

Tutorial: Making a coaster on the CNC router

Insert image of final product

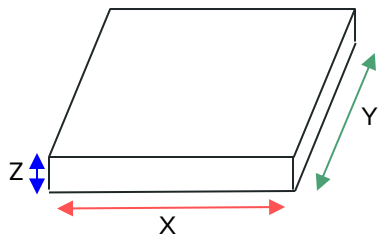
Overview

In this tutorial, we will go over how to:

- Select proper stock and tooling
- 3D model a coaster
- Set up engrave, contour, and pocket toolpaths
- Safely operate the CNC router

Stock Selection

Step 1: Select stock



In Fusion, X refers to the left-right axis, Y refers to the front-back axis, and Z refers to the up-down axis

The CNC router can cut a wide variety of materials, but we mainly stick to the following:

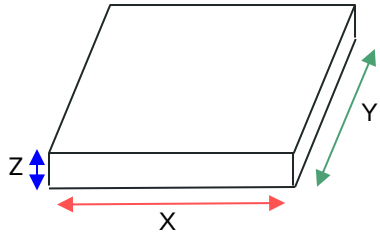
- Acrylic
- Plywood
- MDF
- Hardwood/softwood lumber

When selecting your stock, keep in mind the dimensions of the bed, which is 2' x 3'. Make sure your stock is:

- $X < 2'$ (24" or 610mm)
- $Y < 3'$ (36" or 915mm)
- $Z < 1"$ (25.4mm), although this dimension is more flexible

For our project, pick out a nice piece of scrap plywood that's at least ½" thick.

Step 2: Measure Stock



X: Width
Y: Depth
Z: Thickness

For our purposes, we will be basing our model off of our stock to simplify the process. We need to take precise measurements of our stock.

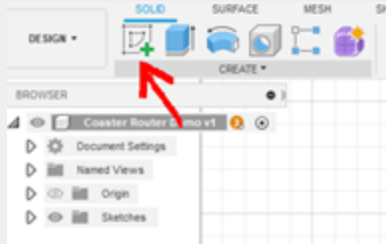
- Use a ruler or tape measure to measure the width (X) and depth (Y) of the stock
- Use calipers to measure the thickness (Z) of your stock, as this dimension will require the highest degree of precision

Be sure to write these measurements down somewhere for future reference.

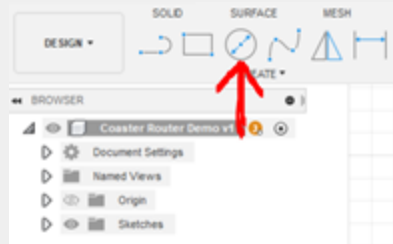
3D Modeling in Fusion 360

Step 3: Create Sketch

Note: For beginner Fusion users, I will put a little bit of extra instruction on the left side of the screen. Disregard if you already have experience with the software

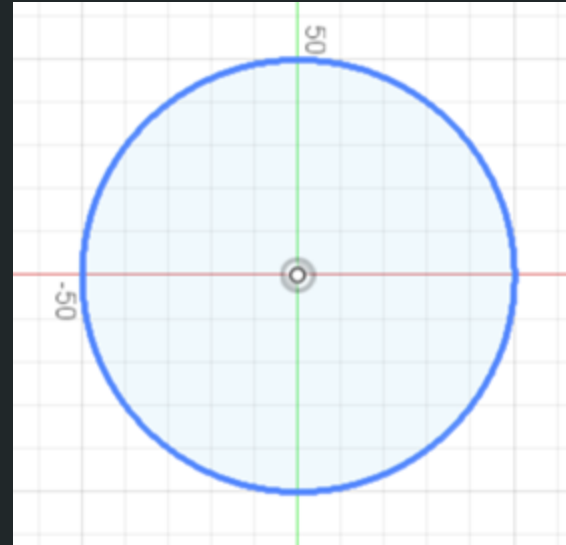


Click "create sketch"



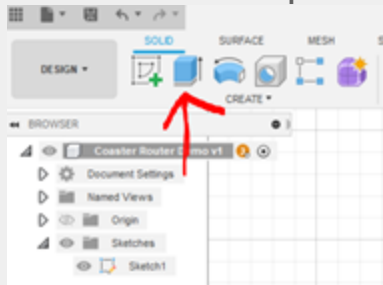
Click "center diameter circle"

In Fusion 360, we start with a two-dimensional sketch. Most coasters fit inside of a 4" x 4" (100mm x 100mm) square, so keep that in mind as we create our outline. A 4" (100mm) circle will do the trick here.



Step 4: Extrude Sketch

Hint: To extrude a sketch, you can just press “E” on your keyboard. The extrude function can also be found in the top left of the toolbar.

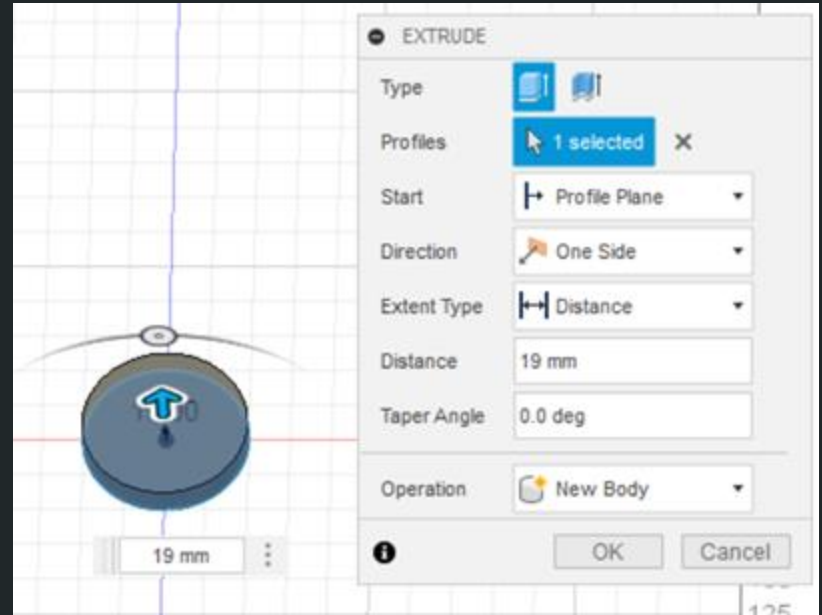


Click “extrude”



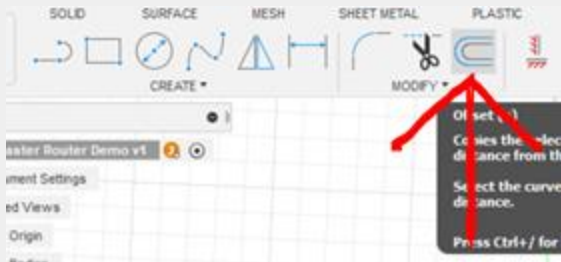
Select your sketch

Next, we will extrude our sketch to create a 3D body. Go get those stock measurements we took before; you want to **extrude to the exact thickness (Z) of your stock.**



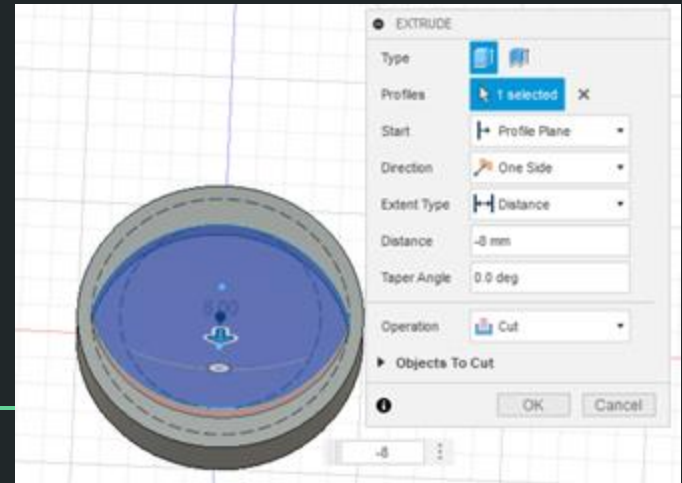
Step 5: Create Pocket

Hint: You can use the offset tool to make a concentric circle that is a set amount larger or smaller than a selected circle.



Within the sketch environment, click “offset” and select the outline circle

Normal, flat coasters can be slippery and downright unsafe. Let's create a little pocket for our glass of choice to sit nicely in. To do this, create a sketch on the top of the body you just created. **Make a circle (or any other shape) that is about 10-20mm (1/2" or so) smaller than the outline.** Finally, extrude your new sketch *negatively* to cut; I am going to **extrude -8mm.**

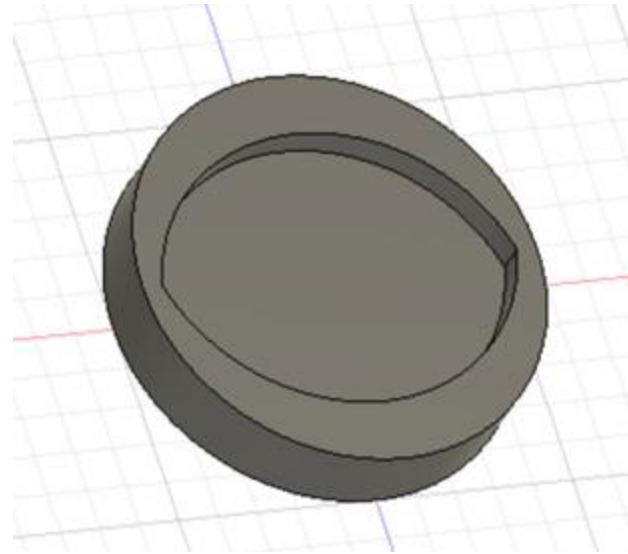


STATUS UPDATE

Let's take a minute to make sure we're all caught up.
We have:

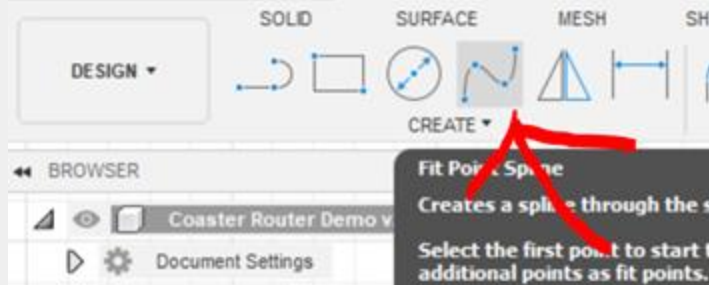
- Created an outline sketch
- Extruded outline into a disk with the same thickness as our stock
- Created a pocket sketch
- Extruded negatively (cut) about halfway into the disk

Your part should look something like this (see image), although feel free to take some creative liberties.



Step 6: Create Engraving Sketch

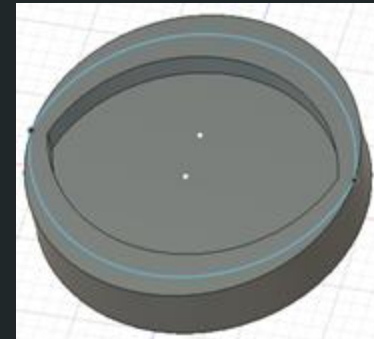
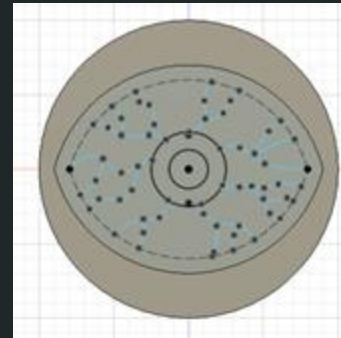
Hint: Try out the fit-point spline function in the sketch dropdown for free form sketching.



I recommend adding "fit point spline" to your toolbar

Finally, let's make this coaster something special. The router is great at engraving; all you have to do is add sketch lines to the top side of your part. Feel free to add engravings to the lowered region or the raised lip.

For the lowered region, **provide at least 5mm clearance from the lip** or else the engraving bit will take chunks out of the lip.



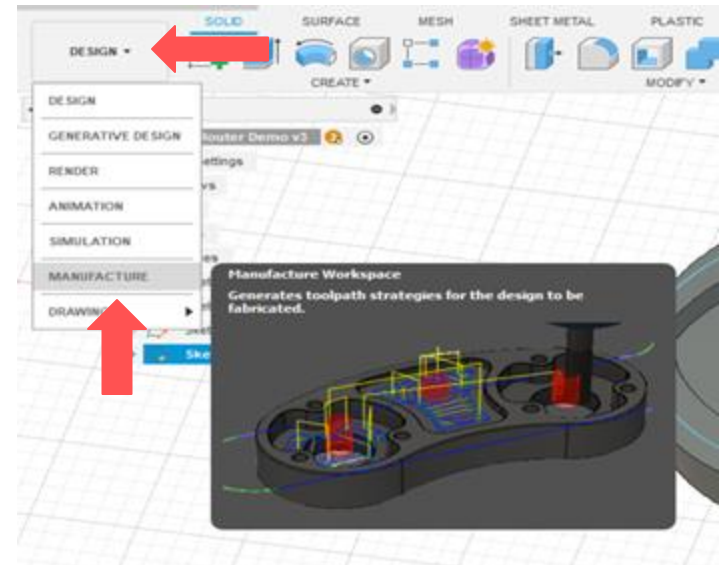
Creating Toolpaths in Fusion 360

Switching Workspaces...

One of the best things about Fusion 360 is that you can design your 3D models and create CAM toolpaths in the same application. In order to do this, though, we need to switch over to the “Manufacturing” workspace.

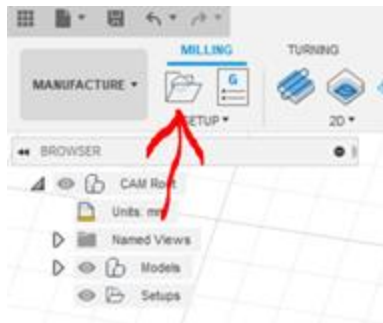
Click on “DESIGN” in the top-left corner. Then, click on “MANUFACTURE”, which should be the second to last option.

Note: In older versions of Fusion, this might be called “CAM”.



Step 7: Create CAM Setup

Once you're in the Manufacture environment, click on the little folder in the top left

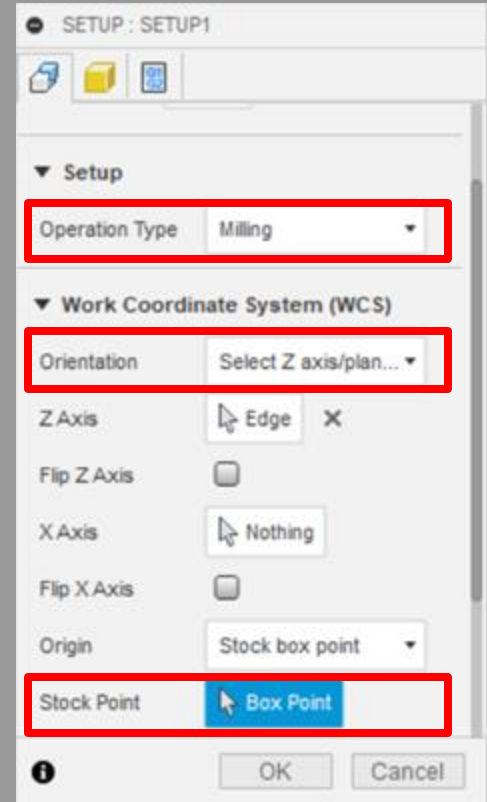


In order to create toolpaths, you need to let Fusion know what size your stock is and what the orientation of your model is.

The next few slides will provide step-by-step detail on how to create a CAM setup.

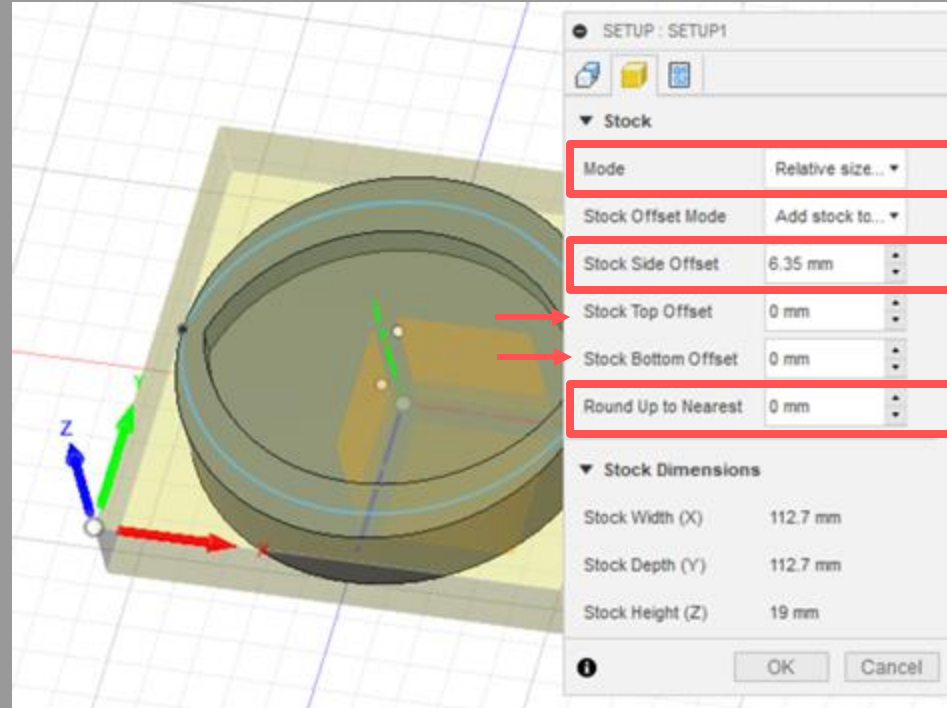
CAM Setup: Setting proper axes

1. Make sure the operation type is set to “milling”.
2. Click the Orientation dropdown and click “Select Z axis/plan and X axis”. You can now select any line that is parallel to the direction you’d like to set an axis to. Note that for the mill, the X axis is left/right, Y axis is front/back, and Z axis is up/down.
3. Now, pick your origin by clicking “Box Point” and selecting the **top, front, left corner**. This will be the easiest point to set your zeroes.



CAM Setup: Stock size

1. Click on the yellow box icon to move to the next tab.
2. Change the mode to “Relative size box”.
3. Set “Stock Side Offset” to the size of the bit we will be using; in our case, this will be ¼” (6.35mm)
4. Set both stock top and bottom offset to 0. Your part should be the exact height as the stock
5. Set “Round Up to Nearest” to 0.
6. You’re all set. Click “OK”.



Step 8: Create Pocket Toolpath



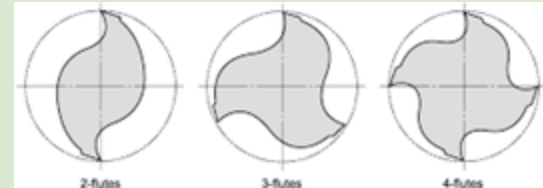
For a simple pocket with flat bottom and sides, **click on the 2D dropdown and select “2D Pocket”**. The next few slides will detail the process for creating a pocket toolpath.

Pocket Toolpath: Tool selection



For most milling/routing, a **flat end mill** is most appropriate. To optimize cutting time and ability to reach tight contours, let's use a 1/4" bit.

1. Click "select" to go to the tool library. Make sure you are in the Fusion 360 library, not your local library.
2. Filter your results by selecting the following filters:
 - a. Type: Flat end mill
 - b. Diameter: enter "1/4 in"
3. There should only be one tool available. Select it and press enter.



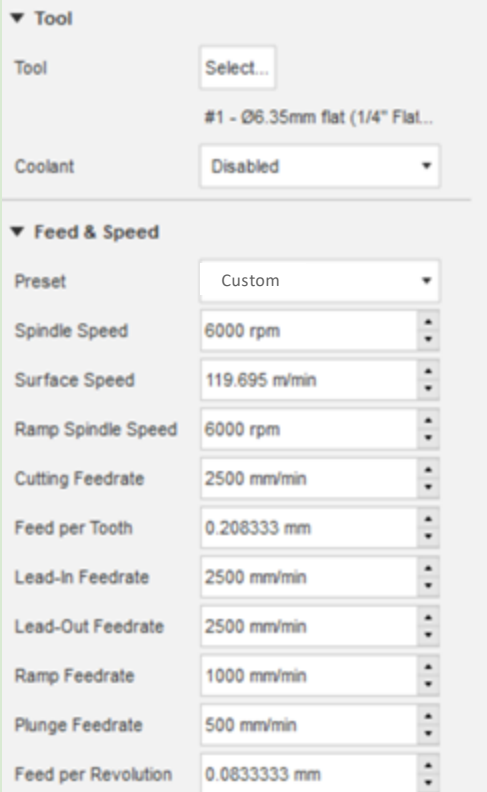
Now, after you've selected your tool, you need to go back and edit the number of flutes. Fusion defaults to 3 flutes, but most of our tools are two flutes.

1. Click on tool select again, right click on your selected tool (now in your local library), and click "edit tool". Go to the "cutter" tab and change flutes to 2.

Pocket Toolpath: Speeds and feeds

Speeds and feeds are the secret to any good machining recipe. It is challenging to develop a full understand of what each term means without actually running the machine. For now, just make sure your values line up with the image on the right. If you hold your mouse over any of the boxes, Fusion provides sufficient descriptions.

Note: the router's spindle speed is actually selected on the machine itself. We keep it at 6000 RPM, and there is rarely a reason to change it. The machine maximum feedrate is 2500mm/min, so don't try to go any faster than that.



The image shows a screenshot of the Fusion 360 software interface, specifically the 'Tool' and 'Feed & Speed' sections. The 'Tool' section is expanded, showing a 'Tool' dropdown menu set to 'Select...' and a 'Coolant' dropdown menu set to 'Disabled'. Below this, the 'Feed & Speed' section is also expanded, showing a 'Preset' dropdown menu set to 'Custom'. The 'Feed & Speed' section contains several input fields with numerical values and up/down arrows for adjustment:

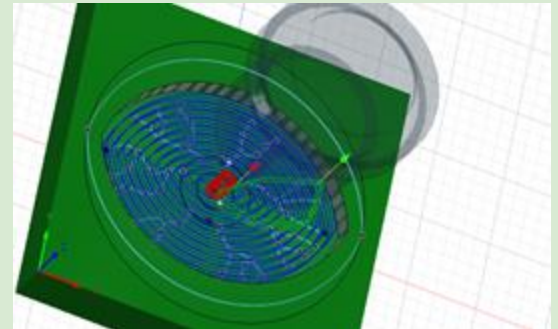
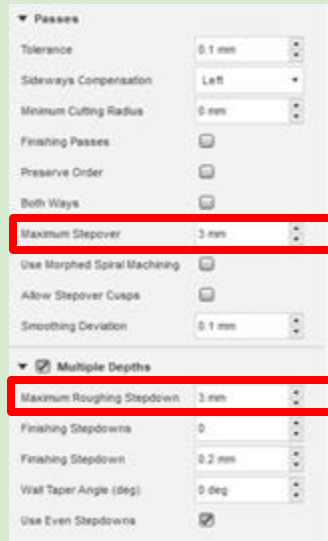
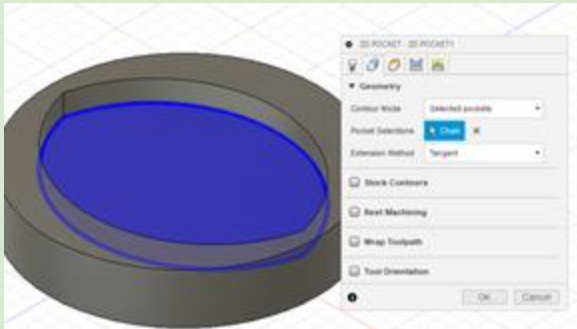
Parameter	Value
Spindle Speed	6000 rpm
Surface Speed	119.695 m/min
Ramp Spindle Speed	6000 rpm
Cutting Feedrate	2500 mm/min
Feed per Tooth	0.208333 mm
Lead-In Feedrate	2500 mm/min
Lead-Out Feedrate	2500 mm/min
Ramp Feedrate	1000 mm/min
Plunge Feedrate	500 mm/min
Feed per Revolution	0.0833333 mm

Pocket Toolpath: The other tabs

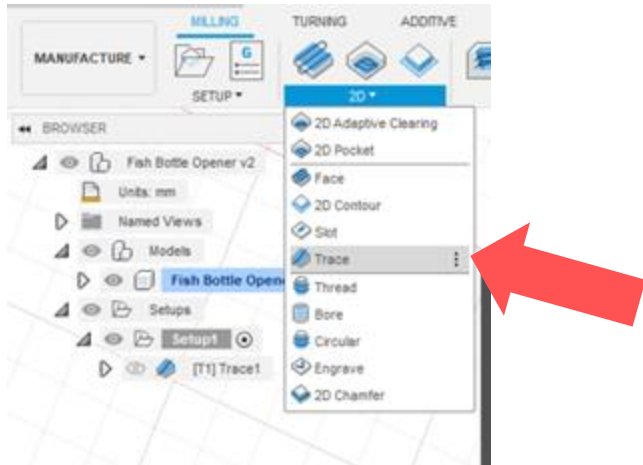
After the first tab, toolpath generation usually requires few additional adjustments.

1. In the **geometry** tab, select the bottom face of your pocket.
2. Everything in the **heights** tab should be fine.
3. In the **passes** tab, set “Maximum Stepover” to 3 mm. Check the “Multiple Depths” box and set “Maximum Roughing Stepdown” to 3 mm. These should always be less than half the diameter of the bit. Uncheck “Stock to Leave”.
4. Everything in the **linking** tab should be fine.

Click enter to generate your trace.



Step 9: Create Engrave Toolpath



To engrave sketch lines, we are actually going to use the “trace” function. **Click on the 2D dropdown and select “trace”**. The next couple of slides will provide a detailed guide for setting up an engraving toolpath using trace.

Engrave (Trace) Toolpath: Tool selection

A great way to engrave is to mill using a V-bit, which is pointed like a drill bit. This allows very fine engraving.

1. Click “select” to go to the tool library. Make sure you are in the Fusion 360 library, not your local library.
2. Filter your results by selecting the following filters:
 - a. Type: Spot Drill
 - b. Diameter: enter “¼ in”
3. There should only be one tool available. Select it and press enter.



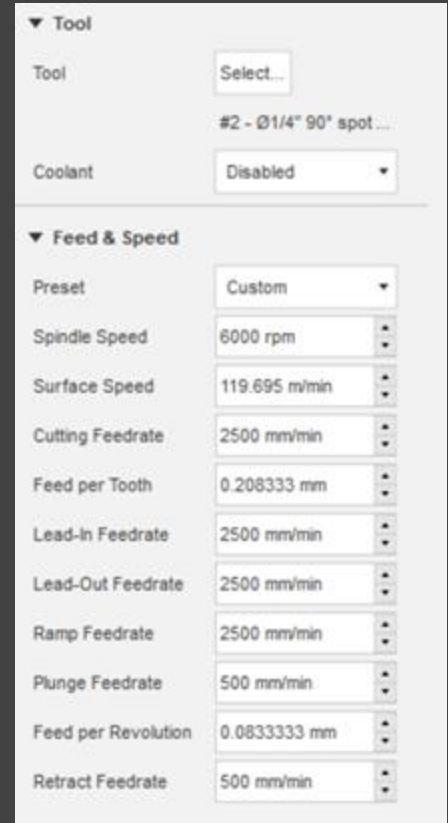
Now, after you’ve selected your tool, you need to go back and edit the number of flutes. Fusion defaults to 3 flutes, but most of our tools are two flutes.

1. Click on tool select again, right click on your selected tool (now in your local library), and click “edit tool”. Go to the “cutter” tab and change flutes to 2.

Engrave (Trace) Toolpath: Speeds and feeds

Speeds and feeds are the secret to any good machining recipe. It is challenging to develop a full understand of what each term means without actually running the machine. For now, just make sure your values line up with the image on the right. If you hold your mouse over any of the boxes, Fusion provides sufficient descriptions.

Note: the router's spindle speed is actually selected on the machine itself. We keep it at 6000 RPM, and there is rarely a reason to change it. The machine maximum feedrate is 2500mm/min, so don't try to go any faster than that.



The image shows a screenshot of the Fusion 360 software interface, specifically the 'Tool' and 'Feed & Speed' sections. The 'Tool' section is expanded, showing a 'Tool' dropdown menu set to 'Select...', a tool description '#2 - Ø1/4" 90° spot...', and a 'Coolant' dropdown menu set to 'Disabled'. The 'Feed & Speed' section is also expanded, showing a 'Preset' dropdown menu set to 'Custom'. Below this, several parameters are listed with their values and up/down arrows for adjustment:

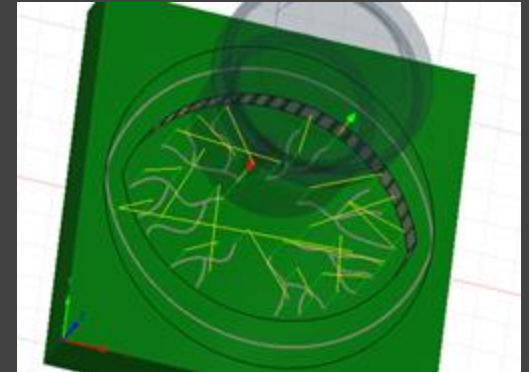
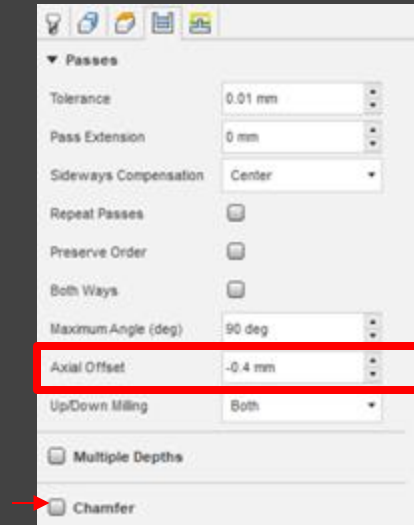
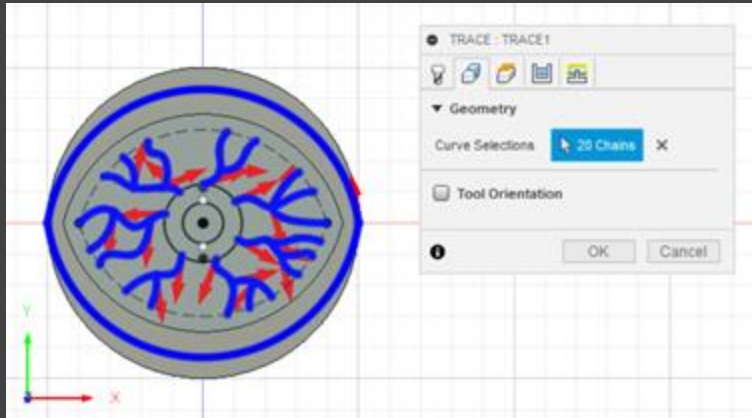
Parameter	Value
Spindle Speed	6000 rpm
Surface Speed	119.695 m/min
Cutting Feedrate	2500 mm/min
Feed per Tooth	0.208333 mm
Lead-In Feedrate	2500 mm/min
Lead-Out Feedrate	2500 mm/min
Ramp Feedrate	2500 mm/min
Plunge Feedrate	500 mm/min
Feed per Revolution	0.0833333 mm
Retract Feedrate	500 mm/min

Engrave (Trace) Toolpath: The other tabs

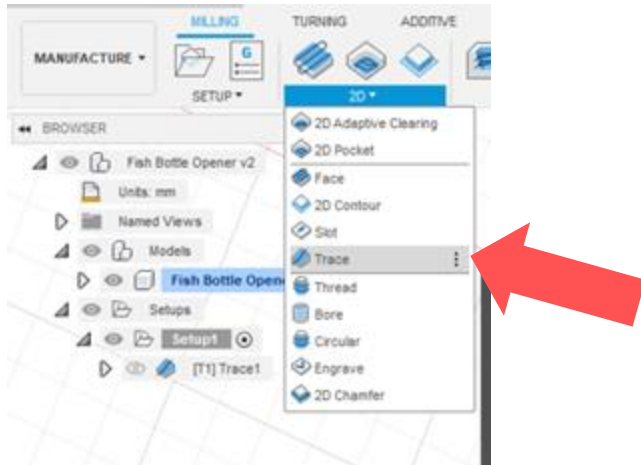
After the first tab, toolpath generation usually requires few additional adjustments.

1. In the **geometry** tab, select all sketch lines you want to engrave.
2. Everything in the **heights** tab should be fine.
3. In the **passes** tab, change the Axial Offset to a depth of your choosing, e.g. -0.4 mm. Uncheck Chamfer. This axial offset sets how deep your V-bit is cutting, which directly correlates with the width of the cut. **Increase depth for thicker engrave lines.**
4. Everything in the **linking** tab should be fine.

Click enter to generate your trace.



Step 10: Create Contour Toolpath



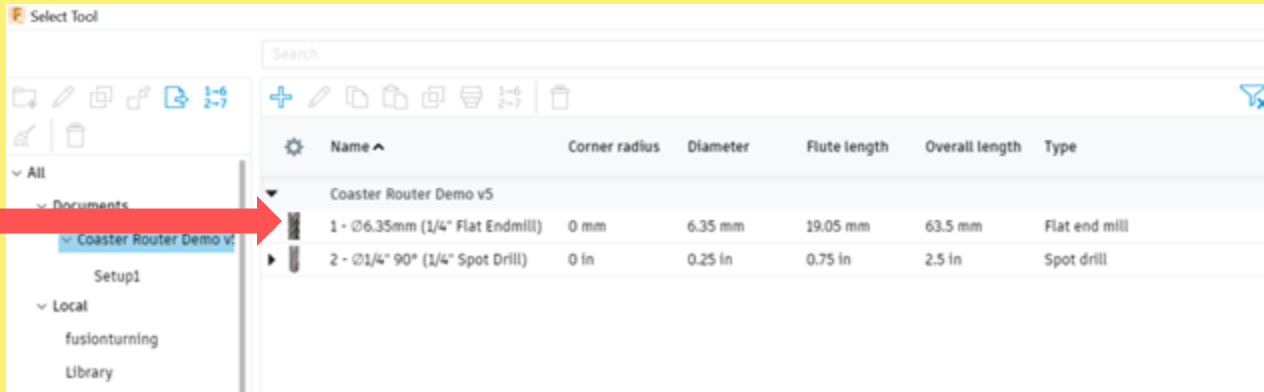
Let's finally cut out our part. We need to find a way to cut the outline of the coaster without it coming loose at the end of the cut. First, **click on the 2D dropdown and select "Contour"**. The next few slides will detail a standard contour toolpath.

Contour Toolpath: Tool selection



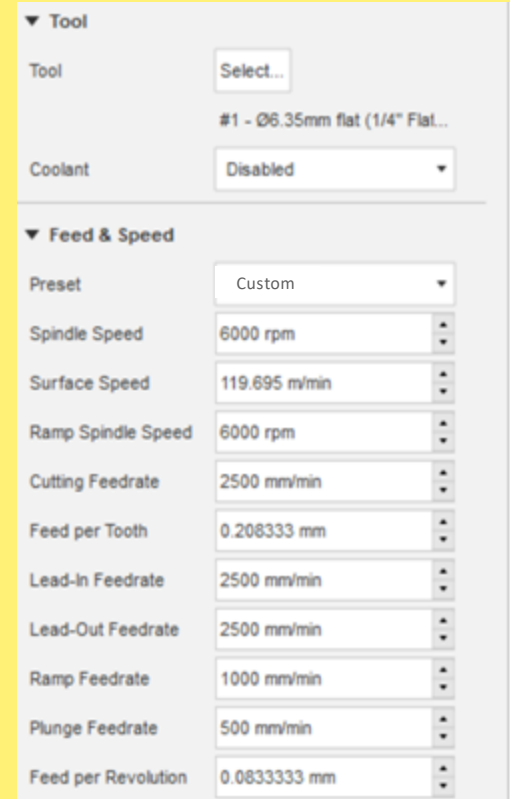
For most milling/routing, a **flat end mill** is most appropriate. To optimize cutting time and ability to reach tight contours, let's use a 1/4" bit.

Luckily, we do not need to go through the whole tool editing process again. Just select the 1/4" flat end mill that's already in your tool library.



Contour Toolpath: Speeds and feeds

Same speeds and feeds as the pocket toolpath.
You may need to re-enter the, so here they are again.



The image shows a software interface for configuring tool and feed & speed parameters. The 'Tool' section is expanded, showing a 'Tool' dropdown menu set to 'Select...', a tool description '#1 - Ø6.35mm flat (1/4" Flat...', and a 'Coolant' dropdown menu set to 'Disabled'. The 'Feed & Speed' section is also expanded, showing a 'Preset' dropdown menu set to 'Custom'. Below this, there are ten input fields for various speed and feed parameters, each with a numeric value and a unit, and a small up/down arrow icon to the right of each field.

▼ Tool	
Tool	Select...
	#1 - Ø6.35mm flat (1/4" Flat...
Coolant	Disabled

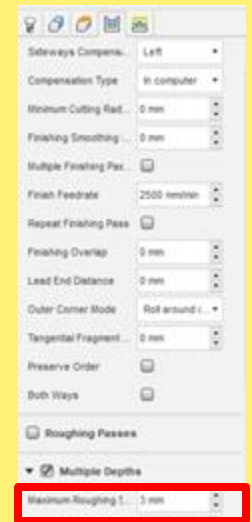
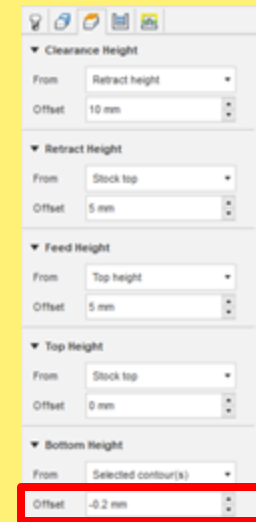
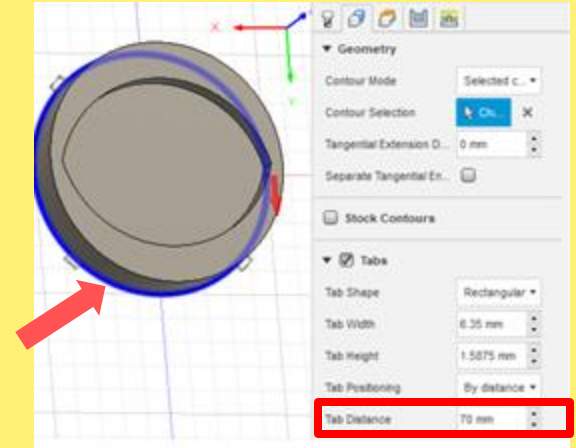
▼ Feed & Speed	
Preset	Custom
Spindle Speed	6000 rpm
Surface Speed	119.695 m/min
Ramp Spindle Speed	6000 rpm
Cutting Feedrate	2500 mm/min
Feed per Tooth	0.208333 mm
Lead-In Feedrate	2500 mm/min
Lead-Out Feedrate	2500 mm/min
Ramp Feedrate	1000 mm/min
Plunge Feedrate	500 mm/min
Feed per Revolution	0.0833333 mm

Pocket Toolpath: The other tabs

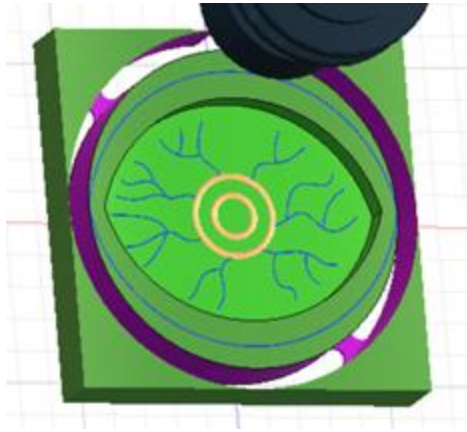
After the first tab, toolpath generation usually requires few additional adjustments.

1. In the **geometry** tab, select the bottom outline of your coaster. Now, check the box for “Tabs” and set the Tab Distance to a value that creates four tabs (should be around 70mm). Tabs are little bits of material that don’t get cut, keeping your part from fully disconnecting after the cut.
2. In the **heights** tab, offset the bottom height about -0.2mm to ensure that the bit cuts all the way through your stock.
3. In the **passes** tab, check the “Multiple Depths” box and set “Maximum Roughing Stepdown” to 3 mm.
4. Everything in the **linking** tab should be fine.

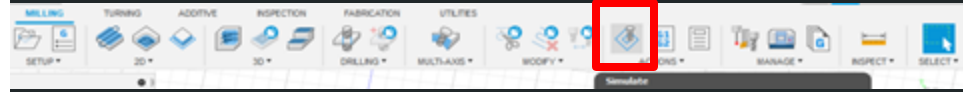
Click enter to generate your trace.



Step 11: Simulate All Toolpaths



Hint: Try unchecking “toolpath” and checking “stock”. The simulation will show you what your stock should like after each toolpath.



Just sit back and watch Fusion do its thing. Click on your setup (which all toolpaths should fall under) and click the Simulate icon at the top of the screen. The simulation checks for errors in your setup and tool selection.

SIMULATE		
Display	Info	Statistics
▼ Statistics		
Machining time	0:04:28	
Machining distance	9.054 m	
Operations	4	
Tool changes	3	

If you click on the “Statistics” tab, you can see the total machining time. While this excludes time spent switching bits, the actual operation of the machine is pretty quick.