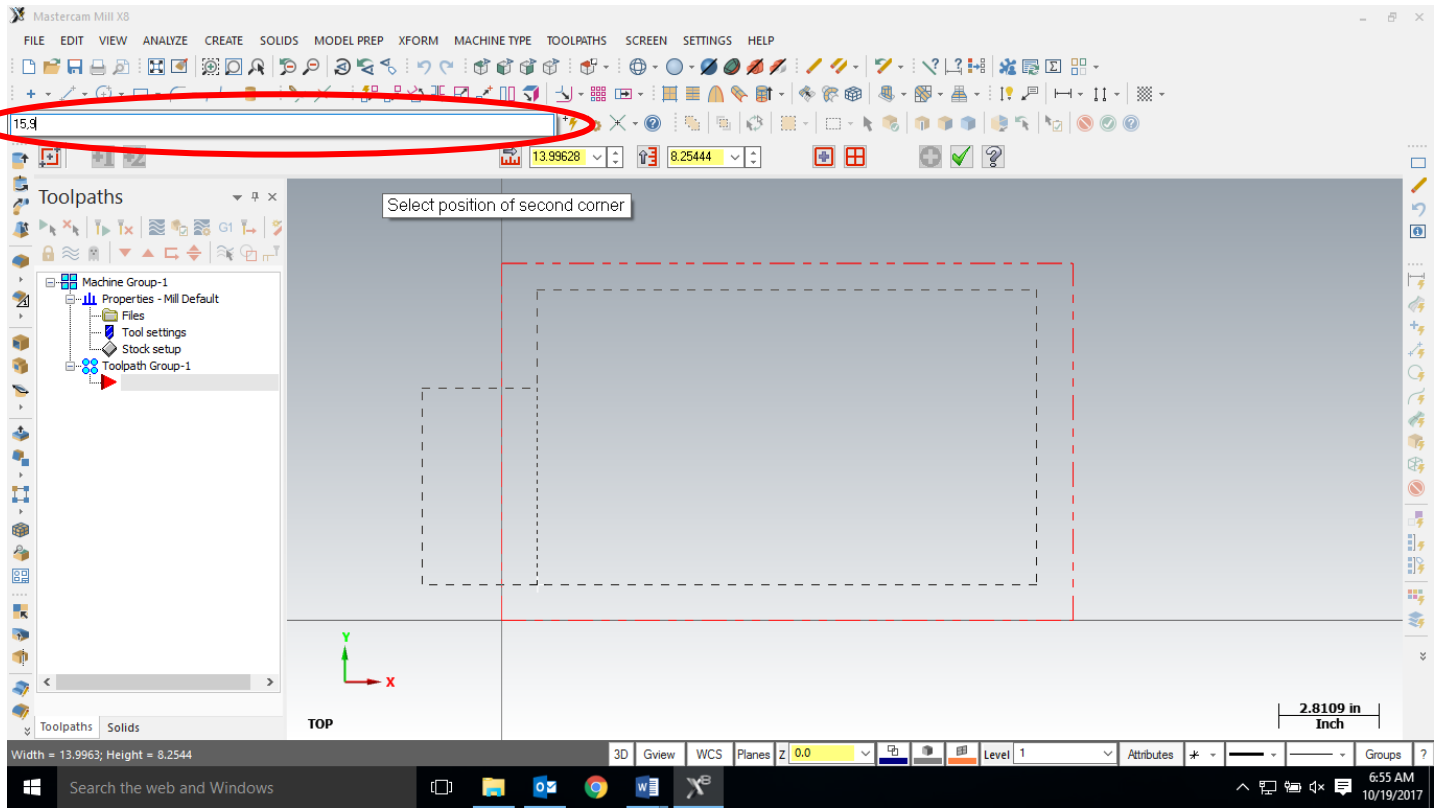
It's time to start drawing some geometry. If you draw what you want on your work piece it helps figure out the (x,y) coordinates. For this operation, the tool bit will follow a rectangle. The rectangle should be 1" smaller all the way around compared to your work piece. The cutting board should be 10" x 16", so the rectangle will be 8" x 14", with a 1" margin all the way around. It starts at (1,1) and the other corner is (15,9). We need the rectangle tool, and we can enter those values for the two corners of the rectangle.



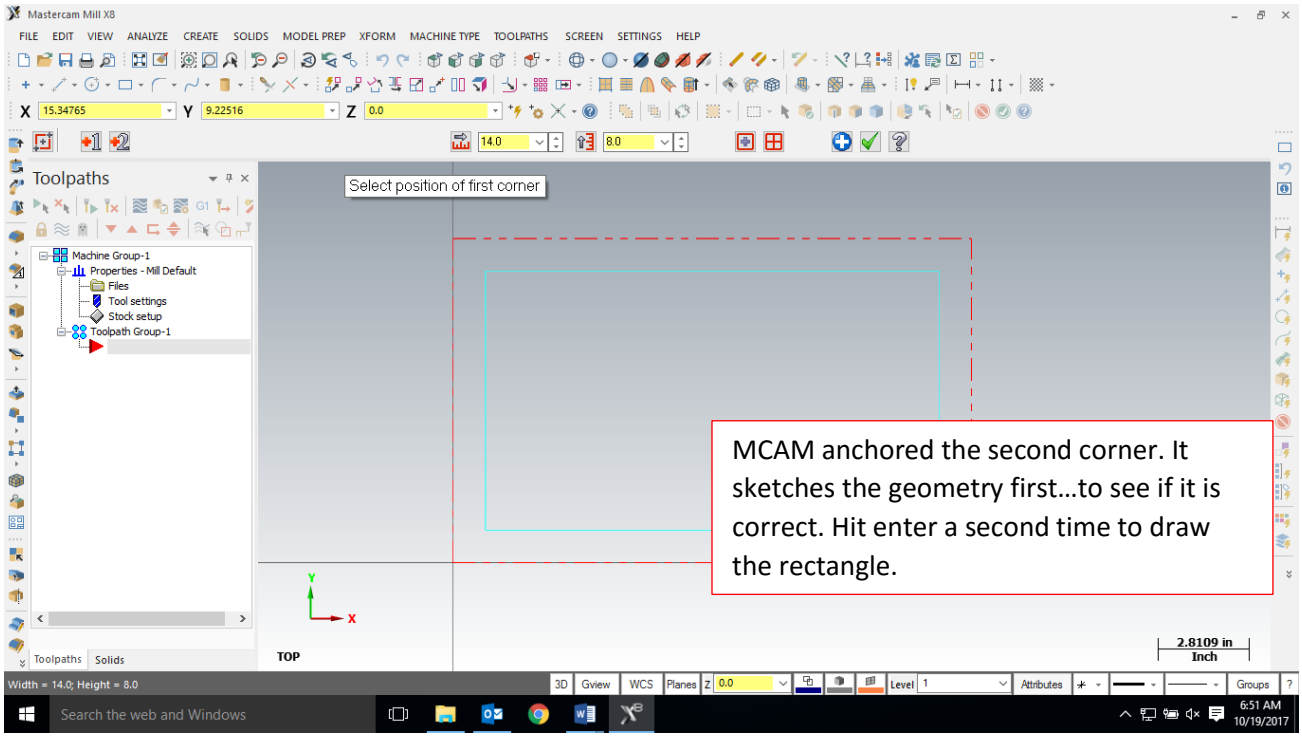


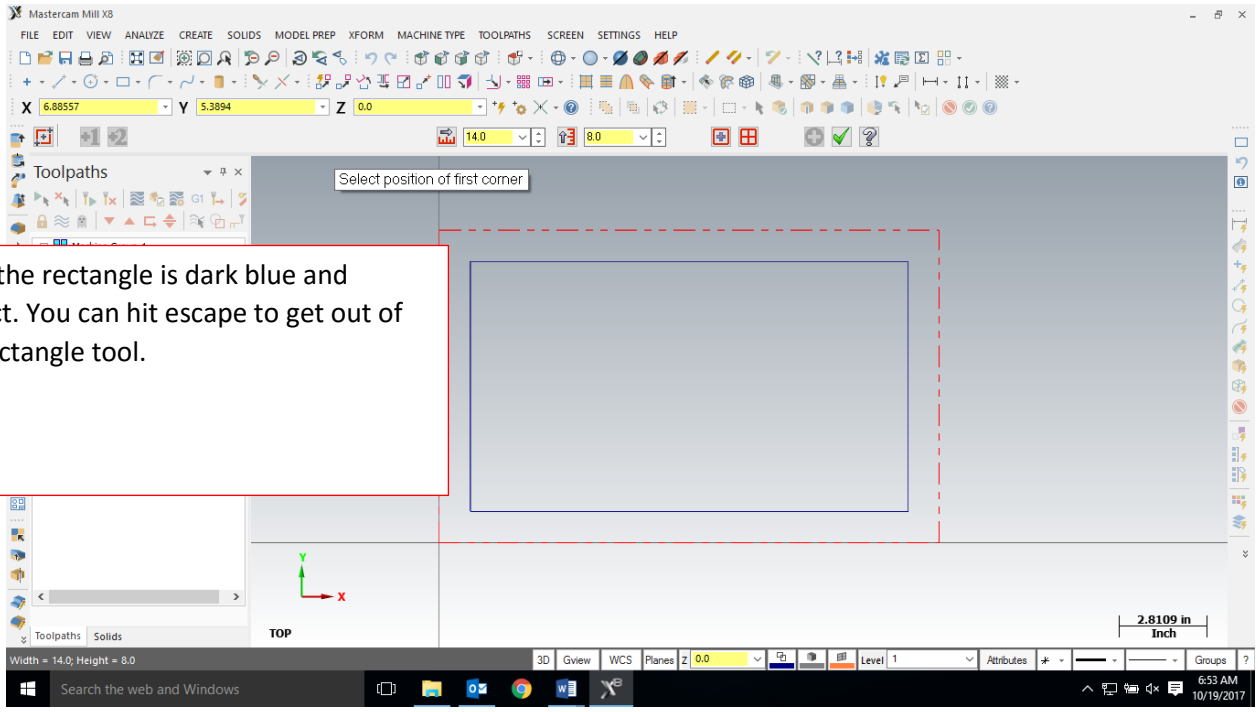
Result:


Mastercam Mill X8
Select position of second corner
MCAM anchored the first corner. It prompts for the second corner. Type (15,9) and hit enter.
TOP
2.8109 in
Inch


Mastercam Mill X8
15,9
Select position of second corner
TOP
2.8109 in
Inch

Result:

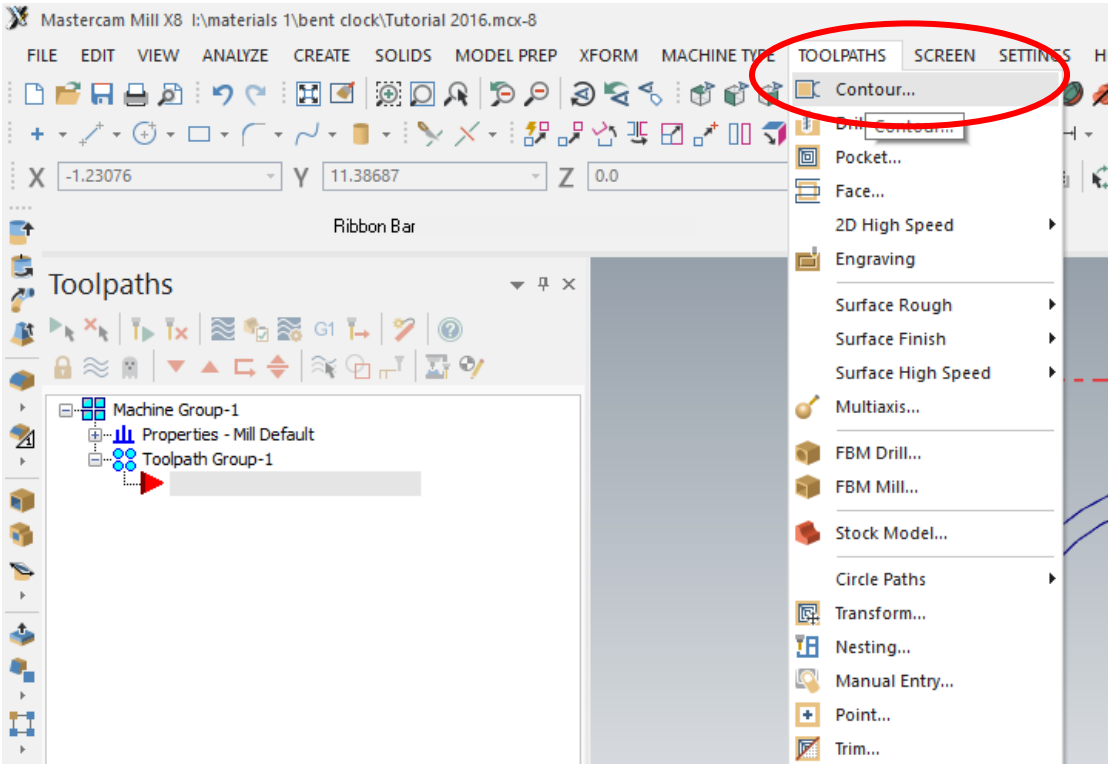

Mastercam Mill X8
Select position of first corner
MCAM anchored the second corner. It sketches the geometry first…to see if it is correct. Hit enter a second time to draw the rectangle.
TOP
2.8109 in
Inch


Mastercam Mill X8
Select position of first corner
Once the rectangle is dark blue and correct. You can hit escape to get out of the rectangle tool.
TOP
2.8109 in
Inch

Toolpaths:

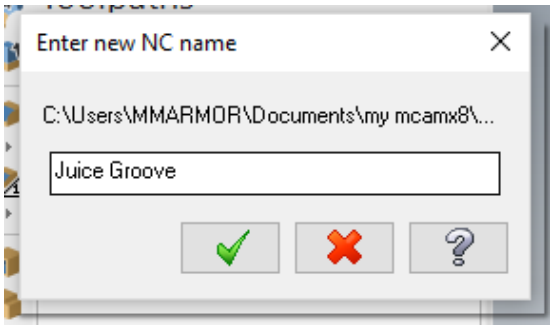
For 2D geometry such as we have, there are 2 main types of tool paths. The first one is a contour. In a contour toolpath, the tool bit will follow a path. The path can be one piece of geometry or multiple pieces of geometry linked together end to end (this is called a chain). When the geometry is selected, 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 three contour toolpaths on the three singular pieces of geometry. The second type of toolpath is a pocket. A pocket toolpath will make a cavity inside the selected geometry. An example of a pocket toolpath would be the handles on the bottom of the cutting board.

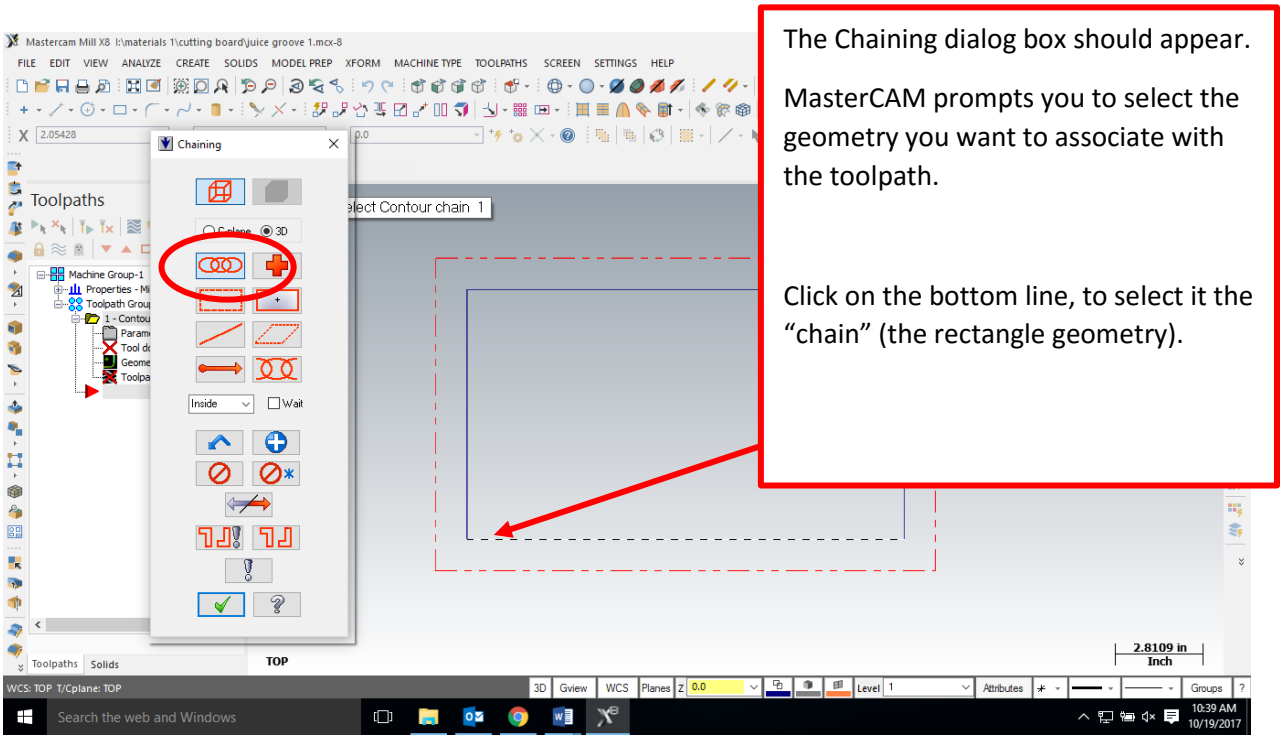
To start the toolpaths, go to Toolpaths/contour



When the new NC dialog box comes up, type in a good file name such as Clock Front.

Click the green check.

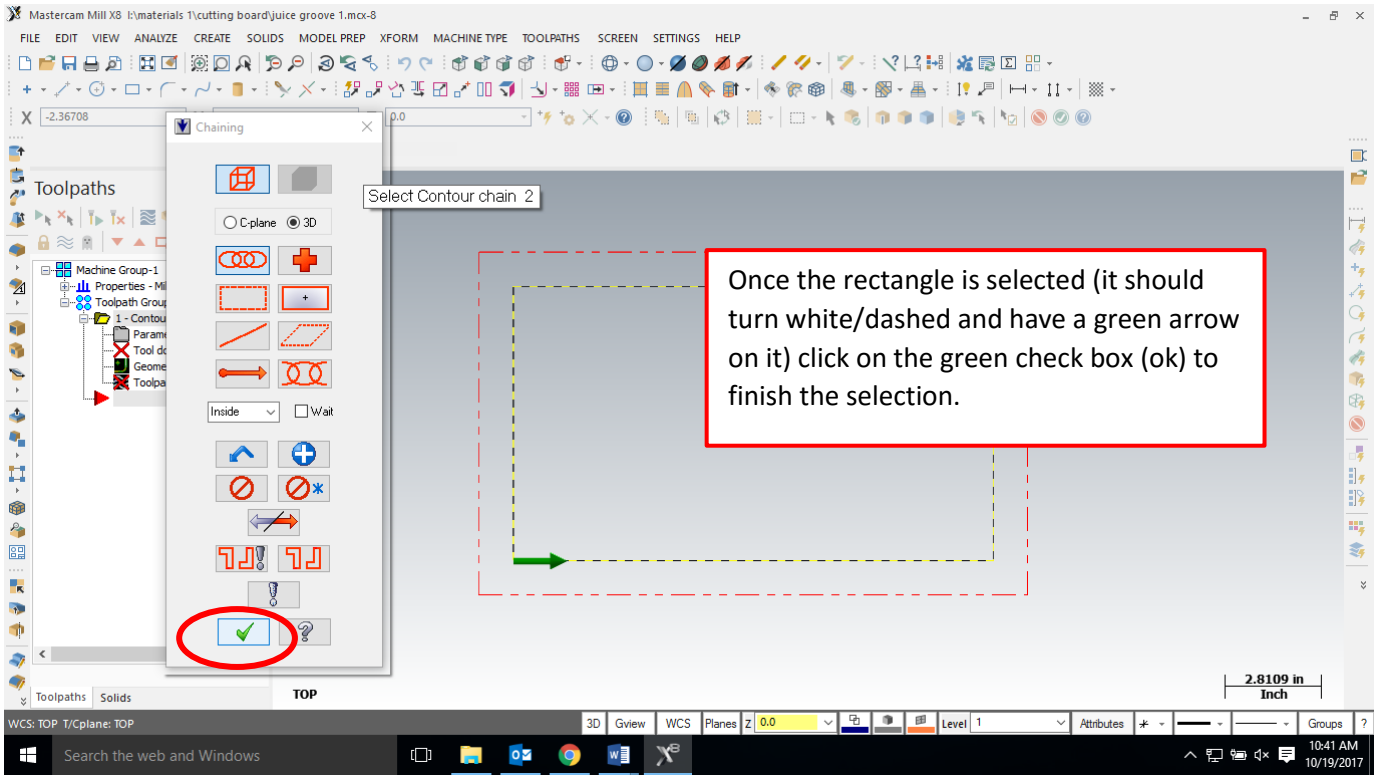




The Chaining dialog box should appear.

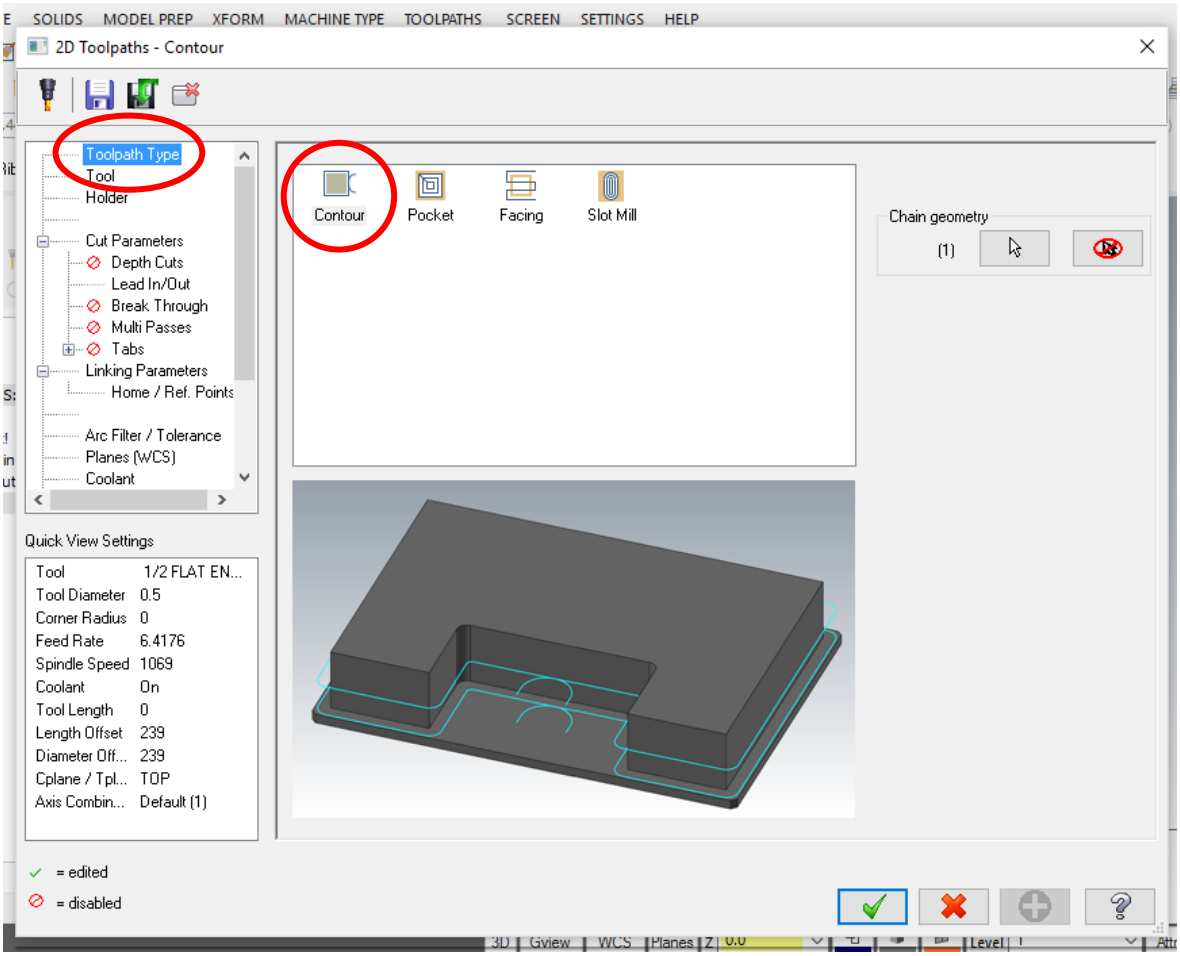
MasterCAM prompts you to select the geometry you want to associate with the toolpath.

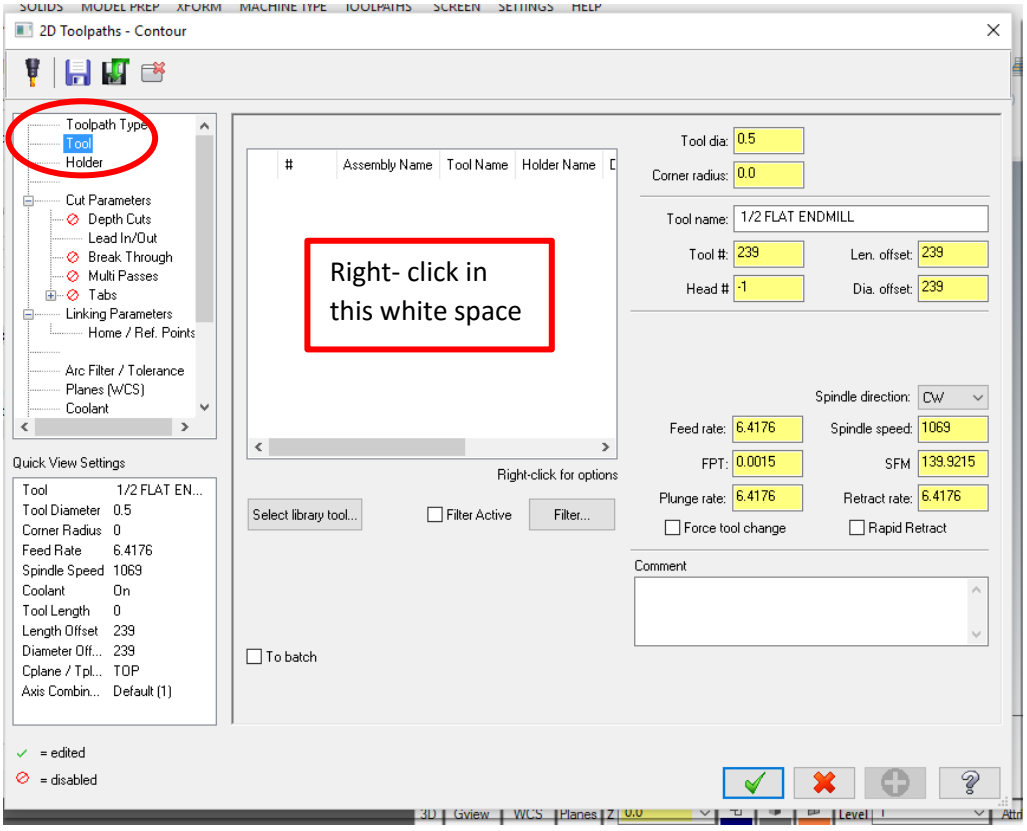
Click on the bottom line, to select it the "chain" (the rectangle geometry).

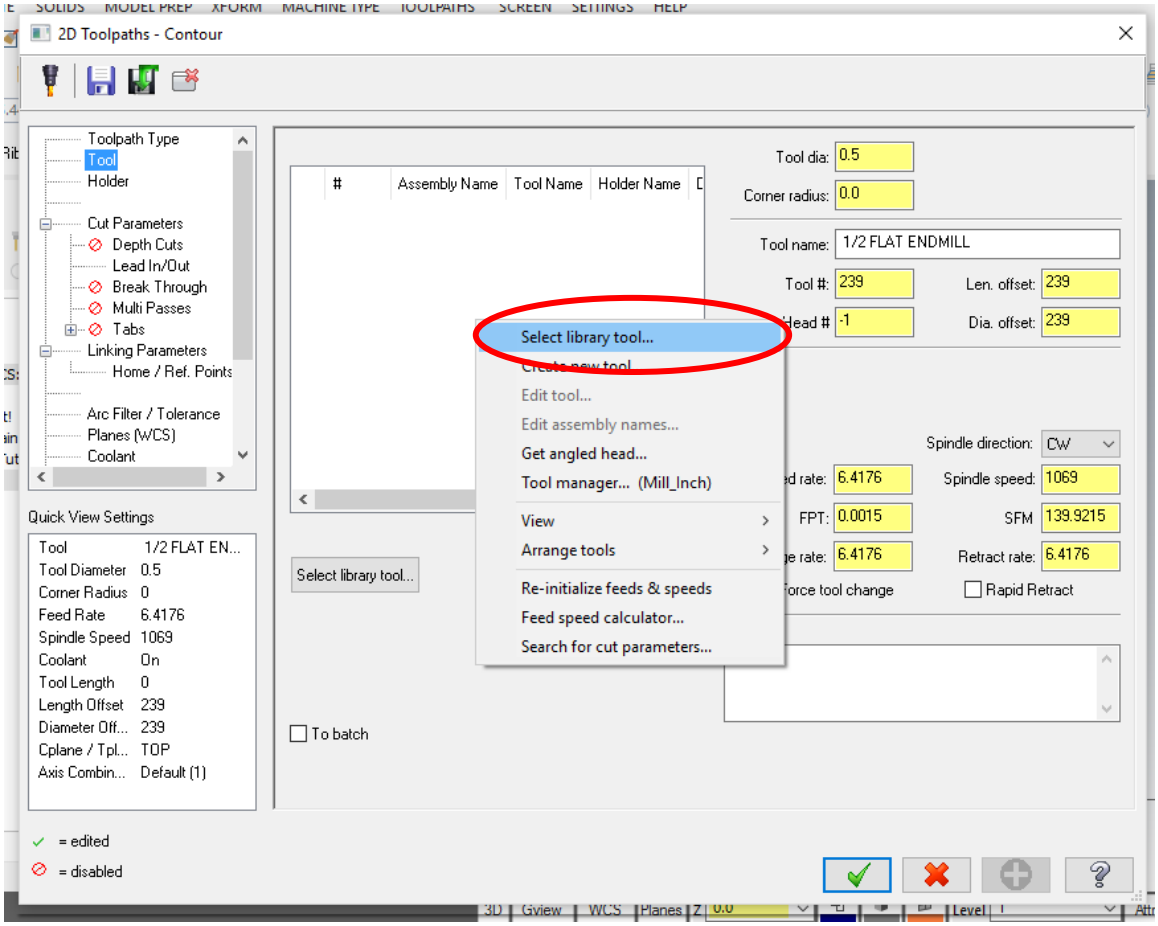


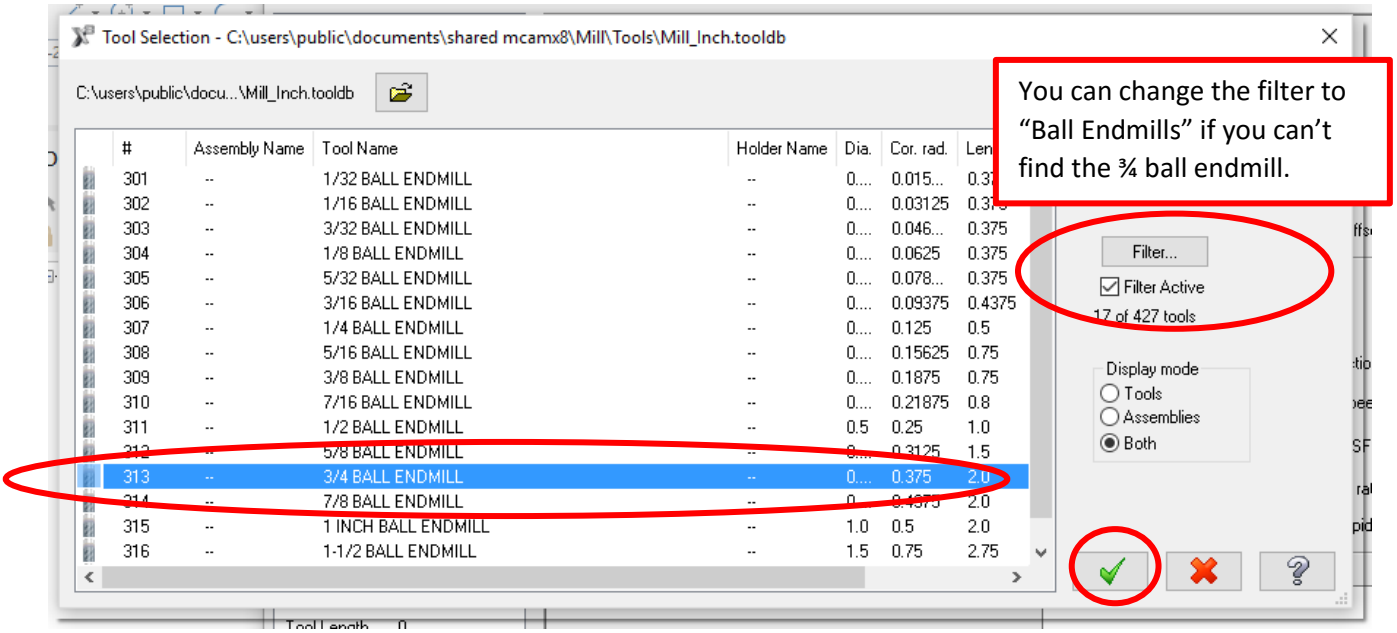
Once the rectangle is selected (it should turn white/dashed and have a green arrow on it) click on the green check box (ok) to finish the selection.

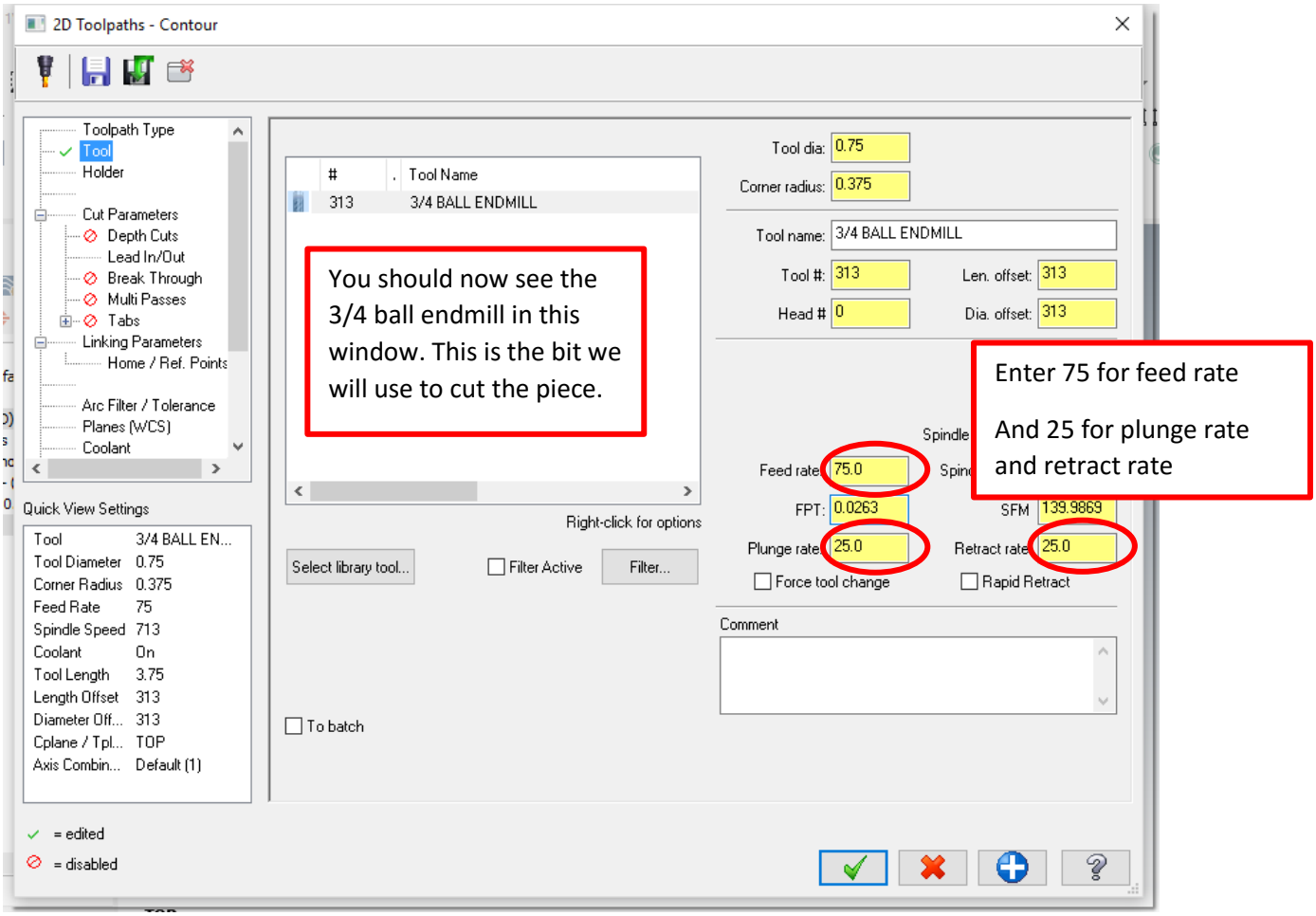
In the 2D Toolpaths – Contour dialog box, please enter the following information:

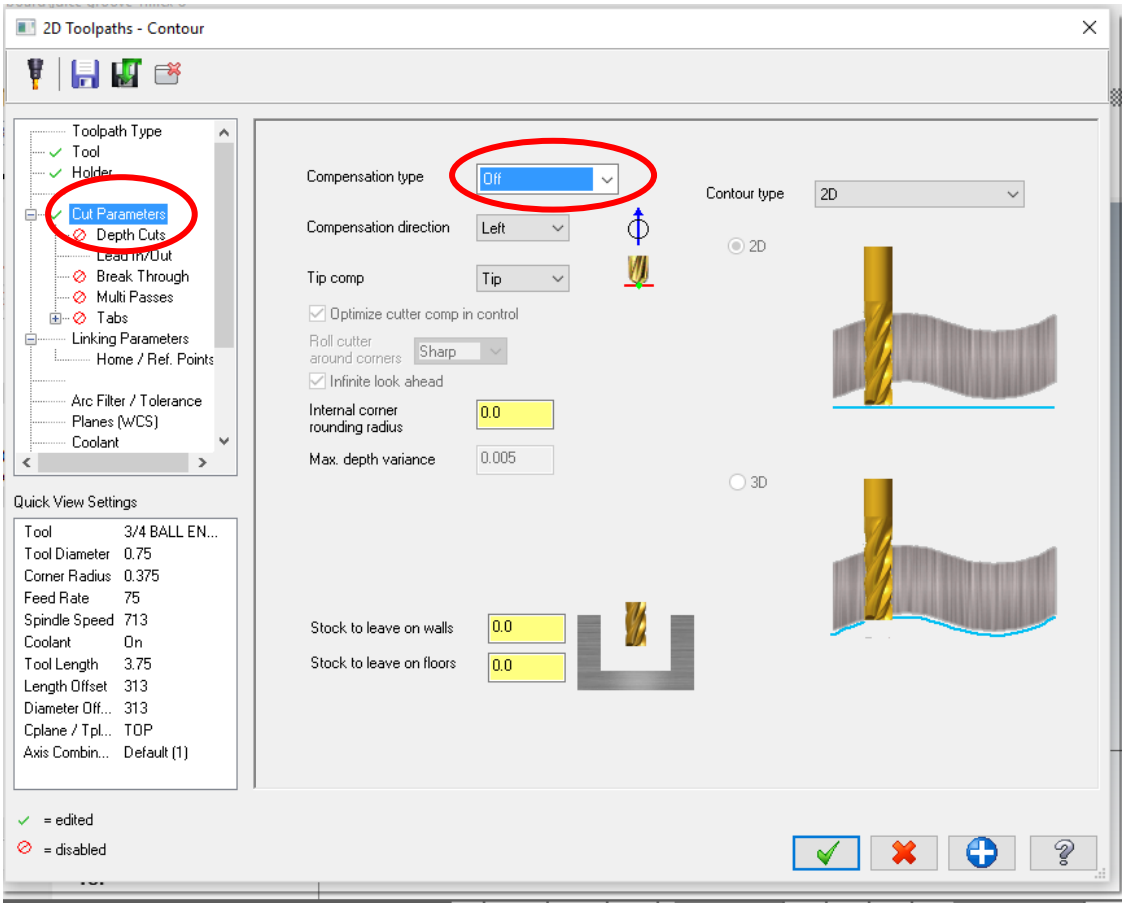


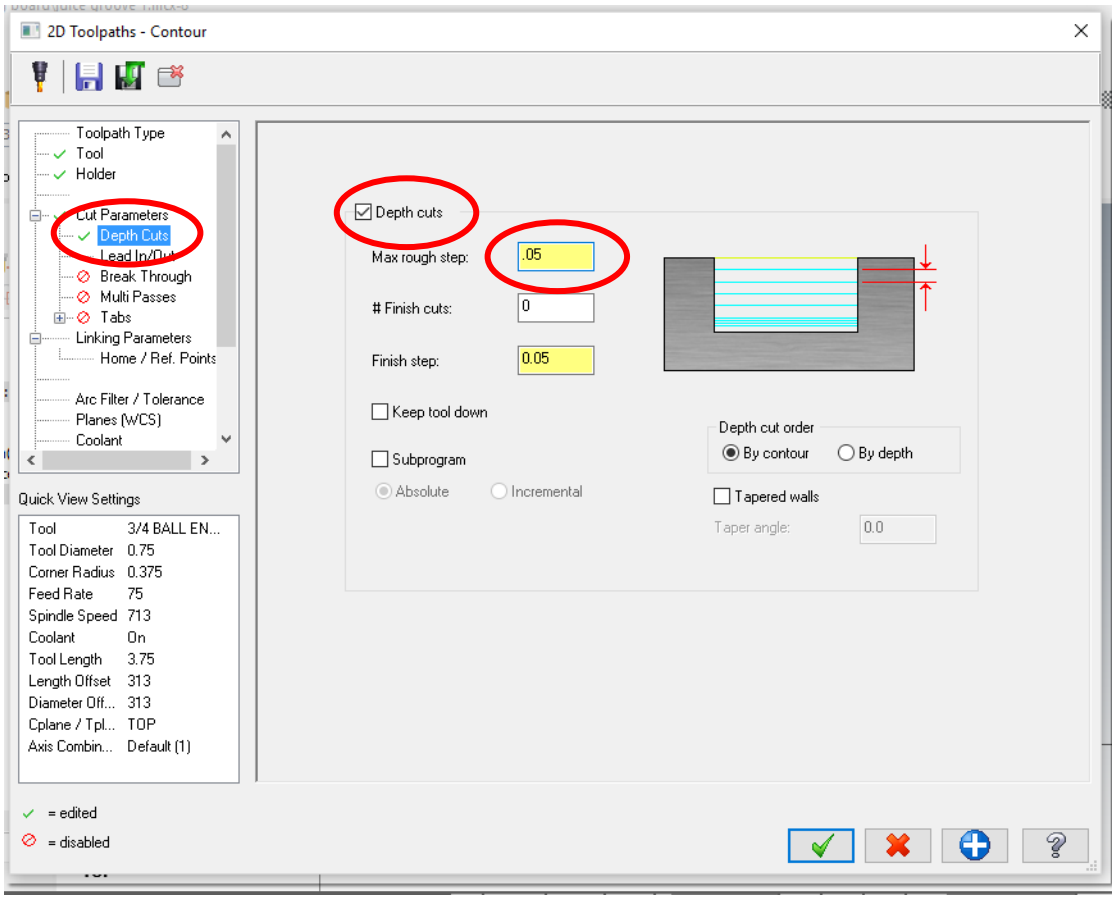


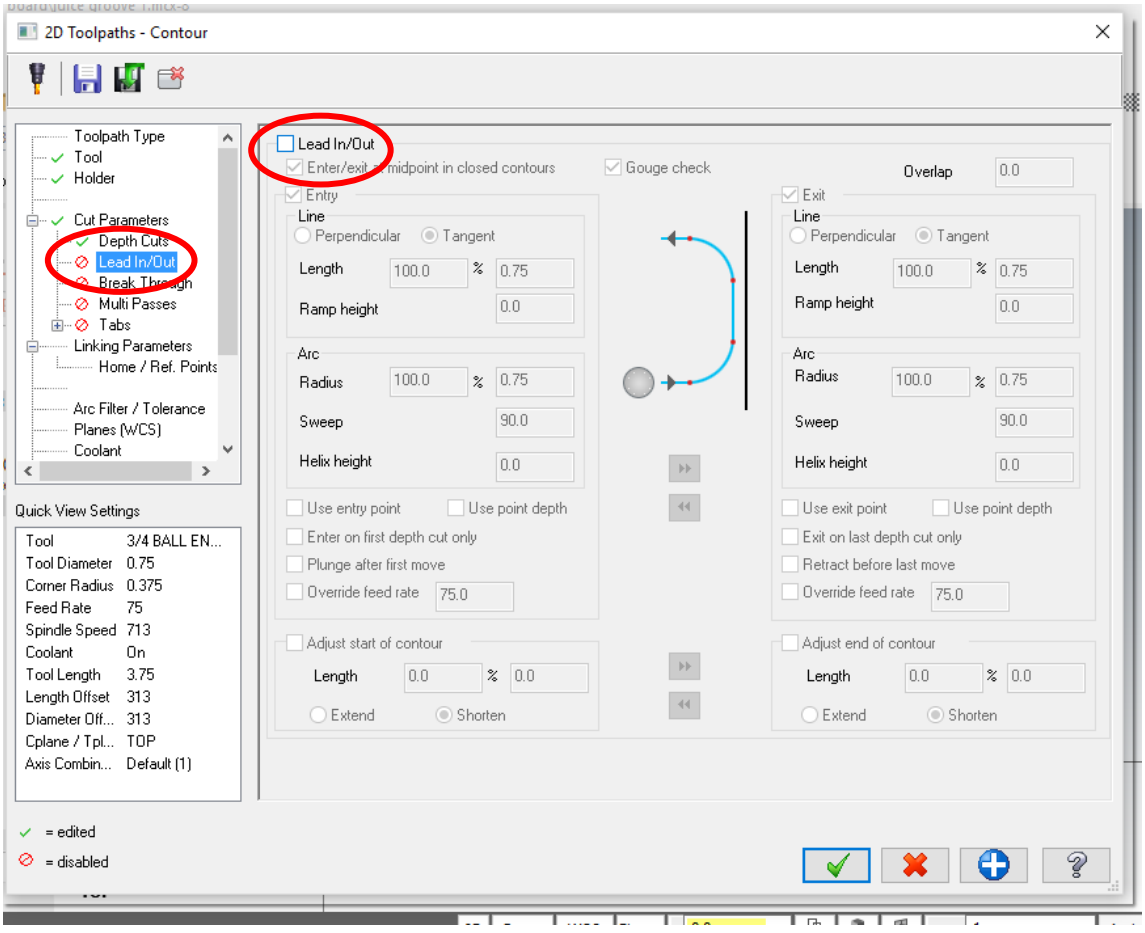

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 6.4176
Spindle Speed 1069
Coolant On
Tool Length 0
Length Offset 239
Diameter Off... 239
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: 0.0
Tool name: 1/2 FLAT ENDMILL
Tool #: 239
Len. offset: 239
Head # -1
Dia. offset: 239
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...
Spindle direction: CW
Spindle speed: 1069
FPT: 0.0015
SFM 139.9215
Retract rate: 6.4176
Rapid Retract

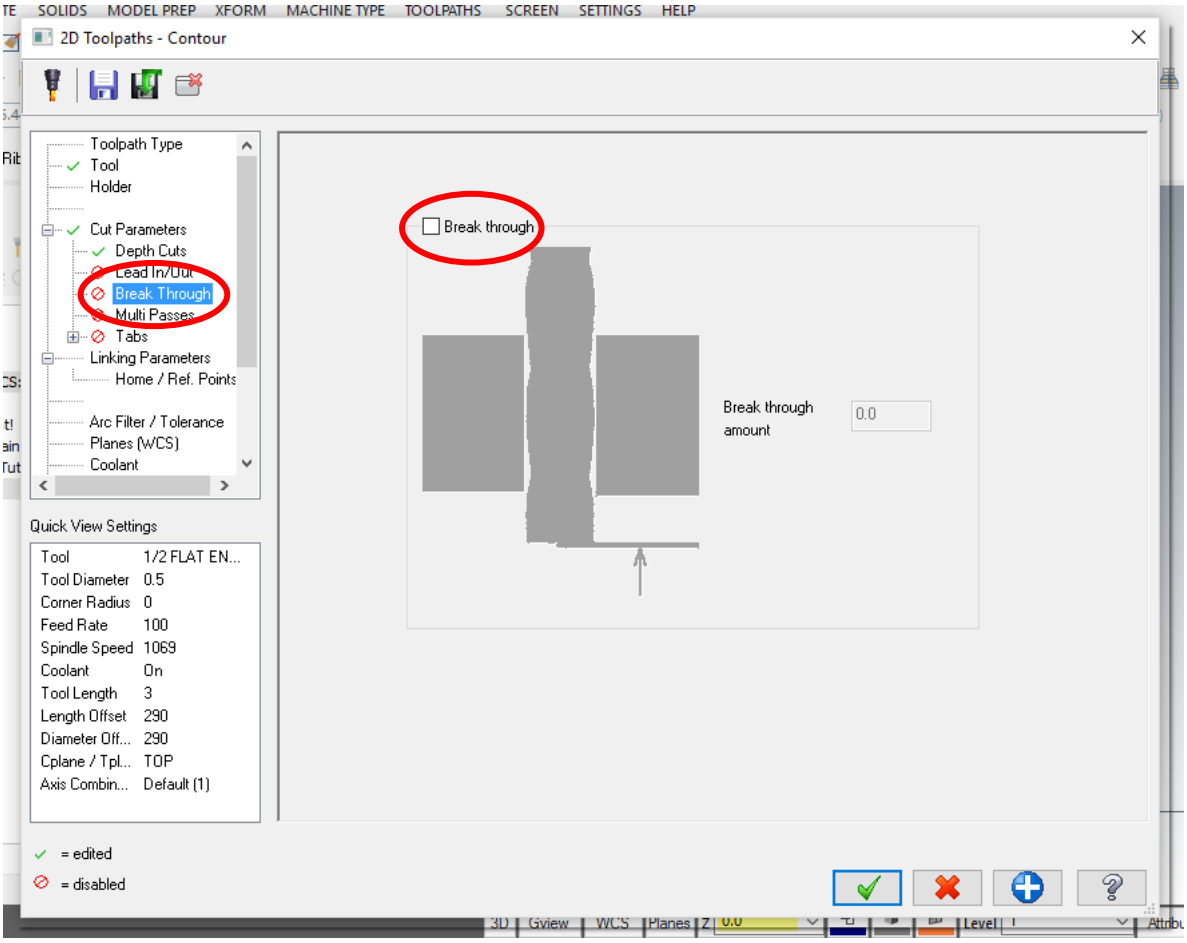

Tool Selection - C:\users\public\documents\shared mcamx8\Mill\Tools\Mill_Inch.tooldb
C:\users\public\docu...\Mill_Inch.tooldb
Assembly Name Tool Name Holder Name Dia. Cor. rad. Len
301 -- 1/32 BALL ENDMILL -- 0.... 0.015... 0.3
302 -- 1/16 BALL ENDMILL -- 0.... 0.03125 0.3
303 -- 3/32 BALL ENDMILL -- 0.... 0.046... 0.375
304 -- 1/8 BALL ENDMILL -- 0.... 0.0625 0.375
305 -- 5/32 BALL ENDMILL -- 0.... 0.078... 0.375
306 -- 3/16 BALL ENDMILL -- 0.... 0.09375 0.4375
307 -- 1/4 BALL ENDMILL -- 0.... 0.125 0.5
308 -- 5/16 BALL ENDMILL -- 0.... 0.15625 0.75
309 -- 3/8 BALL ENDMILL -- 0.... 0.1875 0.75
310 -- 7/16 BALL ENDMILL -- 0.... 0.21875 0.8
311 -- 1/2 BALL ENDMILL -- 0.5 0.25 1.0
312 -- 5/8 BALL ENDMILL -- 0.... 0.3125 1.5
313 -- 3/4 BALL ENDMILL -- 0.... 0.375 2.0
314 -- 7/8 BALL ENDMILL -- 0.... 0.4375 2.0
315 -- 1 INCH BALL ENDMILL -- 1.0 0.5 2.0
316 -- 1-1/2 BALL ENDMILL -- 1.5 0.75 2.75
You can change the filter to “Ball Endmills” if you can’t find the ¾ ball endmill.
Filter...
Filter Active
17 of 427 tools
Display mode
Tools
Assemblies
Both

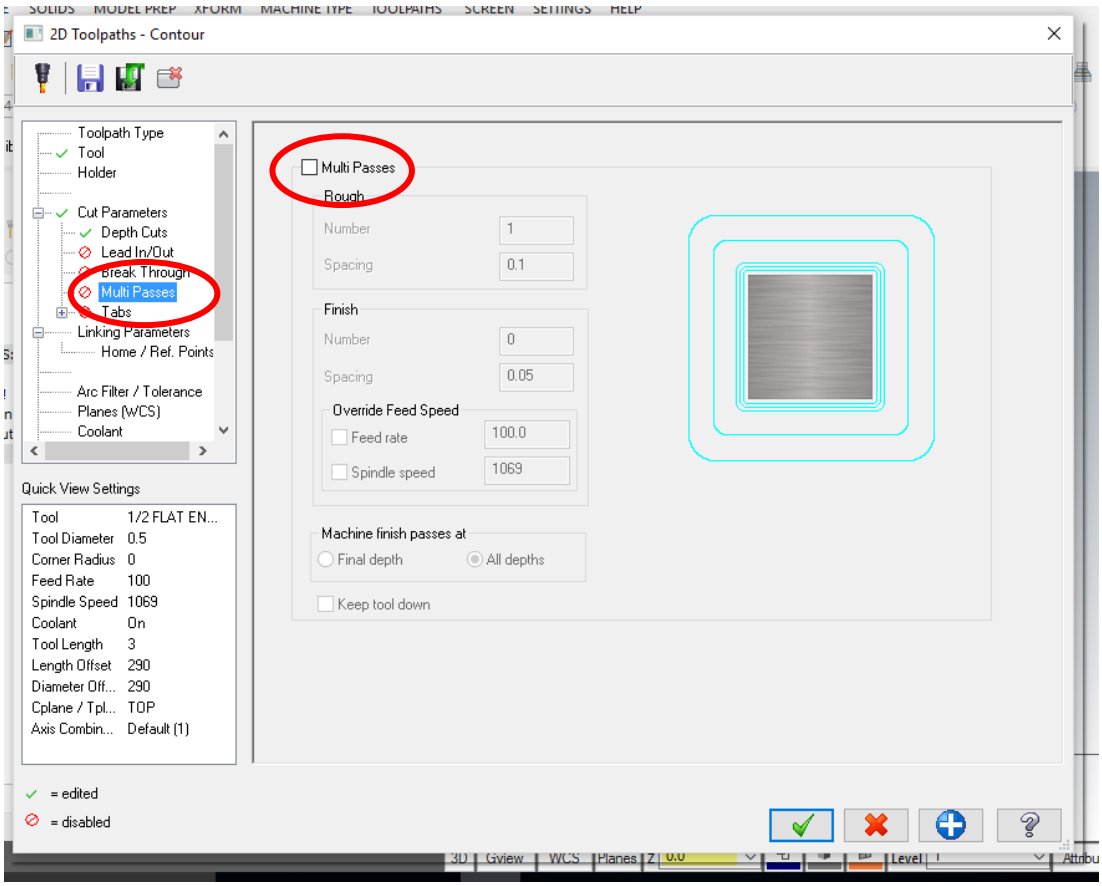

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 3/4 BALL EN...
Tool Diameter 0.75
Corner Radius 0.375
Feed Rate 75
Spindle Speed 713
Coolant On
Tool Length 3.75
Length Offset 313
Diameter Off... 313
Cplane / Tpl... TOP
Axis Combin... Default (1)
. Tool Name
313 3/4 BALL ENDMILL
You should now see the 3/4 ball endmill 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: 0.75
Corner radius: 0.375
Tool name: 3/4 BALL ENDMILL
Tool #: 313
Len. offset: 313
Head #: 0
Dia. offset: 313
Spindle
Feed rate: 75.0
FPT: 0.0263
SFM 139.9869
Plunge rate: 25.0
Retract rate: 25.0
Force tool change
Rapid Retract
Comment
Enter 75 for feed rate
And 25 for plunge rate and retract rate
= edited
= disabled

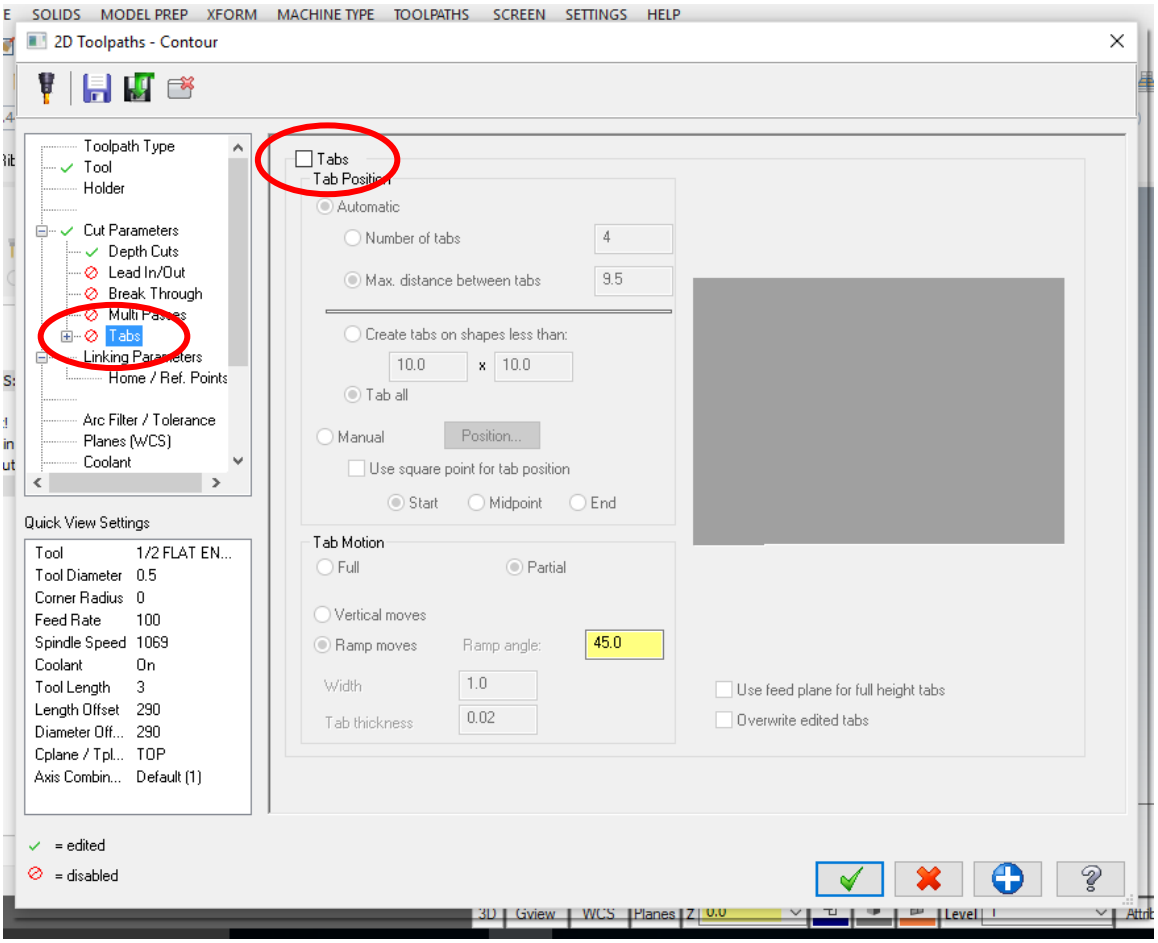

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 3/4 BALL EN...
Tool Diameter 0.75
Corner Radius 0.375
Feed Rate 75
Spindle Speed 713
Coolant On
Tool Length 3.75
Length Offset 313
Diameter Off... 313
Cplane / Tpl... TOP
Axis Combin... Default (1)
Compensation type: Off
Compensation direction: Left
Tip comp: Tip
Optimize cutter comp in control
Roll cutter around corners: Sharp
Infinite look ahead
Internal corner rounding radius: 0.0
Max. depth variance: 0.005
Stock to leave on walls: 0.0
Stock to leave on floors: 0.0
Contour type: 2D
2D
3D
= edited
= disabled

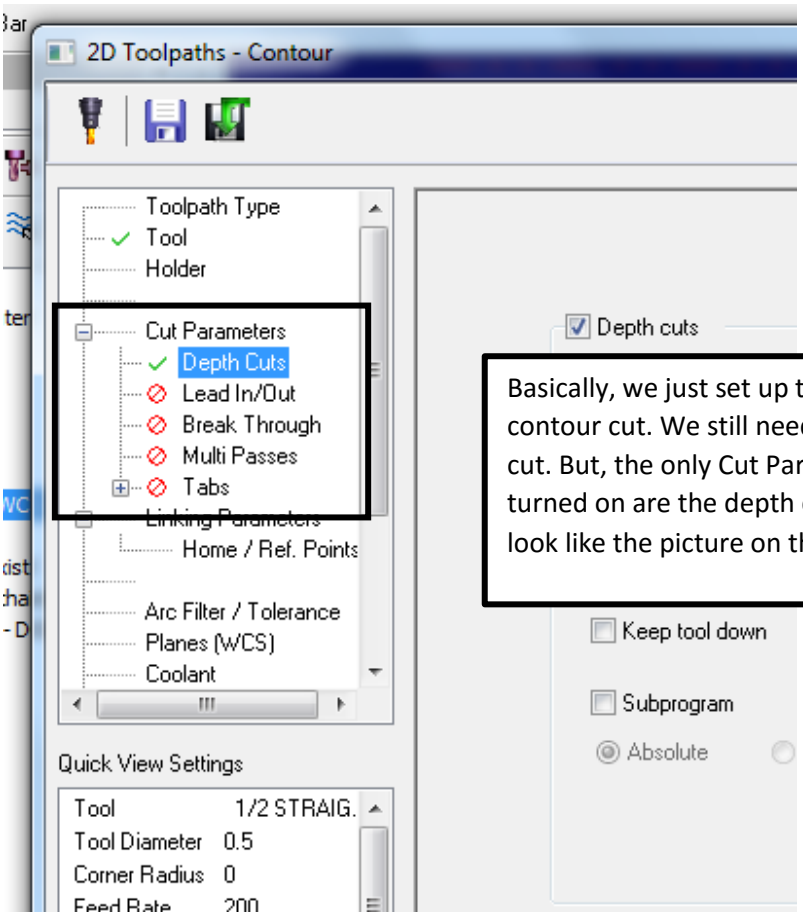

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Depth cuts
Max rough step: .05
Finish cuts: 0
Finish step: 0.05
Keep tool down
Subprogram
Absolute
Incremental
Depth cut order
By contour
By depth
Tapered walls
Taper angle: 0.0
Quick View Settings
Tool 3/4 BALL EN...
Tool Diameter 0.75
Corner Radius 0.375
Feed Rate 75
Spindle Speed 713
Coolant On
Tool Length 3.75
Length Offset 313
Diameter Off... 313
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Lead In/Out
Enter/exit at midpoint in closed contours
Gouge check
Overlap 0.0
Entry
Line
Perpendicular
Tangent
Length 100.0 % 0.75
Ramp height 0.0
Arc
Radius 100.0 % 0.75
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 75.0
Adjust start of contour
Length 0.0 % 0.0
Extend
Shorten
Exit
Line
Perpendicular
Tangent
Length 100.0 % 0.75
Ramp height 0.0
Arc
Radius 100.0 % 0.75
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 75.0
Adjust end of contour
Length 0.0 % 0.0
Extend
Shorten
Quick View Settings
Tool 3/4 BALL EN...
Tool Diameter 0.75
Corner Radius 0.375
Feed Rate 75
Spindle Speed 713
Coolant On
Tool Length 3.75
Length Offset 313
Diameter Off... 313
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled


SOLIDS MODEL PREP XFORM MACHINE TYPE TOOLPATHS SCREEN SETTINGS HELP
2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Break through
Break through amount
0.0
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled

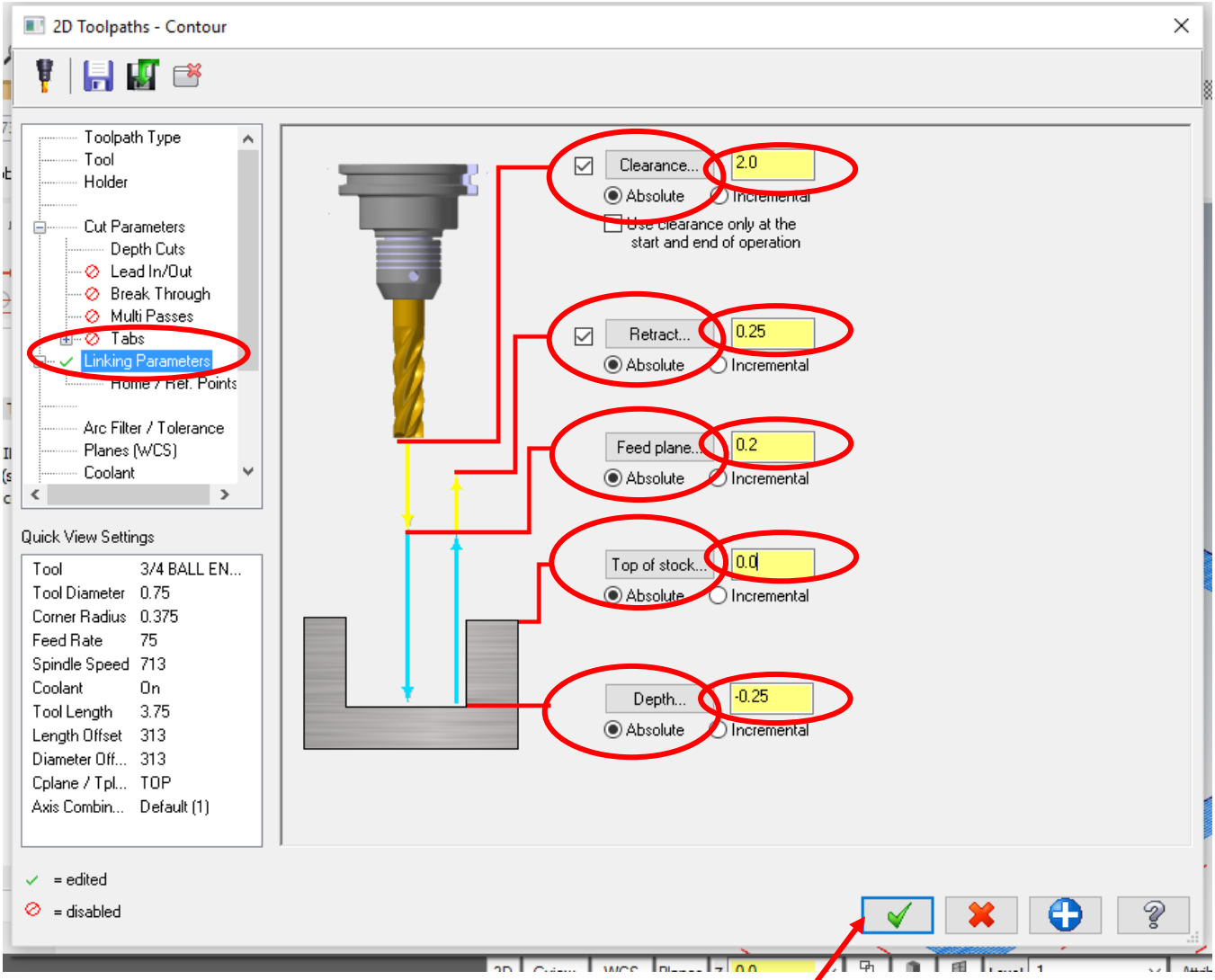

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Multi Passes
Rough
Number 1
Spacing 0.1
Finish
Number 0
Spacing 0.05
Override Feed Speed
Feed rate 100.0
Spindle speed 1069
Machine finish passes at
Final depth
All depths
Keep tool down
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled





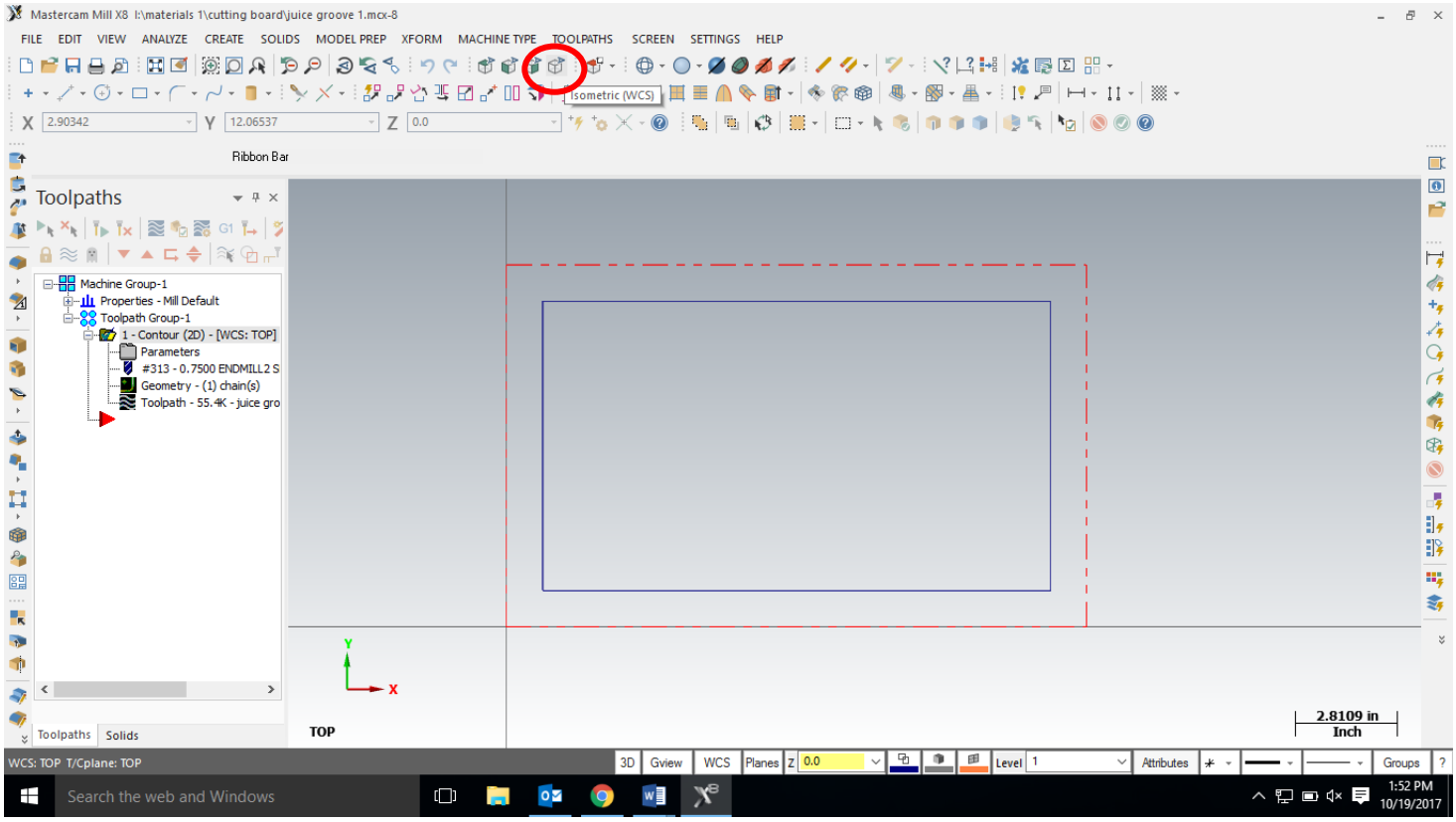
Basically, we just set up the parameters for the contour cut. We still need to set the depth of the cut. But, the only Cut Parameters that should be turned on are the depth cuts (set at .05). It should look like the picture on the left.

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 -.25**

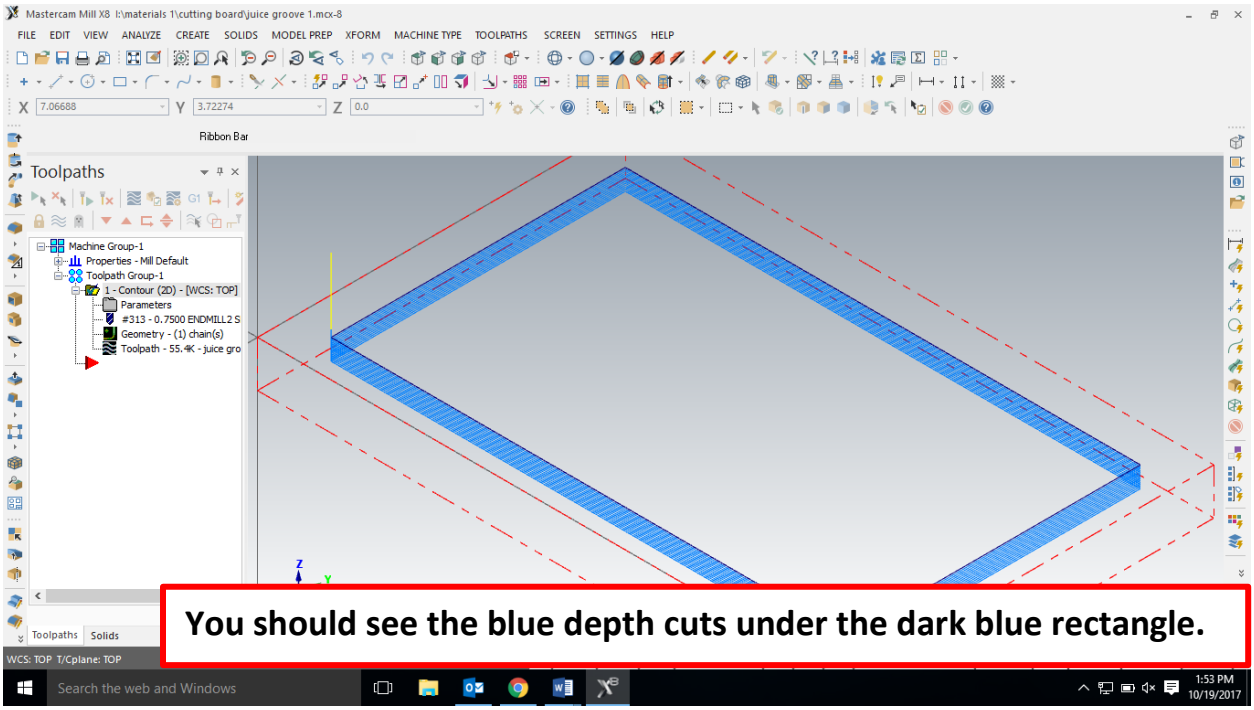


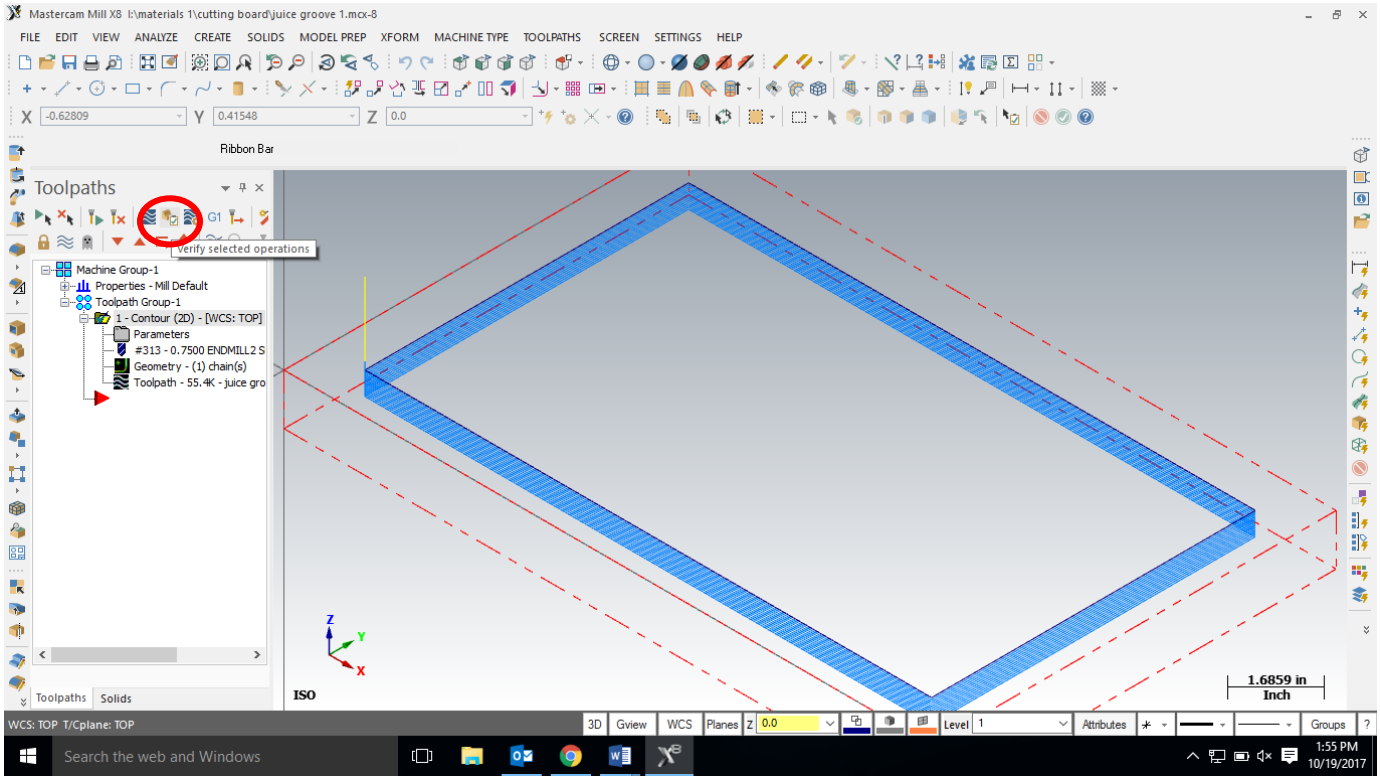
Now we can finally hit the OK check to enter all these values and MasterCAM will draw the contour toolpath on the circle.

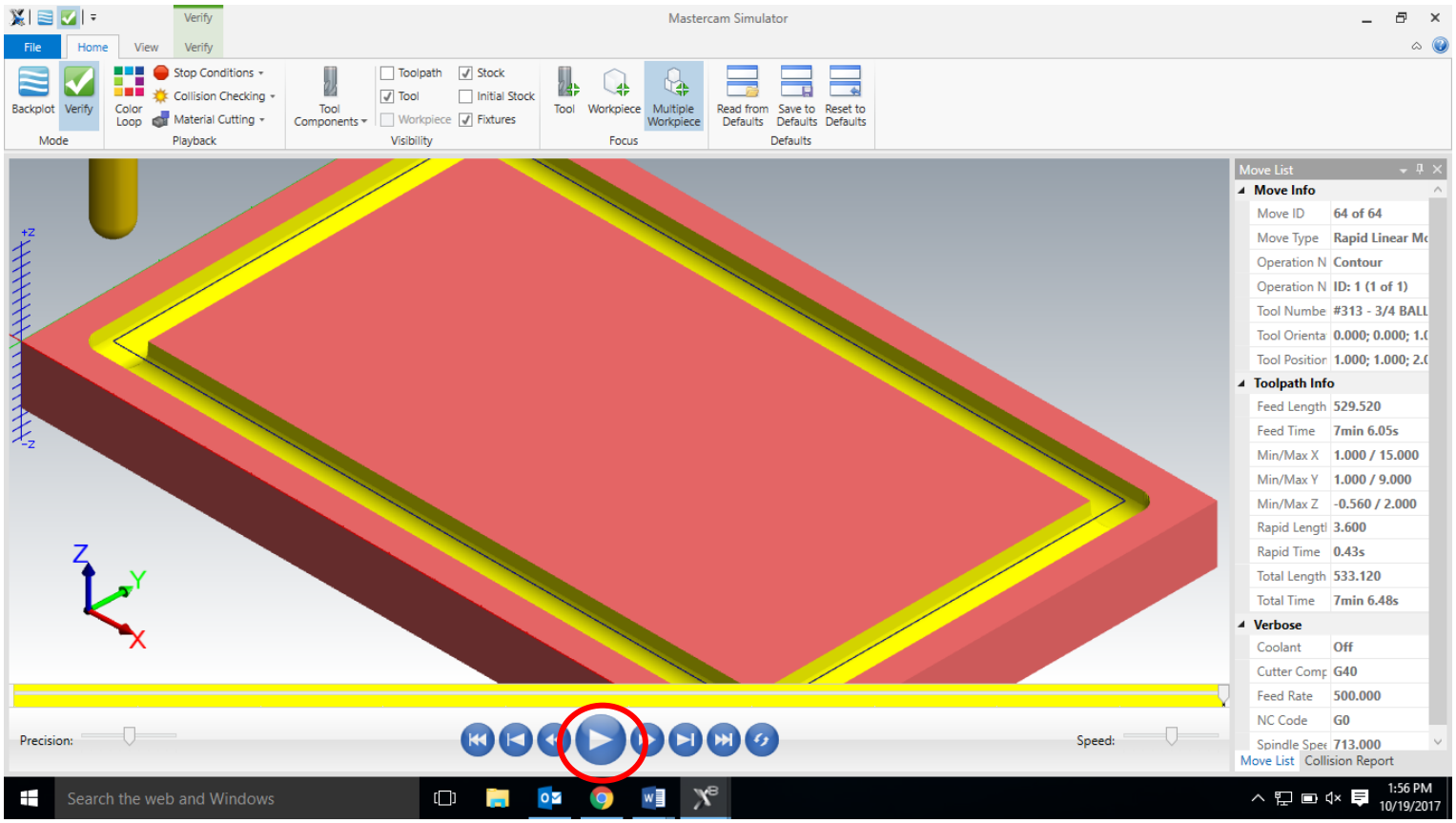
After you hit the green check, go to an isometric view, to see the toolpath.


Mastercam Mill X8 I:\materials 1\cutting board\juice groove 1.mcx-8
FILE EDIT VIEW ANALYZE CREATE SOLIDS MODEL PREP XFORM MACHINE TYPE TOOLPATHS SCREEN SETTINGS HELP
Isometric (WCS)
Ribbon Bar
Toolpaths
Machine Group-1
Properties - Mill Default
Toolpath Group-1
1 - Contour (2D) - [WCS: TOP]
Parameters
#313 - 0.7500 ENDMILL2 S
Geometry - (1) chain(s)
Toolpath - 55.4K - juice gro
TOP
2.8109 in
Inch
WCS: TOP T/Cplane: TOP
Search the web and Windows
1:52 PM
10/19/2017

Result:


Mastercam Mill X8 I:\materials 1\cutting board\juice groove 1.mcx-8
FILE EDIT VIEW ANALYZE CREATE SOLIDS MODEL PREP XFORM MACHINE TYPE TOOLPATHS SCREEN SETTINGS HELP
Ribbon Bar
Toolpaths
Machine Group-1
Properties - Mill Default
Toolpath Group-1
1 - Contour (2D) - [WCS: TOP]
Parameters
#313 - 0.7500 ENDMILL2 S
Geometry - (1) chain(s)
Toolpath - 55.4K - juice gro
You should see the blue depth cuts under the dark blue rectangle.
WCS: TOP T/Cplane: TOP
Search the web and Windows
1:53 PM
10/19/2017





If it looks good, save your work. You have completed the Cutting board juice groove.