

Fusion 360 CAM Guide

Setup

The first thing you are going to do in the Manufacture workspace is create a Setup.

The setup defines two important things,

- the size of your **stock** - the piece of material you are cutting your model out of
- your **origin** - the location of your stock that we are defining as zero.



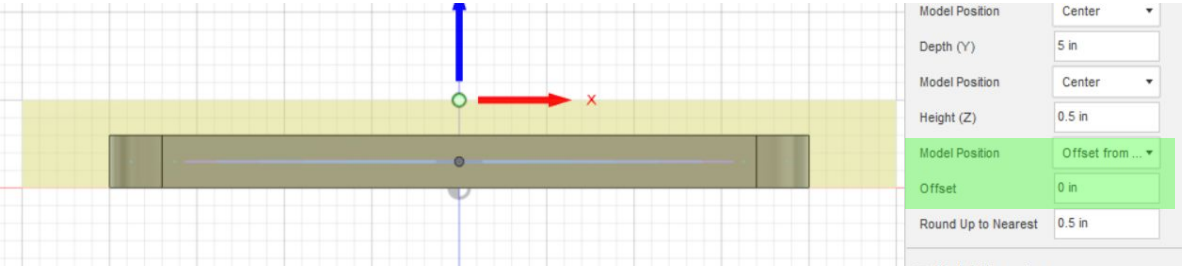
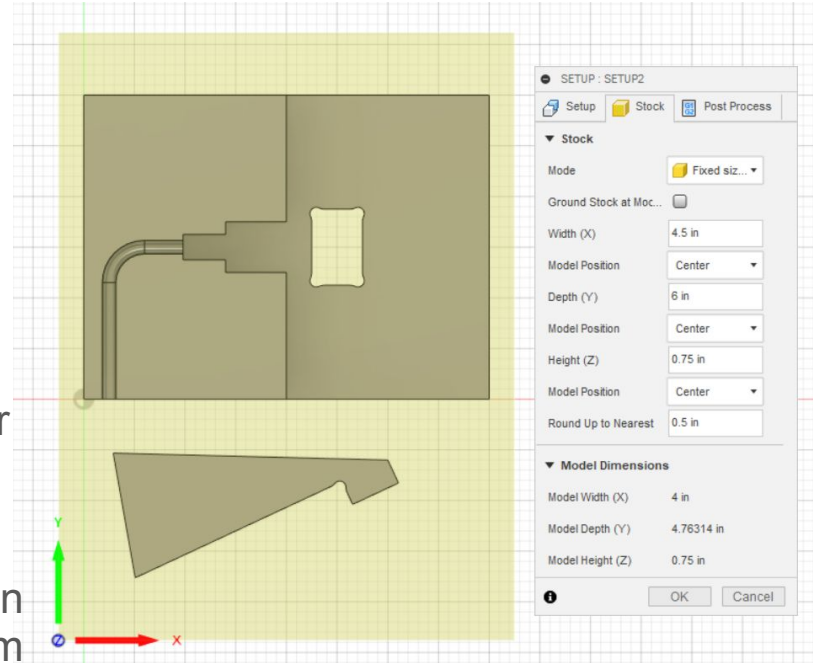
Setup - stock

The stock (piece of material you're cutting from) is represented by the yellow box.

In the Stock tab, for most cases it's best to **change the Mode to Fixed size box**.

Then ensure that the **Width (X)** and **Depth (Y)** of your stock are sized larger than your model.

For the Height (Z) of your stock, make sure it is the thickness of your material. If your model is thinner than your material, typically we want to align it to the bottom of your stock.

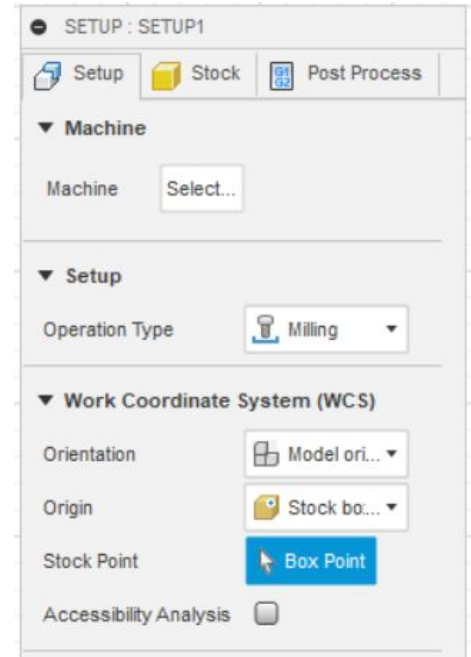
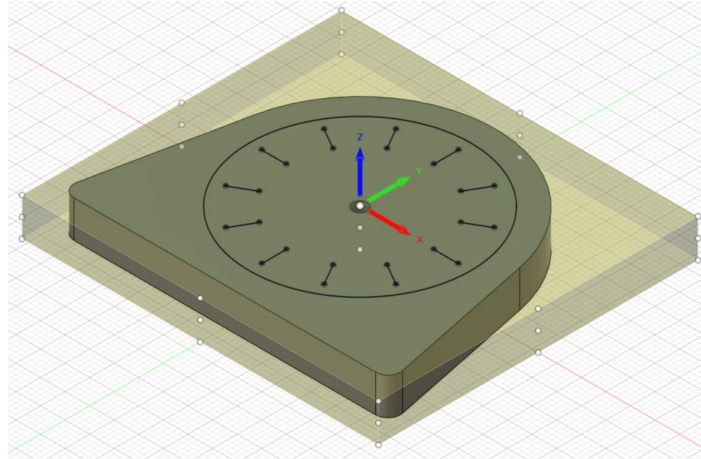
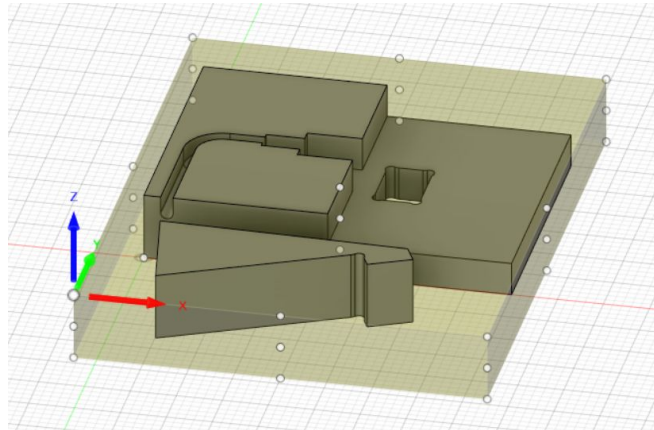


To align it to the bottom, we change Model Position to Offset from Bottom and set the Offset to 0 in.

Setup - origin

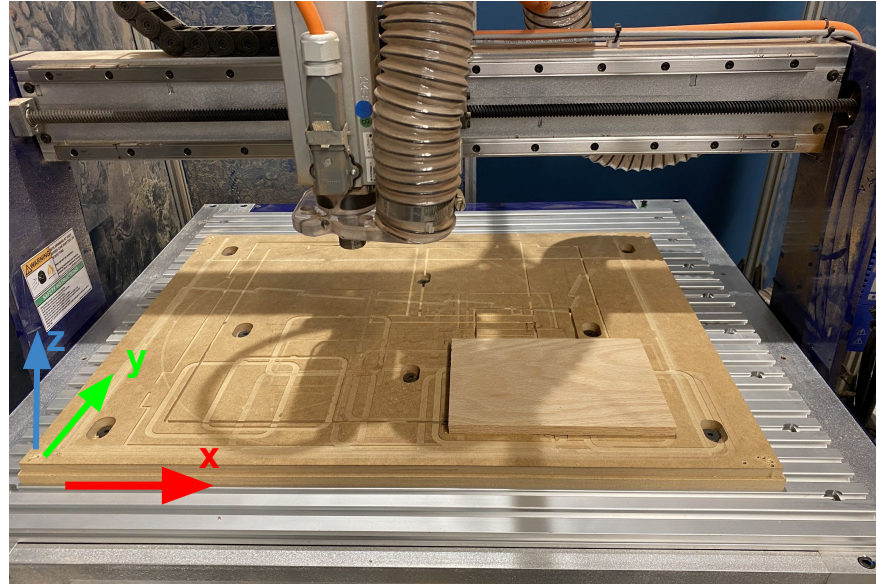
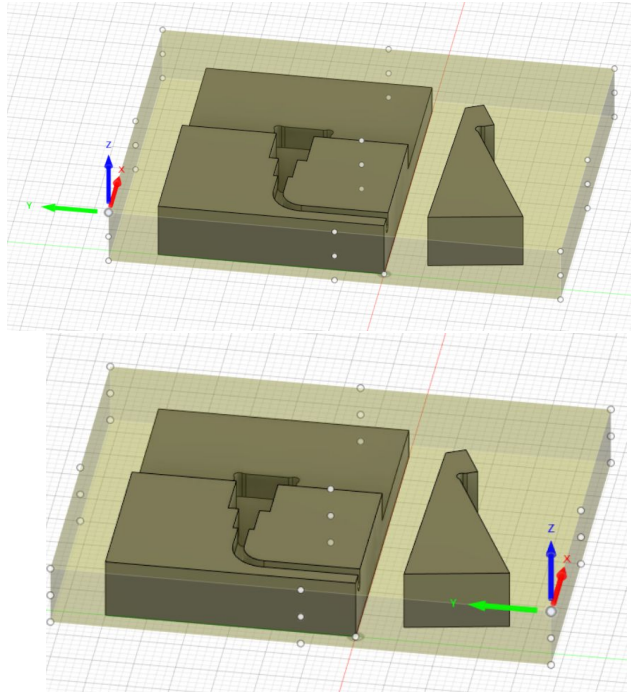
In the Setup tab, we can leave the default settings of Origin and Stock Point but we will have to select one of the dots on the model.

Typically we choose the front left corner to be our origin but occasionally we will choose the center when it is convenient for the part. Select one of the white dots to set the origin.



Setup - origin

It's important to note which direction the axes are located on the machine you are using. For the shopbot, y is located into the machine and x is parallel to the doors. In the Fusion 360 example below, the bottom photo has the correct origin corresponding to the front left corner.

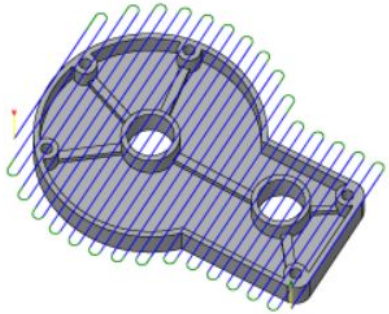


Here are our most common 2D toolpaths

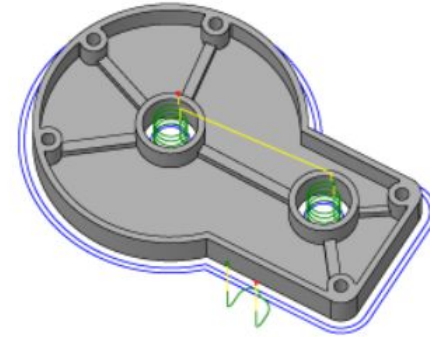
This guide only covers the most common 2D toolpathing tools you might use but not any 3D toolpaths.


The four most common toolpaths that you might use are:

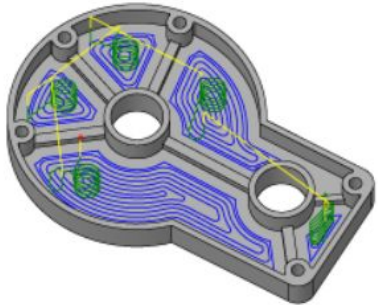
- Face
- 2D Pocket
- 2D Contour
- Trace



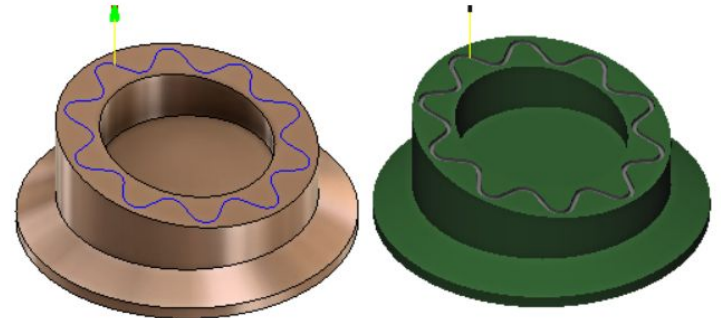
Manufacture > Milling > 2D > Face 



Manufacture > Milling > 2D > 2D Contour 



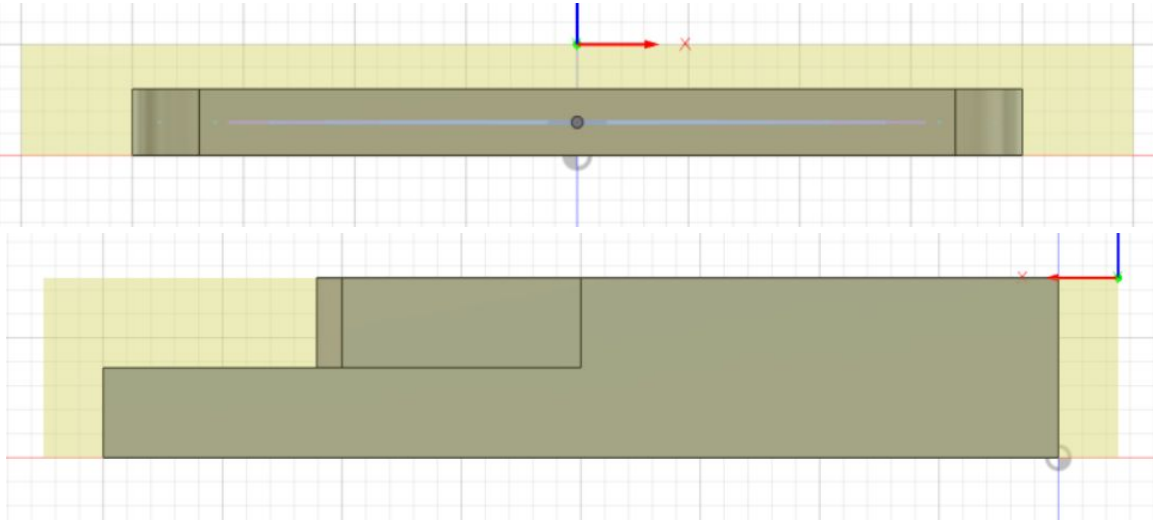
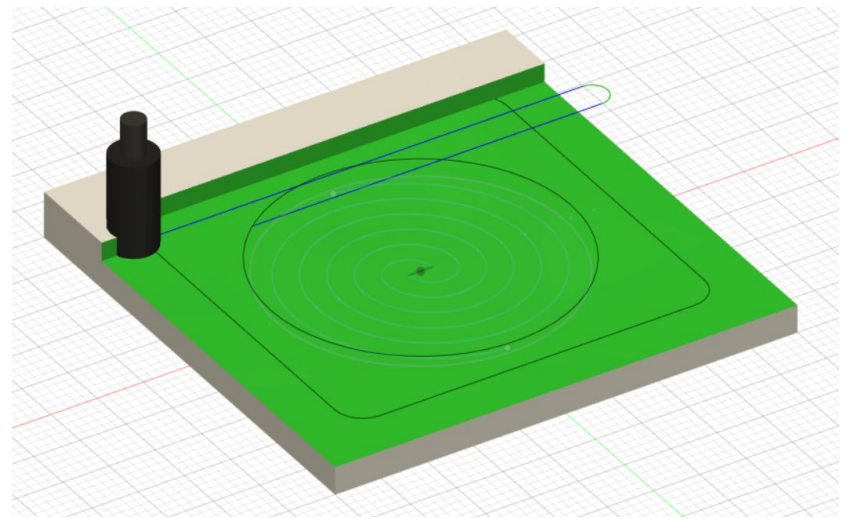
Manufacture > Milling > 2D > 2D Pocket 



Manufacture > Milling > 2D > Trace 

Face

The face operation is used when your entire model is thinner than the stock that you are using.



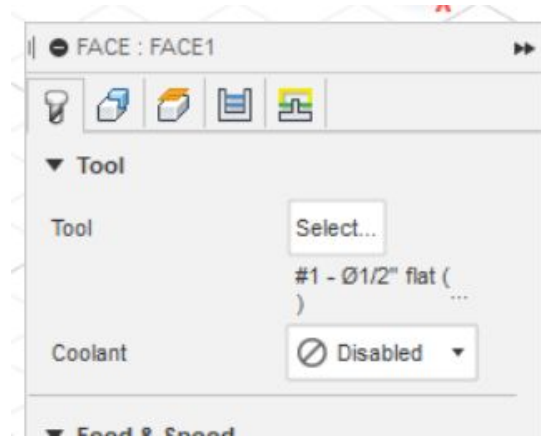
For example this thin coaster will need a facing operation.

The phone stand will not since it is the same thickness as the stock.

Face

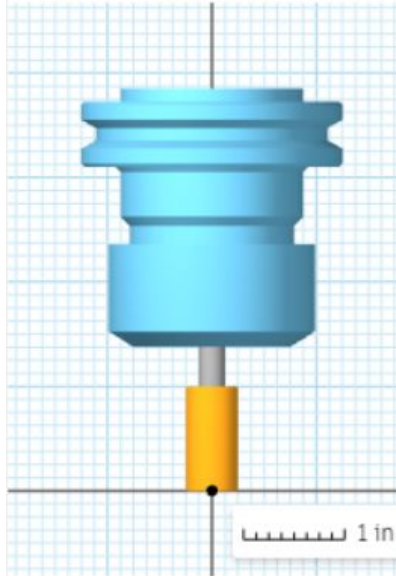
1. Tool Tab:

Typically you will want to use the largest diameter bit possible for the facing operation. For the shopbot that is the 1/2" flat tool.



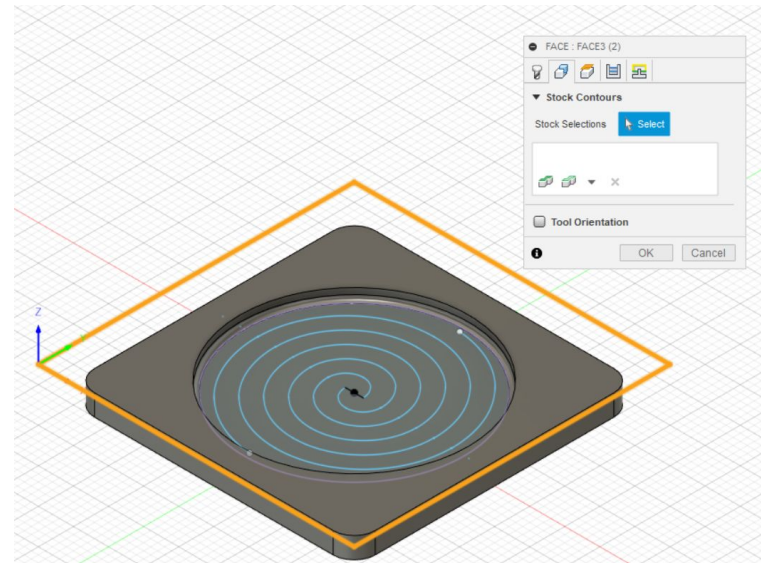
ChapmanShopbot_ToolLibrary

1 - Ø1/2" (45226 Carbide Tipped Straight Plunge High Production 1/2 Dia x 1 Inch x 1/4 Shank)



2. Geometry Tab:

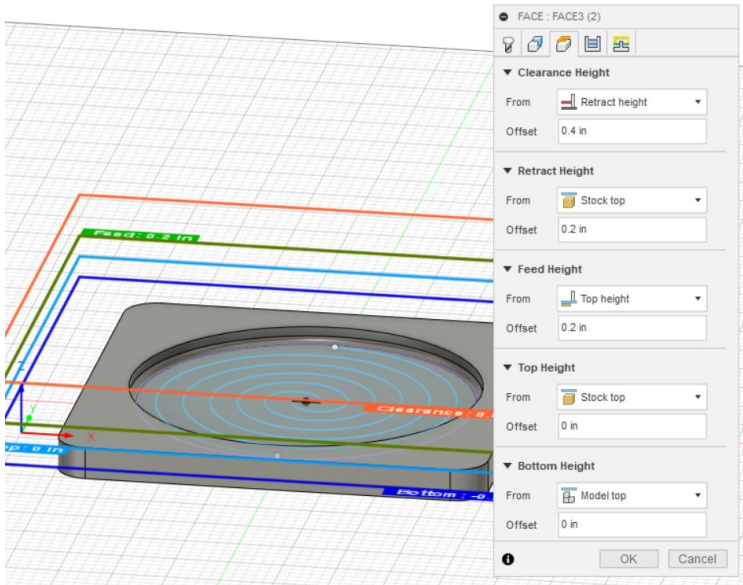
The geometry tab is typically left default as it will cut down the entire stock. You can also select a certain face or edge if you don't want to face the entire stock.



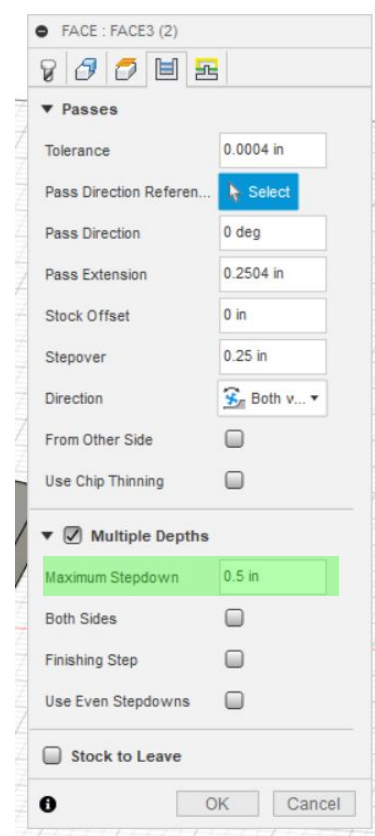
Face

3. Heights Tab:

Typically you will leave all the settings at default on the heights tab for the facing operation.

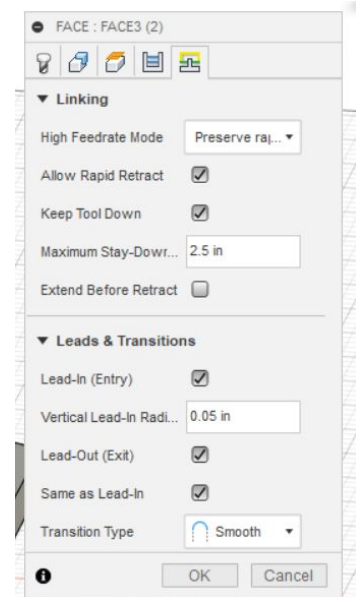


4. **Passes Tab:**
We always want to check the Multiple Depths box, even if you don't think you need it. The Maximum Stepdown should be set to the diameter of the tool (for the shopbot).



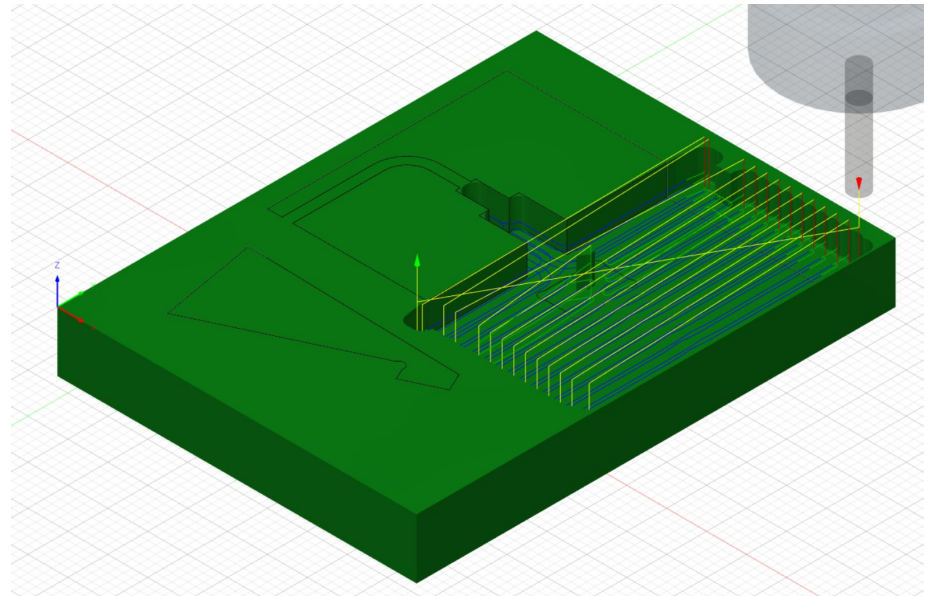
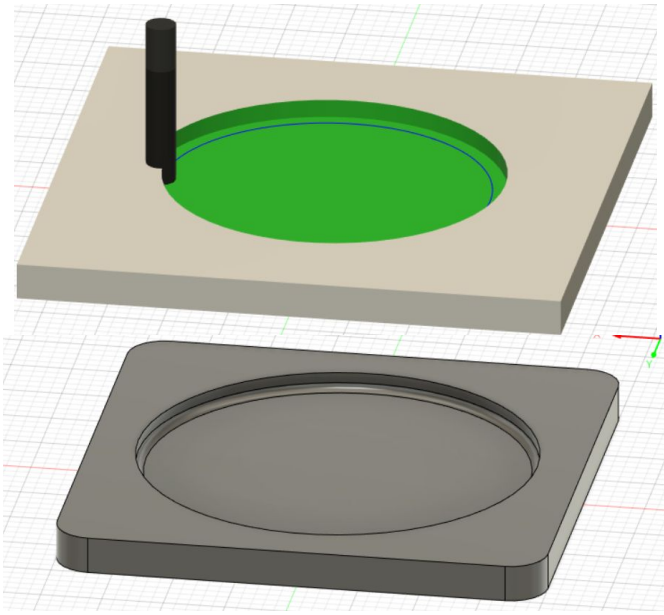
5. Linking Tab:

We can leave this one at default settings.



2D Pocket

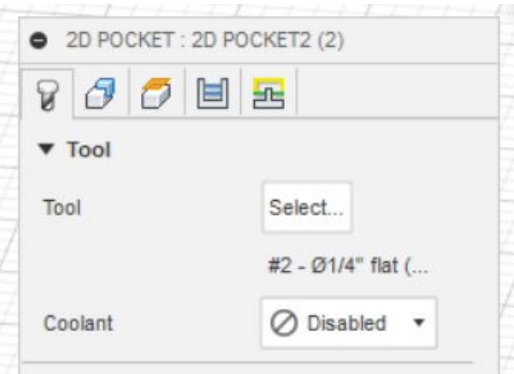
The 2D Pocket operation is used when you have a cavity or a pocket with a flat bottom where the tool typically has to enter the material from above rather than the side.



2D Pocket

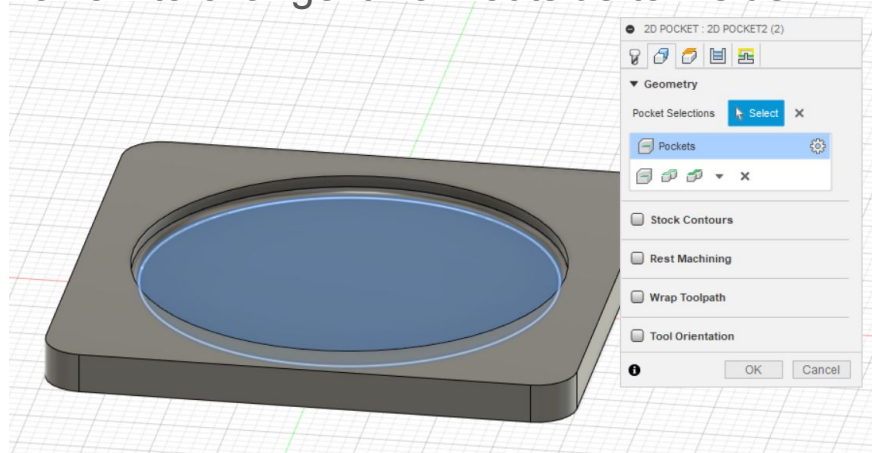
1. Tool Tab:

Typically you will want to use the largest diameter bit that will fit in your pocket to remove the material quickest. But you may want to use a smaller tool if you have inner corners or to avoid extra tool changes.



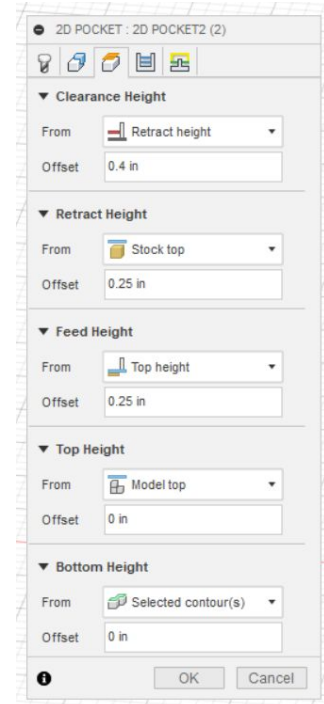
2. Geometry Tab:

You can select a **face** at the bottom of the pocket or an **edge**. If you select the edge make sure it is pocketing the inside rather than the outside. You can select the red arrow to change it from outside to inside.

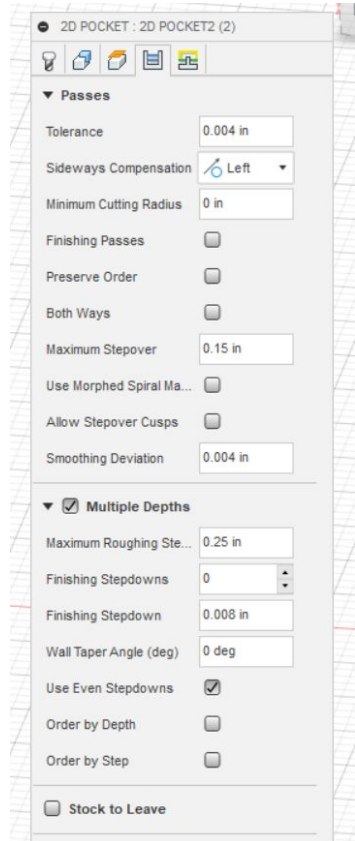


3. Heights Tab:

Normally you'll leave settings at default unless you already removed material from above the pocket (i.e. from a facing operation). In that case you will want to change the top height to where it needs to start cutting the material. (i.e. model top)



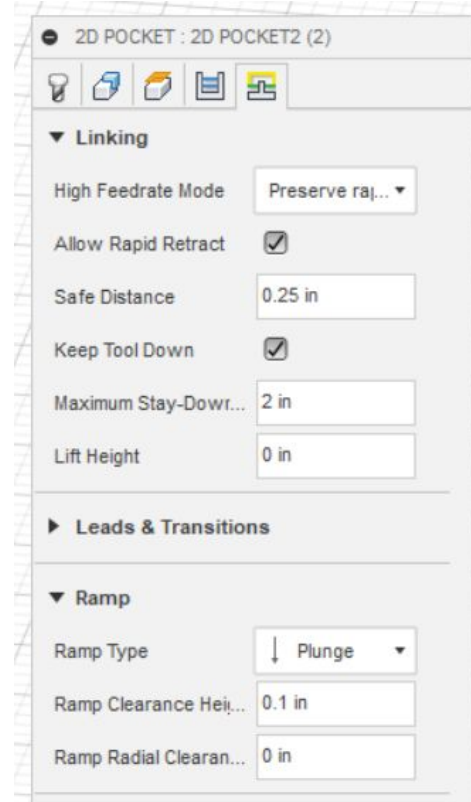
2D Pocket



4. Passes Tab:

We always want to check the Multiple Depths box, even if you don't think you need it. The Maximum Stepdown should be set to the diameter of the tool (for the shopbot).

Typically we want to uncheck Stock to Leave unless you are planning to come back and remove the small amount of stock later.



5. Linking Tab:

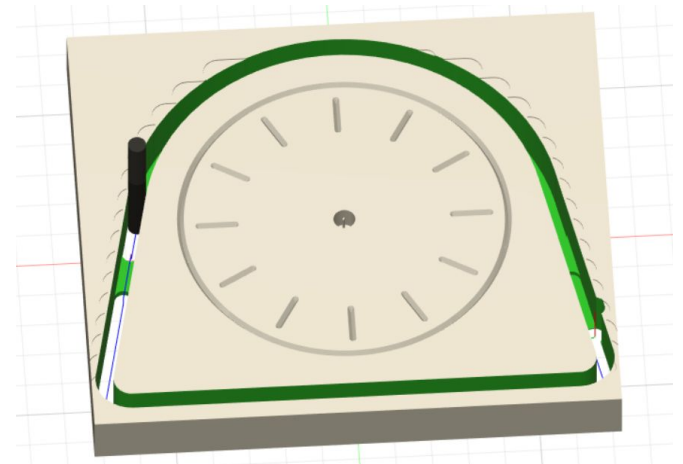
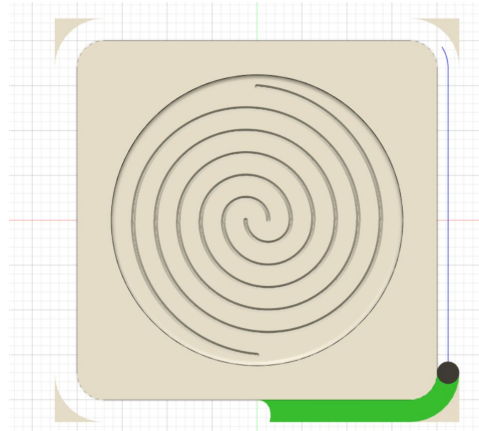
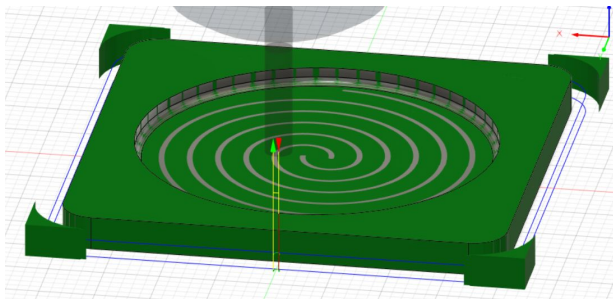
Ramping defines how the tool enters the material. For soft materials like wood/plastic Plunge will work and is the fastest ramping type.

*A helical toolpath may be necessary to enter the material from above and not ruin the tool for metal cutting

2D Contour

The 2D Contour operation is used when you want to cut on the inside or outside of an edge/chain. It is commonly used for cutting out your final part.

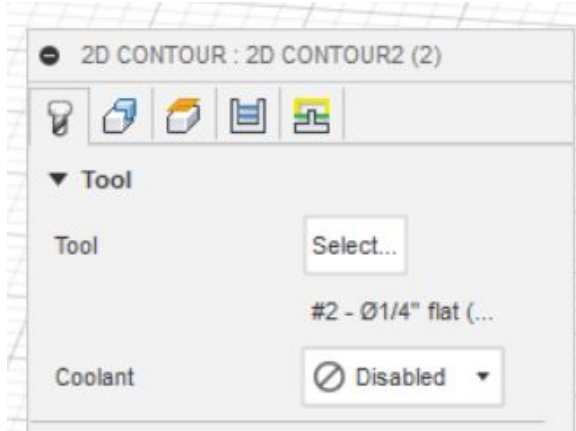
I will almost always use this last after everything else has been cut.



2D Contour

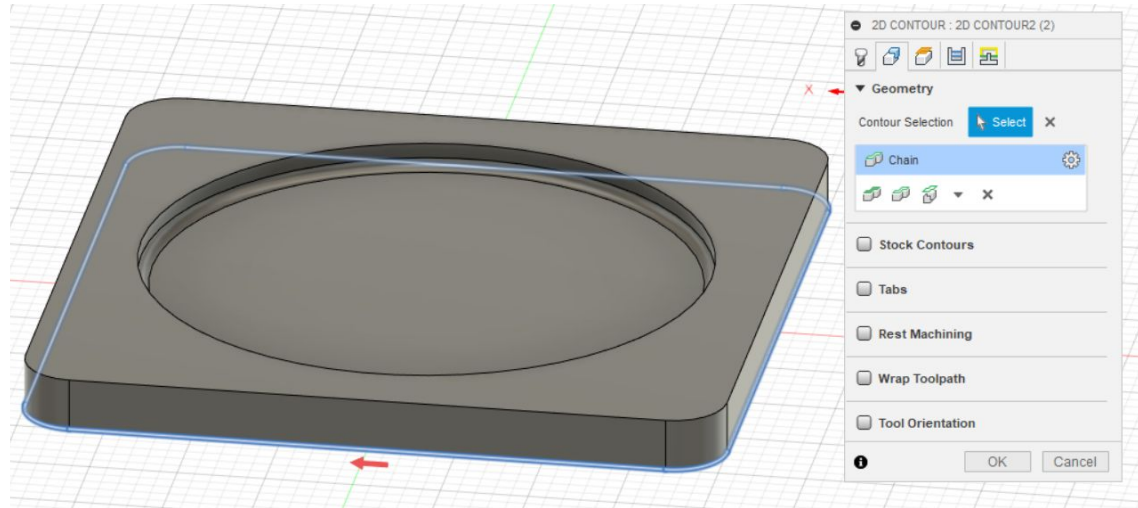
1. Tool Tab:

Typically you will want to use a larger diameter bit to remove the material quickest. But you may want to use a smaller tool if you have inner corners or to avoid extra tool changes.



2. Geometry Tab:

It is generally best to select an edge/chain and the red arrow will indicate whether it is on the inside or outside. To change it from one to the other, click on the red arrow.



2D Contour

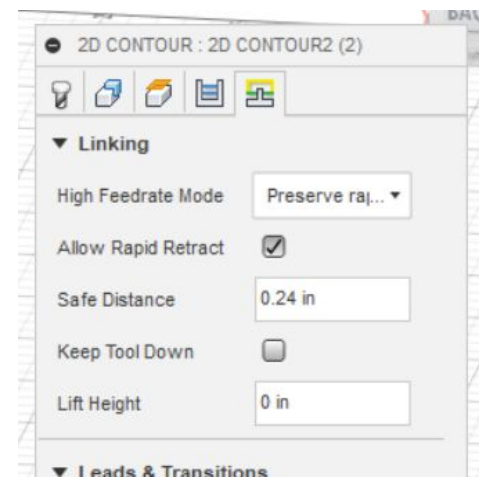
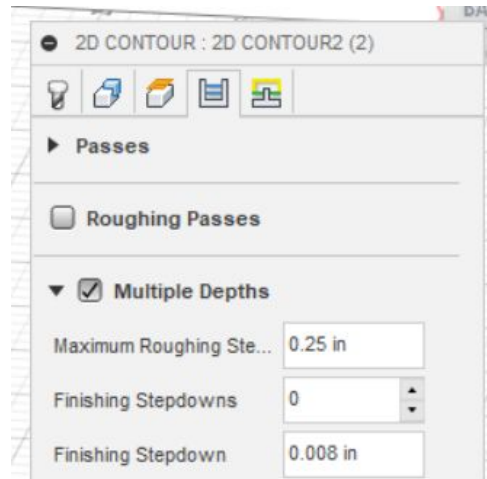
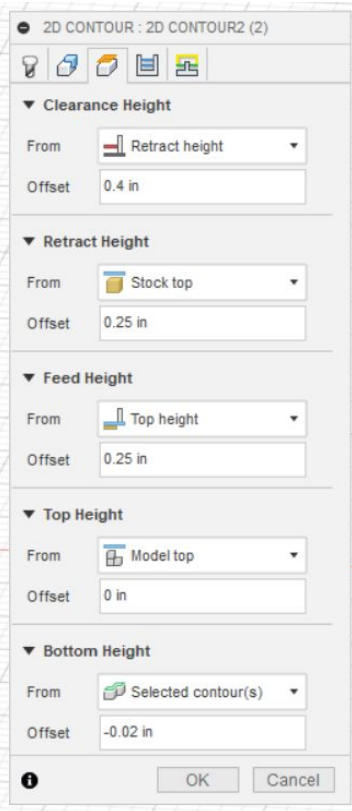
3. Heights Tab:
Normally you'll leave settings at default unless you already removed material from above the contour (i.e. from a facing operation). In that case you will want to *change the top height* to where it needs to start cutting the material. (i.e. model top). If you are cutting to the bottom of your part, then it's best to *offset the Bottom Height -0.03 inches*.

4. Passes Tab:

We always want to check the Multiple Depths box, even if you don't think you need it. The Maximum Stepdown should be set to the diameter of the tool (for the shopbot).

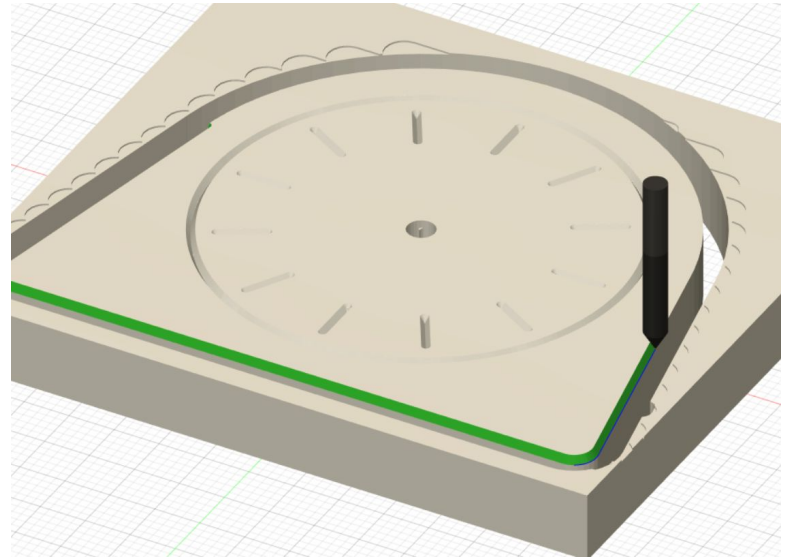
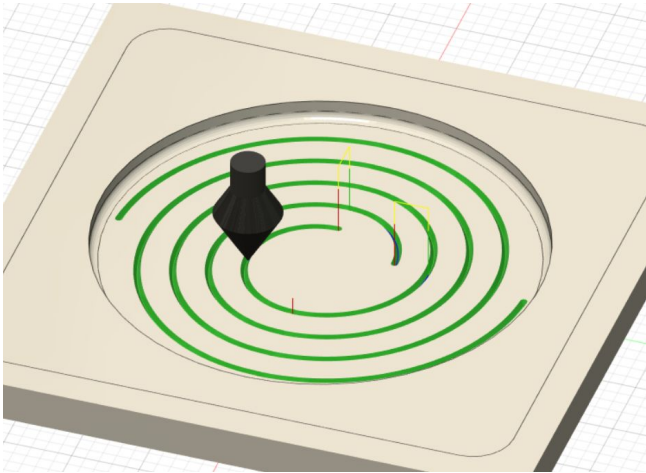
5. Linking Tab:

We can leave this one at default settings.



Trace

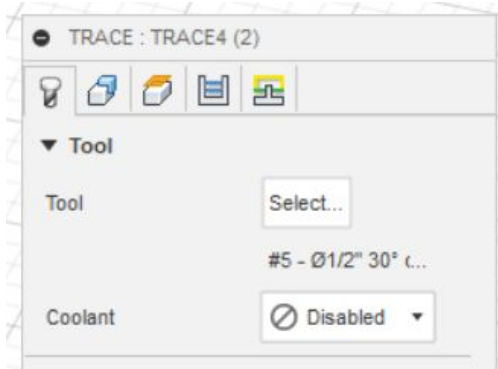
The Trace toolpath is used when you want a tool to trace the center of an edge/chain (unlike the contour which is on the side). This is used typically for designs, chamfers, or slots.



Trace

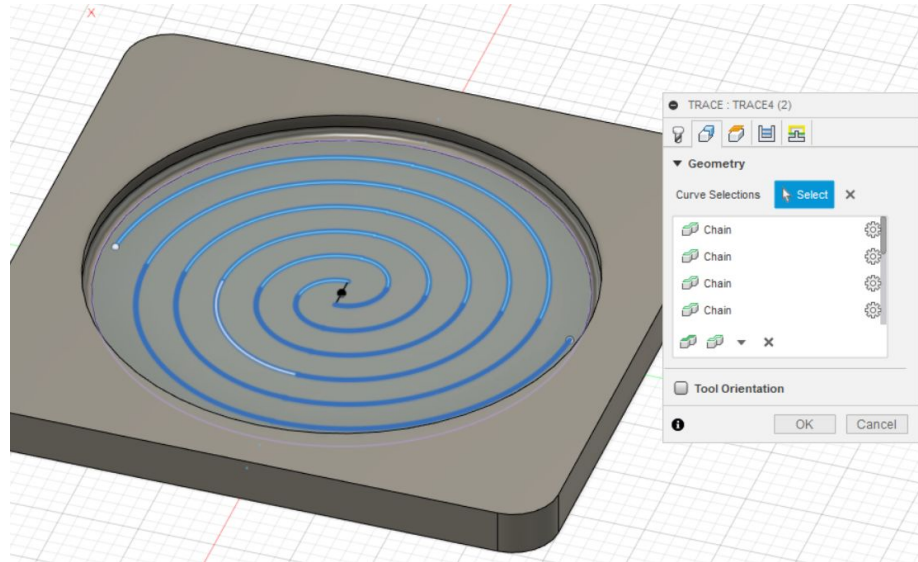
1. Tool Tab:

The tool you choose will depend on what geometry you are trying to achieve. It is common to use a V groove bit with this toolpath.



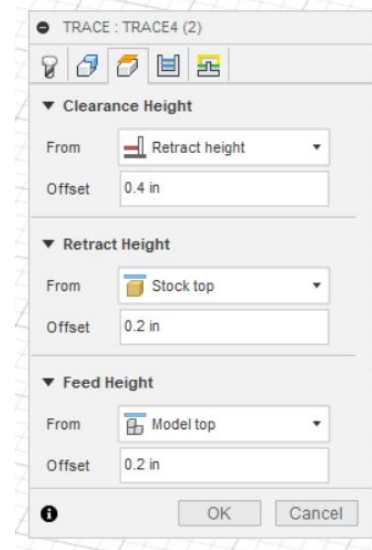
2. Geometry Tab:

You will have to select an edge/chain for the geometry. It is common to use a sketch for the trace.



3. Heights Tab:

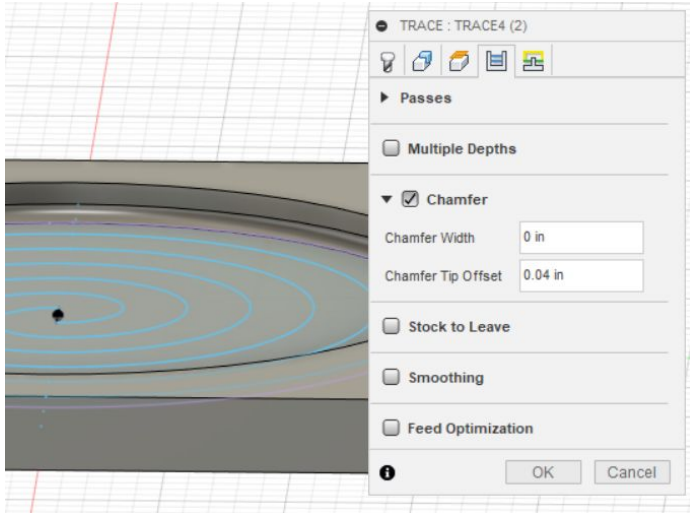
We can leave this one at default settings.



Trace

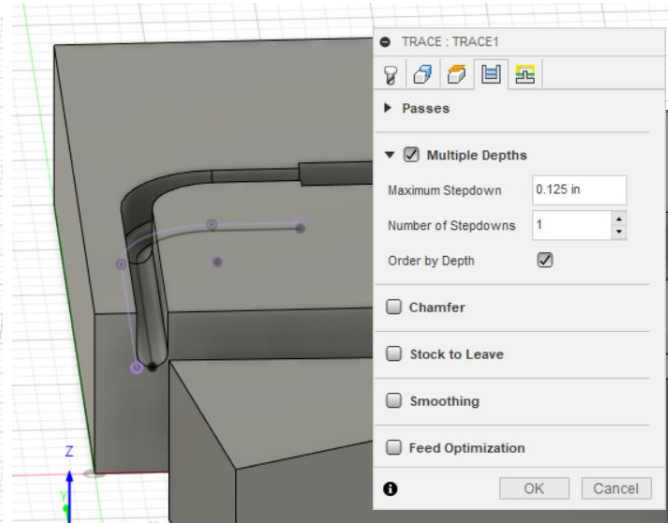
4. Passes Tab:

When our sketch is on the top of the surface we want to cut, we can use the Chamfer Tip Offset to move the tool tip down into the part.



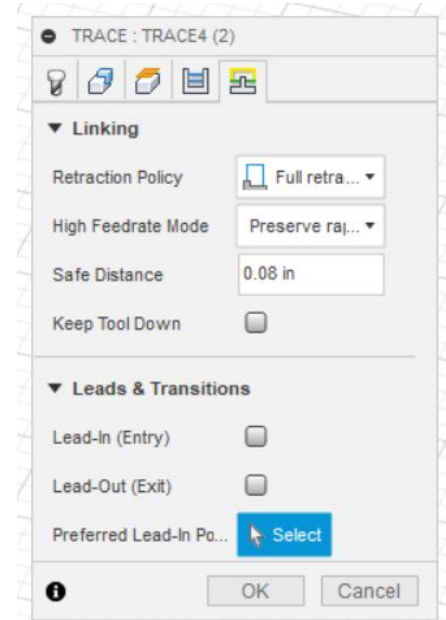
4. Passes Tab:

If our sketch is at the height we want to cut and deep into our part, then we want to select Multiple Depths and set Maximum Stepdown to the tool diameter.



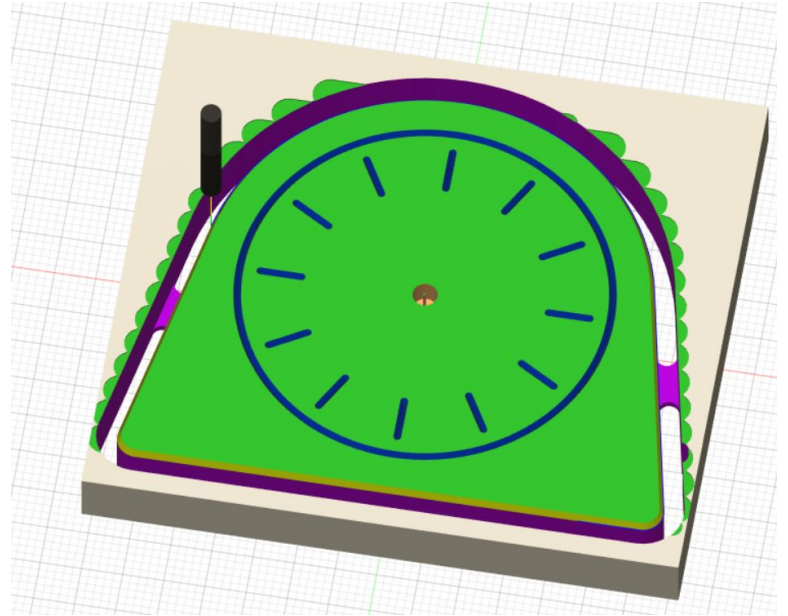
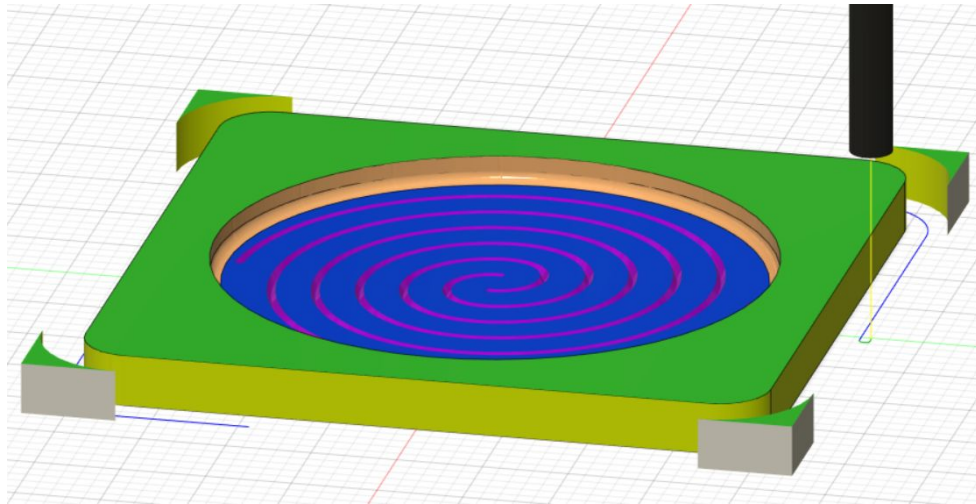
5. Linking Tab:

We can leave this one at default settings.



Simulate!

Always simulate and check everything is ok! It's important to watch the entire process (even if it sped up) and check for anything that looks weird, looks like it is wasting time, or anything that is not cutting out properly.



Post Process

You can right click on your Setup1 and select Post Process to get all the gcode in one file.

When everything looks good it's time to post process and create the gcode (.spb file)!

If you want to post each tool separately or just certain parts of the toolpath, you can select multiple of the toolpaths, right click and select Post Process.

Ensure you are using the correct Post for your machine. Rename, select the output folder and select Post.

