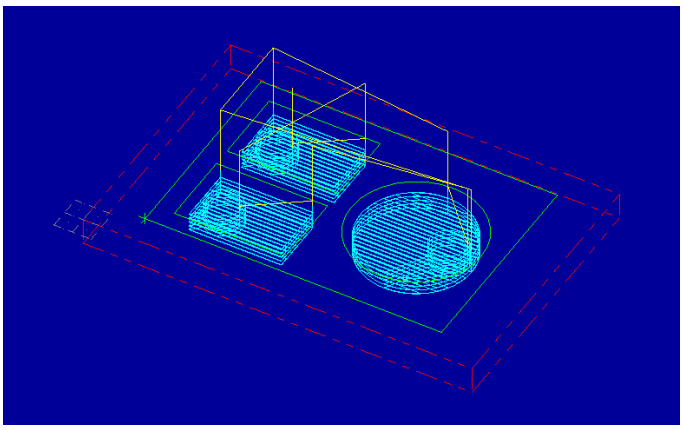
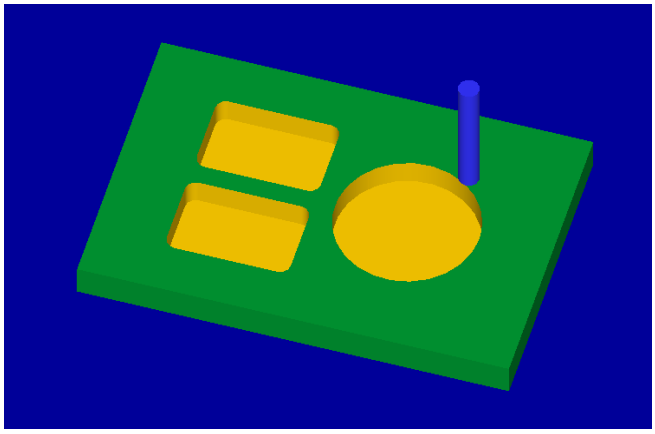
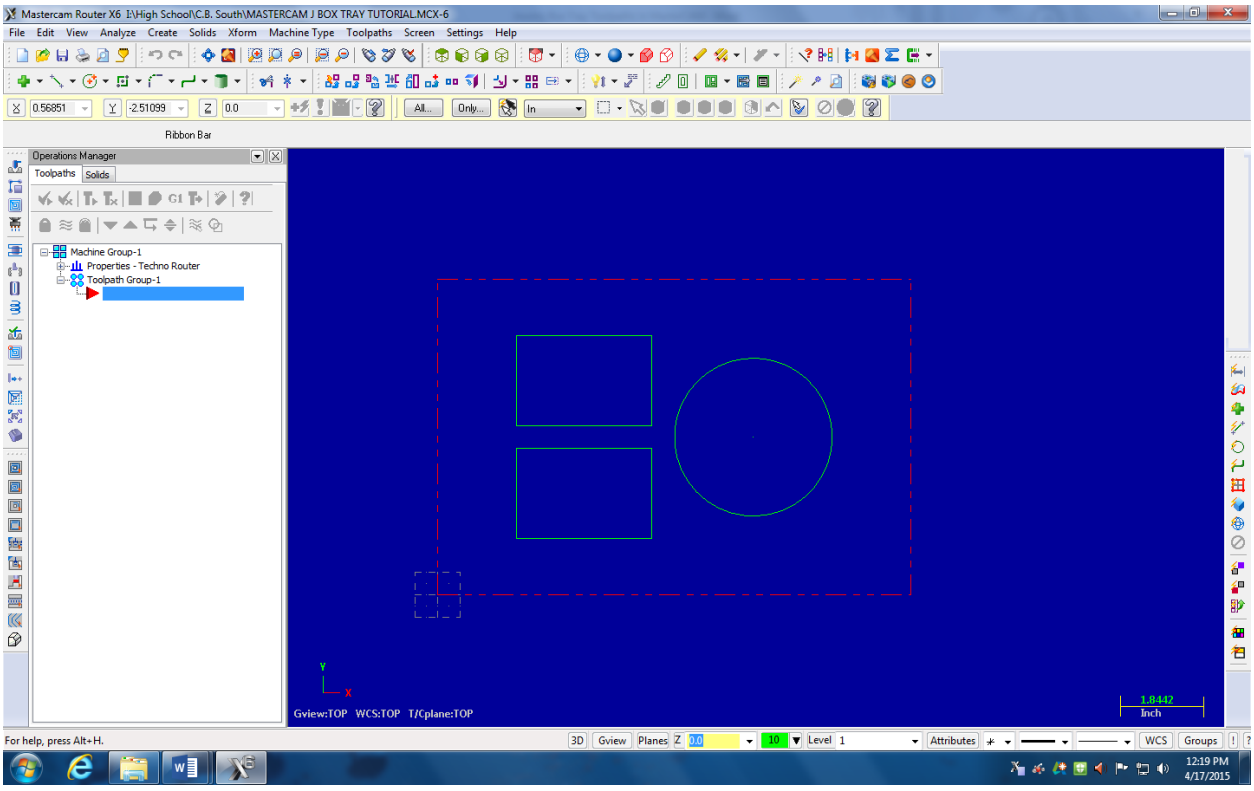
Jewelry Box Tray Toolpaths in MasterCAM





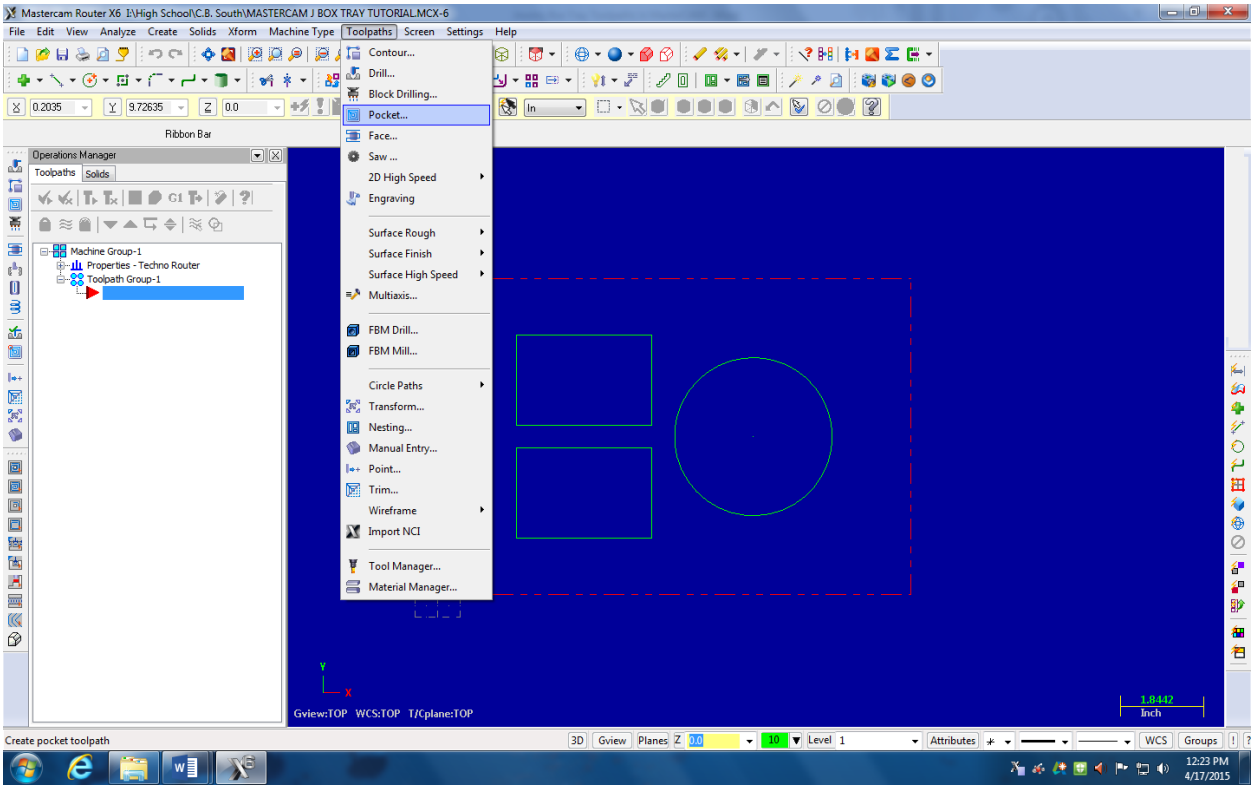
Open the MasterCAM application and open your jewelry box tray geometry file.



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. When the geometry is selected you must either pick the single option or the chaining option (multiple objects laid out

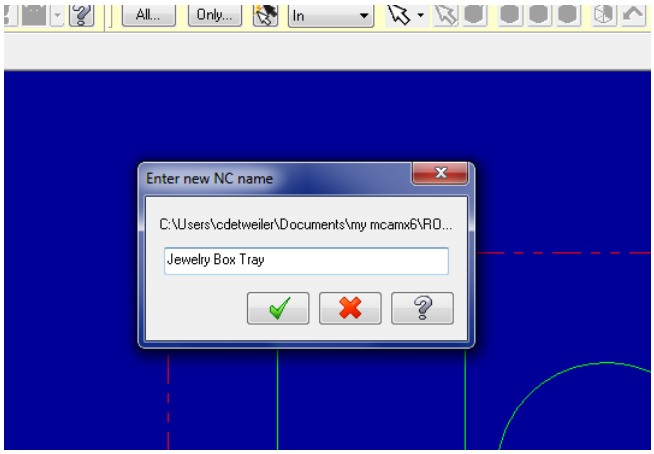
end to end) before you select the geometry. The second type of toolpath is a pocket. A pocket toolpath will make a cavity inside the selected geometry. Pocket toolpath is what will be used for machining the tray for your jewelry box. When we select that geometry, we will use the chaining option, since the rectangles are really made from four lines each.

To start the toolpaths, go to Toolpaths / pocket:



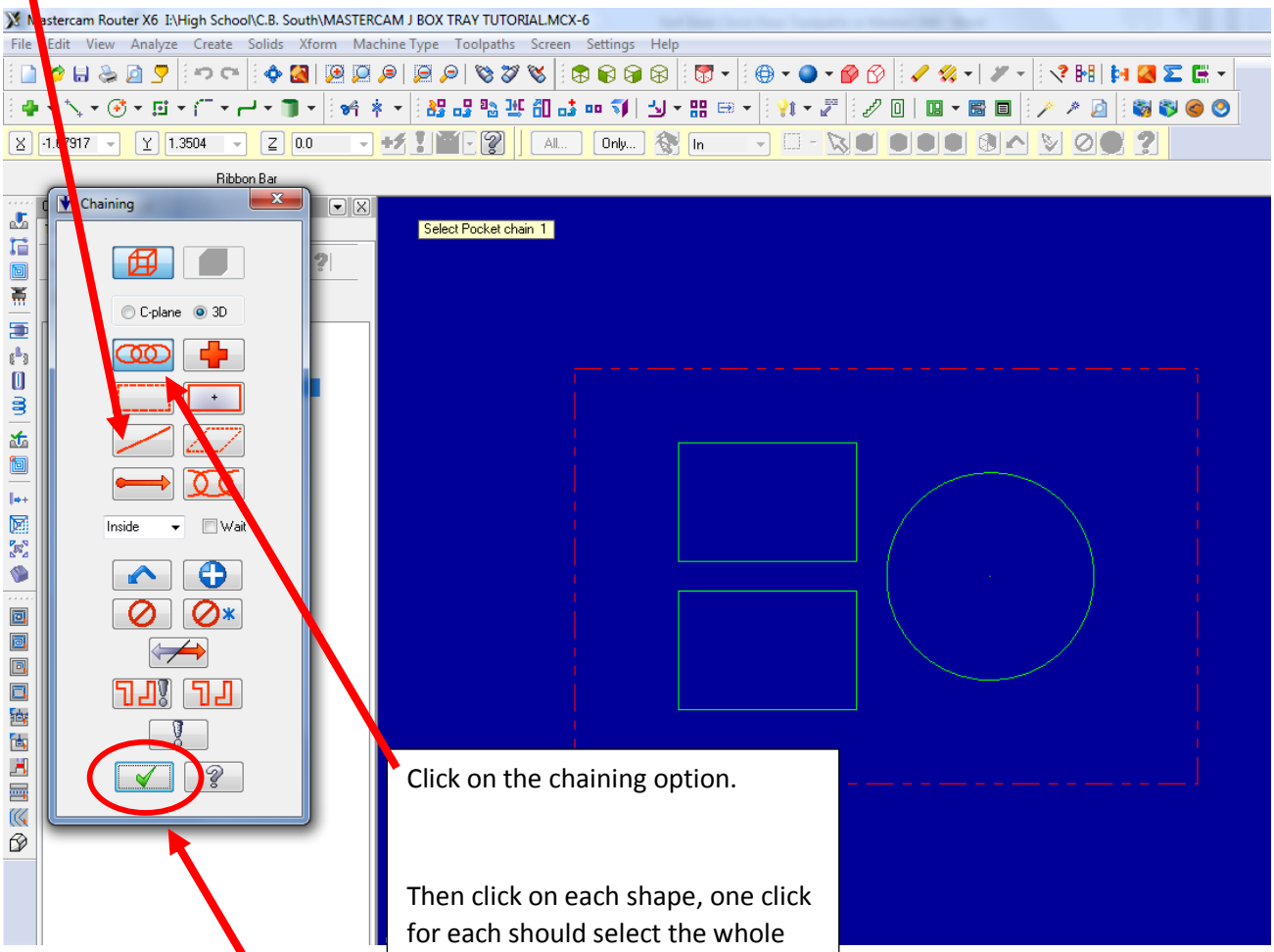
When the new NC dialog box comes up, type in a good file name such as Jewelry Box Tray.

Click the green check.



Now it's time to select the geometry which we want to assign toolpaths to. The rectangles and circle I have in my tray design can be chained together since they will have the same style and depth of cut. We can complete this operation with one toolpath. Each pocket will be a different chain of geometry. Each rectangle is actually four lines that are placed end to end and a circle is once continuous line. To select the geometry, we will use the chaining function and select all of the different chains then apply one toolpath to all of those geometry chains.

If my design had pockets at different depths I would use the single chaining command a select / assign individual toolpaths to each piece of geometry.



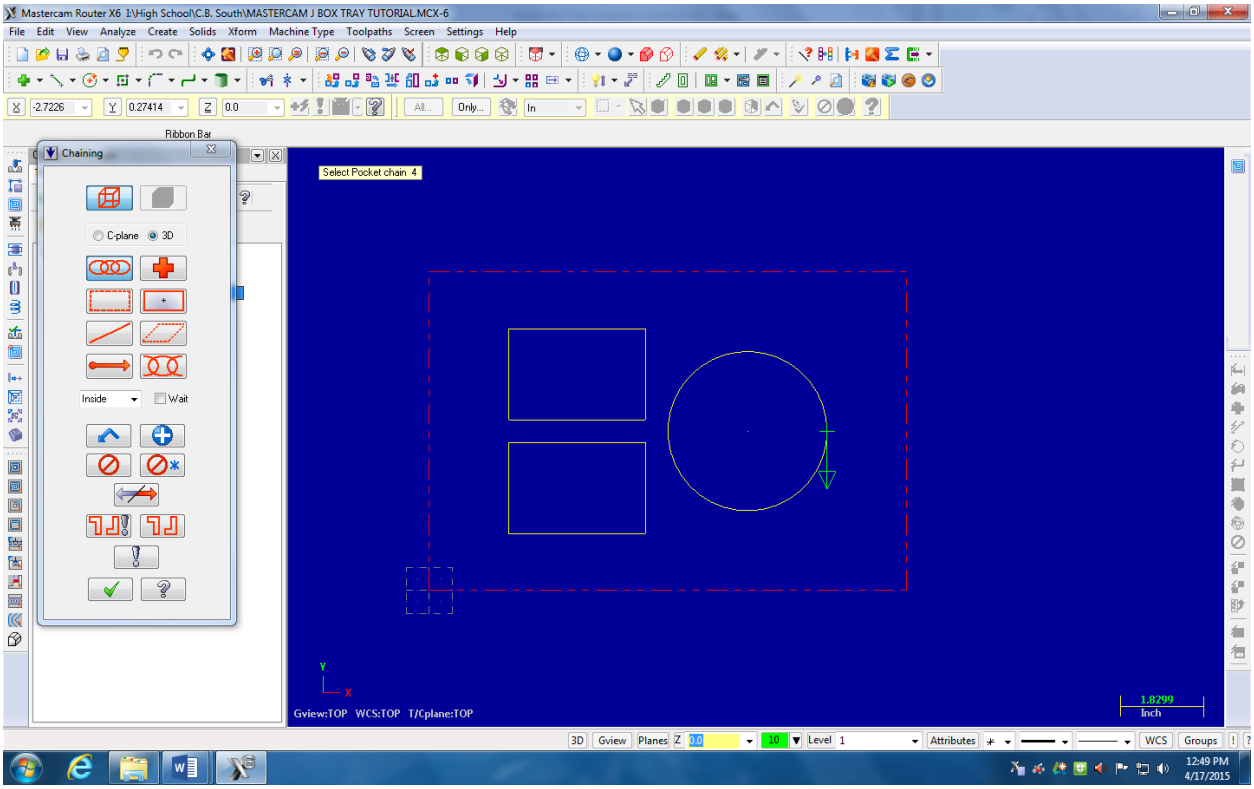
Click on the chaining option.

Then click on each shape, one click for each should select the whole shape

If you follow the prompts, chain 1 is one rectangle, chain 2 is the other and chain 3 is my circle.

Then hit OK

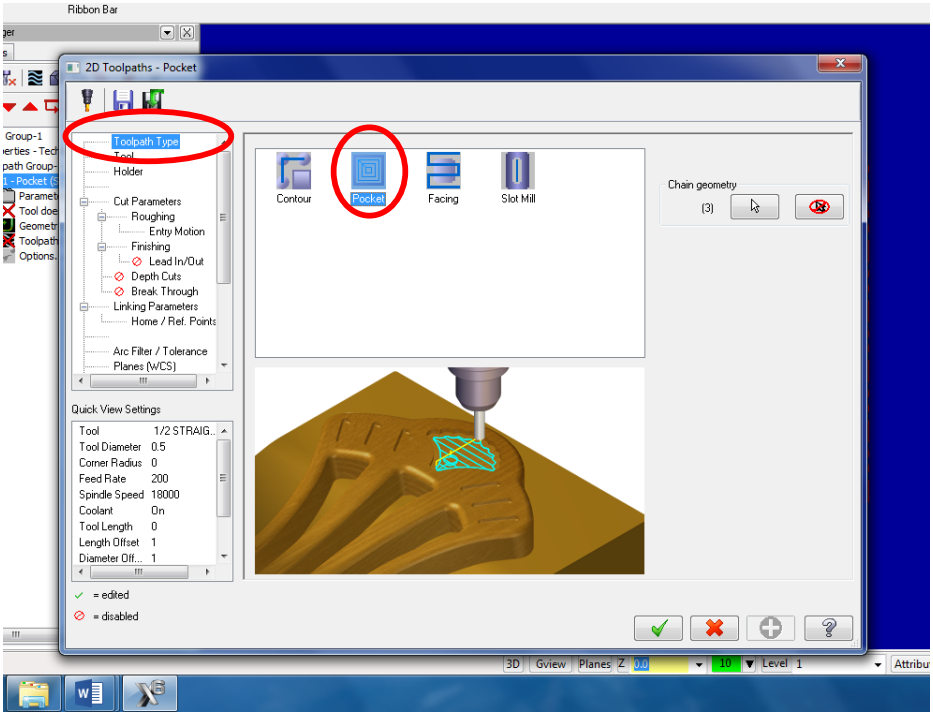
After you have selected all of the shapes / geometry they should change to a white color and have an arrow on them indicating the direction of cutter.

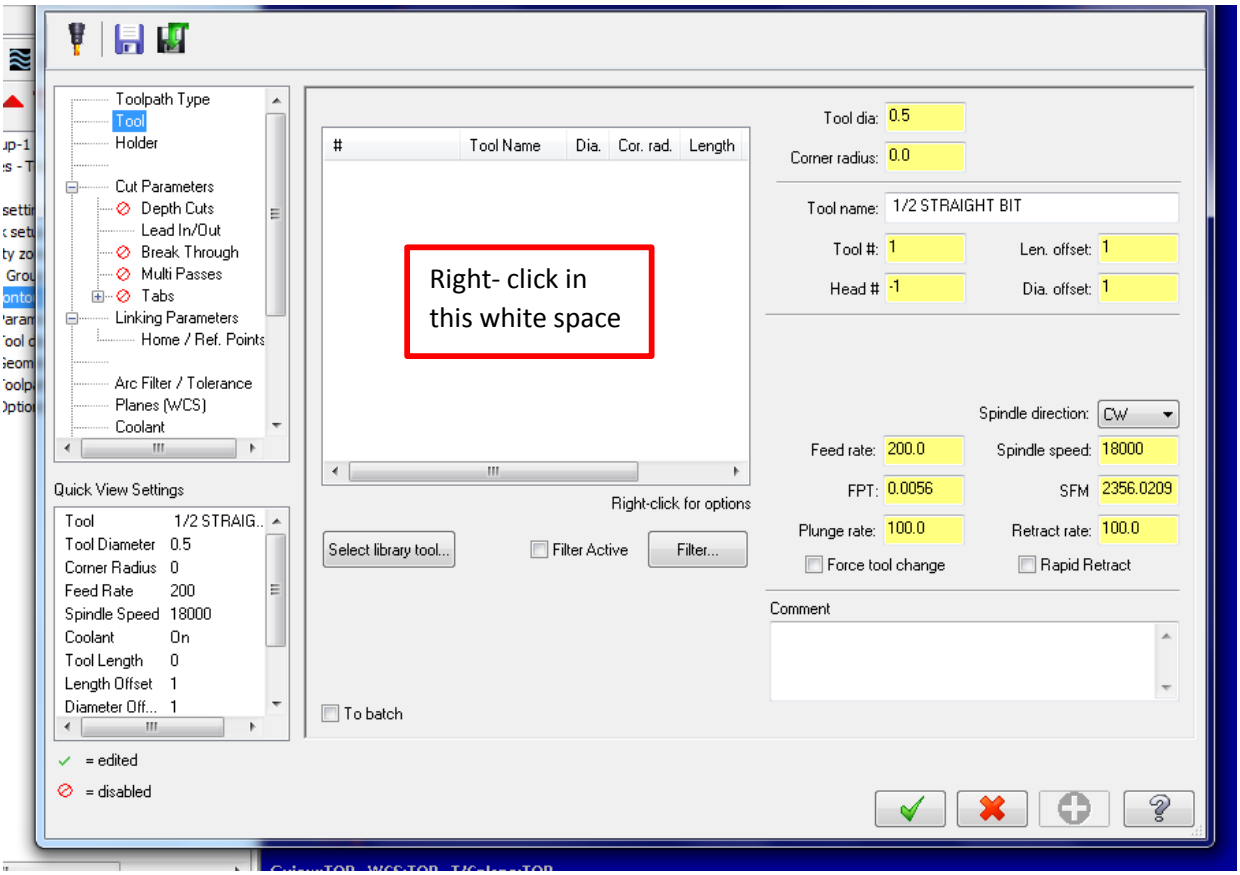


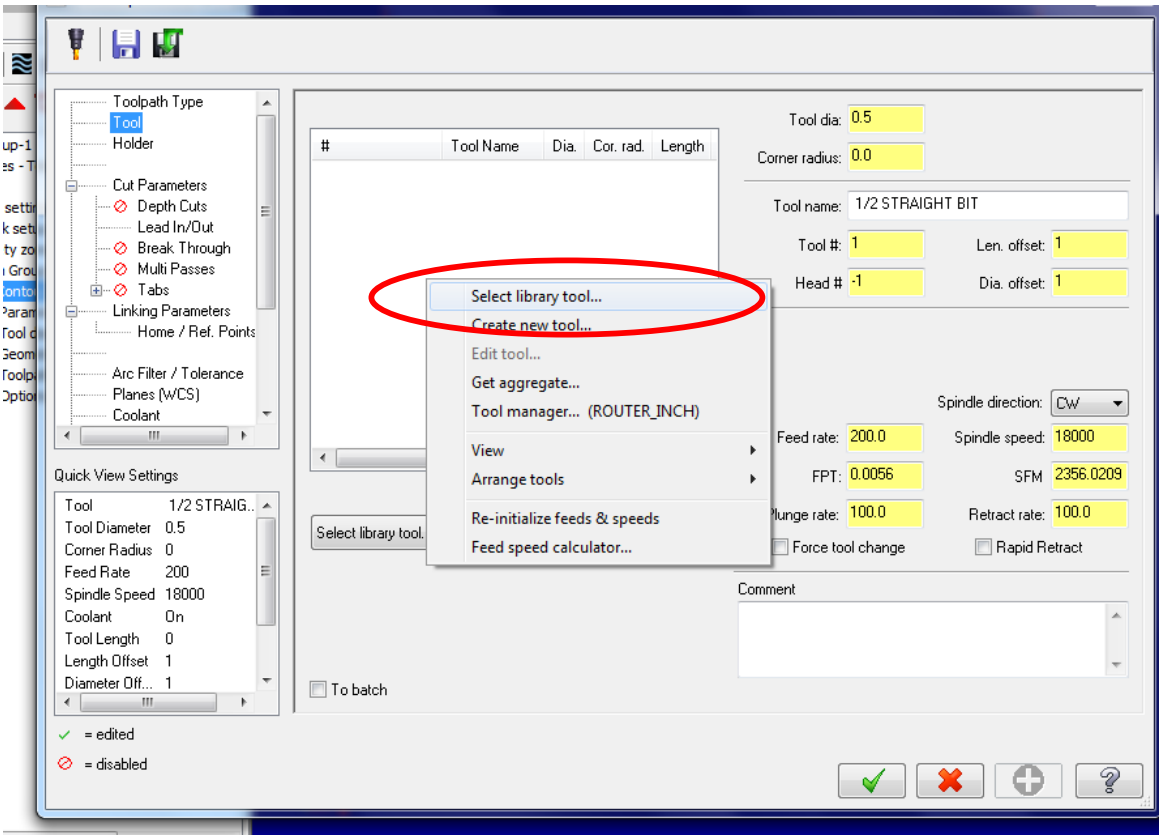
After you have selected all of your geometry, hit the green check to move on.

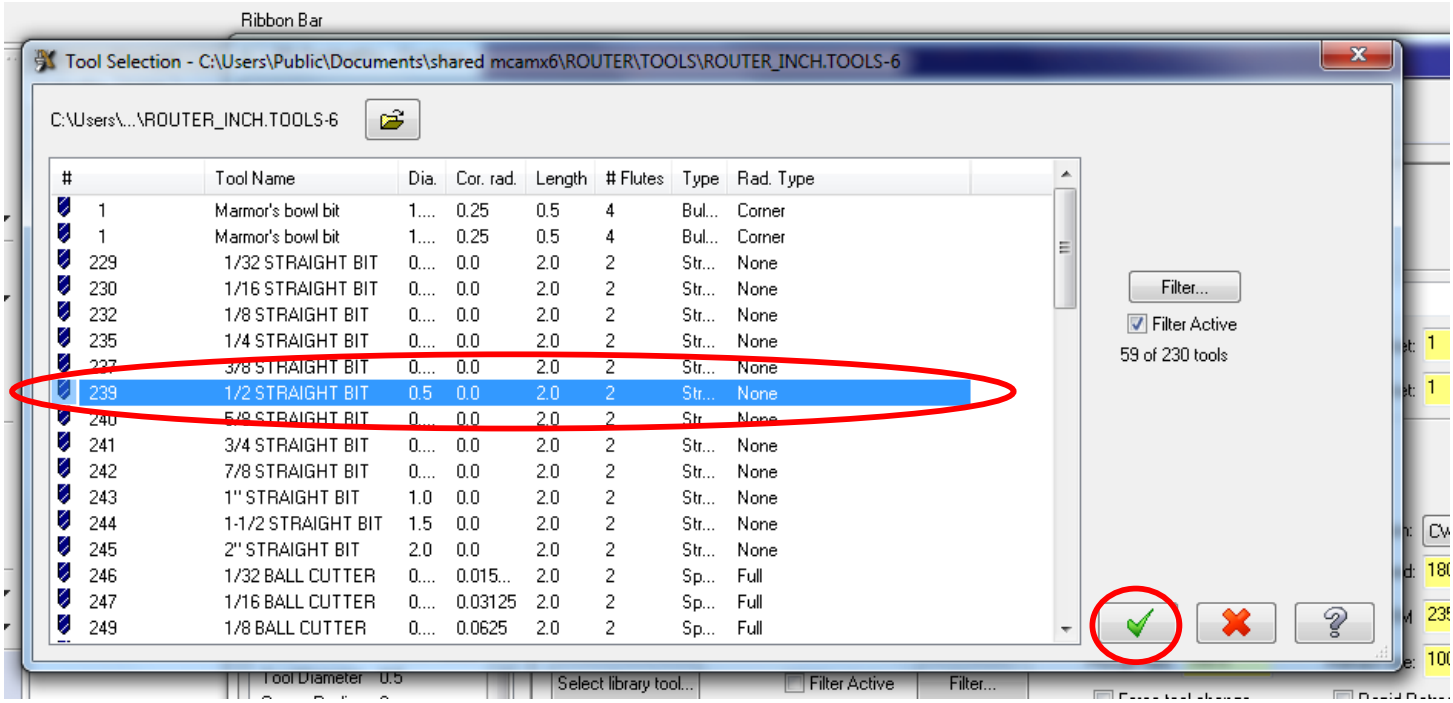
After clicking the green check this screen will appear.

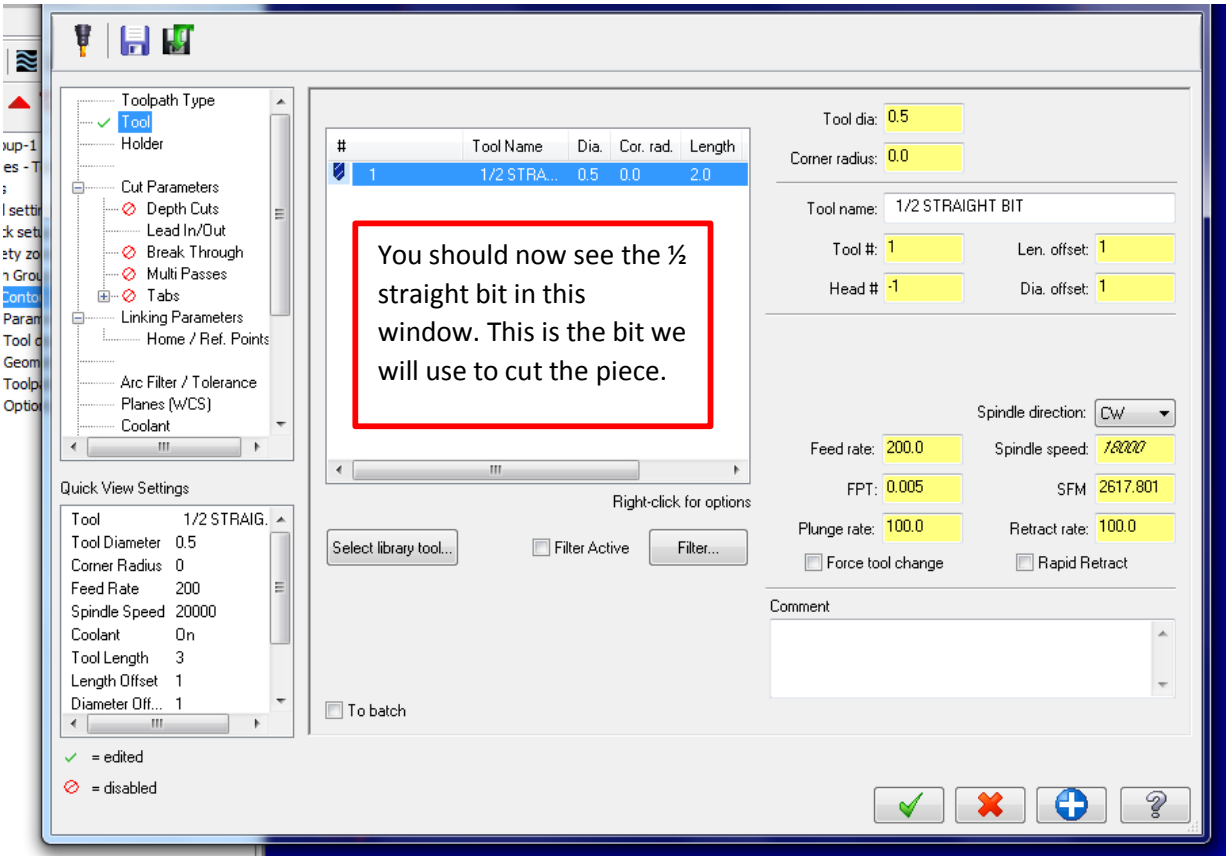
Under toolpath type click on Pocket.

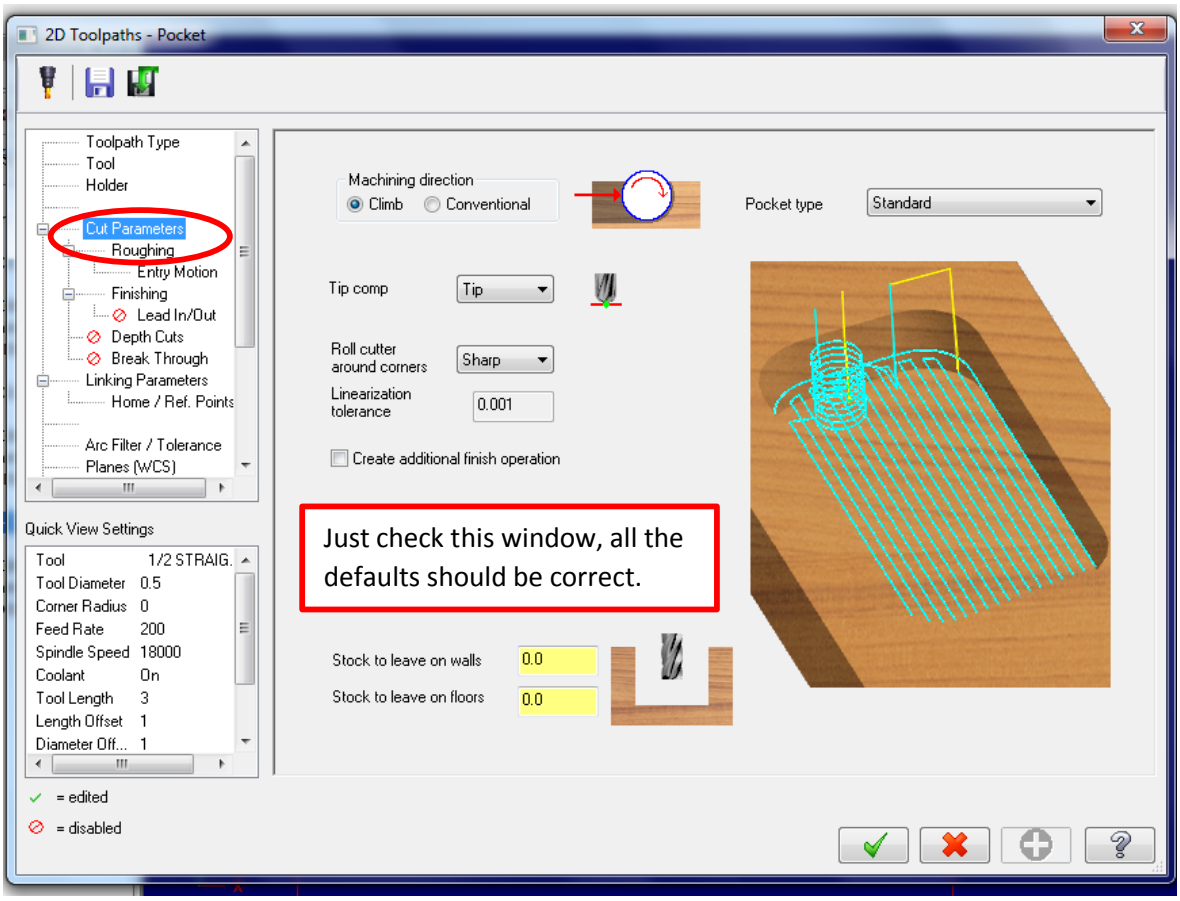


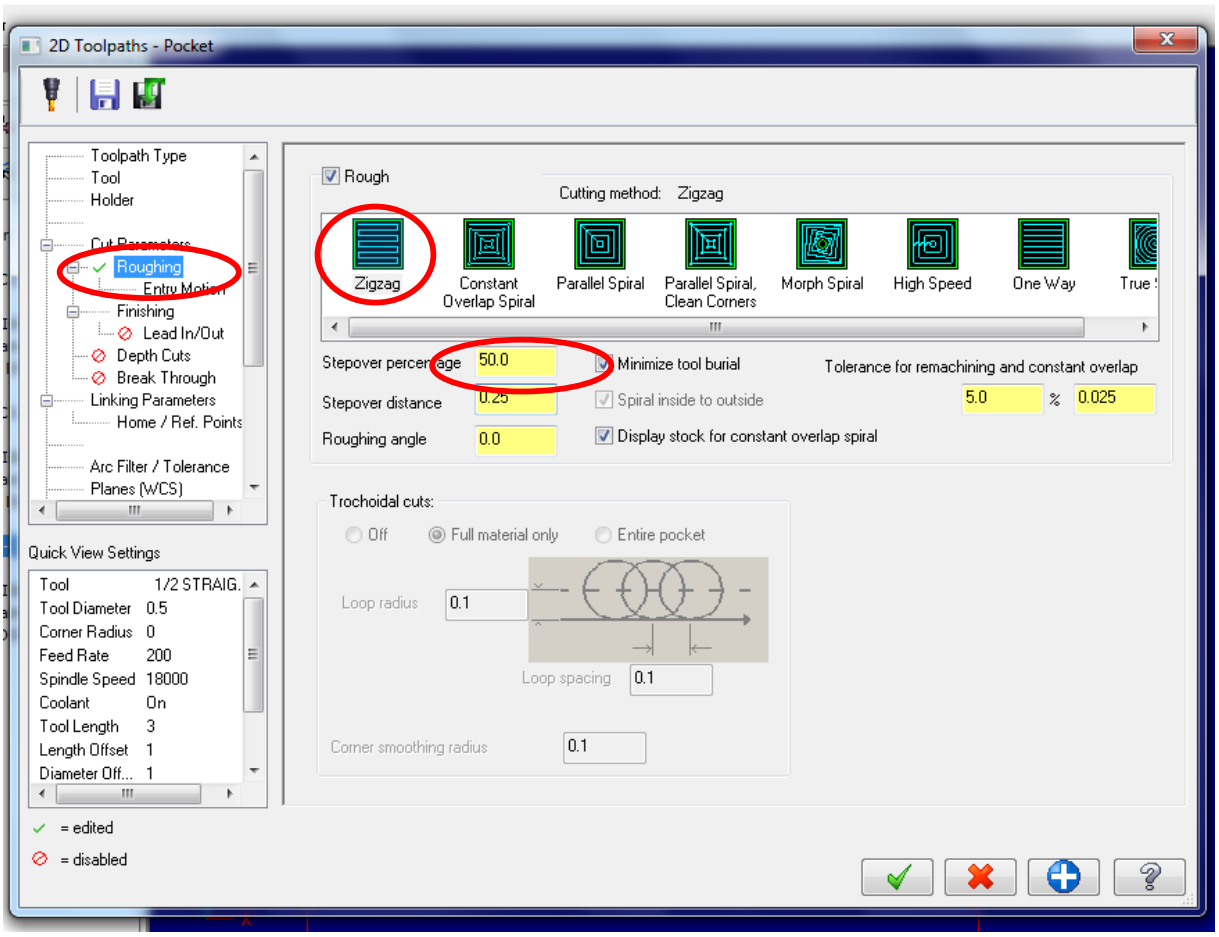

Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 STRAIG..
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 0
Length Offset 1
Diameter Off... 1
= edited
= disabled
Tool Name Dia. Cor. rad. Length
Right- click in this white space
Right-click for options
Select library tool...
Filter Active
Filter...
To batch
Tool dia: 0.5
Corner radius: 0.0
Tool name: 1/2 STRAIGHT BIT
Tool #: 1
Len. offset: 1
Head #: -1
Dia. offset: 1
Spindle direction: CW
Feed rate: 200.0
Spindle speed: 18000
FPT: 0.0056
SFM 2356.0209
Plunge rate: 100.0
Retract rate: 100.0
Force tool change
Rapid Retract
Comment

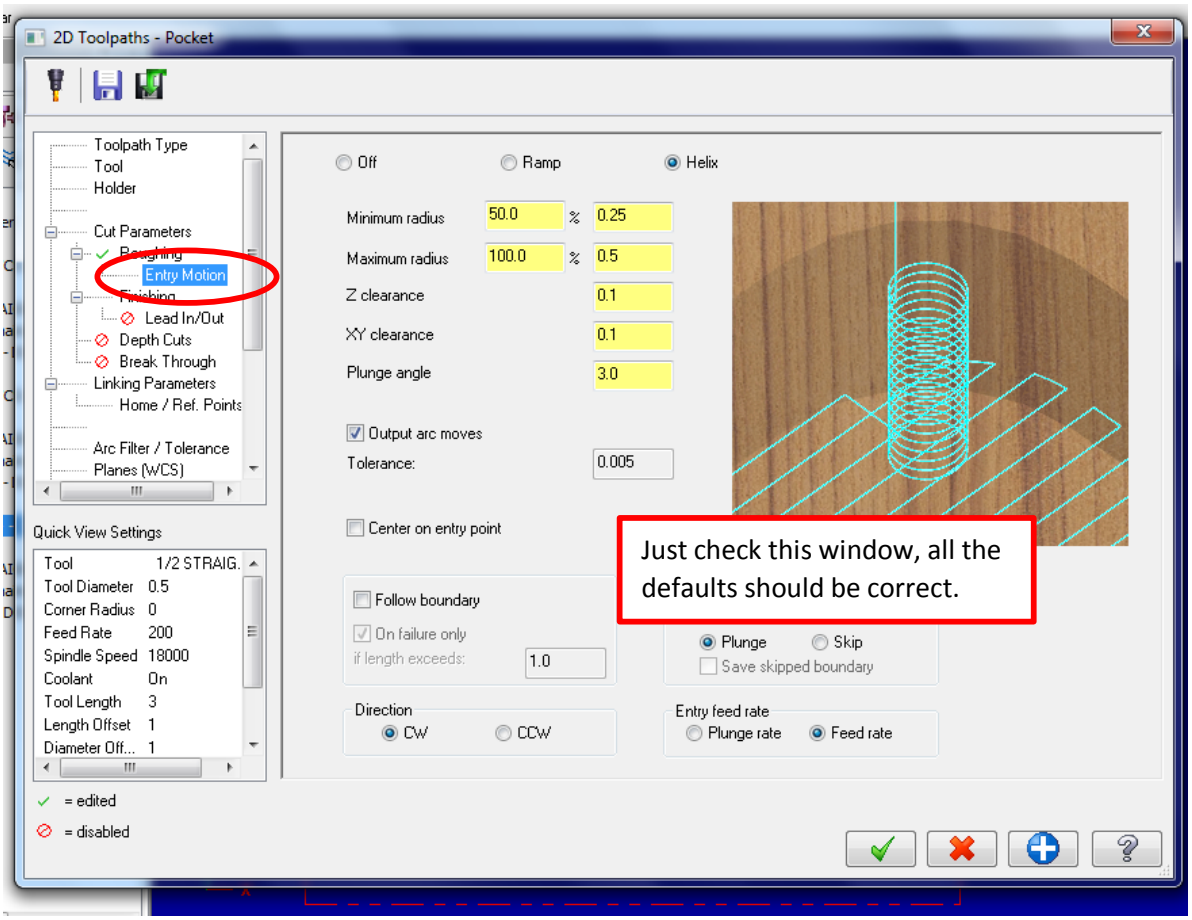

Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 STRAIG..
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 0
Length Offset 1
Diameter Off... 1
= edited
= disabled
Tool Name Dia. Cor. rad. Length
Select library tool...
Create new tool...
Edit tool...
Get aggregate...
Tool manager... (ROUTER_INCH)
View
Arrange tools
Re-initialize feeds & speeds
Feed speed calculator...
Select library tool.
To batch
Tool dia: 0.5
Corner radius: 0.0
Tool name: 1/2 STRAIGHT BIT
Tool #: 1
Len. offset: 1
Head #: -1
Dia. offset: 1
Spindle direction: CW
Feed rate: 200.0
Spindle speed: 18000
FPT: 0.0056
SFM 2356.0209
Plunge rate: 100.0
Retract rate: 100.0
Force tool change
Rapid Retract
Comment

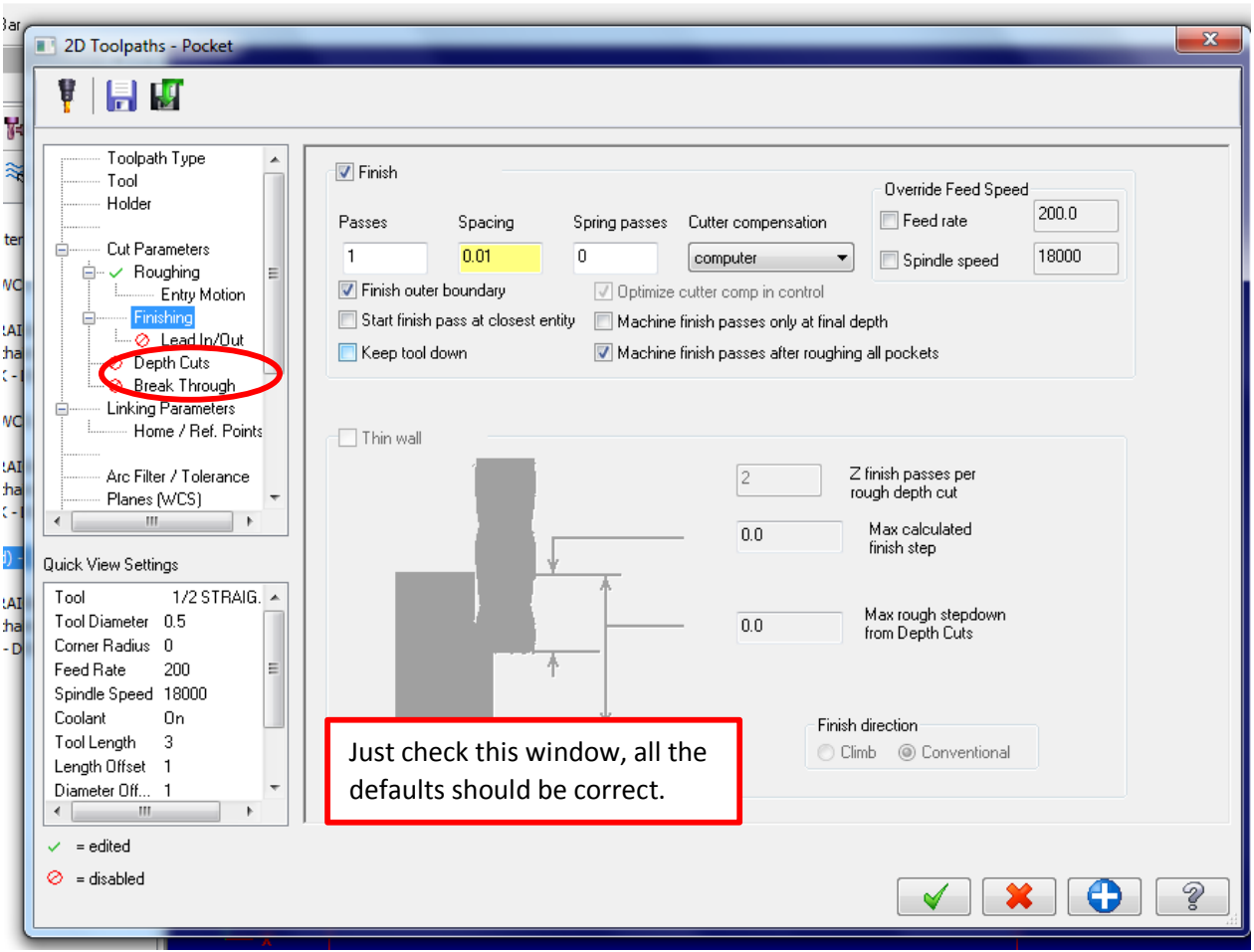

Ribbon Bar
Tool Selection - C:\Users\Public\Documents\shared mcamx6\ROUTER\TOOLS\ROUTER_INCH.TOOLS-6
C:\Users\...\ROUTER_INCH.TOOLS-6
Tool Name Dia. Cor. rad. Length # Flutes Type Rad. Type
1 Marmor's bowl bit 1.... 0.25 0.5 4 Bul... Corner
1 Marmor's bowl bit 1.... 0.25 0.5 4 Bul... Corner
229 1/32 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
237 3/8 STRAIGHT BIT 0.... 0.0 2.0 2 Str... None
239 1/2 STRAIGHT BIT 0.5 0.0 2.0 2 Str... None
240 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
59 of 230 tools
Select library tool...
Filter Active
Filter...

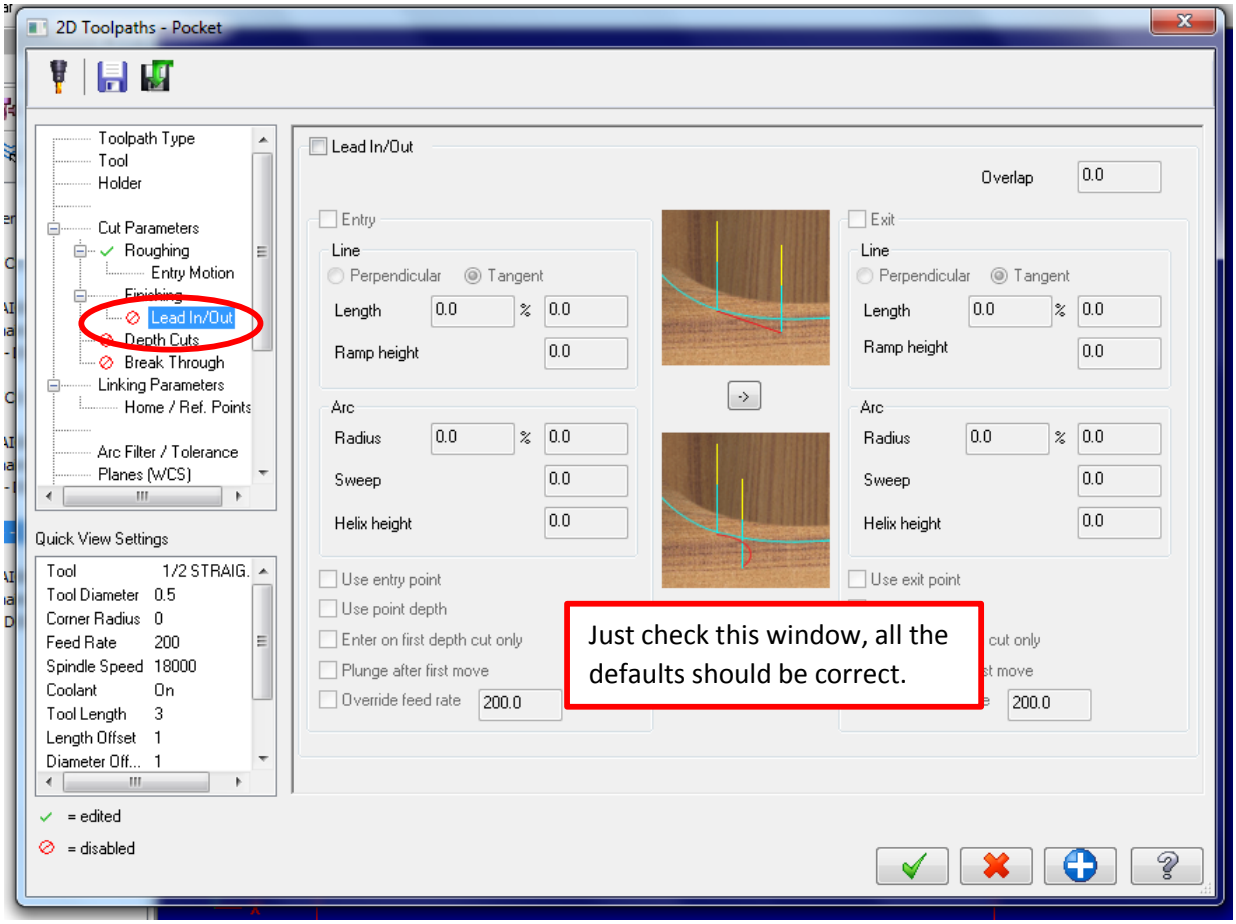

Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Coolant
Quick View Settings
Tool 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 20000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled
Tool Name Dia. Cor. rad. Length
1 1/2 STRA... 0.5 0.0 2.0
You should now see the ½ 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 STRAIGHT BIT
Tool #: 1
Len. offset: 1
Head #: -1
Dia. offset: 1
Spindle direction: CW
Feed rate: 200.0
Spindle speed: 18000
FPT: 0.005
SFM: 2617.801
Plunge rate: 100.0
Retract rate: 100.0
Force tool change
Rapid Retract
Comment

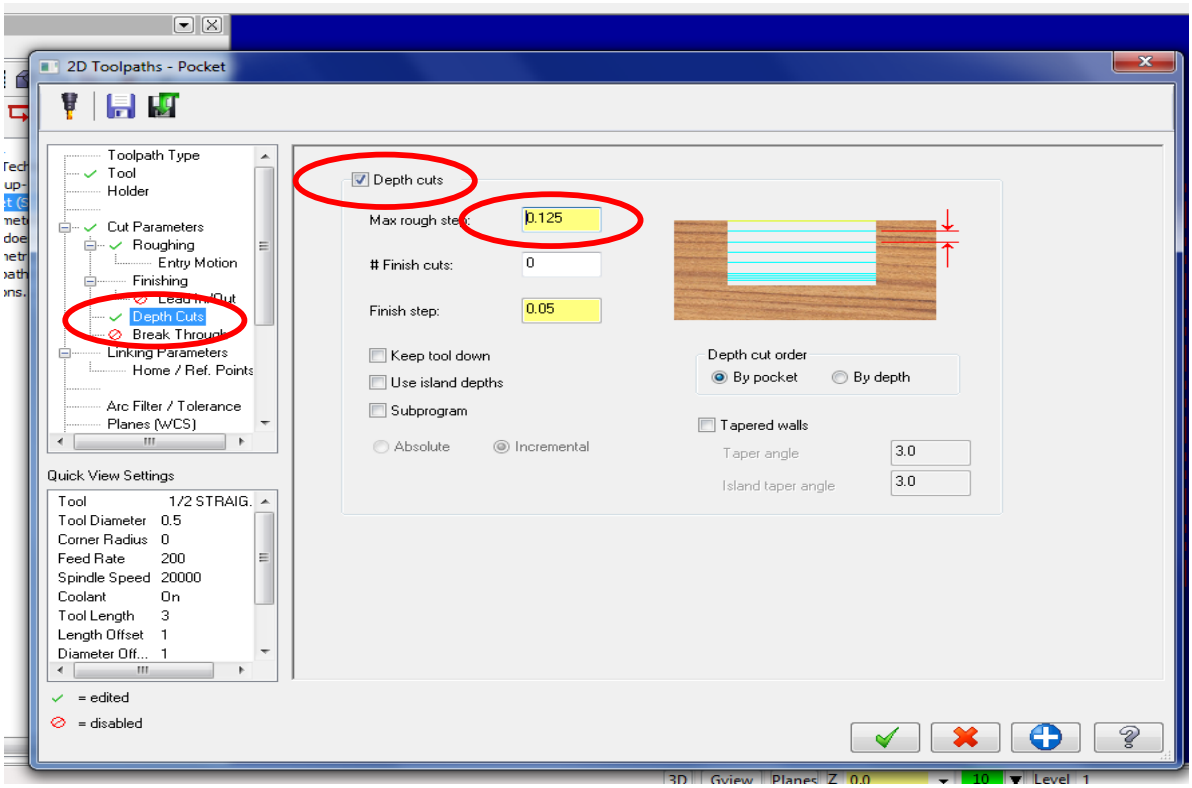

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Machining direction
Climb
Conventional
Pocket type
Standard
Tip comp
Tip
Roll cutter around corners
Sharp
Linearization tolerance
0.001
Create additional finish operation
Just check this window, all the defaults should be correct.
Stock to leave on walls
0.0
Stock to leave on floors
0.0
Quick View Settings
Tool 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled

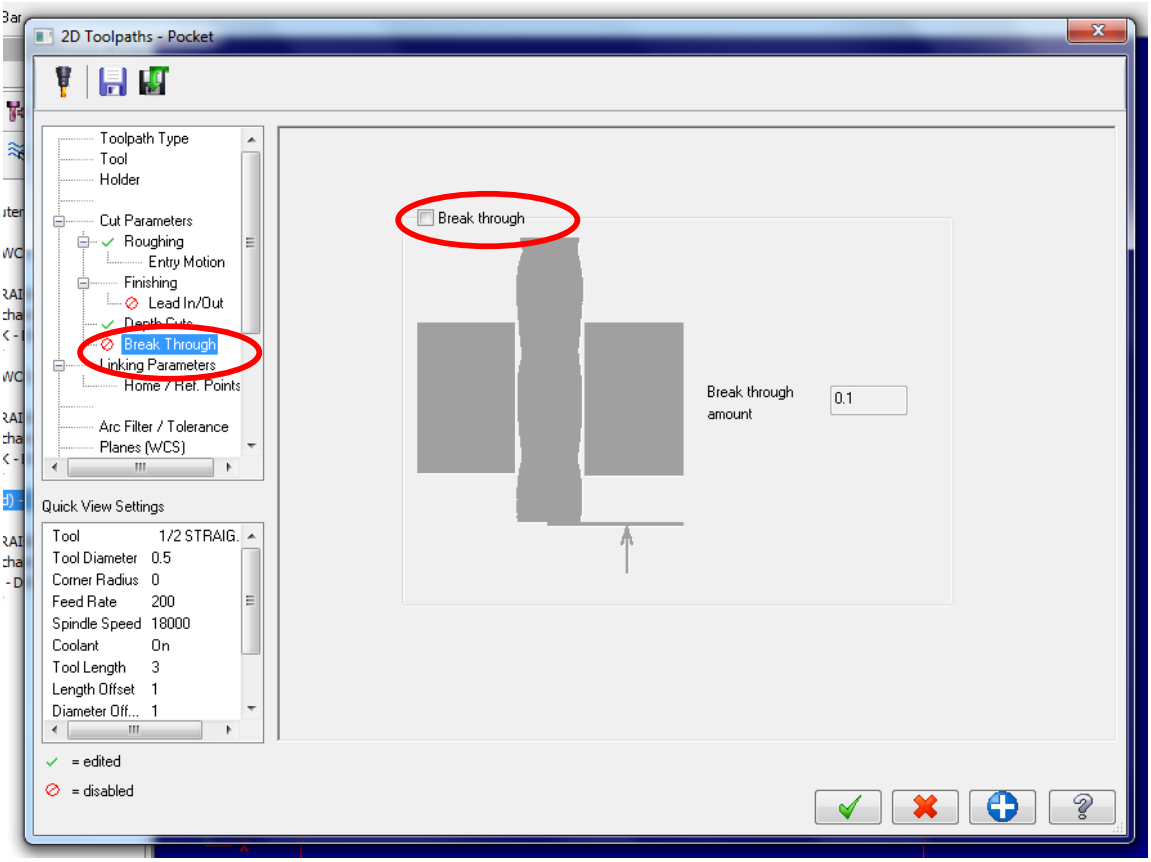

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Rough
Cutting method: Zigzag
Zigzag
Constant Overlap Spiral
Parallel Spiral
Parallel Spiral, Clean Corners
Morph Spiral
High Speed
One Way
Stepover percentage
50.0
Minimize tool burial
Tolerance for remachining and constant overlap
Stepover distance
0.25
Spiral inside to outside
5.0
%
0.025
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 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 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 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled
Off
Ramp
Helix
Minimum radius 50.0 % 0.25
Maximum radius 100.0 % 0.5
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
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.

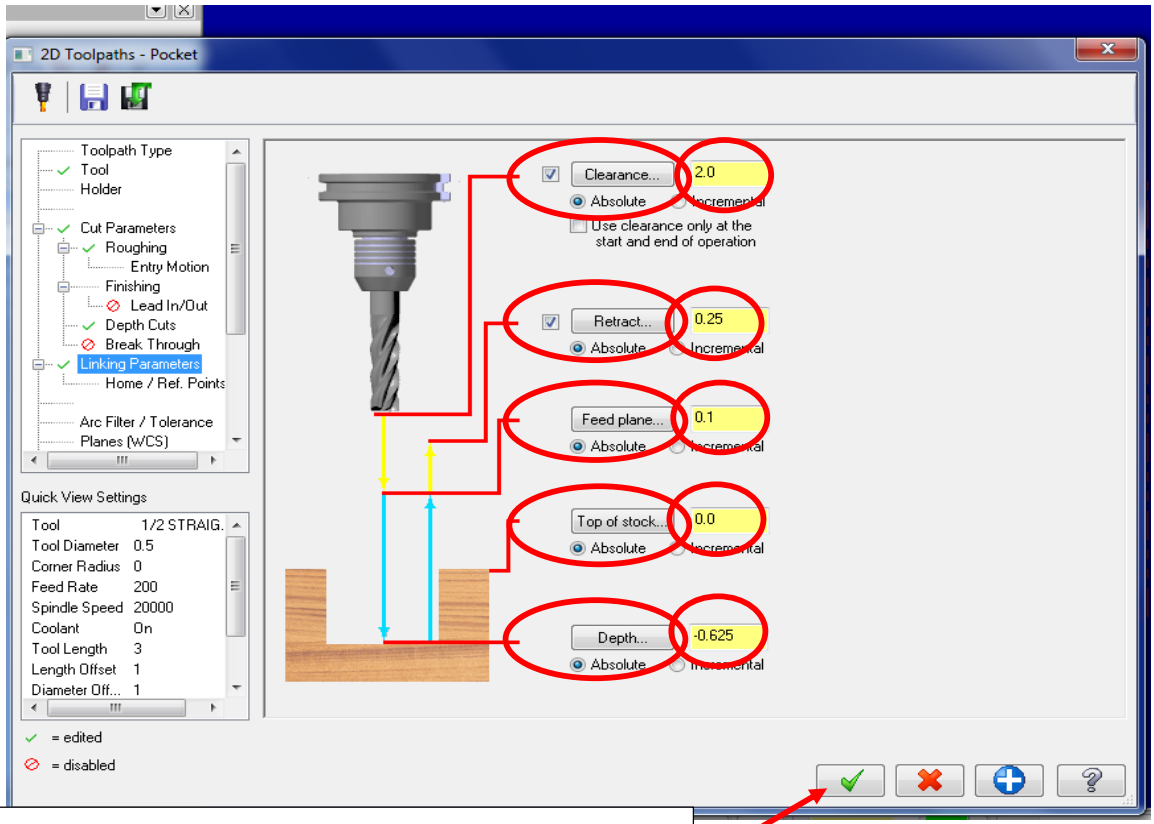

2D Toolpaths - Pocket
Toolpath Type
Tool
Holder
Cut Parameters
Roughing
Entry Motion
Finishing
Lead In/Out
Depth Cuts
Break Through
Linking Parameters
Home / Ref. Points
Arc Filter / Tolerance
Planes (WCS)
Quick View Settings
Tool 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled
Finish
Passes
Spacing
Spring passes
Cutter compensation
1
0.01
0
computer
Override Feed Speed
Feed rate 200.0
Spindle speed 18000
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.


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 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 18000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled
Lead In/Out
Overlap 0.0
Entry
Line
Perpendicular
Tangent
Length 0.0 % 0.0
Ramp height 0.0
Arc
Radius 0.0 % 0.0
Sweep 0.0
Helix height 0.0
Use entry point
Use point depth
Enter on first depth cut only
Plunge after first move
Override feed rate 200.0
Exit
Line
Perpendicular
Tangent
Length 0.0 % 0.0
Ramp height 0.0
Arc
Radius 0.0 % 0.0
Sweep 0.0
Helix height 0.0
Use exit point
cut only
st move
200.0
Just check this window, all the defaults should be correct.


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)
Quick View Settings
Tool 1/2 STRAIG.
Tool Diameter 0.5
Corner Radius 0
Feed Rate 200
Spindle Speed 20000
Coolant On
Tool Length 3
Length Offset 1
Diameter Off... 1
= edited
= disabled
Depth cuts
Max rough step 0.125
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

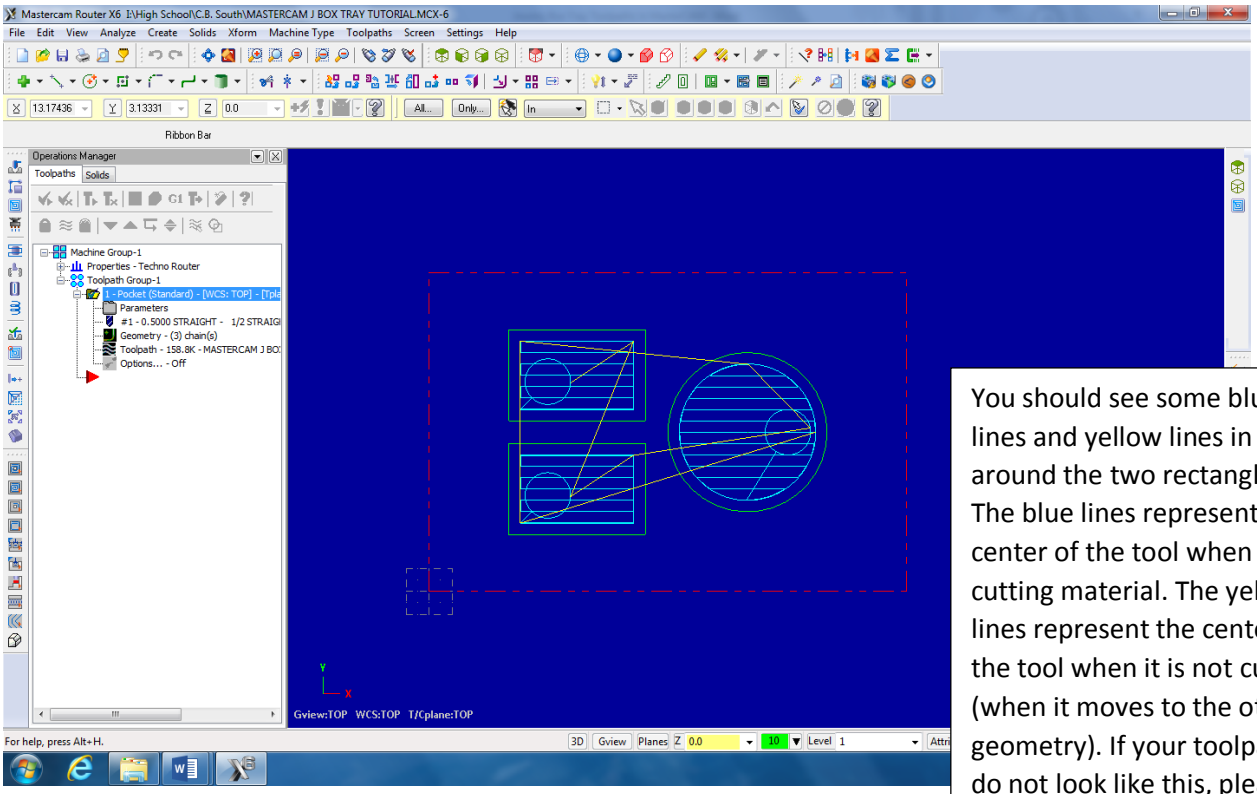


To set the depth of the cut, please enter the below values. In the Linking Parameters tab. Notice that all the values are **"Absolute" and the depth is a -0.625.** This will indicate the depth of each of my pocket cuts. -0.625 = 5/8" which will leave me with an 1/8" thick bottom of my tray since my stock is ¾" thick.



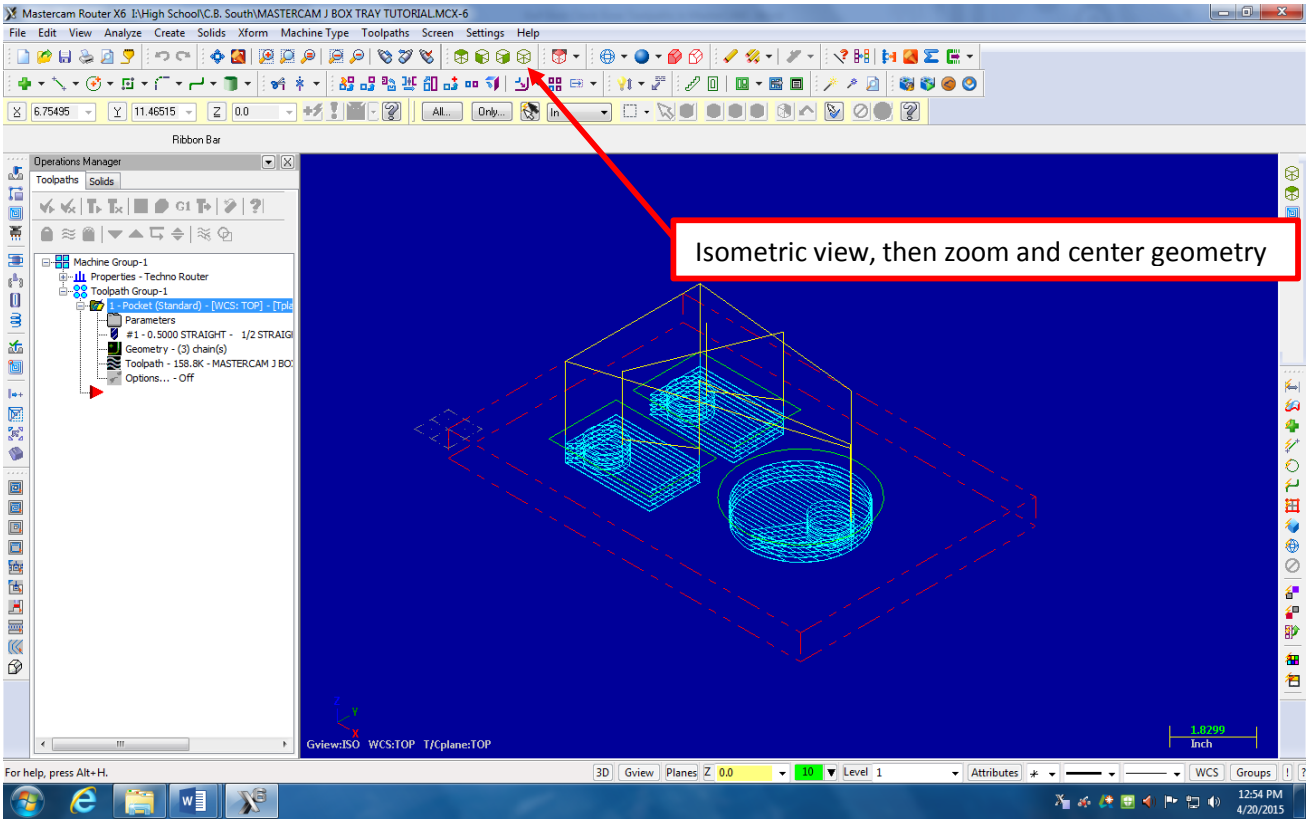
Now we can finally hit the OK check to enter all of these values and MasterCAM will draw the pocket toolpaths.

Resulting In:

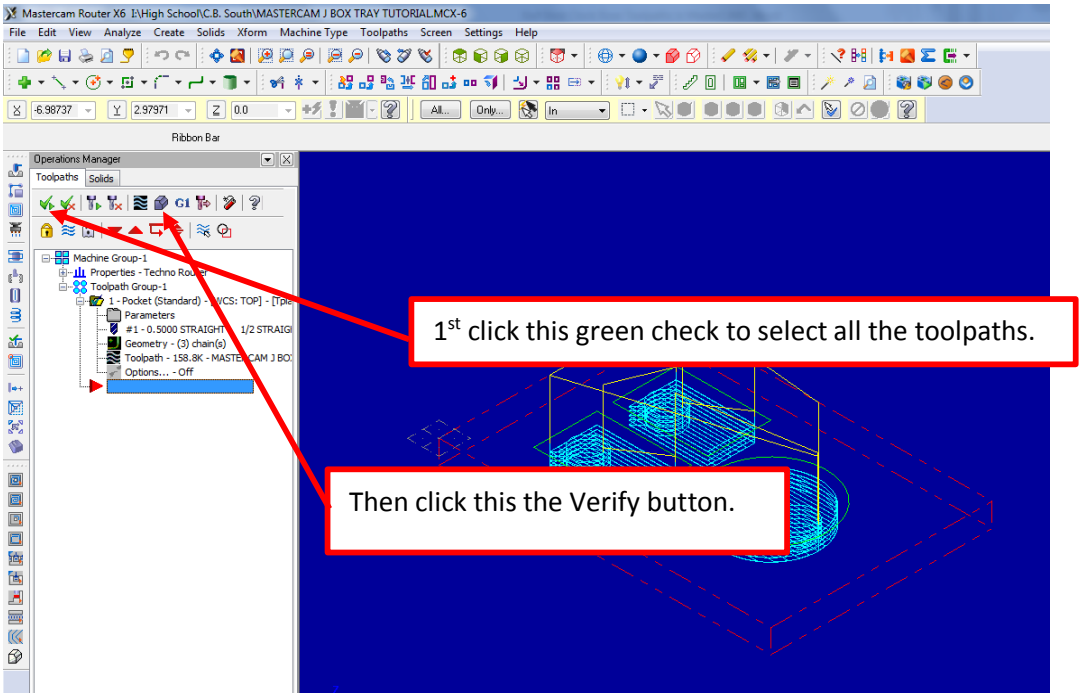


You should see some blue lines and yellow lines in and around the two rectangles. The blue lines represent the center of the tool when it is cutting material. The yellow lines represent the center of the tool when it is not cutting (when it moves to the other geometry). If your toolpaths do not look like this, please get assistance.

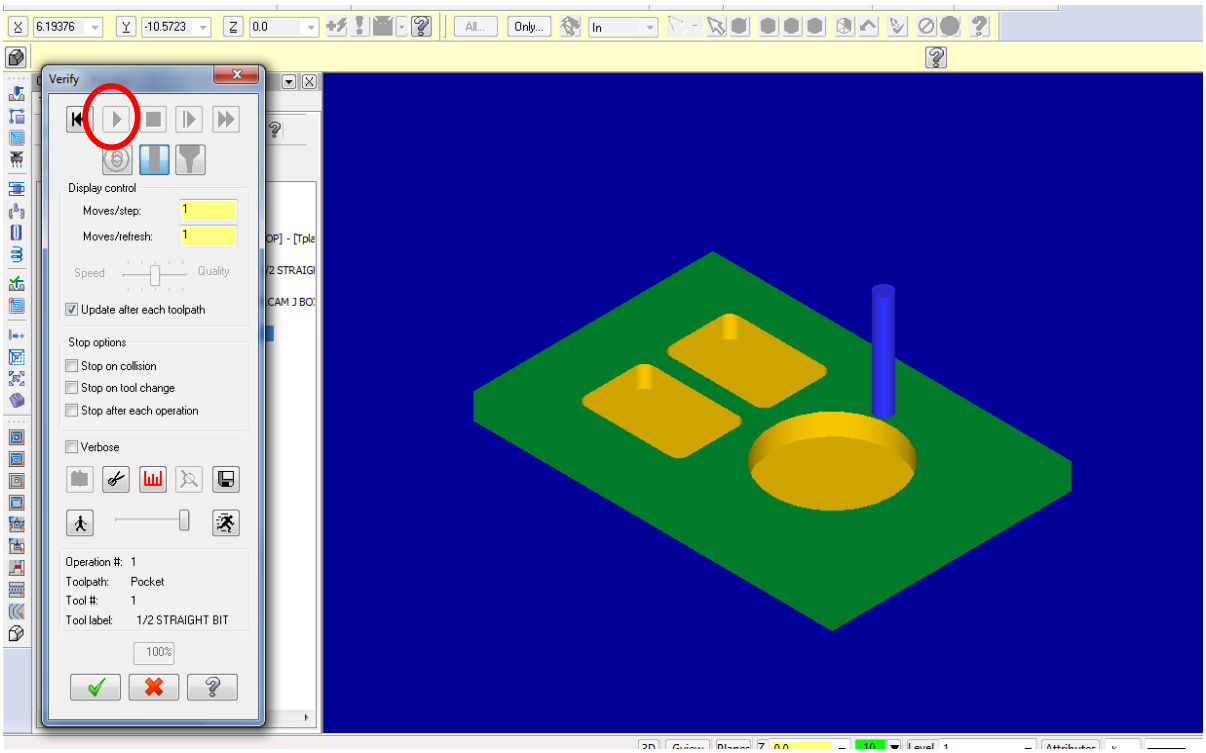
For the next procedure we want to verify the toolpath, basically we are going to virtually cut the piece on the computer. So we are going to look at the geometry and toolpaths in a 3D view so we can see what is going to happen better. Go to an isometric view, zoom in\out, and center the work piece so it looks something like what is below.



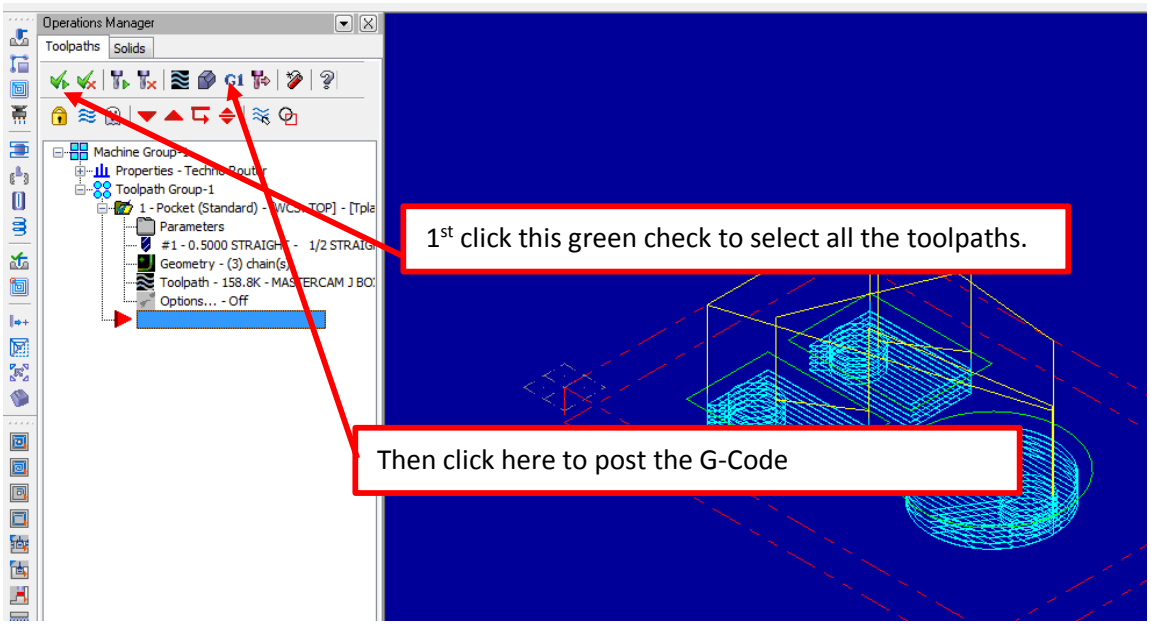
To verify the toolpaths and virtually machine the piece do the following:

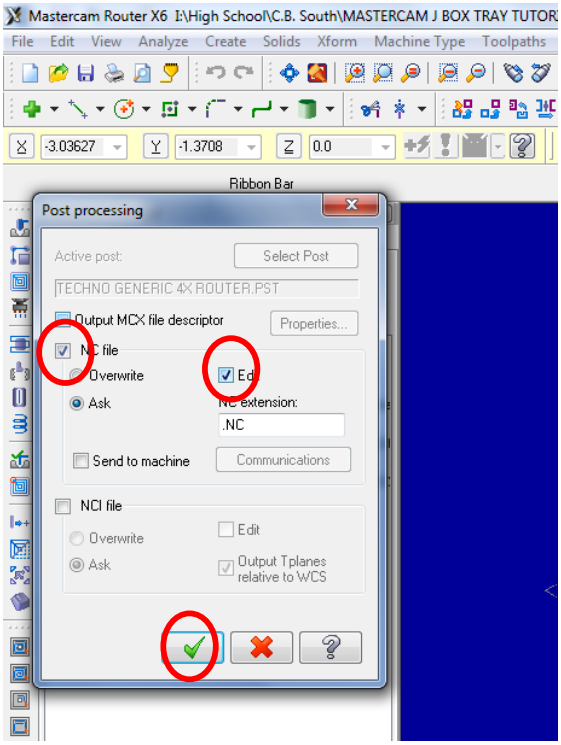


To verify, click the play button, and your work piece should look like below. Please show your instructor to receive credit.



After you have verified the toolpaths you are ready to post the G-Code. This code is was the CNC router reads and we need to post it as an NC (Numerical Code) file so that we can pull it up at the CNC router under the Techno Interface.





This box will appear. The defaults should be correct. Check to make sure they are and then hit the green check mark.

Then save your file to your I-Drive so that we can access it when we go to the CNC router.