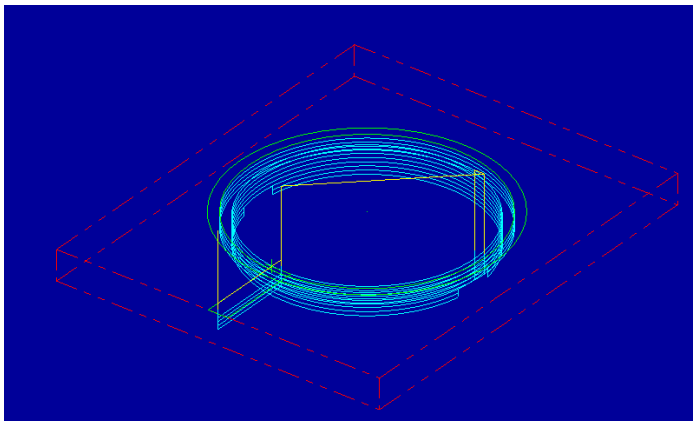
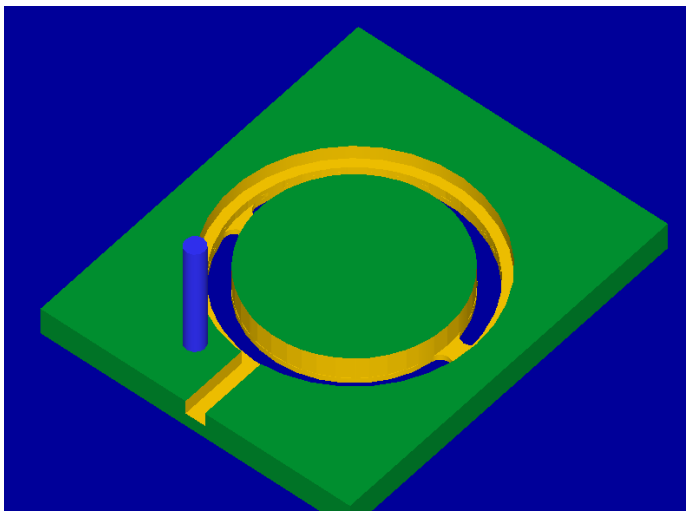
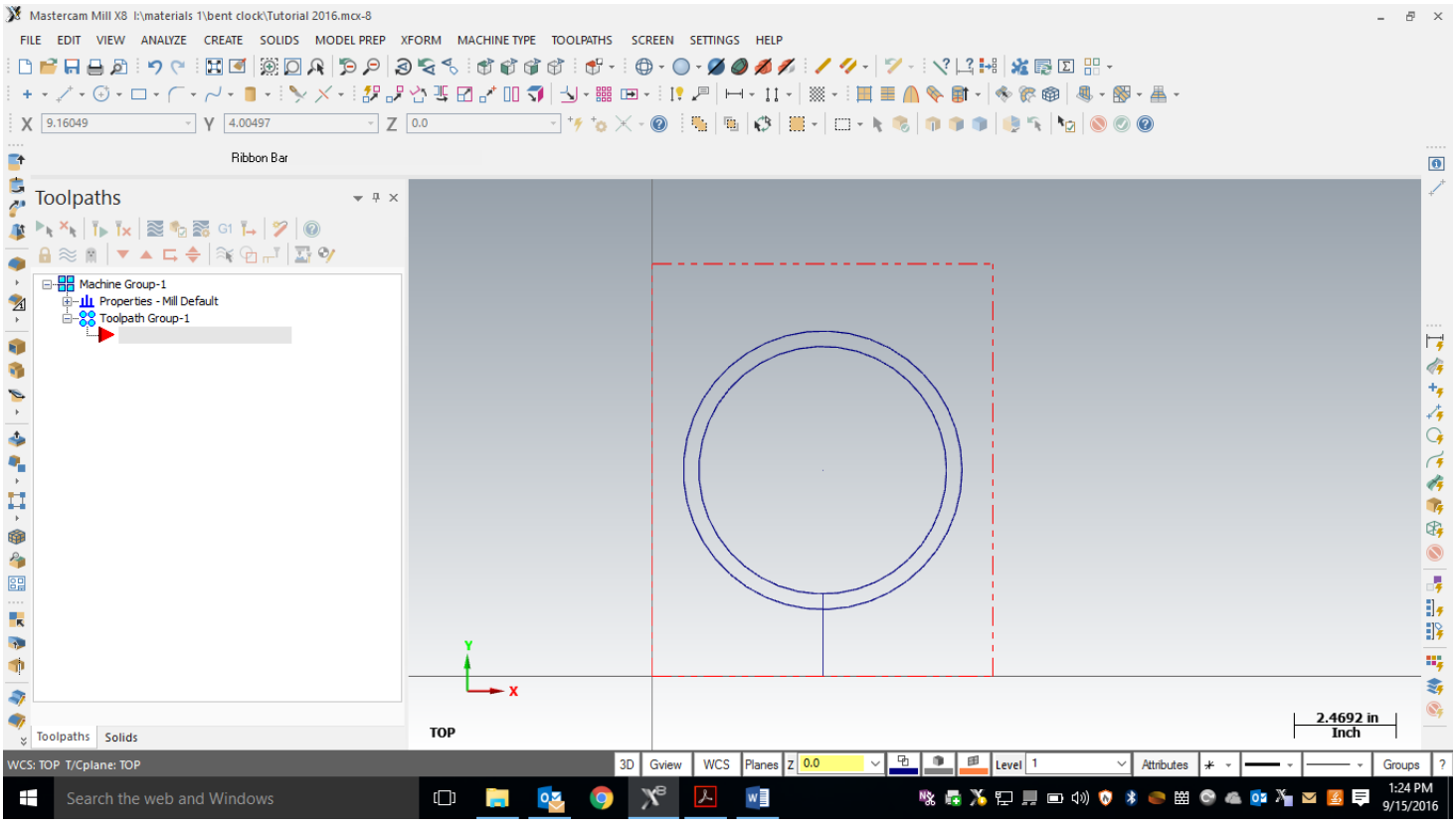
Kerf Bent Clock Front Toolpaths in MasterCAM



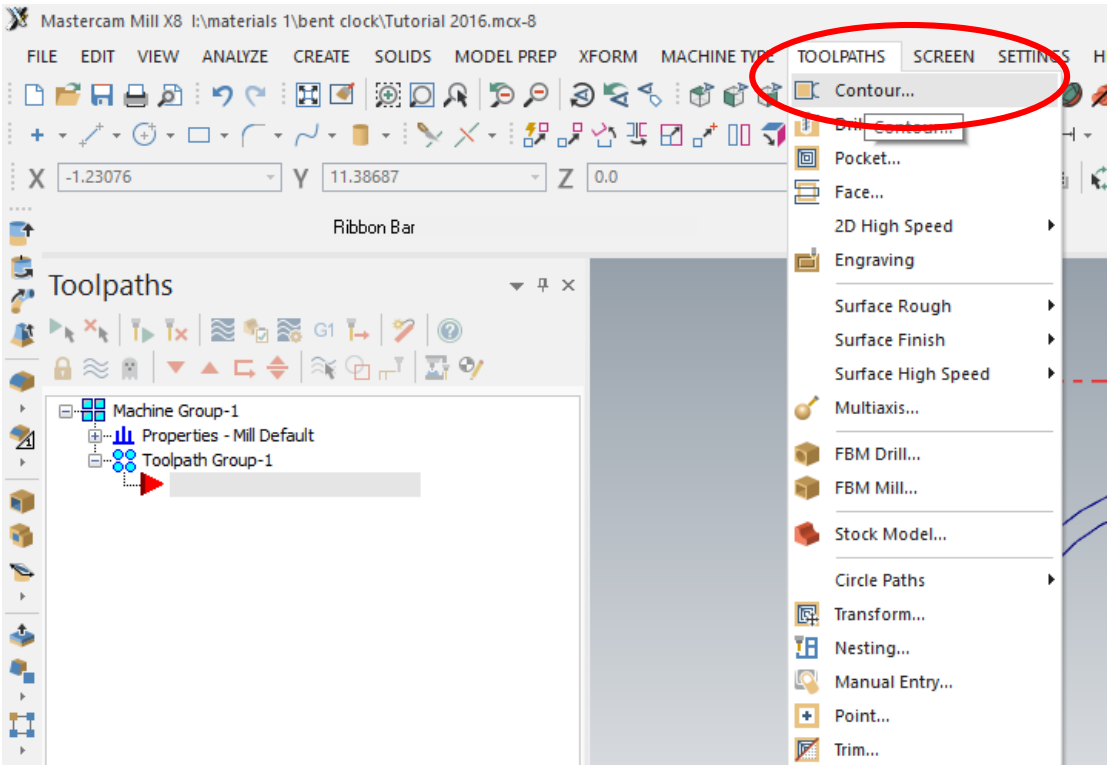


Open the MasterCAM application and open your clock front geometry file.



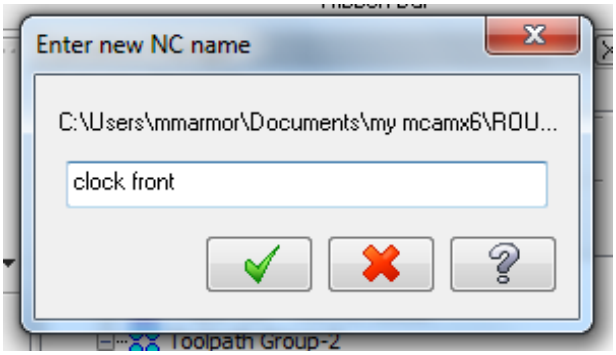
For 2D geometry such as we have, there are 2 main types of tool paths. The first one is a contour. In a contour toolpath the tool bit will follow a path. The path can be one piece of geometry or multiple pieces of geometry linked together end to end (this is called a chain). When the geometry is selected 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. For the clock front, we do not need a pocket toolpath, but you will use them when we get to the box.

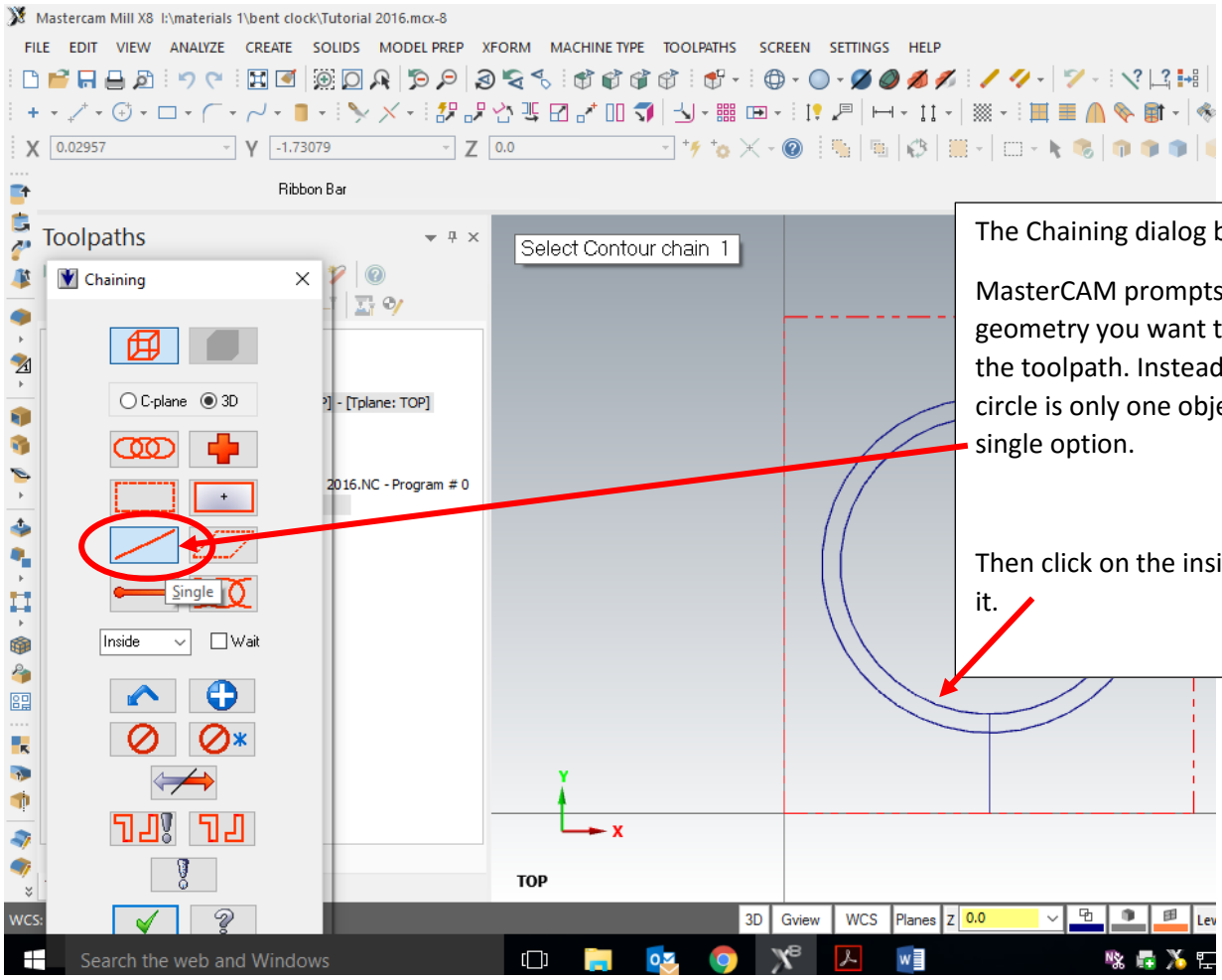
To start the toolpaths, go to Toolpaths/contour



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

Click the green check.

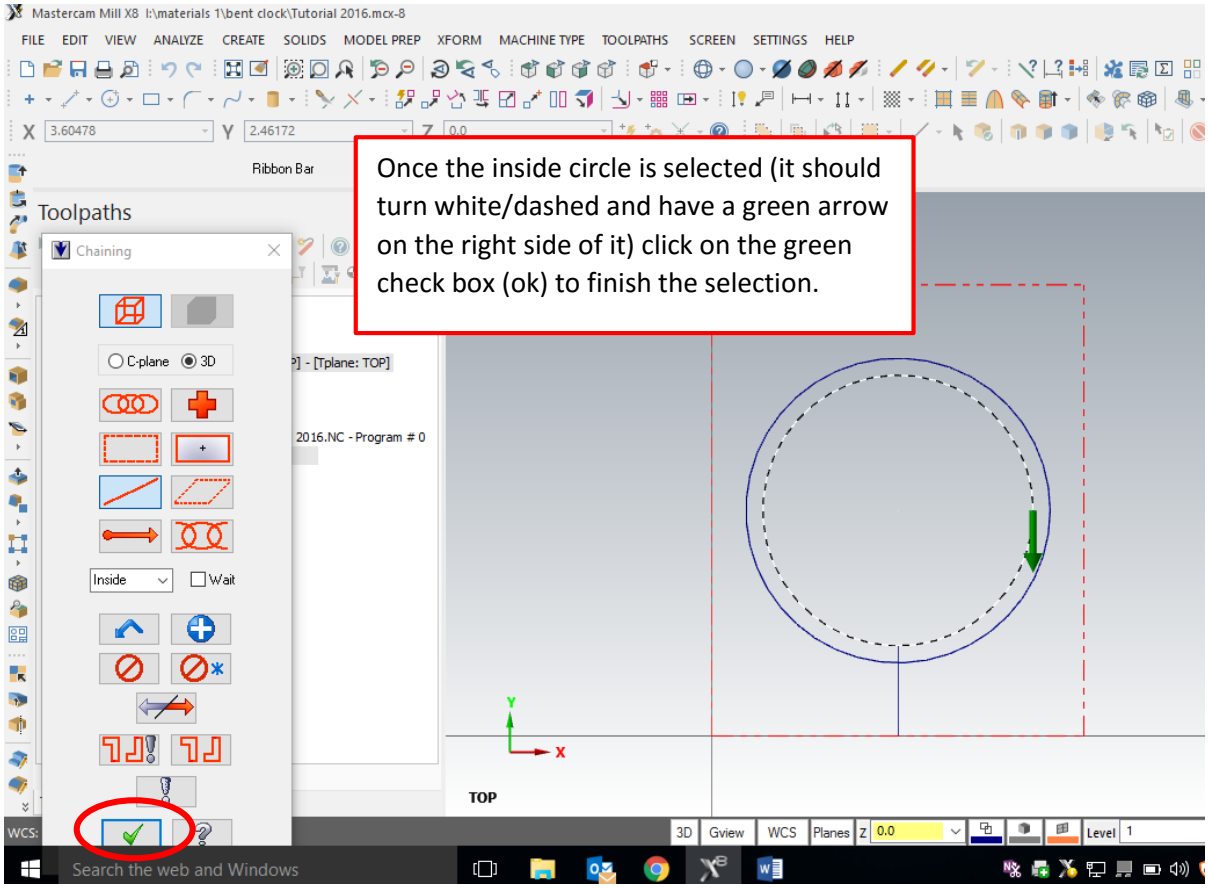




The Chaining dialog box should appear.

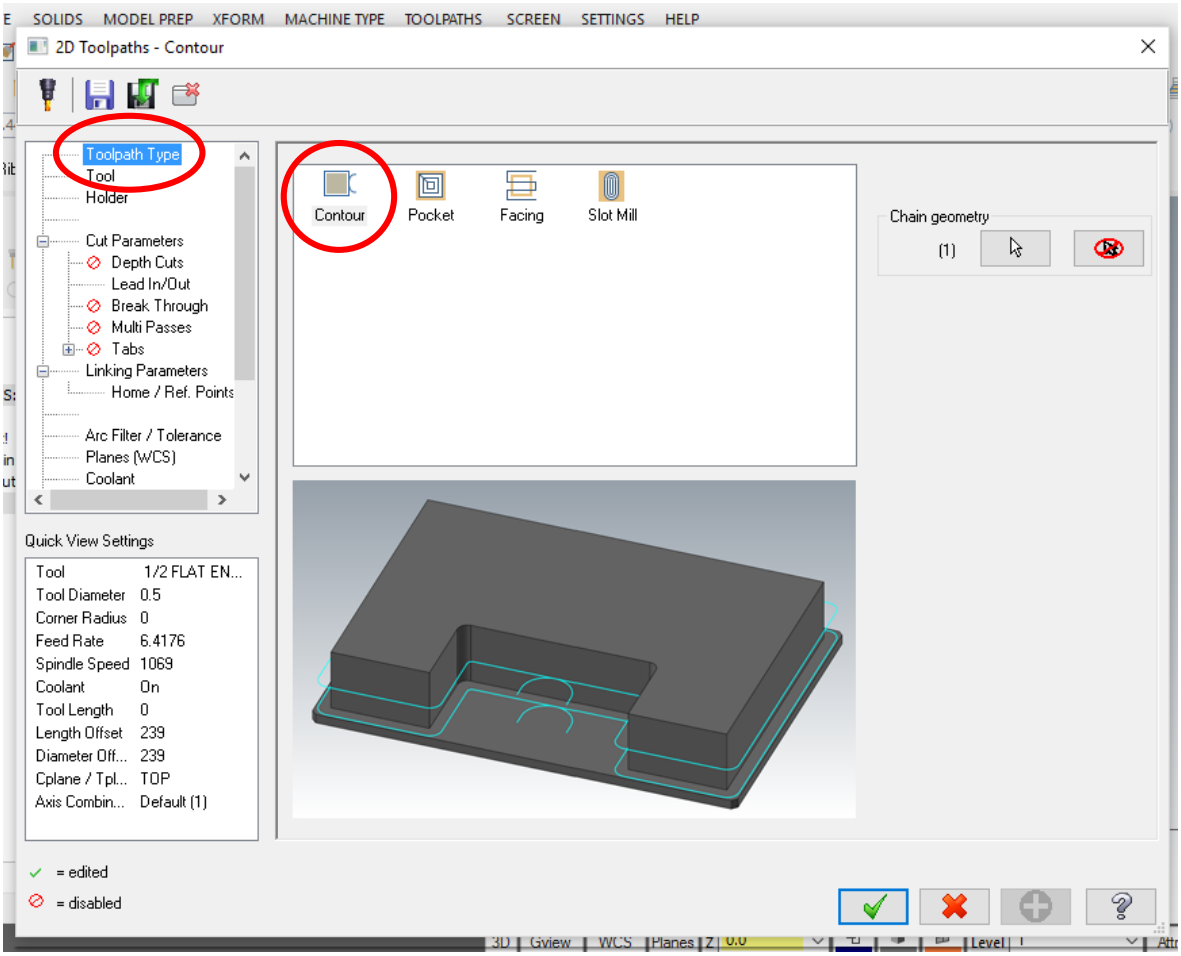
MasterCAM prompts you to select the geometry you want to associate with the toolpath. Instead of chaining, the circle is only one object, so click on the single option.

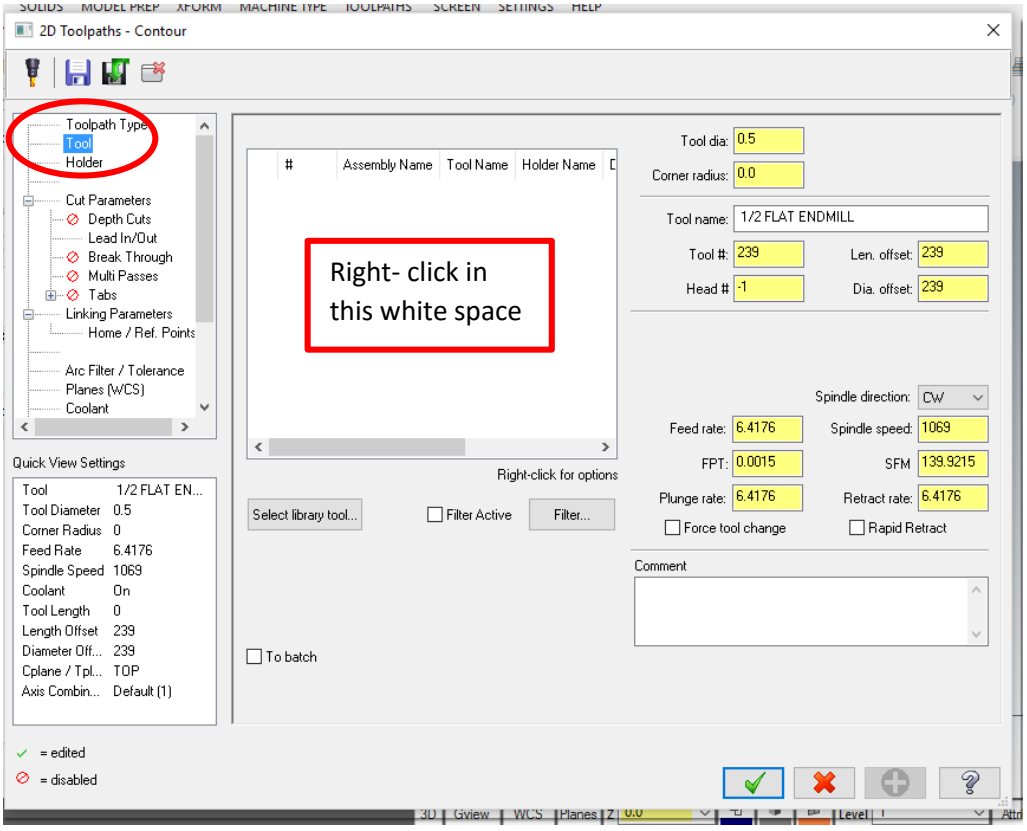
Then click on the inside circle to select it.

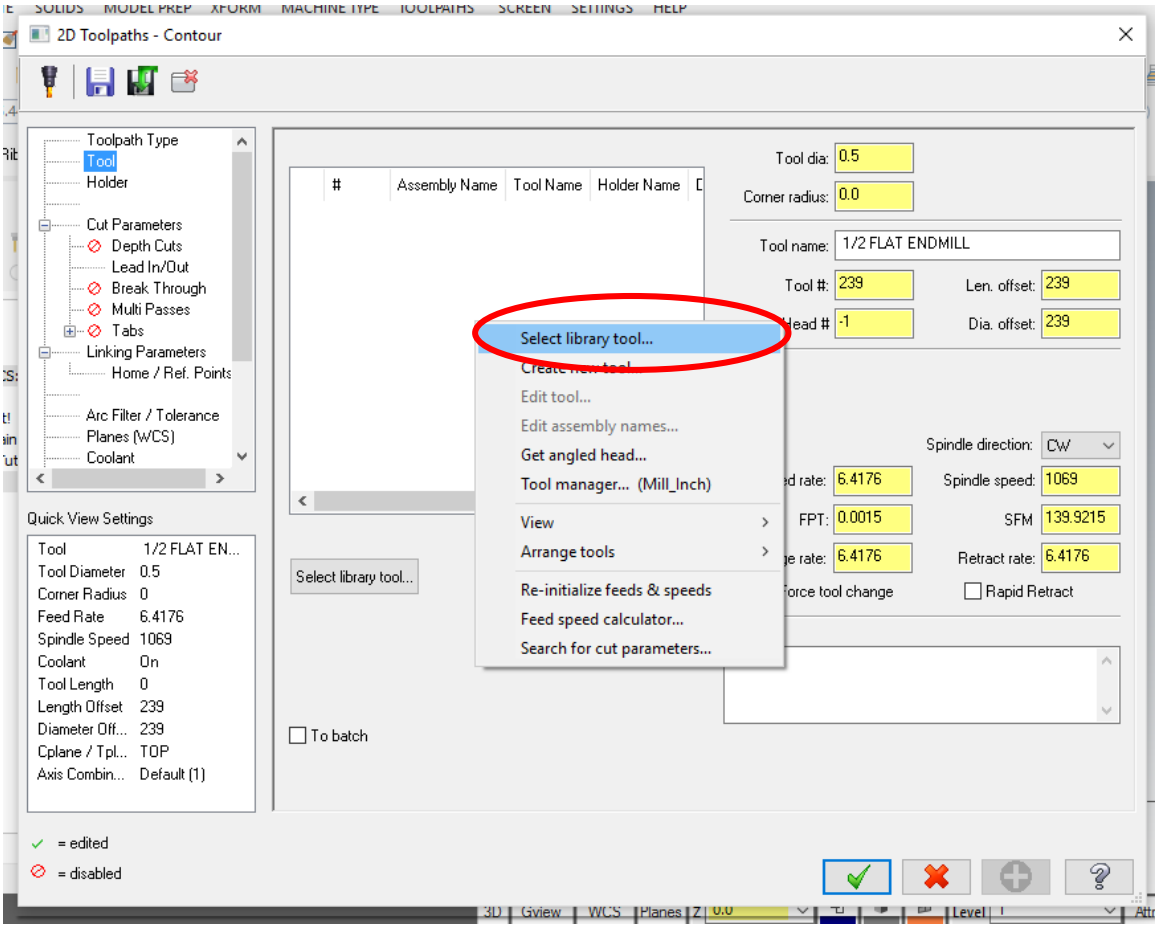


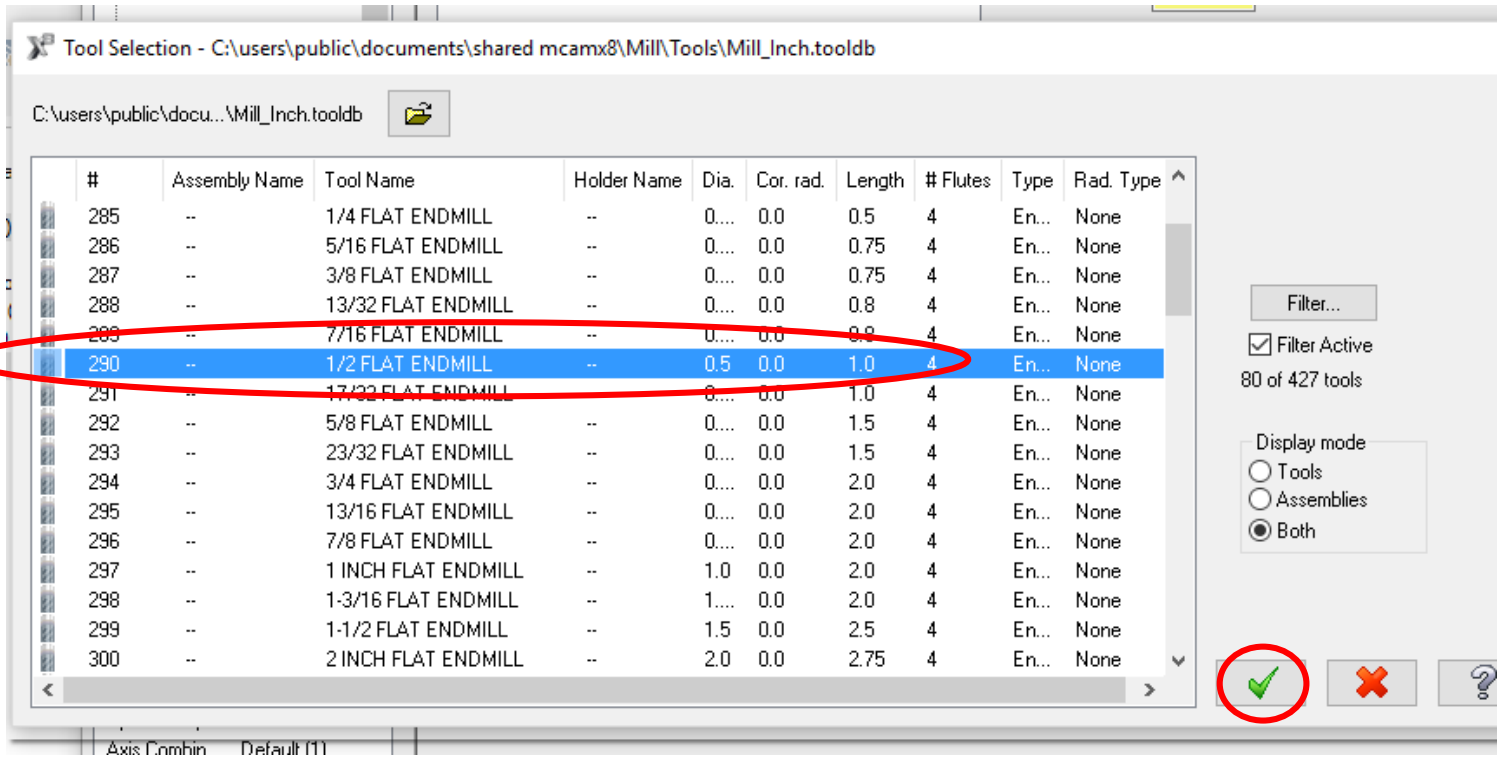
Once the inside circle is selected (it should turn white/dashed and have a green arrow on the right side of it) click on the green check box (ok) to finish the selection.

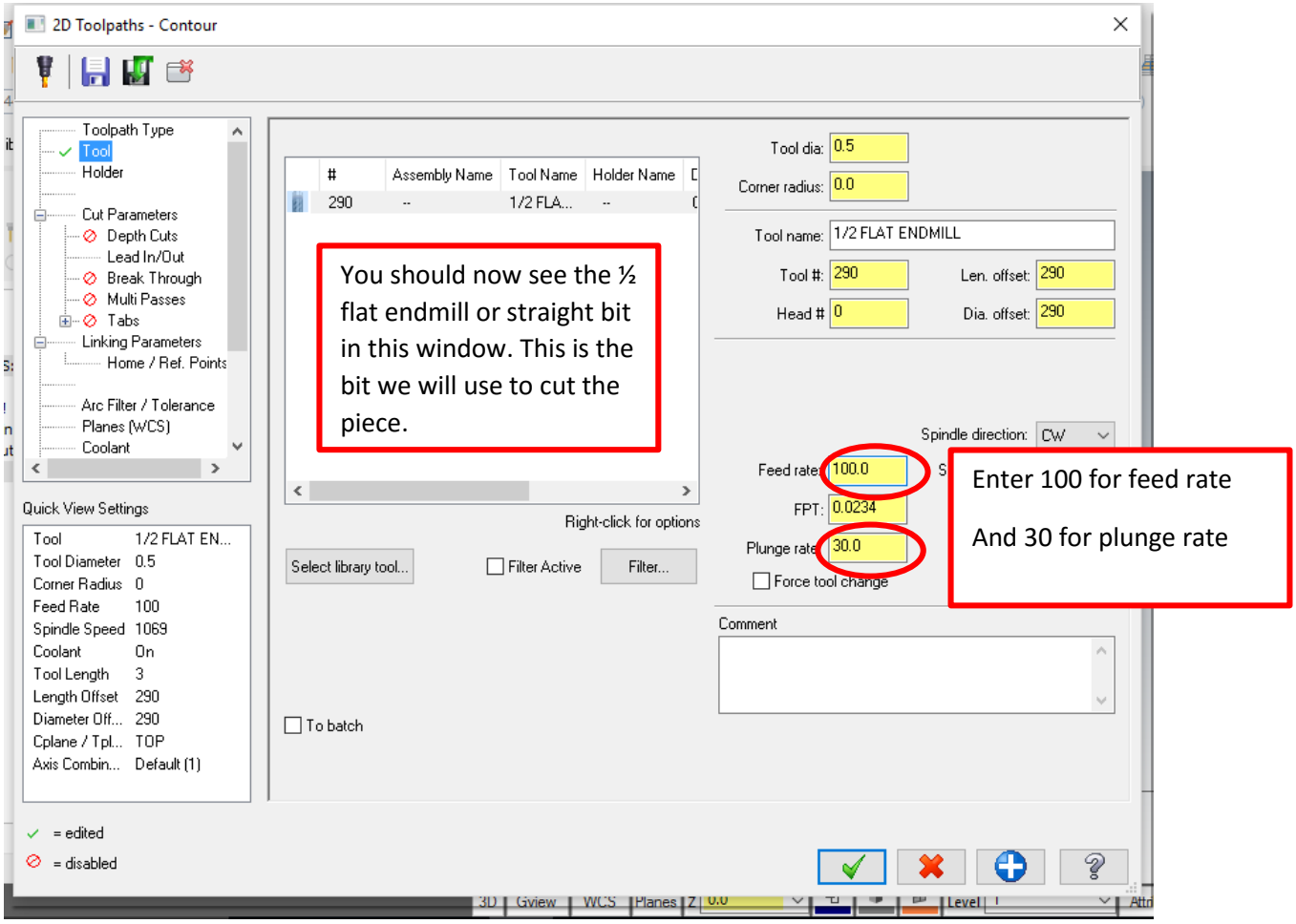
In the 2D Toolpaths – Contour dialog box, please enter the following information:

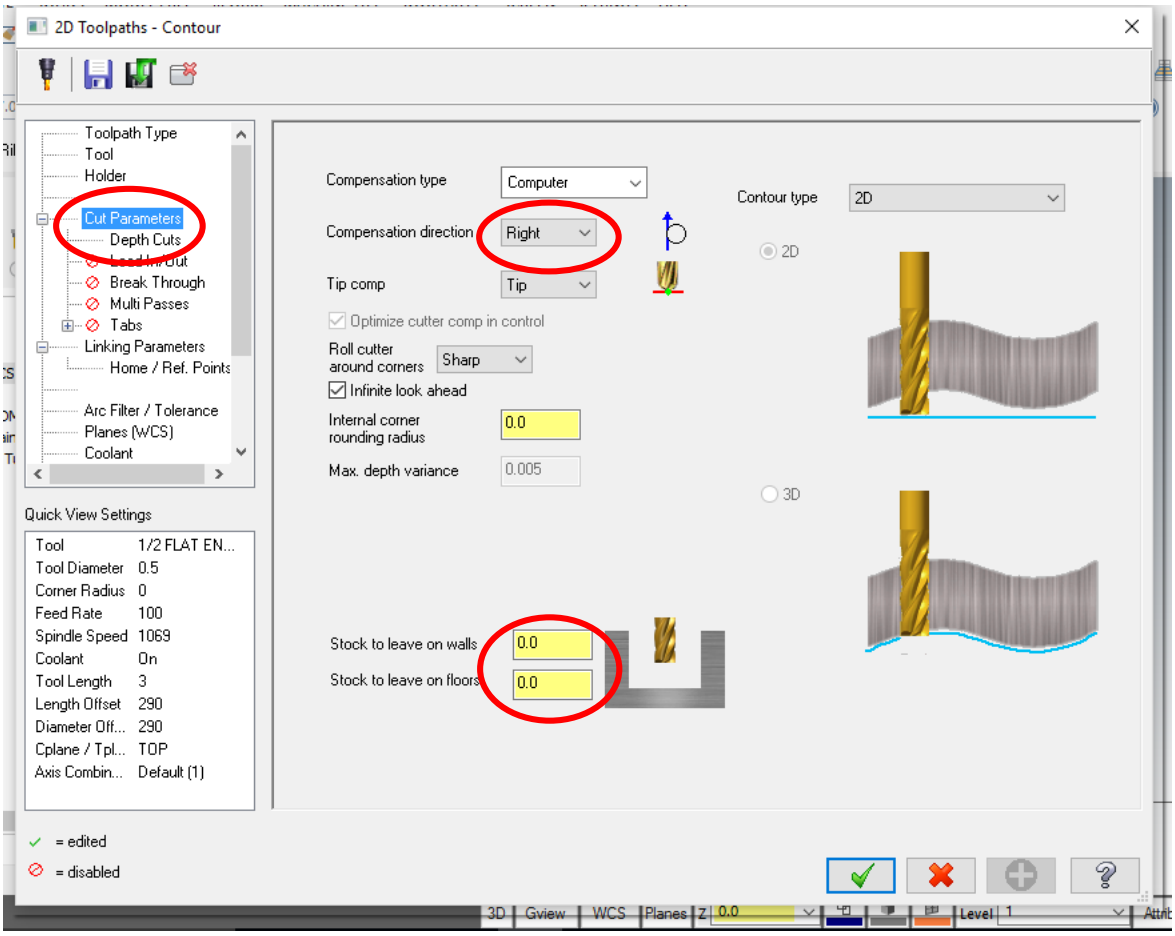


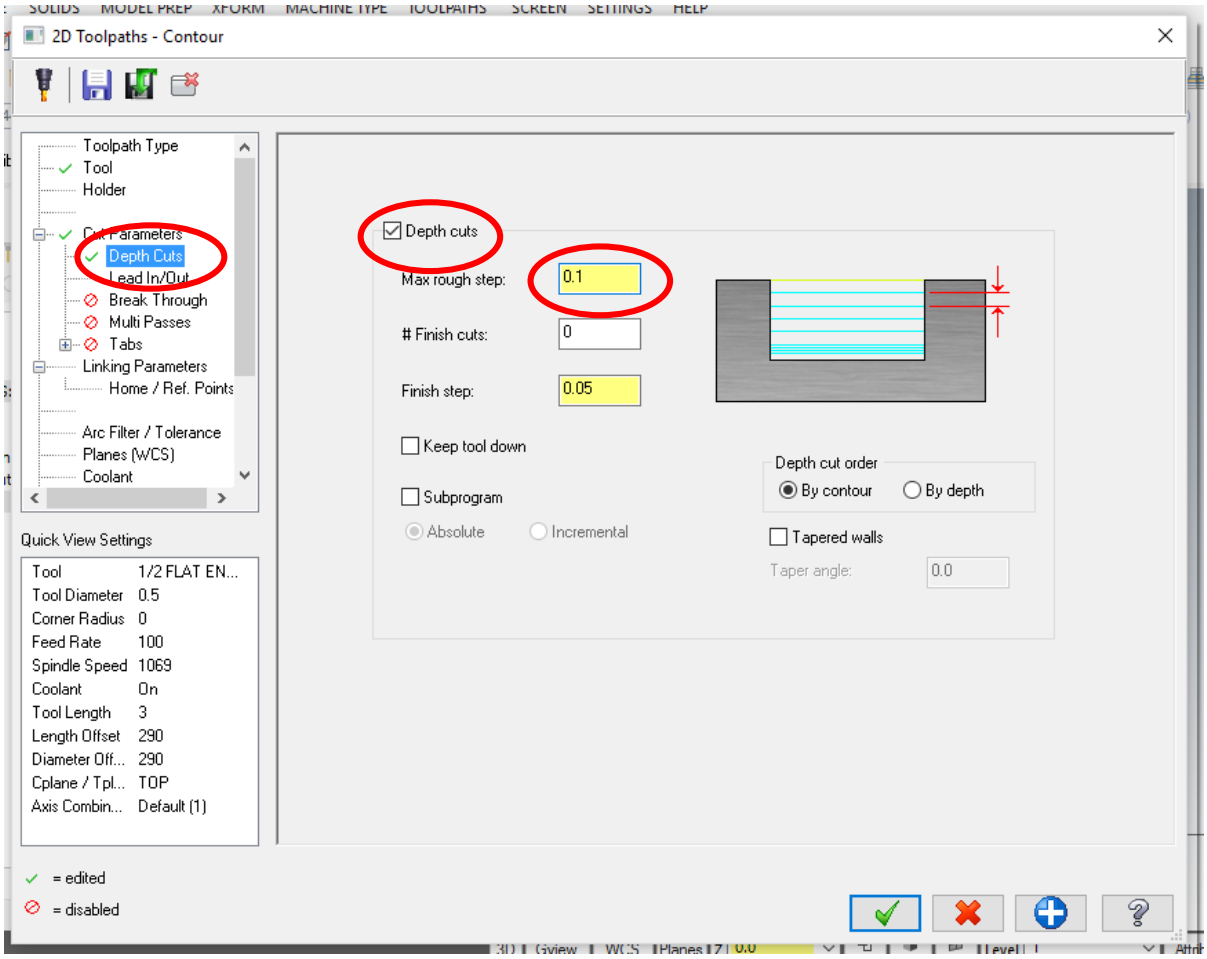


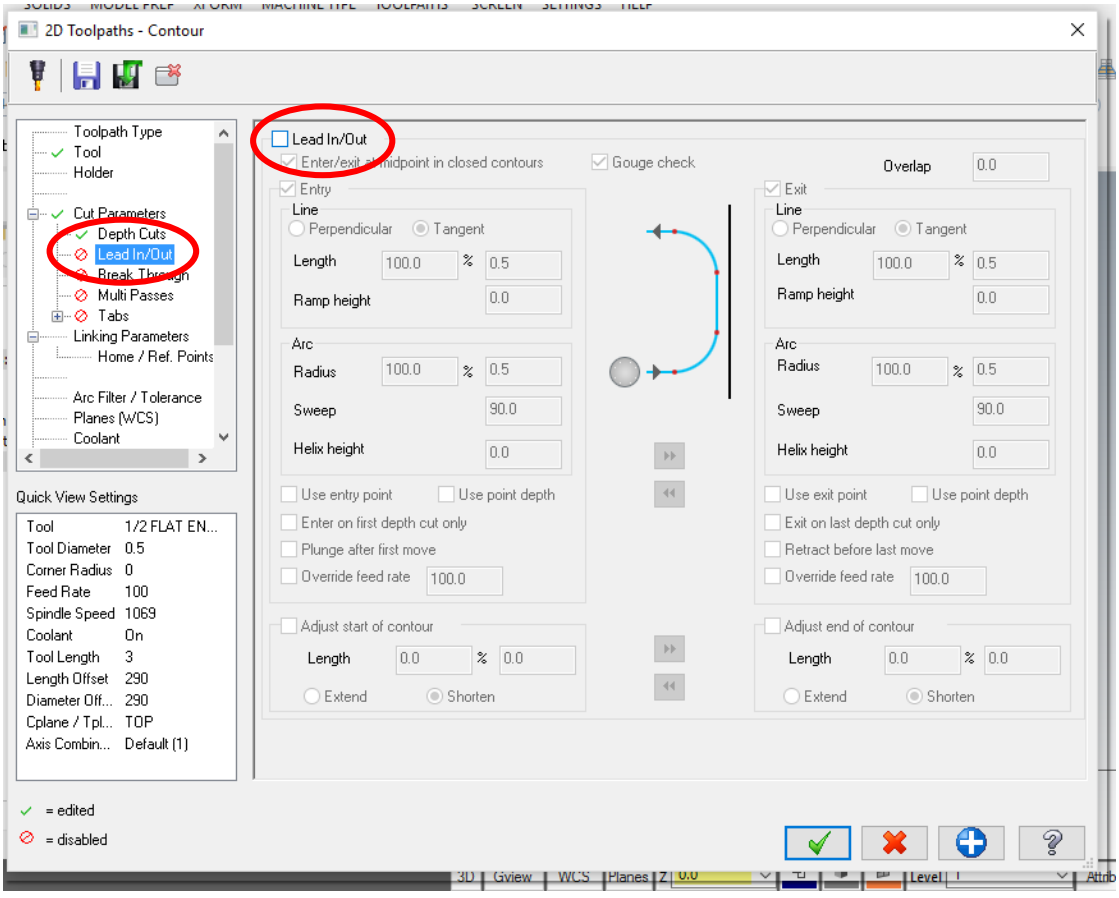

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 6.4176
Spindle Speed 1069
Coolant On
Tool Length 0
Length Offset 239
Diameter Off... 239
Cplane / Tpl... TOP
Axis Combin... Default (1)
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
Dia. offset: 239
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...
= edited
= disabled

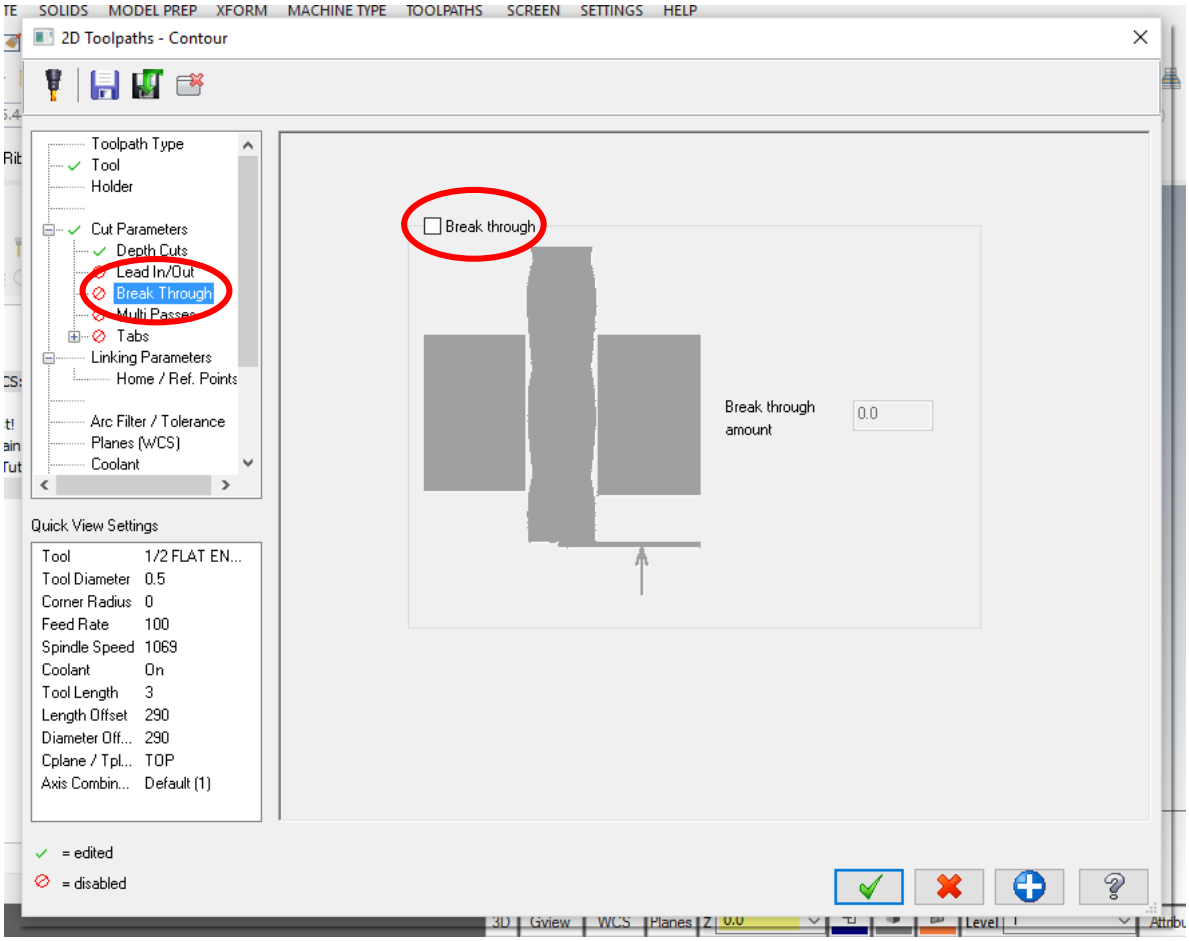

Tool Selection - C:\users\public\documents\shared mcamx8\Mill\Tools\Mill_Inch.tooldb
C:\users\public\docu...\Mill_Inch.tooldb
Assembly Name Tool Name Holder Name Dia. Cor. rad. Length # Flutes Type Rad. Type
285 -- 1/4 FLAT ENDMILL -- 0.... 0.0 0.5 4 En... None
286 -- 5/16 FLAT ENDMILL -- 0.... 0.0 0.75 4 En... None
287 -- 3/8 FLAT ENDMILL -- 0.... 0.0 0.75 4 En... None
288 -- 13/32 FLAT ENDMILL -- 0.... 0.0 0.8 4 En... None
289 -- 7/16 FLAT ENDMILL -- 0.... 0.0 0.8 4 En... None
290 -- 1/2 FLAT ENDMILL -- 0.5 0.0 1.0 4 En... None
291 -- 17/32 FLAT ENDMILL -- 0.... 0.0 1.0 4 En... None
292 -- 5/8 FLAT ENDMILL -- 0.... 0.0 1.5 4 En... None
293 -- 23/32 FLAT ENDMILL -- 0.... 0.0 1.5 4 En... None
294 -- 3/4 FLAT ENDMILL -- 0.... 0.0 2.0 4 En... None
295 -- 13/16 FLAT ENDMILL -- 0.... 0.0 2.0 4 En... None
296 -- 7/8 FLAT ENDMILL -- 0.... 0.0 2.0 4 En... None
297 -- 1 INCH FLAT ENDMILL -- 1.0 0.0 2.0 4 En... None
298 -- 1-3/16 FLAT ENDMILL -- 1.... 0.0 2.0 4 En... None
299 -- 1-1/2 FLAT ENDMILL -- 1.5 0.0 2.5 4 En... None
300 -- 2 INCH FLAT ENDMILL -- 2.0 0.0 2.75 4 En... None
Filter...
Filter Active
80 of 427 tools
Display mode
Tools
Assemblies
Both

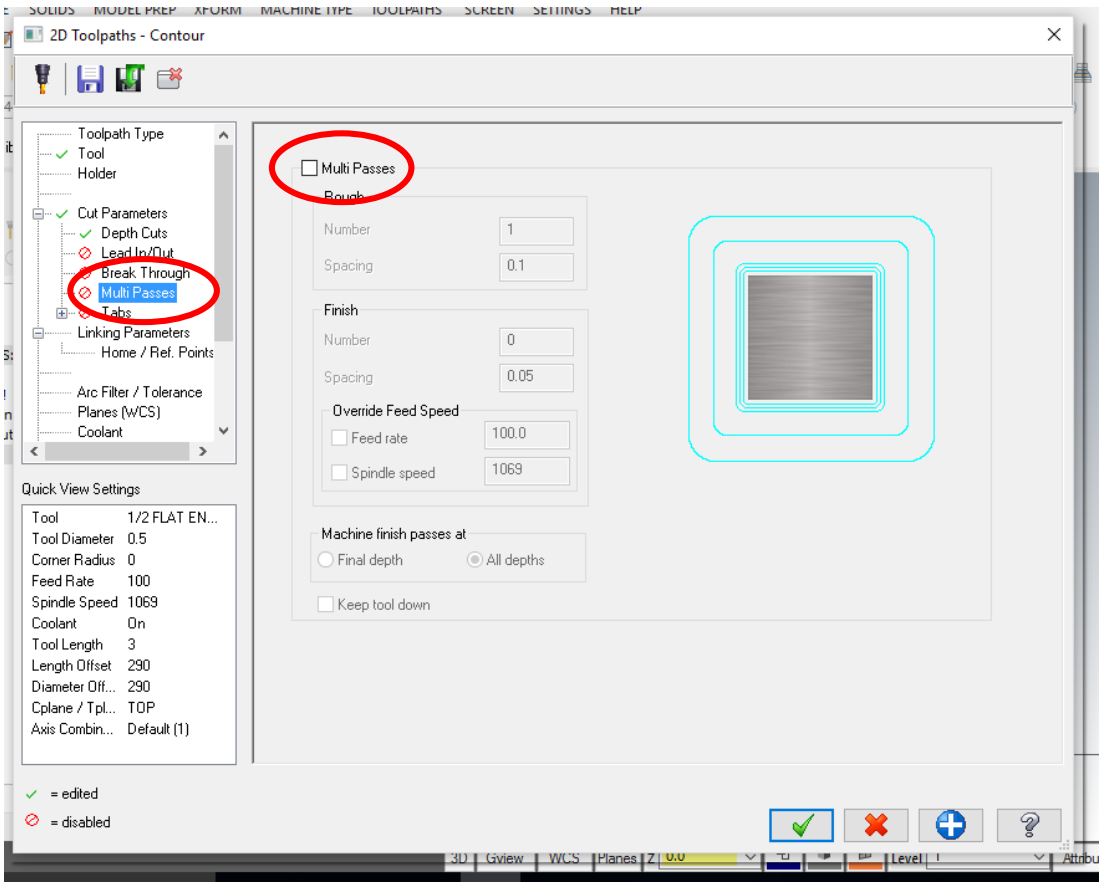

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
Assembly Name Tool Name Holder Name
290 -- 1/2 FLA... --
You should now see the ½ flat endmill or straight 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: 0.5
Corner radius: 0.0
Tool name: 1/2 FLAT ENDMILL
Tool #: 290
Len. offset: 290
Head #: 0
Dia. offset: 290
Spindle direction: CW
Feed rate: 100.0
FPT: 0.0234
Plunge rate: 30.0
Force tool change
Enter 100 for feed rate
And 30 for plunge rate
Comment
= edited
= disabled

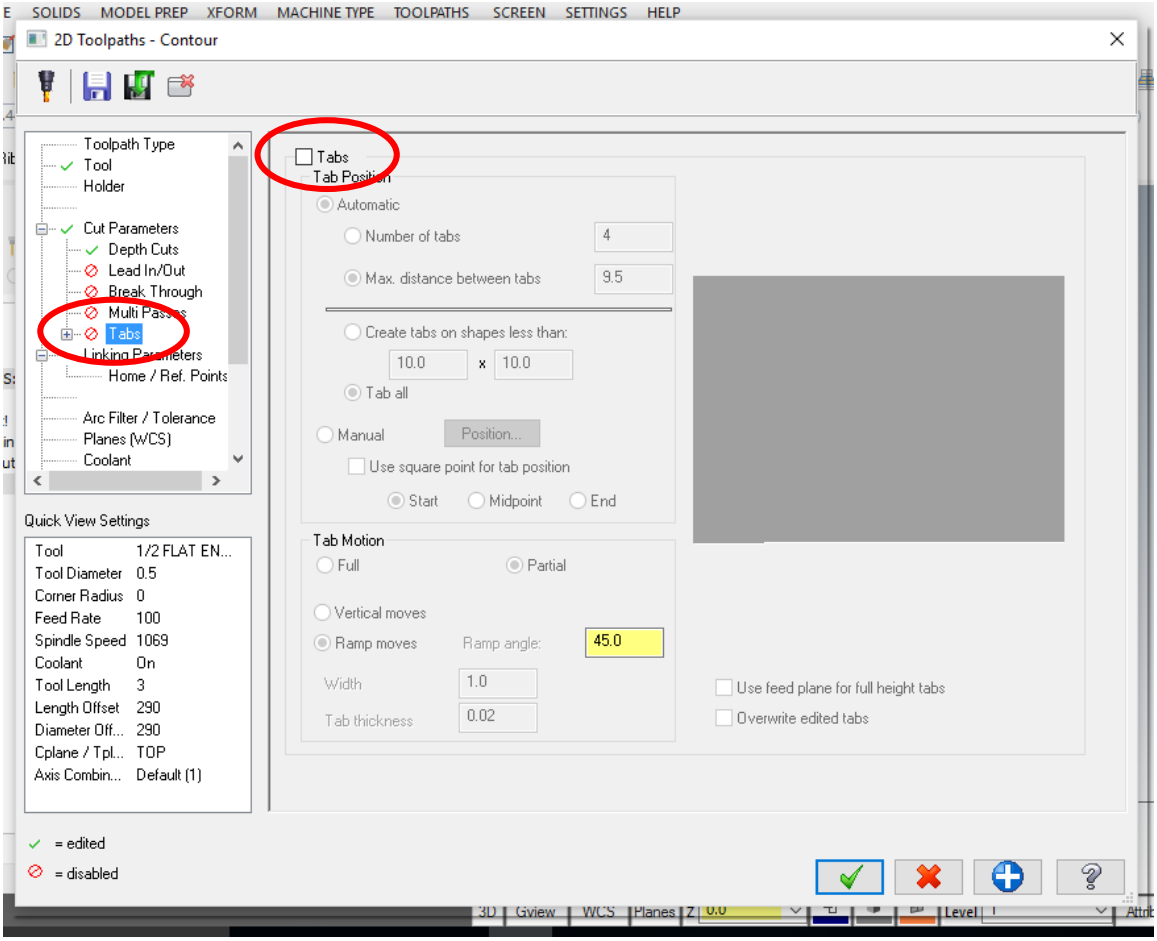

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
Compensation type Computer
Compensation direction Right
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

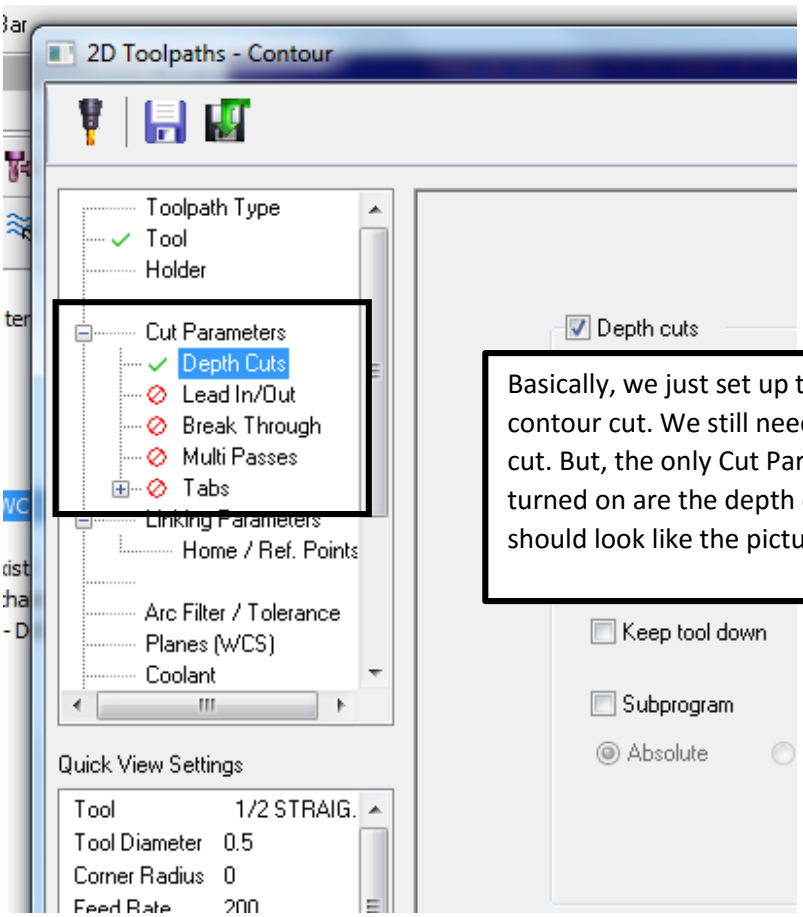

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Depth cuts
Max rough step: 0.1
Finish cuts: 0
Finish step: 0.05
Keep tool down
Subprogram
Absolute
Incremental
Depth cut order
By contour
By depth
Tapered walls
Taper angle: 0.0
Quick View Settings
Tool 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
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 % 0.5
Ramp height 0.0
Arc
Radius 100.0 % 0.5
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
Adjust start of contour
Length 0.0 % 0.0
Extend
Shorten
Exit
Line
Perpendicular
Tangent
Length 100.0 % 0.5
Ramp height 0.0
Arc
Radius 100.0 % 0.5
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
Adjust end of contour
Length 0.0 % 0.0
Extend
Shorten
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


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

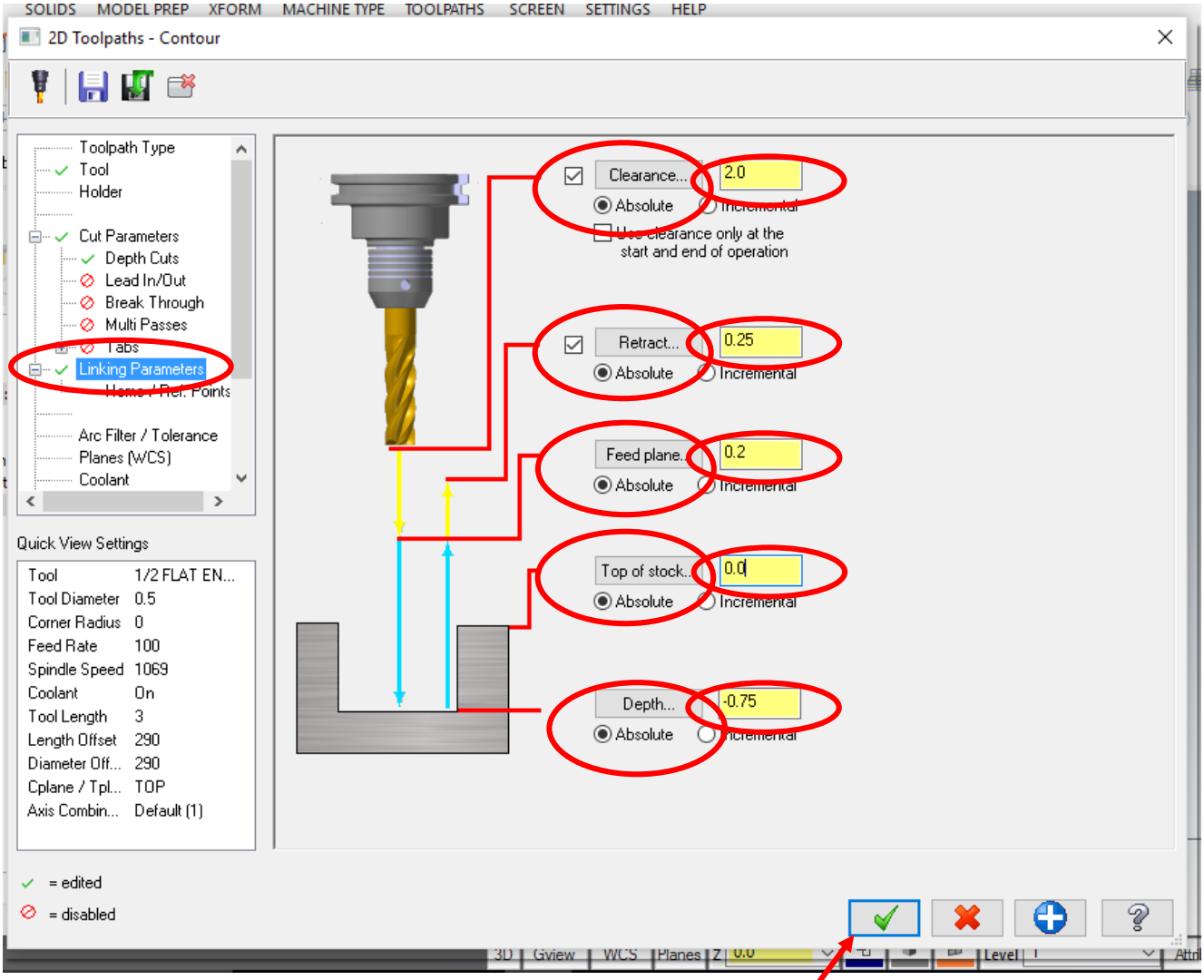

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Multi Passes
Number 1
Spacing 0.1
Finish
Number 0
Spacing 0.05
Override Feed Speed
Feed rate 100.0
Spindle 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





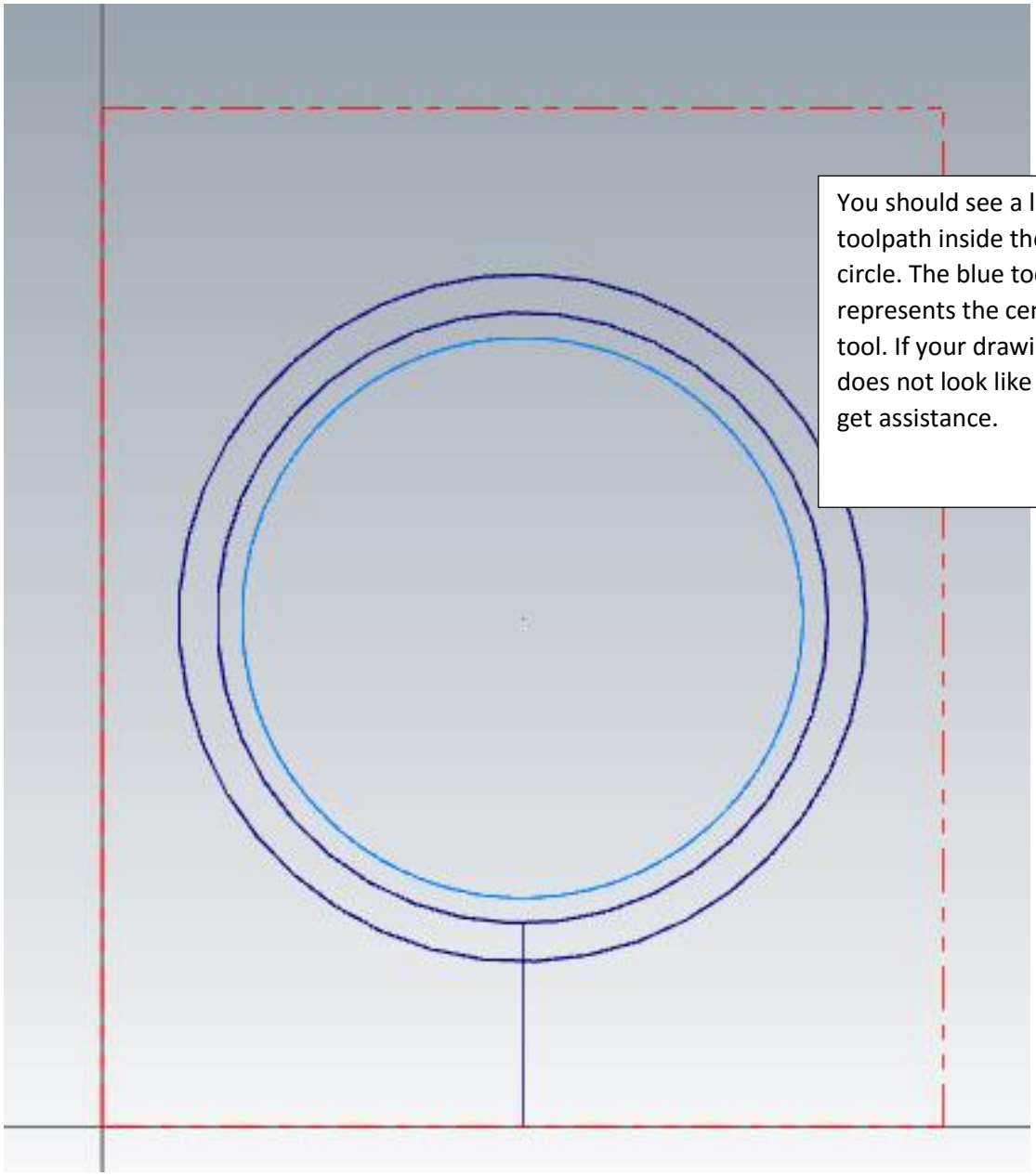
Basically, we just set up the parameters for the contour cut. We still need to set the depth of the cut. But, the only Cut Parameters that should be turned on are the depth cuts (set at .125). It should look like the picture on the left.

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 -.75.**



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

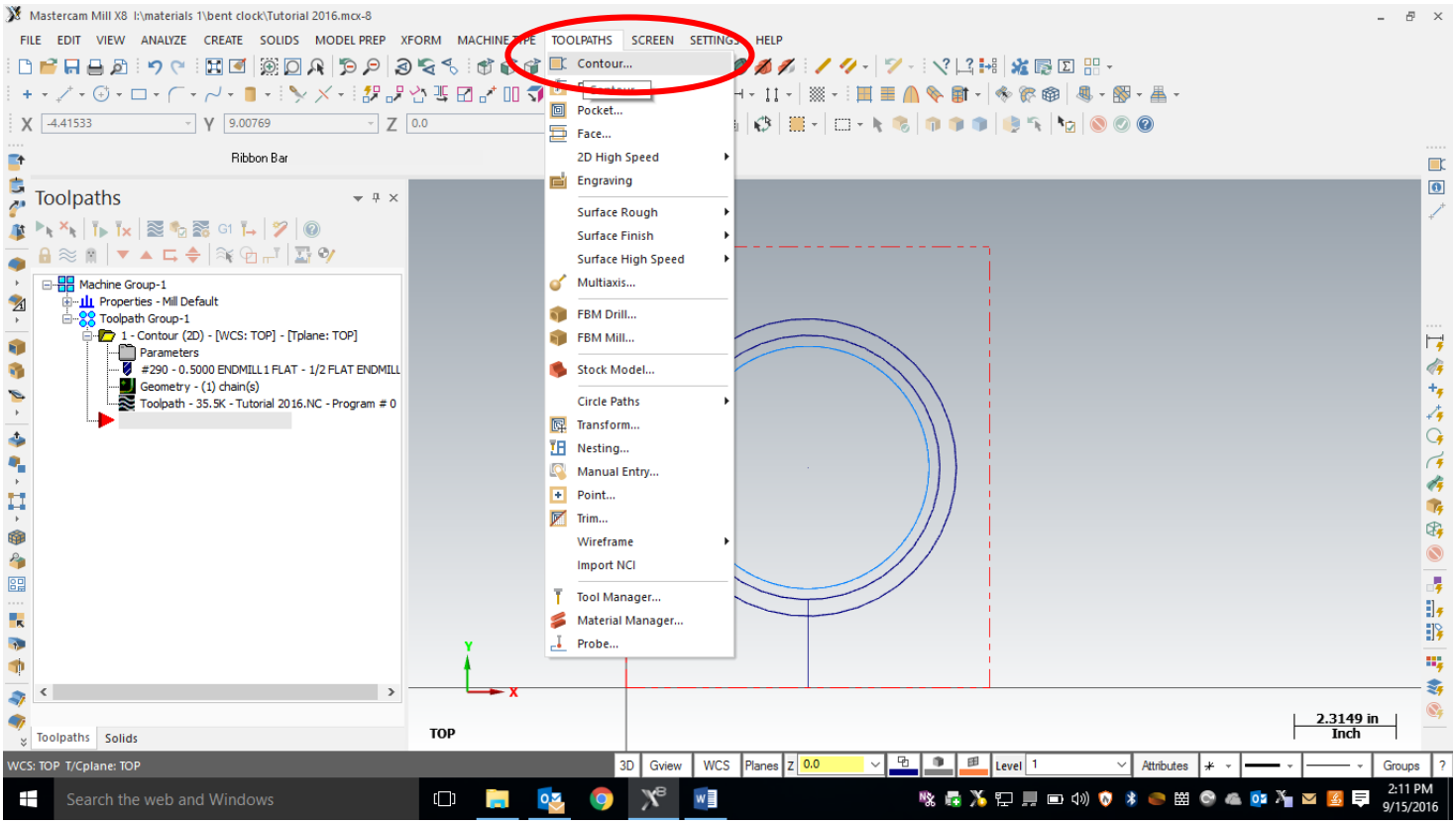
Result:



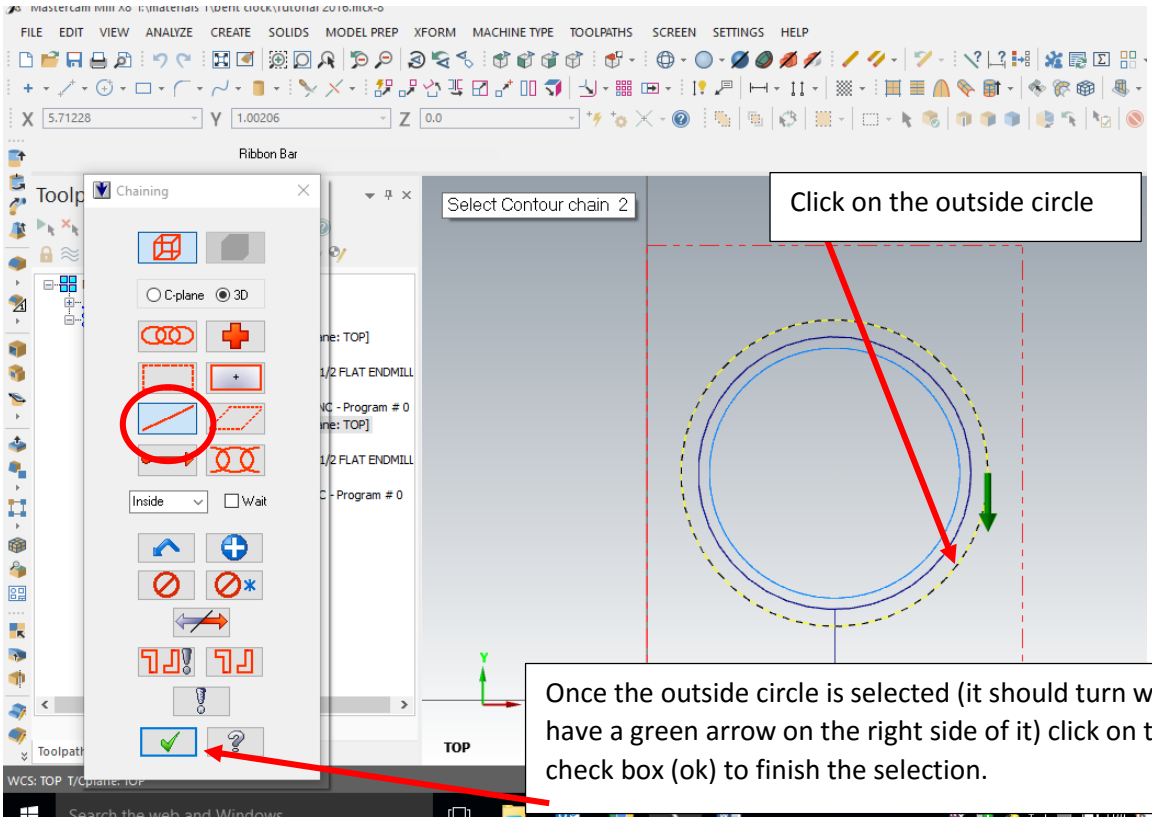
You should see a light blue toolpath inside the inside circle. The blue toolpath represents the center of the tool. If your drawing/toolpath does not look like this, please get assistance.

The second toolpath is similar to the first. It will be a contour with the same cut parameters and same tool, but we will not go all the way through the piece. This time we will make a rabbit, so the depth will be -3/8” or -.375.

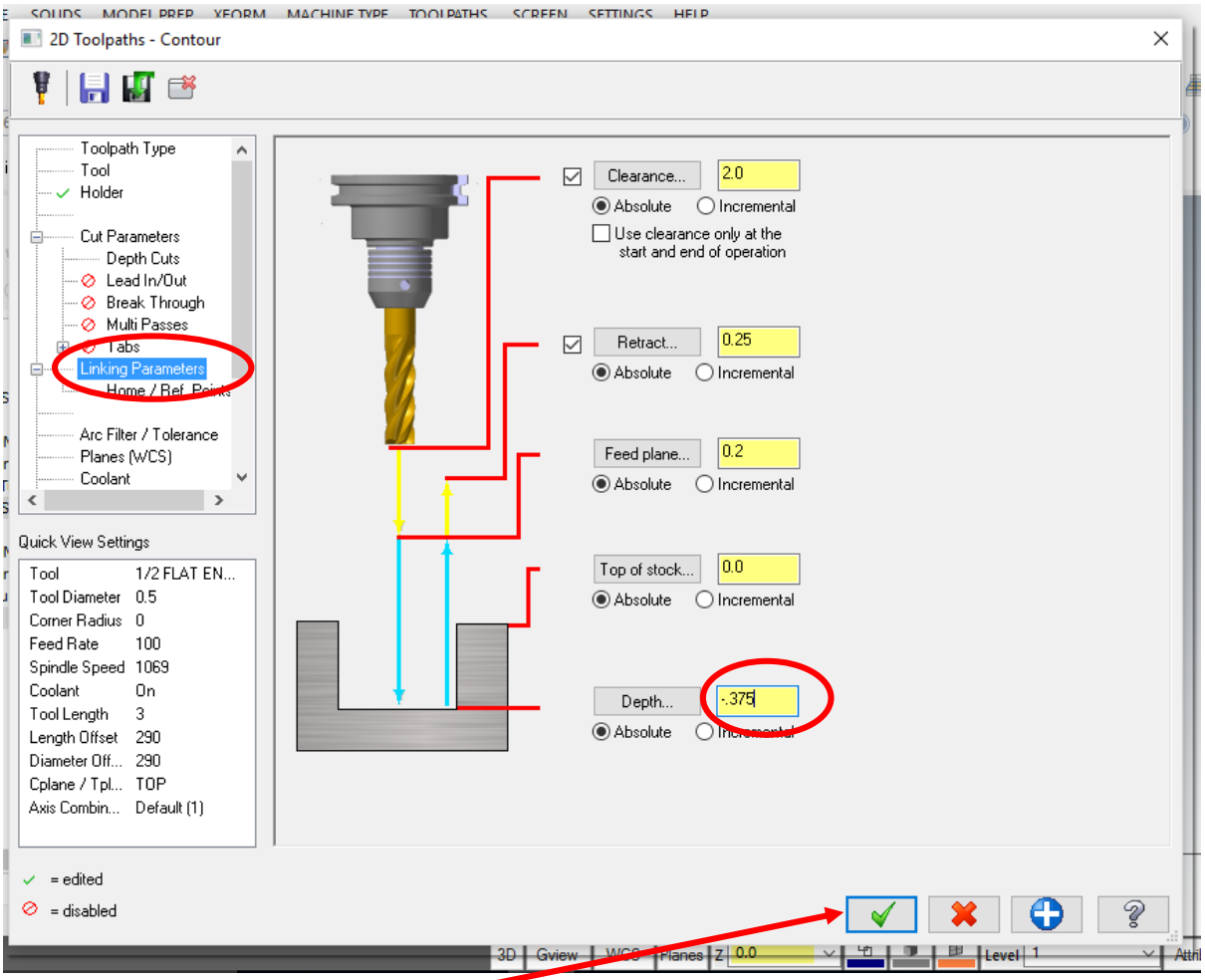
Go to toolpaths/contour.



Again, go to single, and click on the outside circle this time.

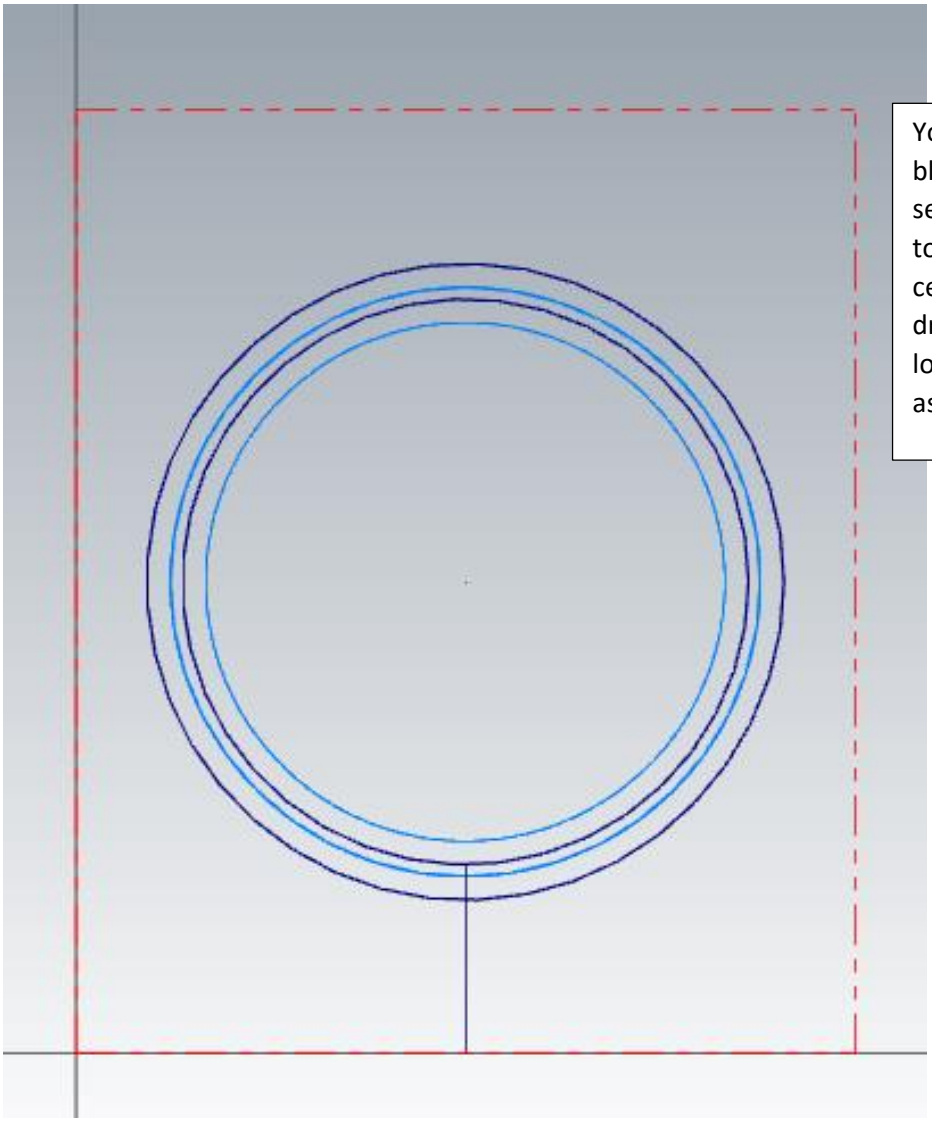


After you hit the OK button for selecting geometry for the toolpath, the 2D contour toolpath window pops up. This should have all the same settings we set for the first toolpath except for the depth. So you can check all the cut parameters and see if they are the same (they should be, if not just enter the same information for the last contour). Once you check the cut parameters, go to linking parameters and enter the **depth of -.375.**



Then hit OK.

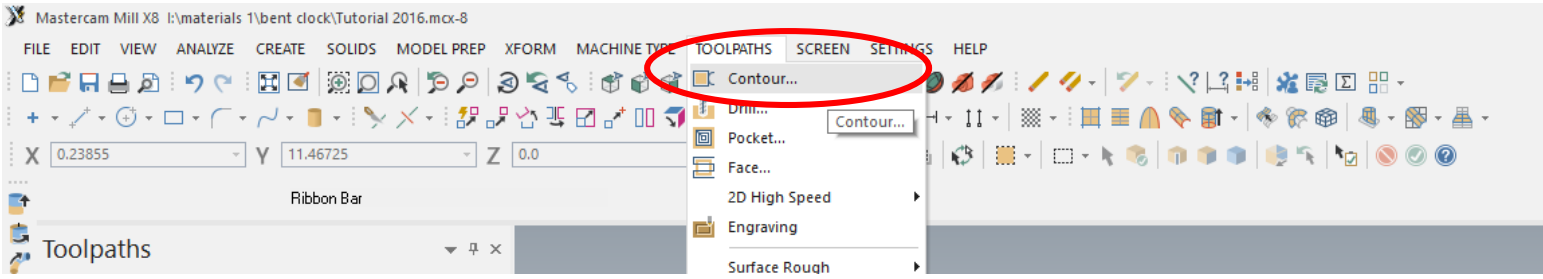
Result:

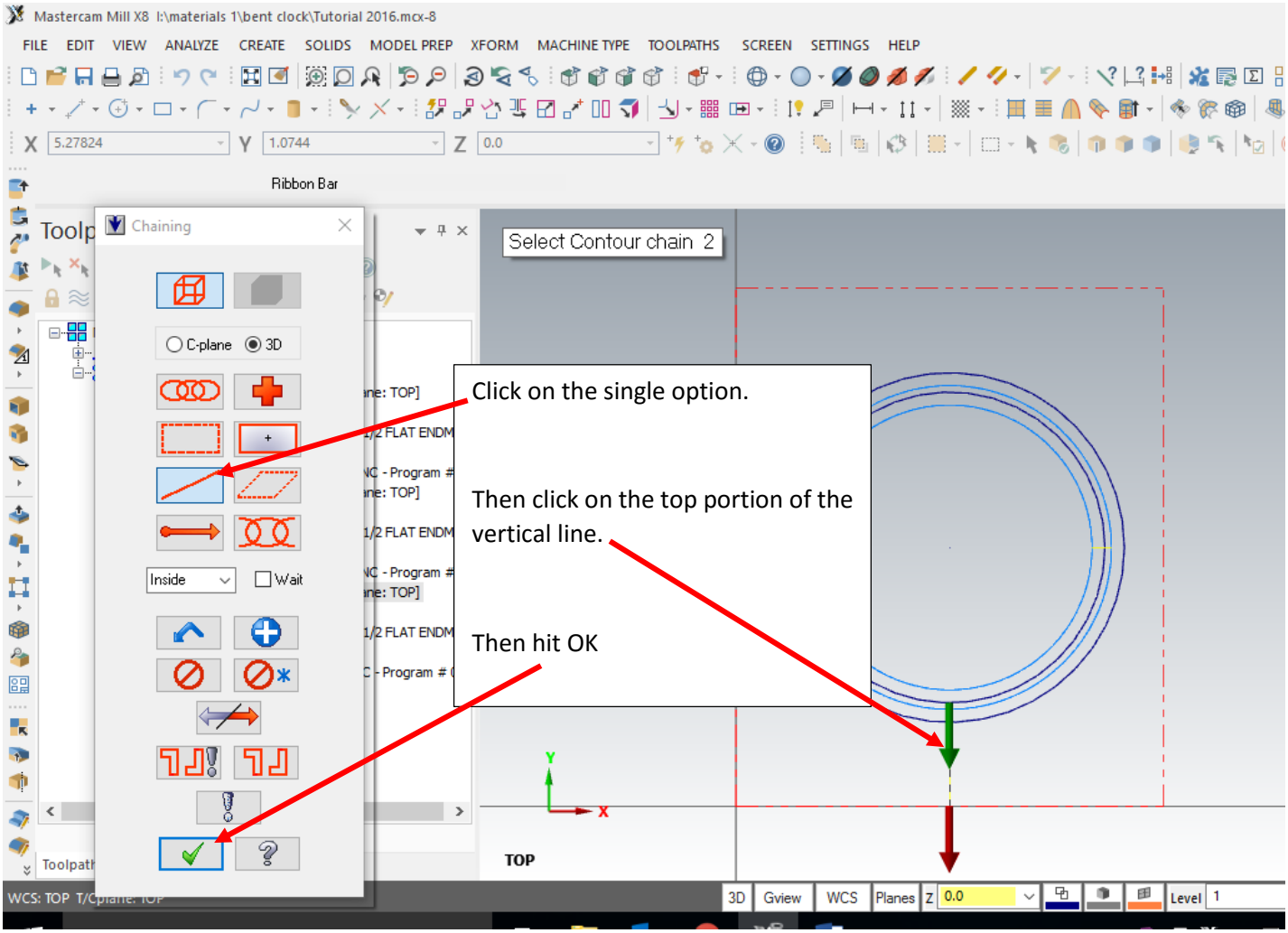


You should see a second light blue toolpath inside the second circle. The blue toolpath represents the center of the tool. If your drawing/toolpath does not look like this, please get assistance.

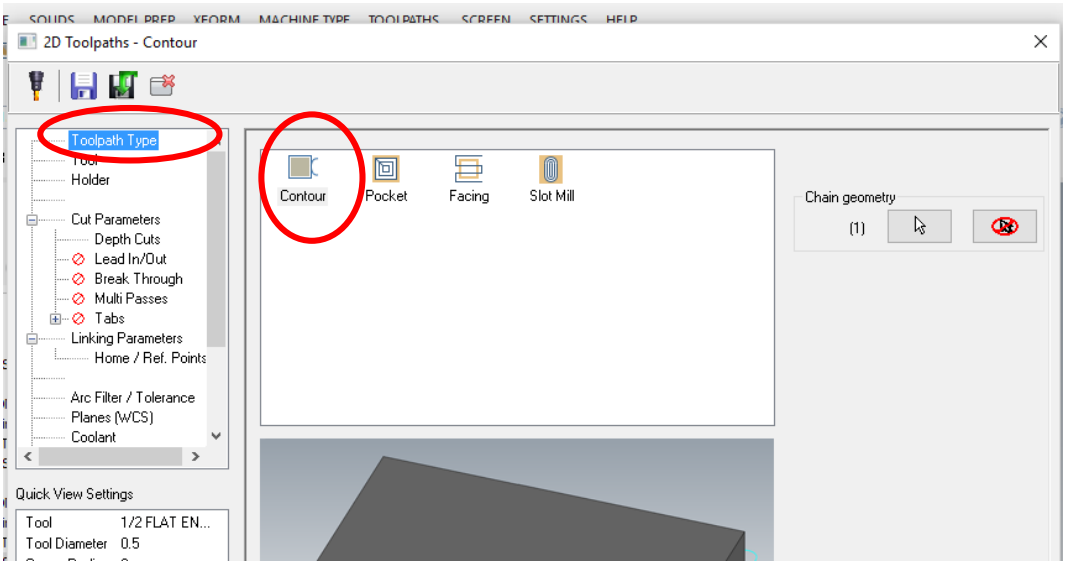
Now it's time for the plow for the dial and clock movement. This plow is represented by the small vertical line in the center of the clock front geometry. It is still a contour toolpath, so the procedure is similar. The only difference is the cutter compensation needs to be turned off, so the center of the cutter follows the line.

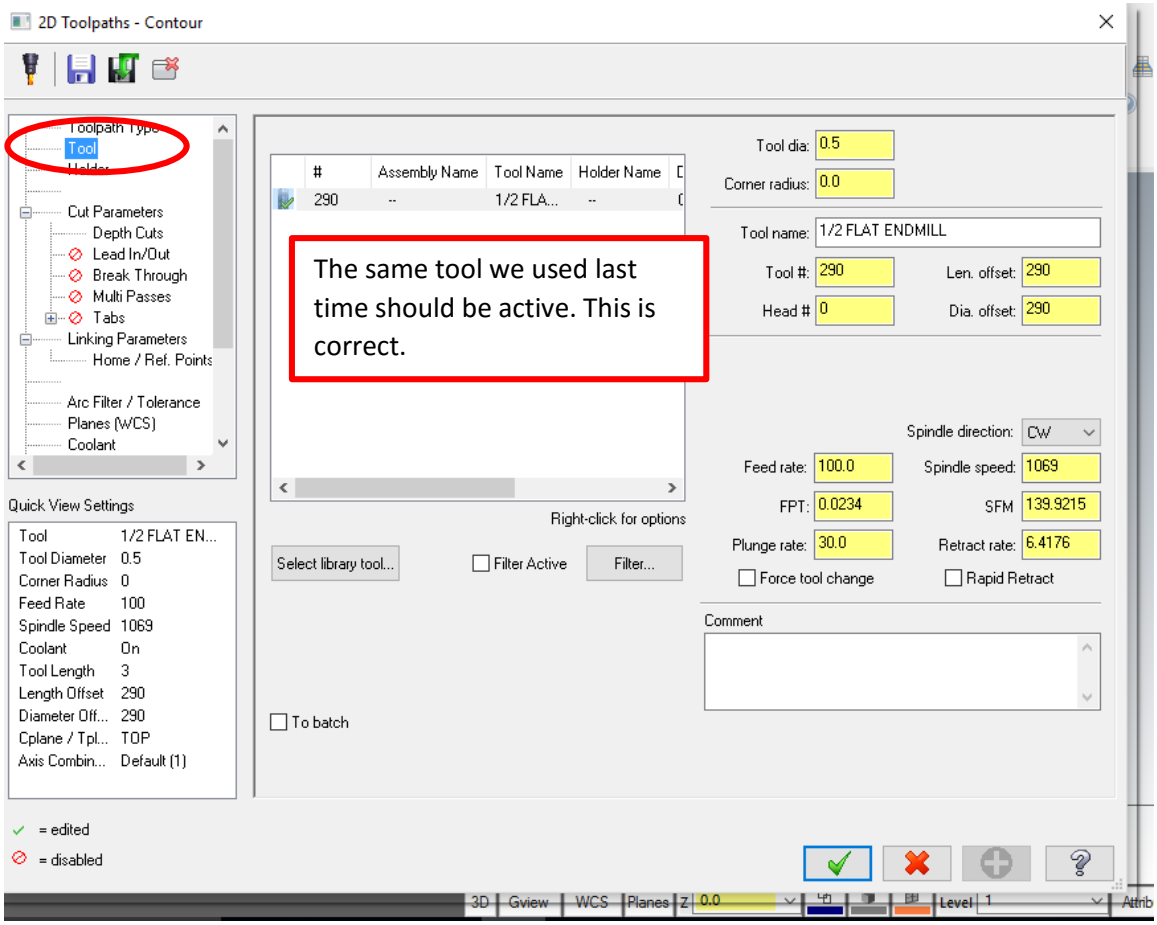
Go to Toolpaths/Countour

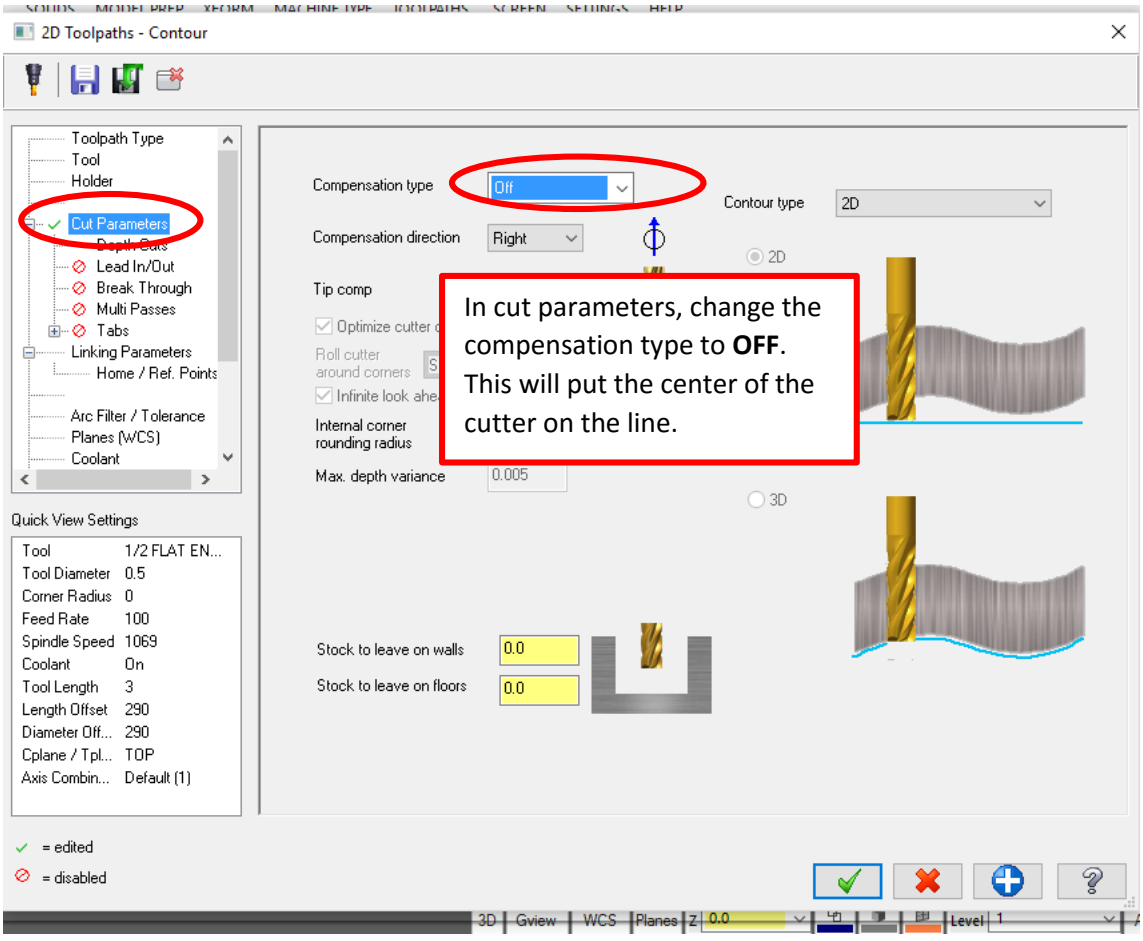


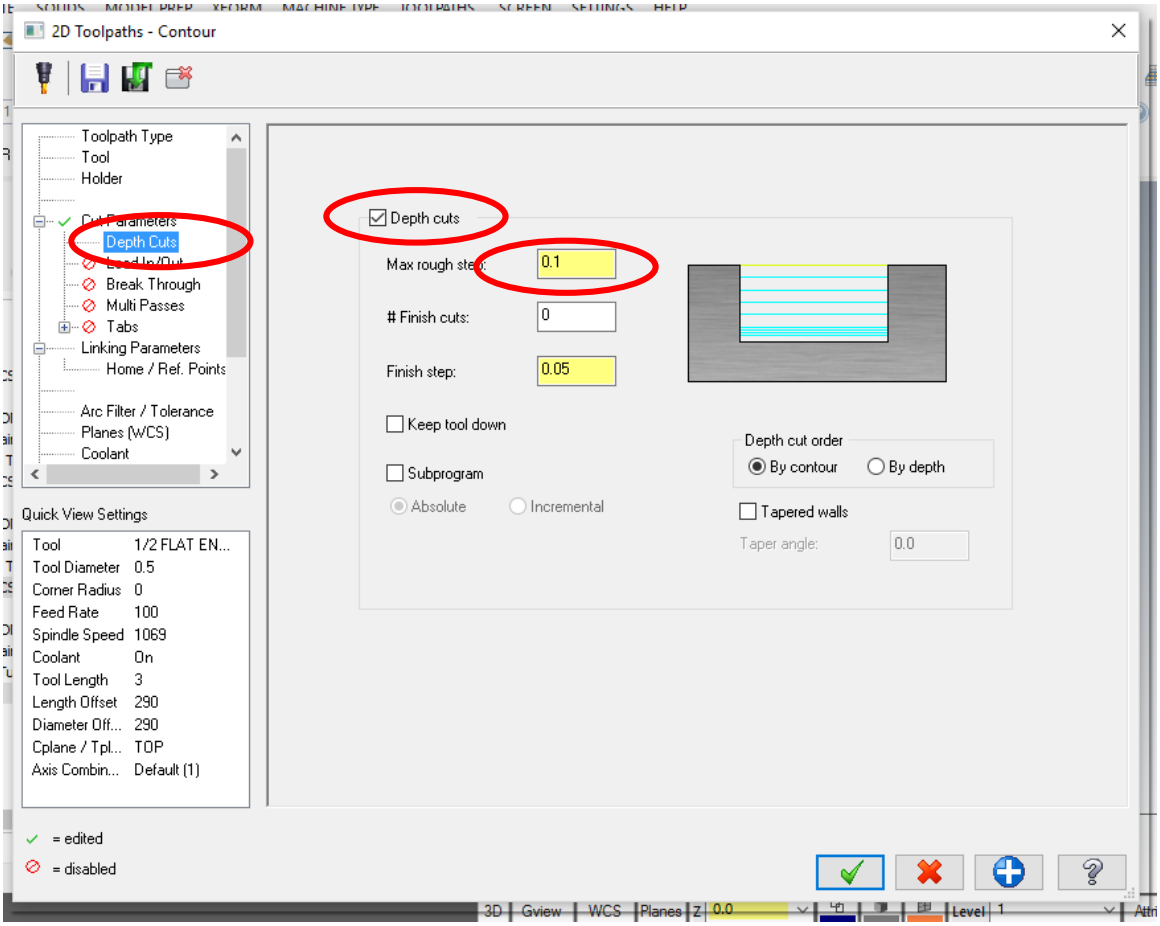


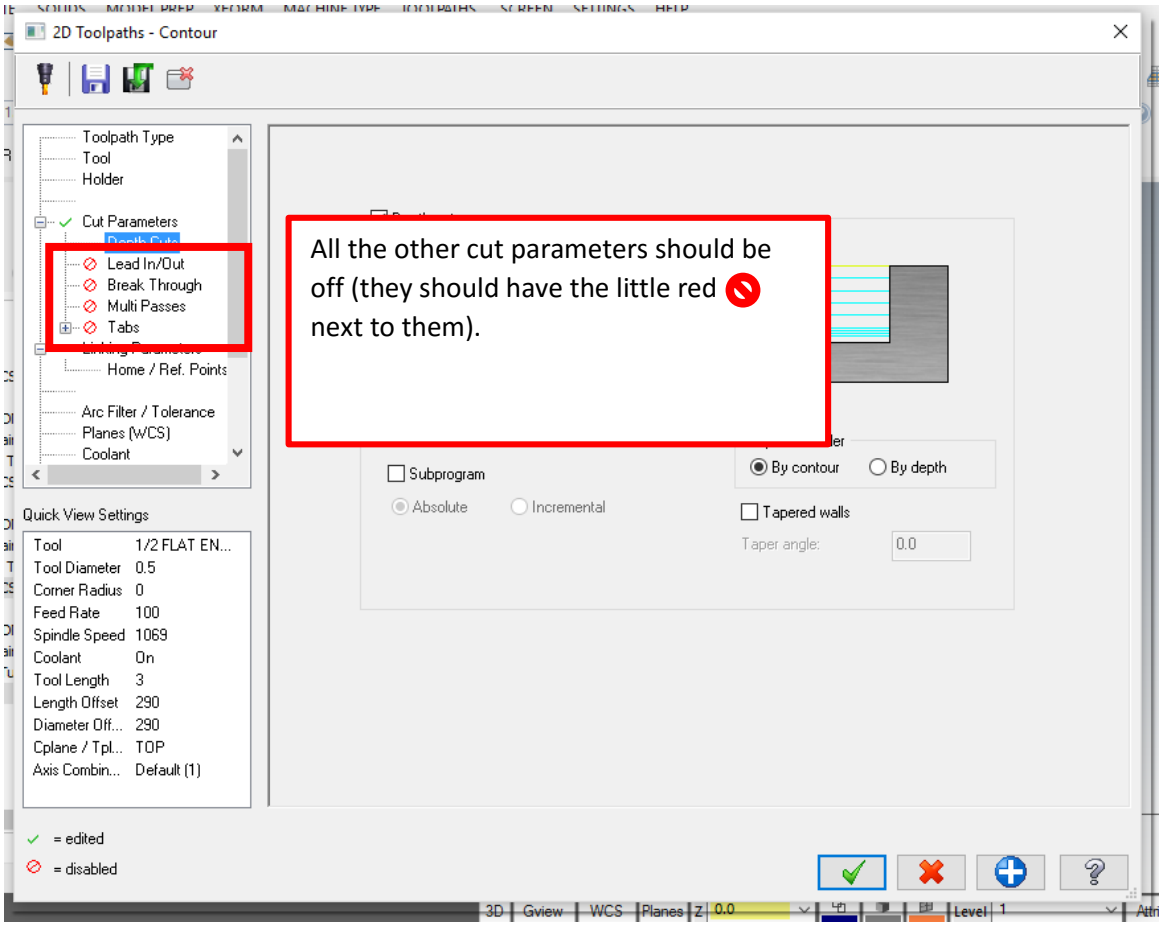
This contour tool path properties are similar to the other ones, but we need a couple of changes. So follow along and enter the following information into all the windows.

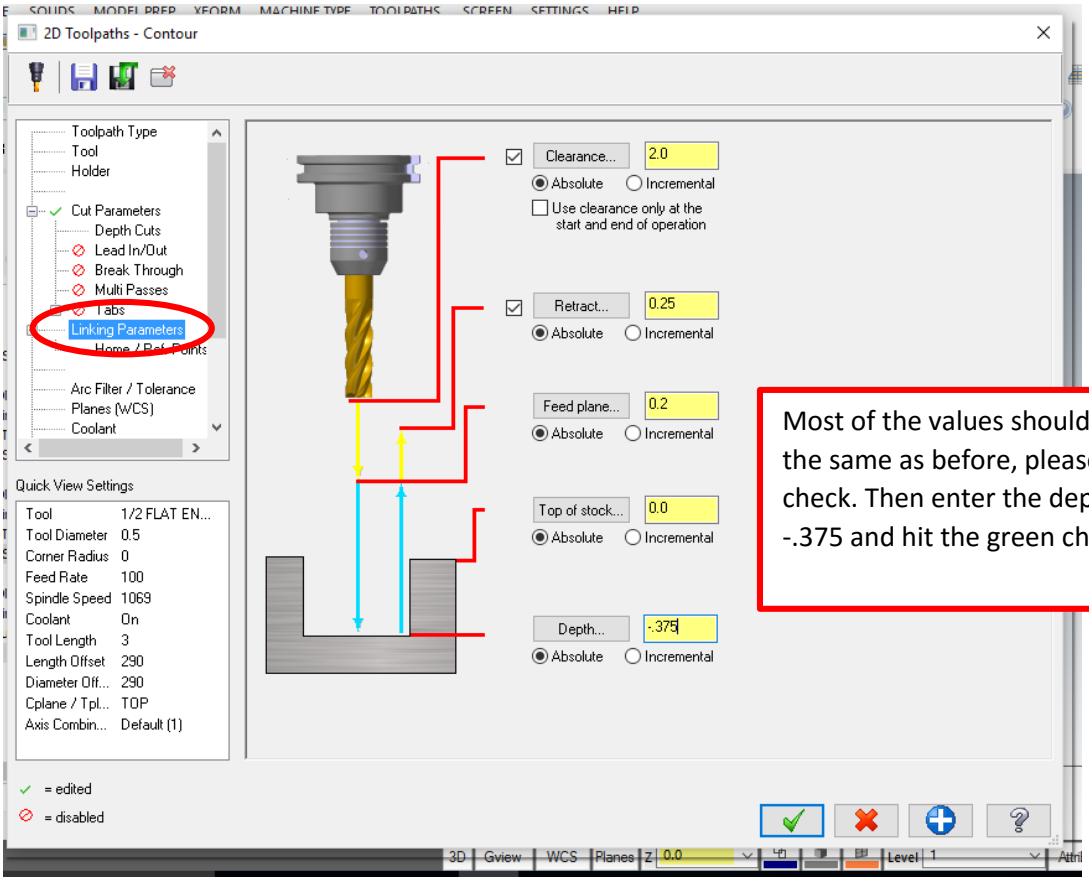



2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 FLAT EN...
Tool Diameter 0.5
Corner Radius 0
Feed Rate 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
Assembly Name Tool Name Holder Name
290 -- 1/2 FLA... --
The same tool we used last time should be active. This is correct.
Right-click for options
Select library tool...
Filter Active
Filter...
To batch
Tool dia: 0.5
Corner radius: 0.0
Tool name: 1/2 FLAT ENDMILL
Tool #: 290
Len. offset: 290
Head #: 0
Dia. offset: 290
Spindle direction: CW
Feed rate: 100.0
Spindle speed: 1069
FPT: 0.0234
SFM 139.9215
Plunge rate: 30.0
Retract rate: 6.4176
Force tool change
Rapid Retract
Comment
3D Gview WCS Planes Z 0.0 Level 1 Attrib


2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
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 100
Spindle Speed 1069
Coolant On
Tool Length 3
Length Offset 290
Diameter Off... 290
Cplane / Tpl... TOP
Axis Combin... Default (1)
= edited
= disabled
Compensation type Off
Compensation direction Right
Contour type 2D
Tip comp
2D
3D
In cut parameters, change the compensation type to OFF. This will put the center of the cutter on the line.
Internal corner rounding radius
Max. depth variance 0.005
Stock to leave on walls 0.0
Stock to leave on floors 0.0
3D Gview WCS Planes Z 0.0 Level 1

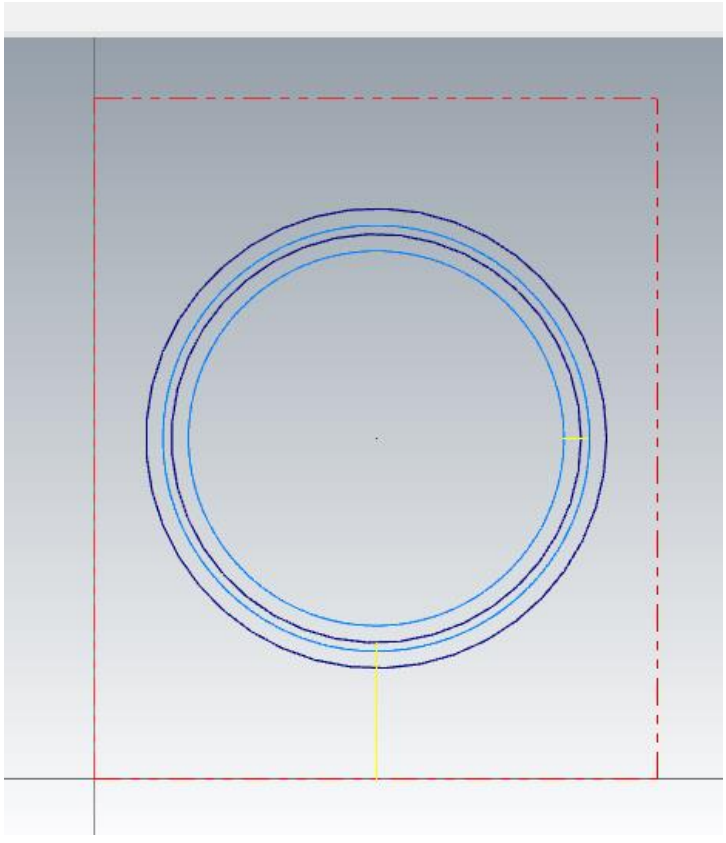

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Depth cuts
Max rough step: 0.1
Finish cuts: 0
Finish step: 0.05
Keep tool down
Subprogram
Absolute
Incremental
Depth cut order
By contour
By depth
Tapered walls
Taper angle: 0.0
Quick View Settings
Tool 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
3D
Gview
WCS
Planes
Z 0.0
Level 1

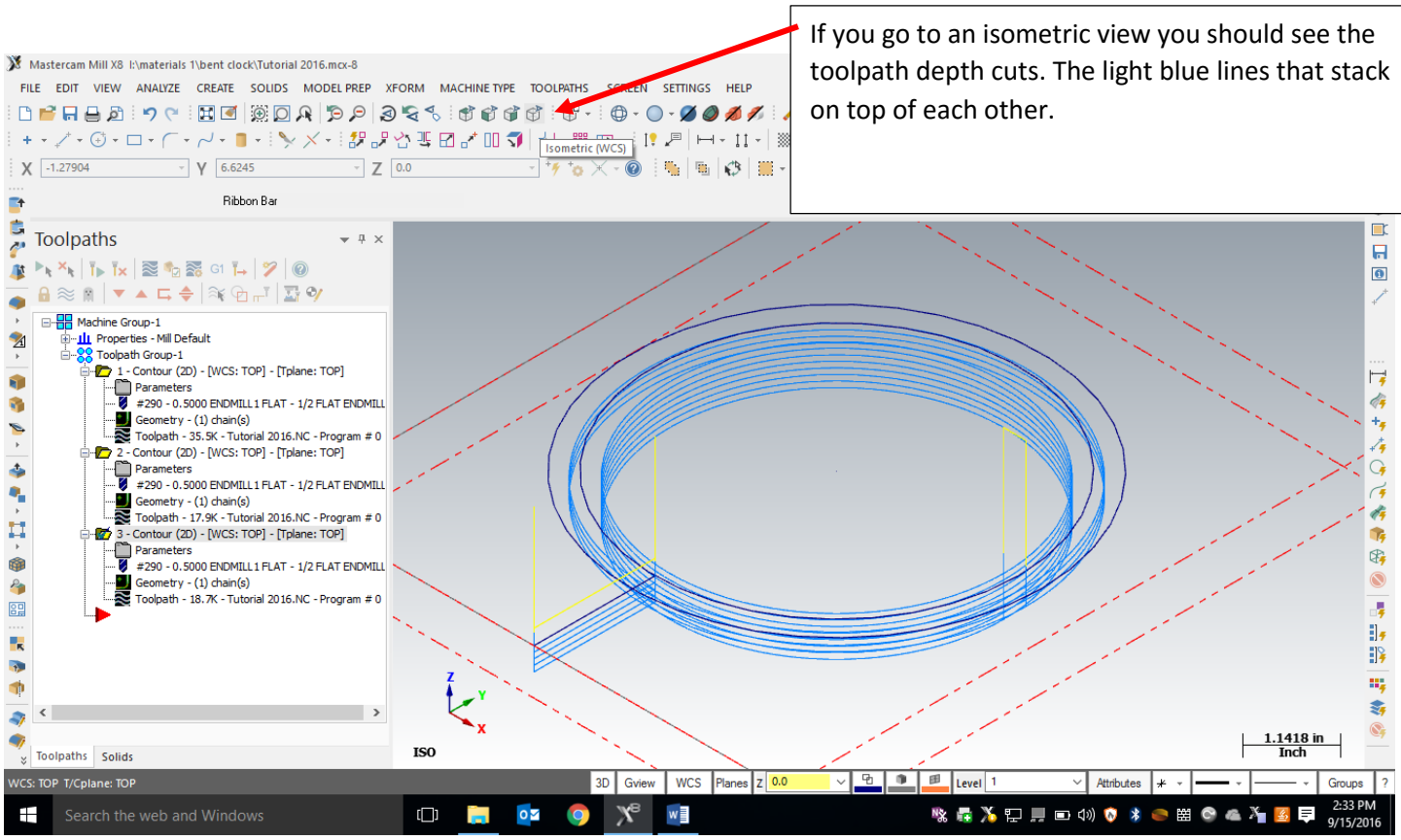

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Lead In/Out
Break Through
Multi Passes
Tabs
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
All the other cut parameters should be off (they should have the little red ⊘ next to them).
Subprogram
Absolute
Incremental
By contour
By depth
Tapered walls
Taper angle: 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
3D
Gview
WCS
Planes
Z 0.0
Level 1



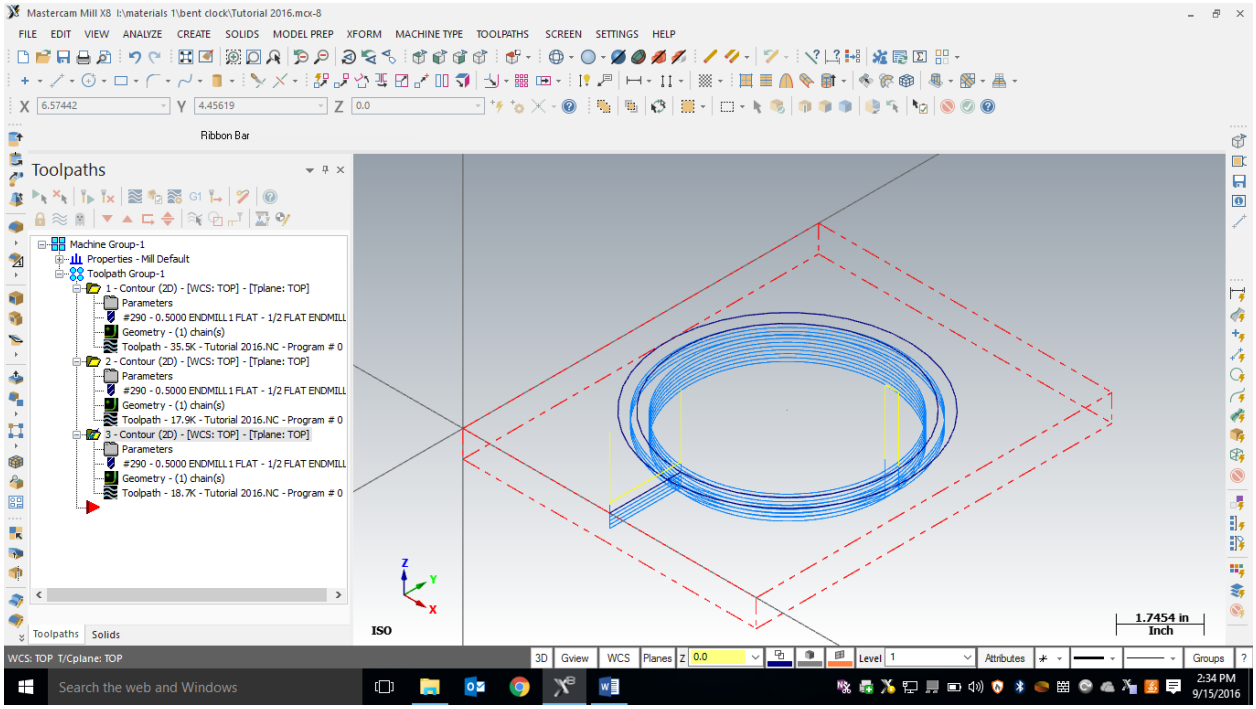
This should draw the toolpath, you probably will not see the toolpath since the center of the cutter follows the line. So you probably just see the line unless you are in a different view.

Result:

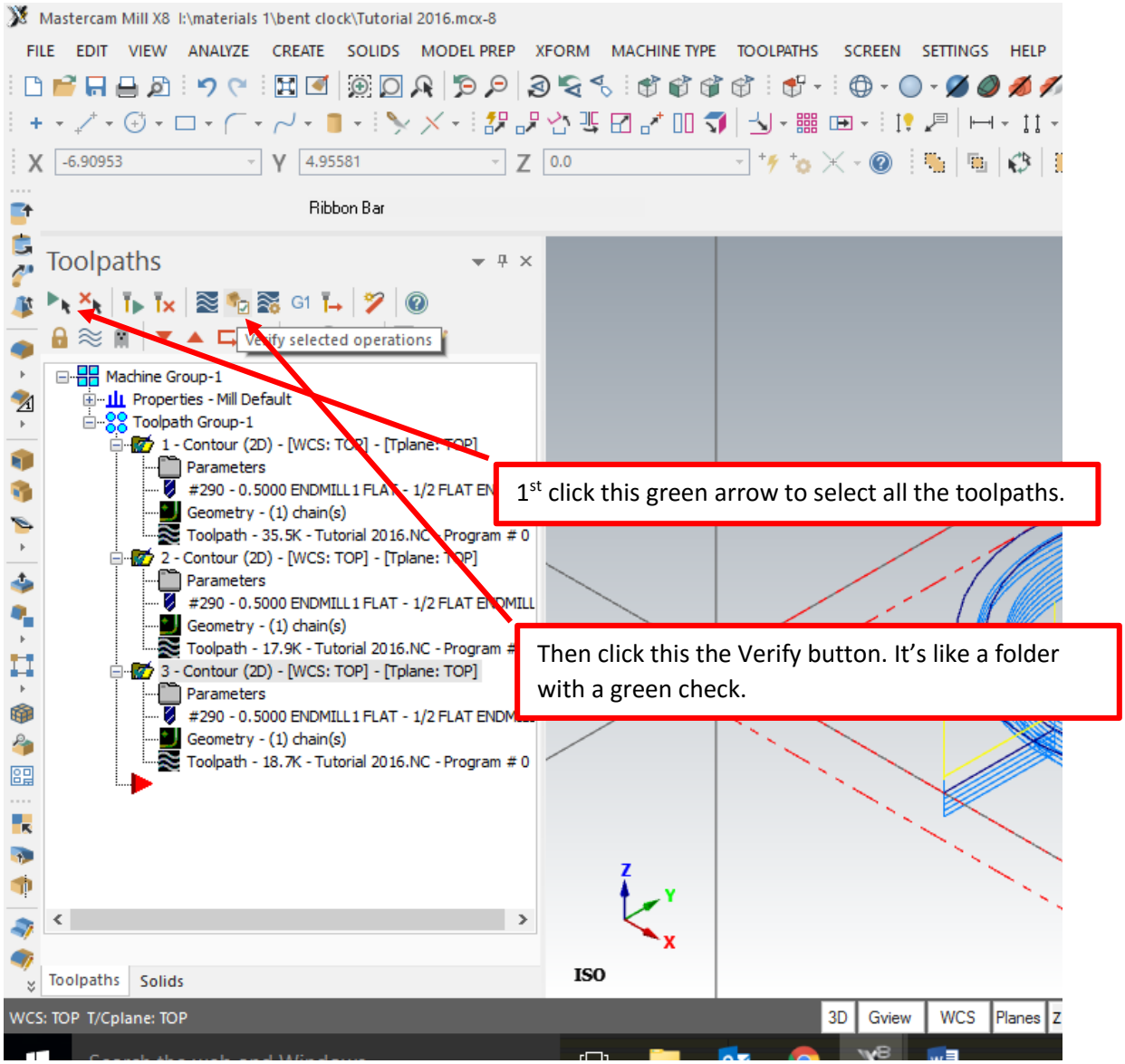




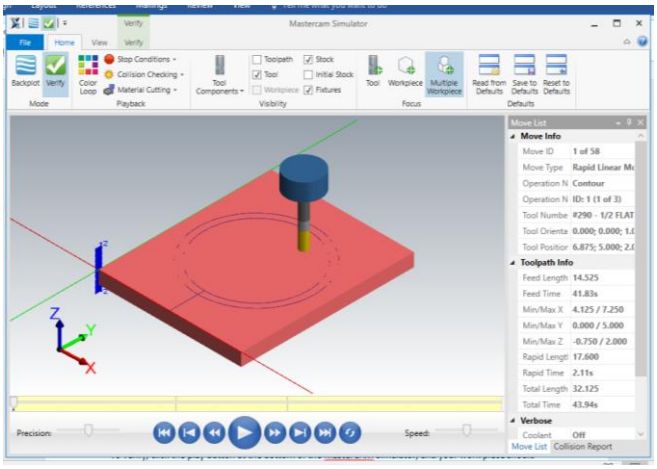
For the next procedure we want to verify the toolpath, basically we are going to virtually cut the piece on the computer. So we are going to look at the geometry and toolpaths in a 3D view so we can see what is going to happen better. Go to an isometric view if you did not already, zoom in\out, and center the work piece so it looks something like what is below.



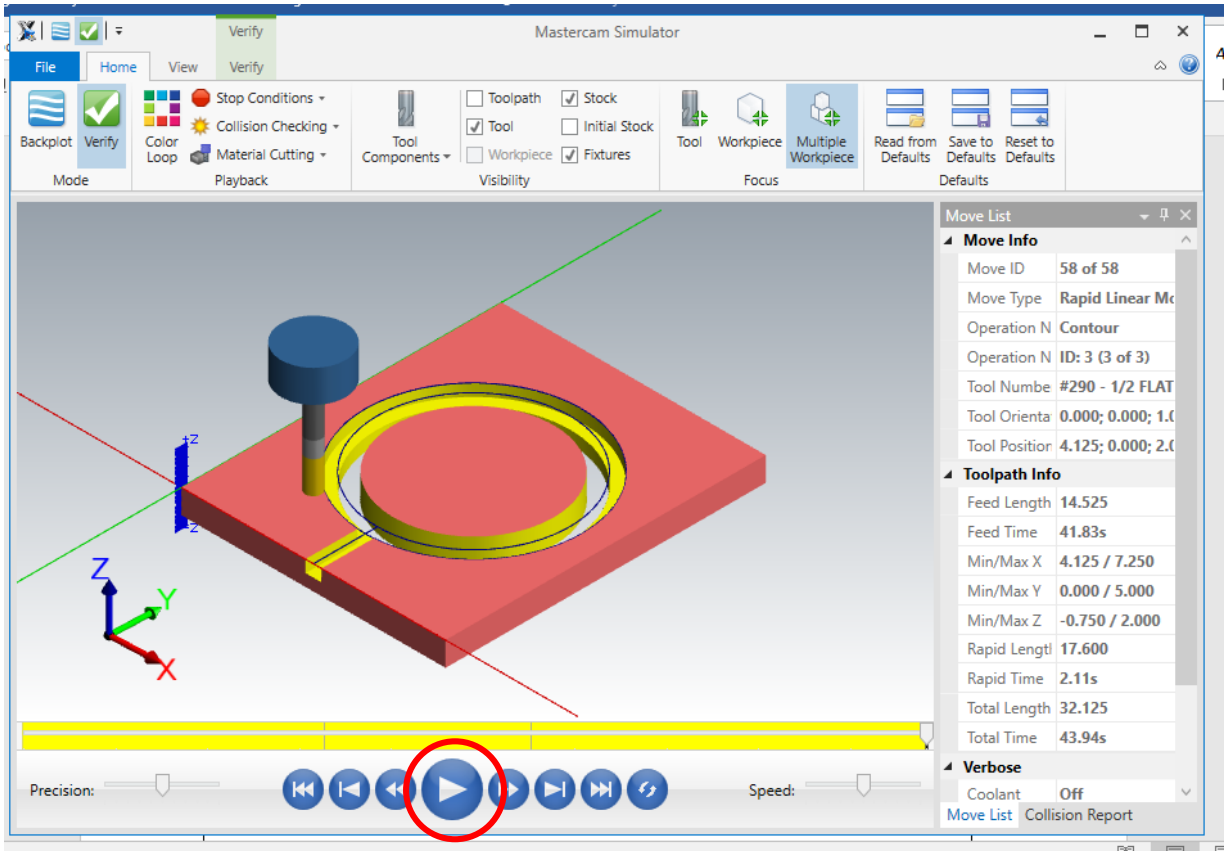
To verify the toolpaths, do the following:



The MasterCAM Toolpath Simulator should open up in a new window. Like below.



To verify, click the play button at the bottom of the MasterCAM Simulator, and your work piece should look like below. Please show your instructor to receive credit.



Congratulations, you drew all the toolpaths you need for the clock front**. Please show Mr. Marmor so he can sign off on your completion of the process.**.