Parametric Design using Fusion 360

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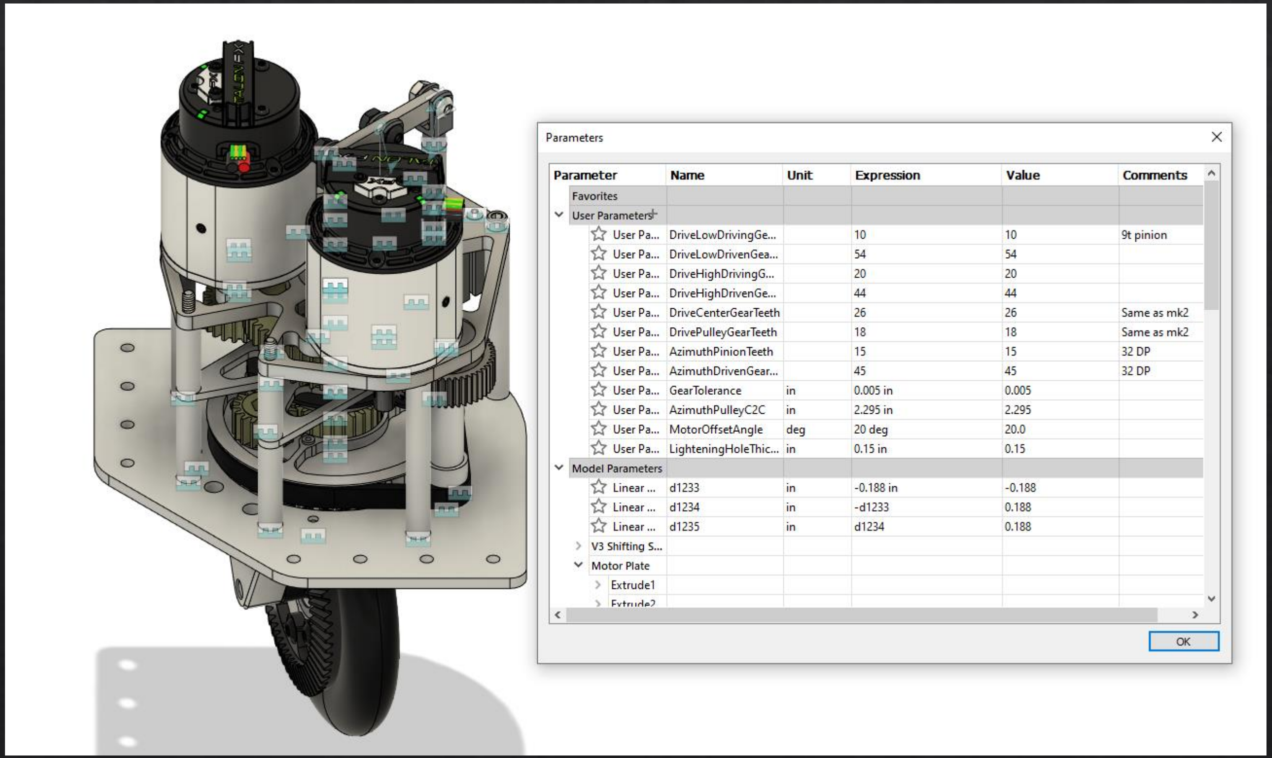
Why use parametric design?

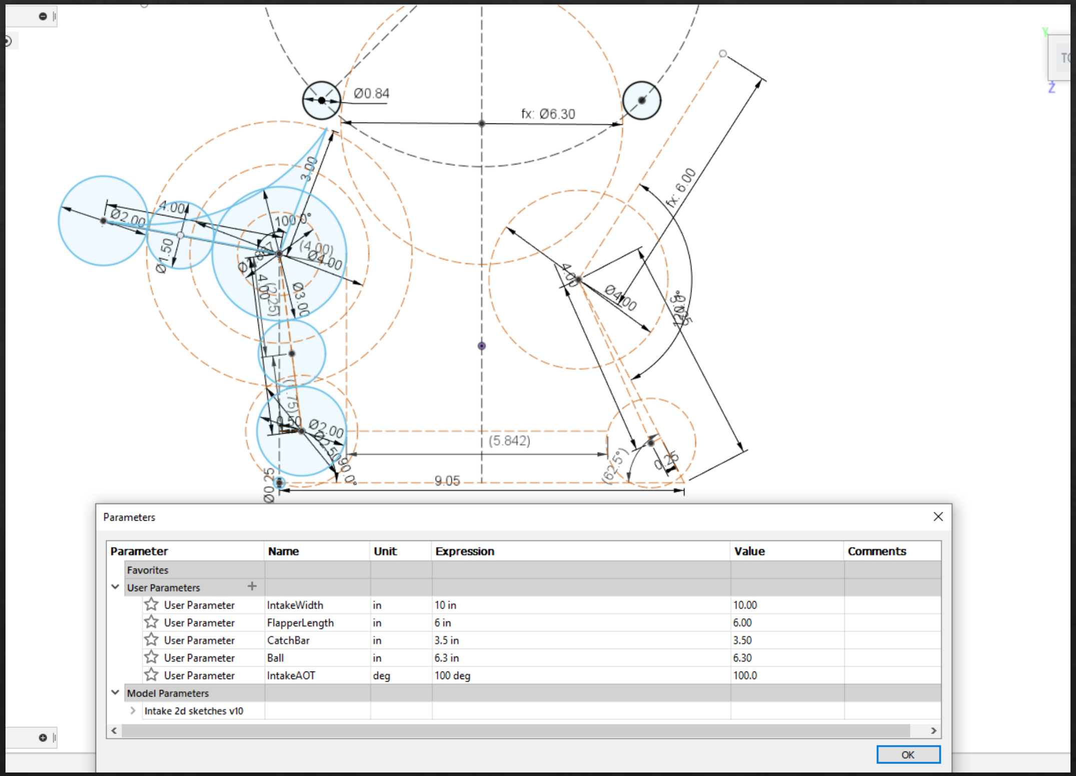
- Parametric design allows for rapid changes in models based on a table of common values.
- Parameters can be used to change many values at once.
- Parameters can reference sketches, extrudes, or nearly anywhere that numbers can be used.
- Using parameters saves time because common values are automatically available to each new sketch.



Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameters					
User Pa...	DriveLowDrivingGe...		10	10	9t pinion
User Pa...	DriveLowDrivenGea...		54	54	
User Pa...	DriveHighDrivingG...		20	20	
User Pa...	DriveHighDrivenGe...		44	44	
User Pa...	DriveCenterGearTeeth		26	26	Same as mk2
User Pa...	DrivePulleyGearTeeth		18	18	Same as mk2
User Pa...	AzimuthPinionTeeth		15	15	32 DP
User Pa...	AzimuthDrivenGear...		45	45	32 DP
User Pa...	GearTolerance	in	0.005 in	0.005	
User Pa...	AzimuthPulleyC2C	in	2.295 in	2.295	
User Pa...	MotorOffsetAngle	deg	20 deg	20.0	
User Pa...	LighteningHoleThic...	in	0.15 in	0.15	
Model Parameters					
Linear ...	d1233	in	-0.188 in	-0.188	
Linear ...	d1234	in	-d1233	0.188	
Linear ...	d1235	in	d1234	0.188	
V3 Shifting S...					
Motor Plate					
Extrude1					
Extrude2					

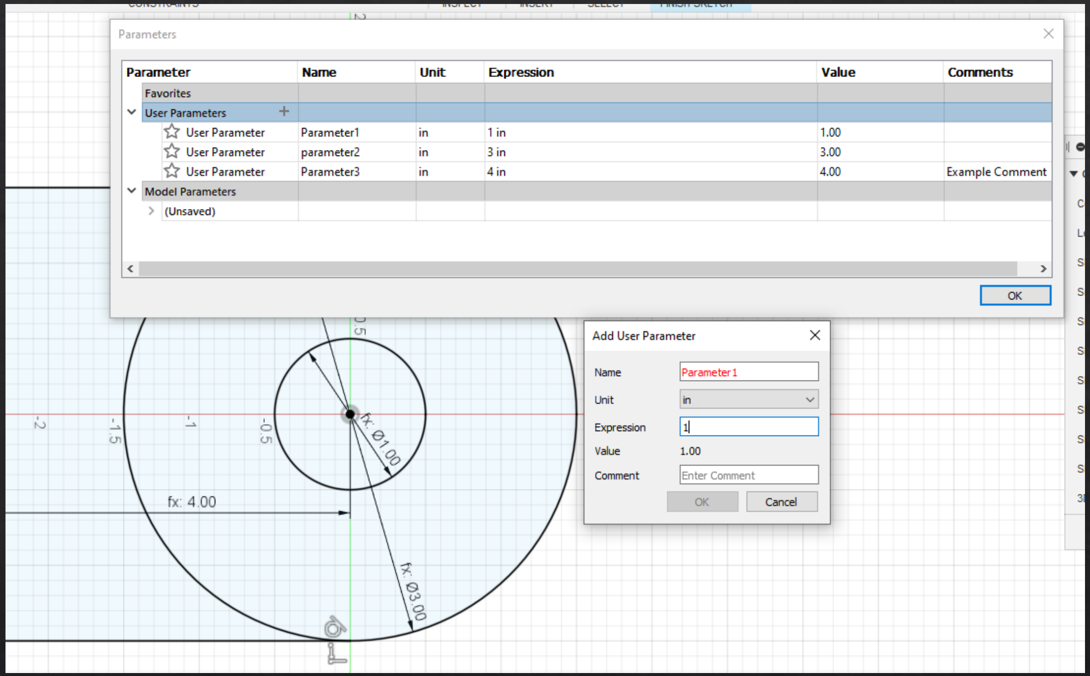
When to use parametric design

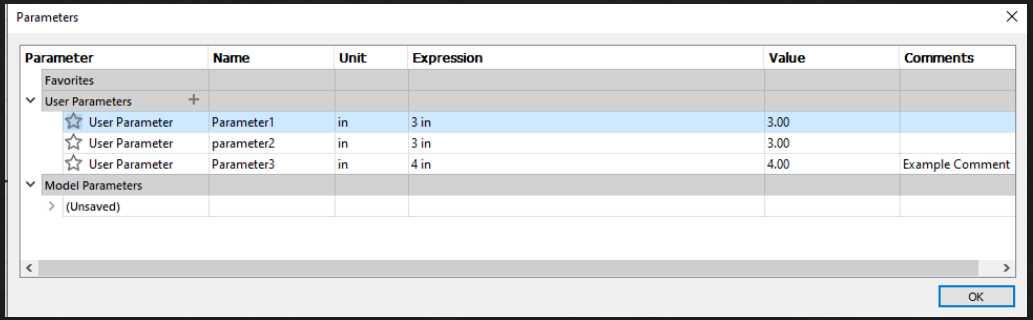
- High level design waterfall
- To simplify assembly work
- When there are unknowns
- Documentation and identification



Getting started with Parameters

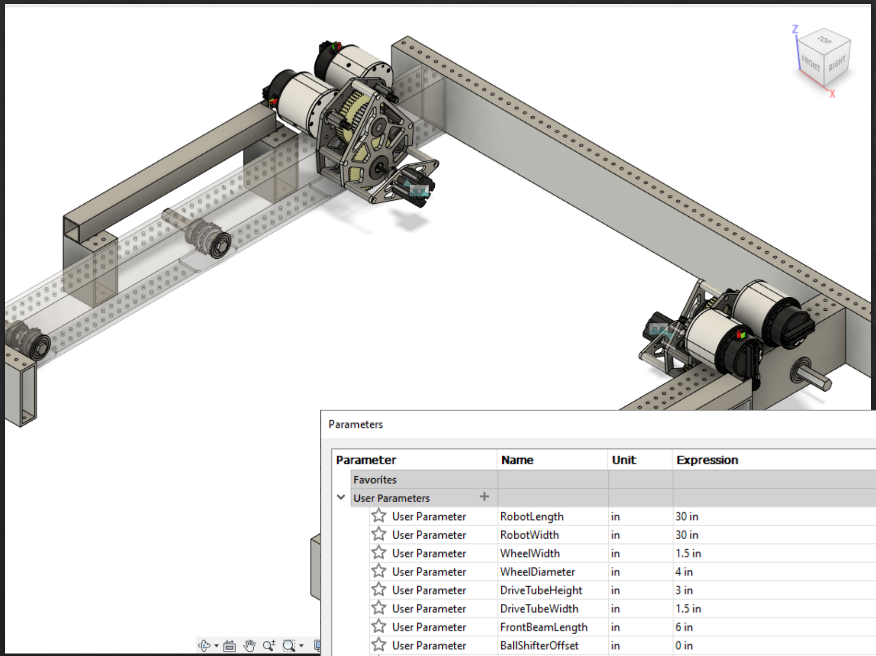
- Open a new design in fusion 360
- Create a sketch of anything
- Select "Modify" then "parameters"
- Create a new parameter called Parameter1 and set it to 1"
- Dimension your part by simply typing the parameter name into the dimension value
- To test this parameter, click on the expression and change 1" to 3"
- This is very basic parametric design and unpractical
- This is very similar to using global variables when programming your robot





An example of advanced parameterized design

- In 2019, my FRC team wanted to have a drivetrain designed by the end of day 1
- We did not know the perimeter of the robot, so we used parameters to make a universal bot
- We did not know our speeds, so we parameterized our gearing ratios
- By the end of day 1, we had a legal and ideal robot drivetrain ready to be built



Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameters					
User Parameter	RobotLength	in	30 in	30.00	
User Parameter	RobotWidth	in	30 in	30.00	
User Parameter	WheelWidth	in	1.5 in	1.50	
User Parameter	WheelDiameter	in	4 in	4.00	
User Parameter	DriveTubeHeight	in	3 in	3.00	
User Parameter	DriveTubeWidth	in	1.5 in	1.50	
User Parameter	FrontBeamLength	in	6 in	6.00	
User Parameter	BallShifterOffset	in	0 in	0.00	DistanceFromCer
User Parameter	WheelOffset	in	0.45 in	0.45	
User Parameter	CDrop	in	0.1 in	0.10	
User Parameter	PinionToothCount		10	10	8t
User Parameter	HighDrivingGearTo...		60	60	
User Parameter	HighDriveGearToot...		44	44	
User Parameter	LowDrivingGearToo...		44	44	
User Parameter	LowDrivenGearToot...		60	60	
User Parameter	PlateOutsideThickn...	in	0.125 in	0.125	
User Parameter	GearTolerance	in	0.005 in	0.005	
User Parameter	MotorOffsetFromT...	in	(1 / 16) * 1 in	0.063	
User Parameter	BallShifterC2CBack	in	3 in + 0.18 in	3.18	24 Lengths
User Parameter	WheelC2C	in	11 in + 0.18 in	11.18	38 Lengths
User Parameter	BallShifterC2CFront	in	8 in + 0.18 in	8.18	30 Lengths
User Parameter	AccessHole	in	2.5 in	2.50	
User Parameter	BellyPanThiccness	in	(1 / 16) * 1 in	0.063	
User Parameter	FrameTubeWidth	in	1 in	1.00	
User Parameter	FrameTubeHeight	in	3 in	3.00	

OK

Assessment

- What's the main benefit of parametric design?
- What would a good example of parametric design be?
- Where is the parameter button located?

Assessment answers

- What's the main benefit of parametric design?
  Makes design process easier and saves time
- What would a good example of parametric design be?
  Anything with unknowns
- Where is the parameter button located?
  Under the modify tab in either the 3d or sketch bar

Further Learning

- Tutorial: Parametric Modeling With Fusion 360 - (autodesk.com)
- Parametric modeling in Fusion360 - Prusa Printers

CAM in Fusion 360

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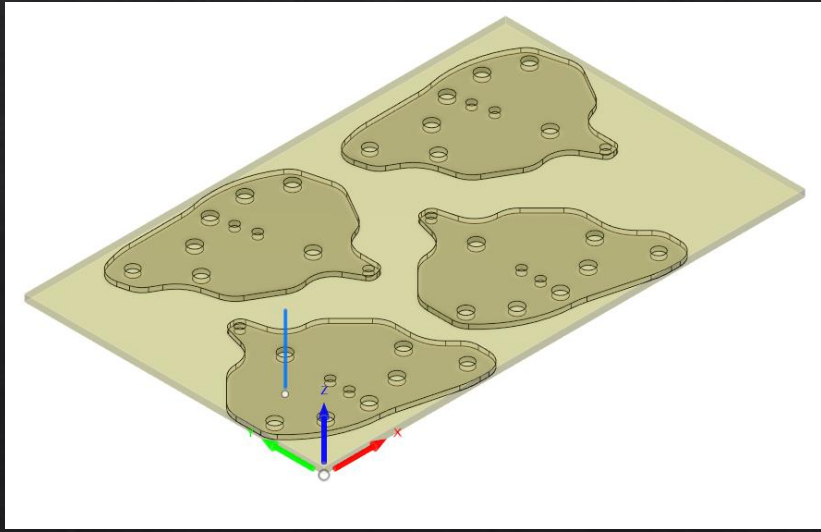
Designing Sheets of parts for Fusion:

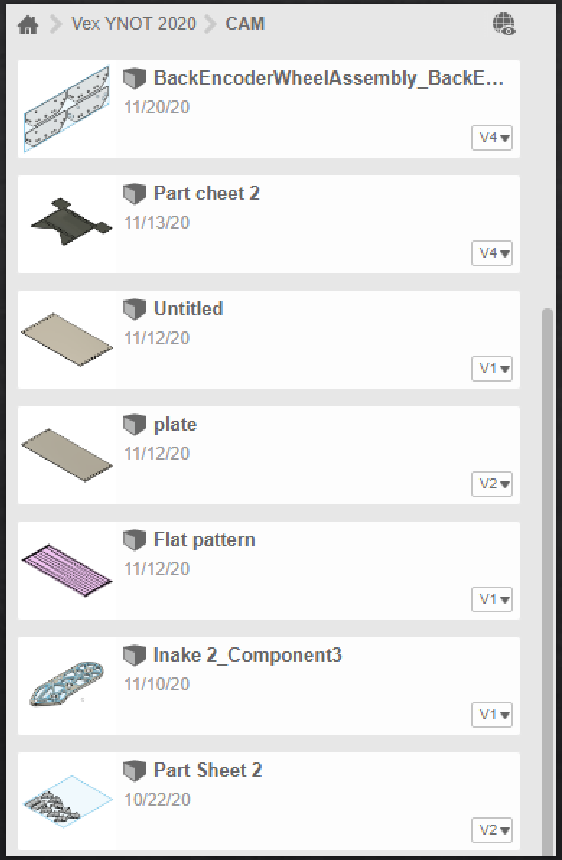
- This tutorial is an overview of turning flat part designs into reality
- Flat parts can include metal, plastic, or wood parts cut from sheets
- The example used in this tutorial will be made from 1/16" polycarbonate

- Estimated time to complete this lesson is 20 minutes



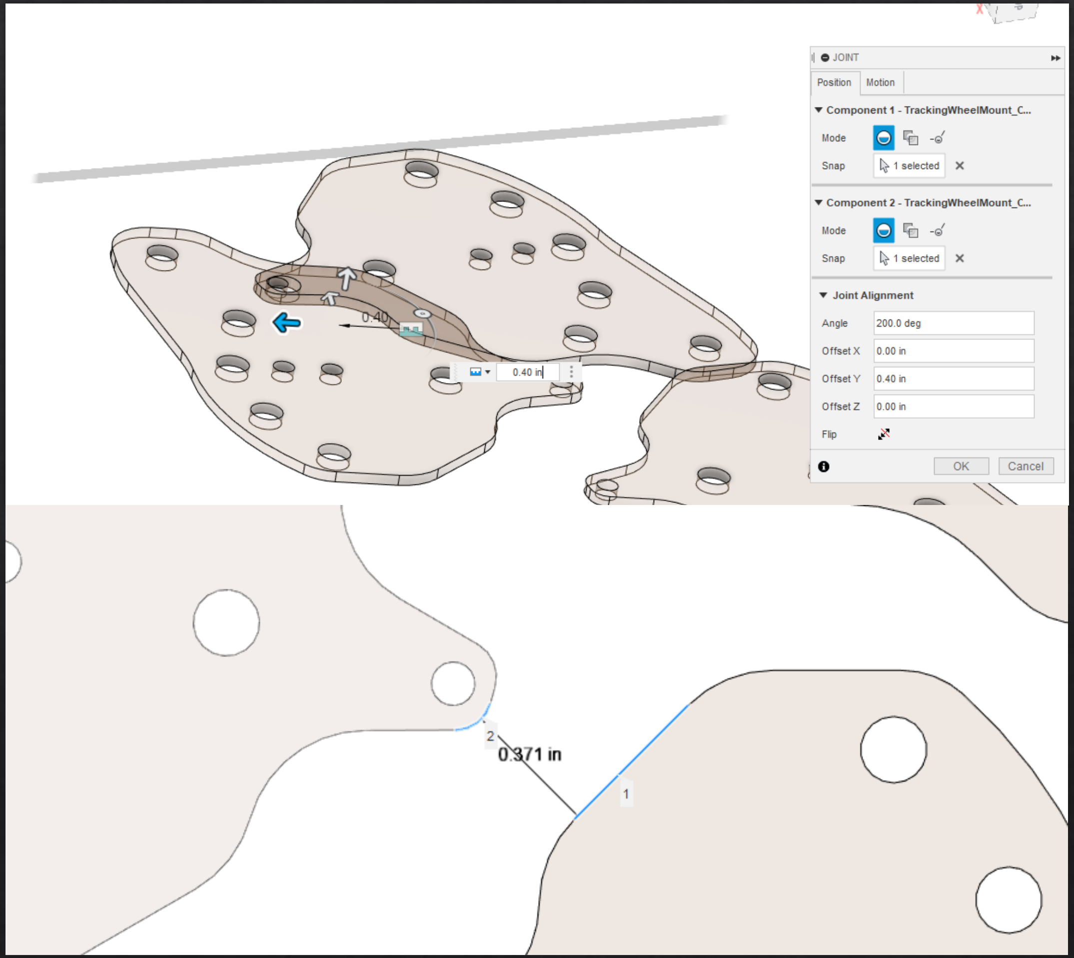
Picking the right parts

- When cutting parts on a CNC, you can choose to do many parts at once to save time or do a single part to protect yourself from potential costly failures
- If you are unfamiliar with a material or machine, it can be helpful to do a practice part
- Create a folder for your cam on Fusion 360 to avoid clutter



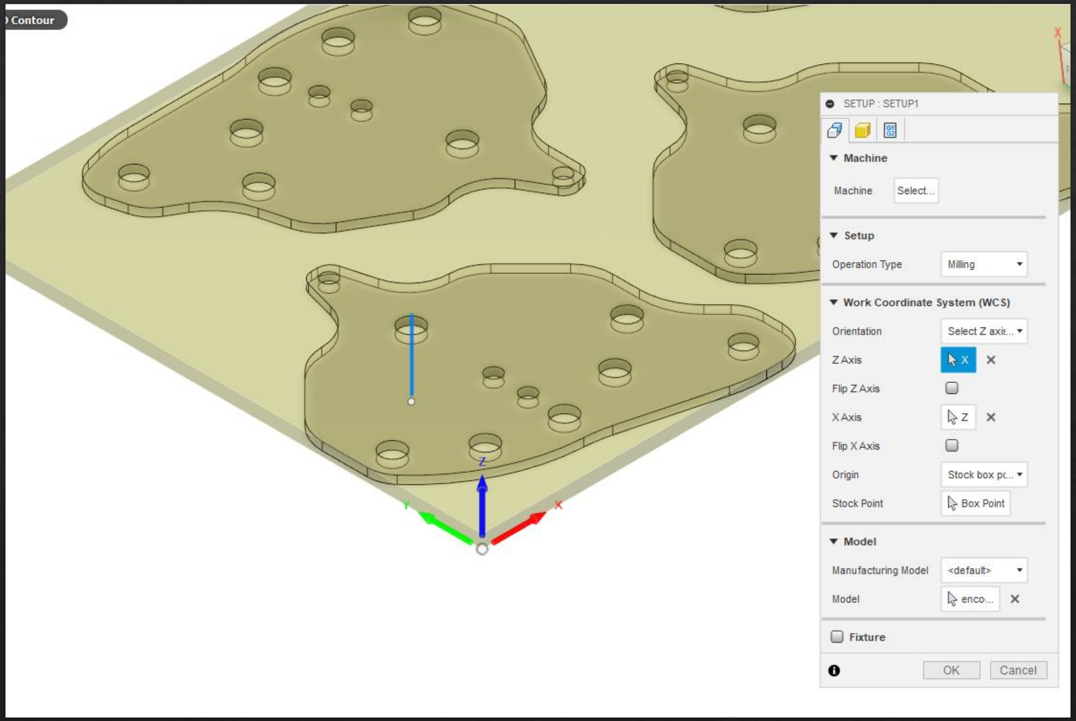
Step 1: Place parts

- Insert all of your parts into a new design. Give it a descriptive name
- Joint the parts to each other so that they are all on an equal plane
- Leave space for the mill bit to pass in between parts
- Place parts in a layout which will fit onto your machine



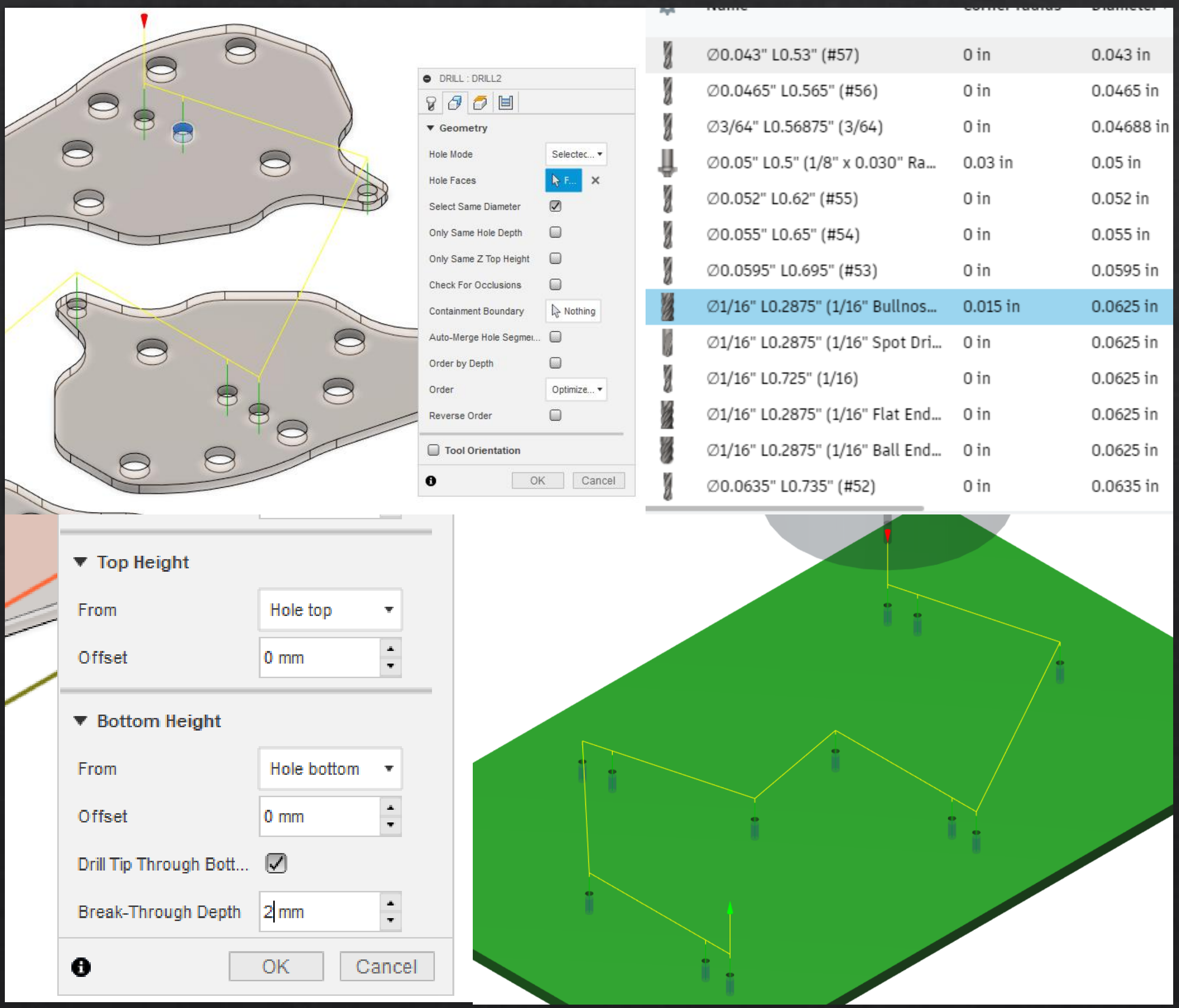
Step 2: Origin

- Its important to know what your machine's axes are and which direction is positive
- Generally try to set the origin to a corner of the part sheet so you can easily measure how much bed space you will need
- Select your axis directions accordingly
- Select a stock box point for the origin on the top of the part



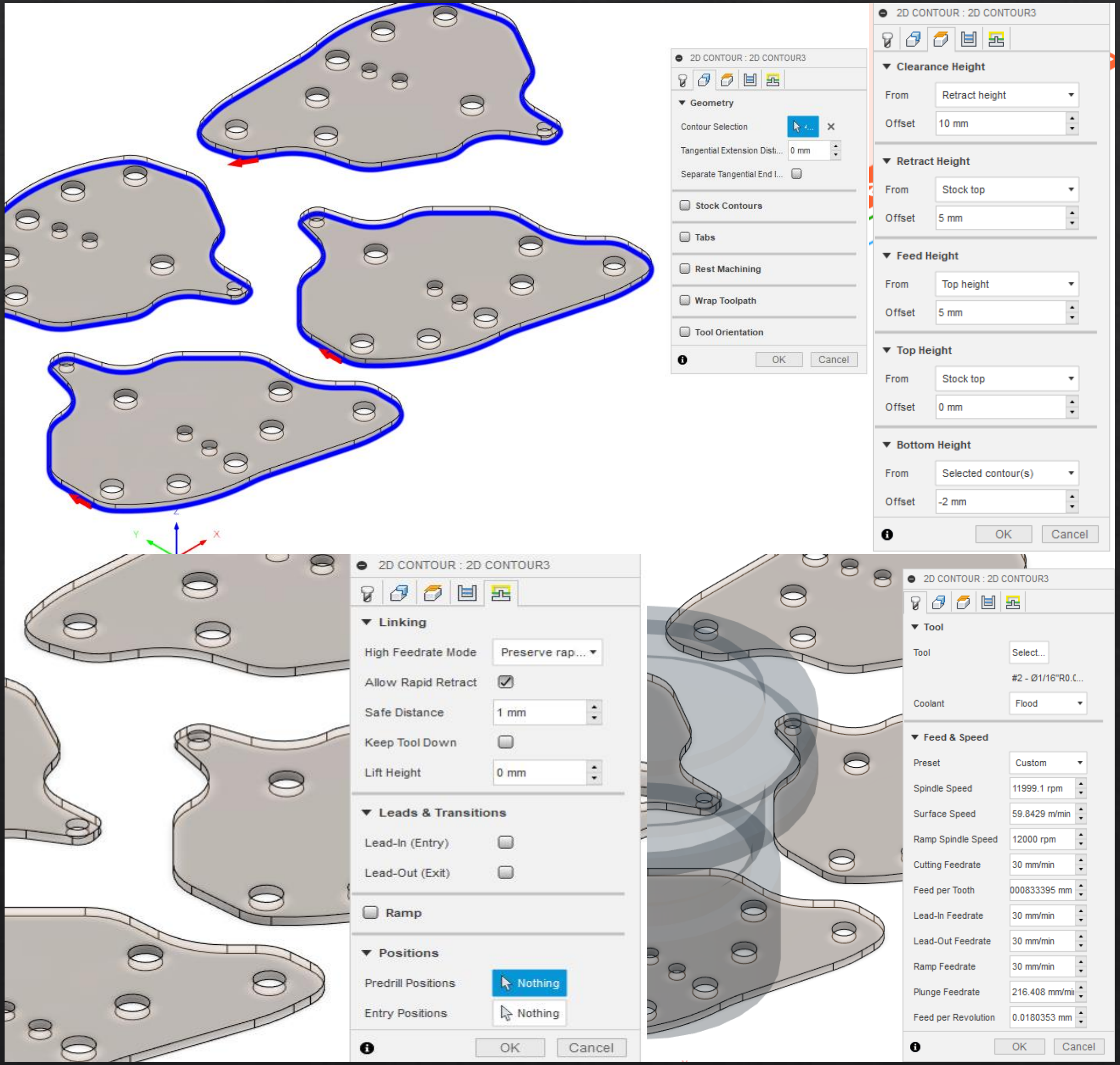
Step 3: Drill Operations

- Use drill operations to plunge into materials quickly and safely
- Select the function from the hot bar
- Select the drill bit from the lib you will use
- Select the hole faces you want to drill
- A shortcut is to select “Select same diameter” to quickly select holes
- Adjust feeds and speeds according to the material and machine capabilities
- Select “Drill through bottom” and add a breakthrough depth based on the bed level tolerance



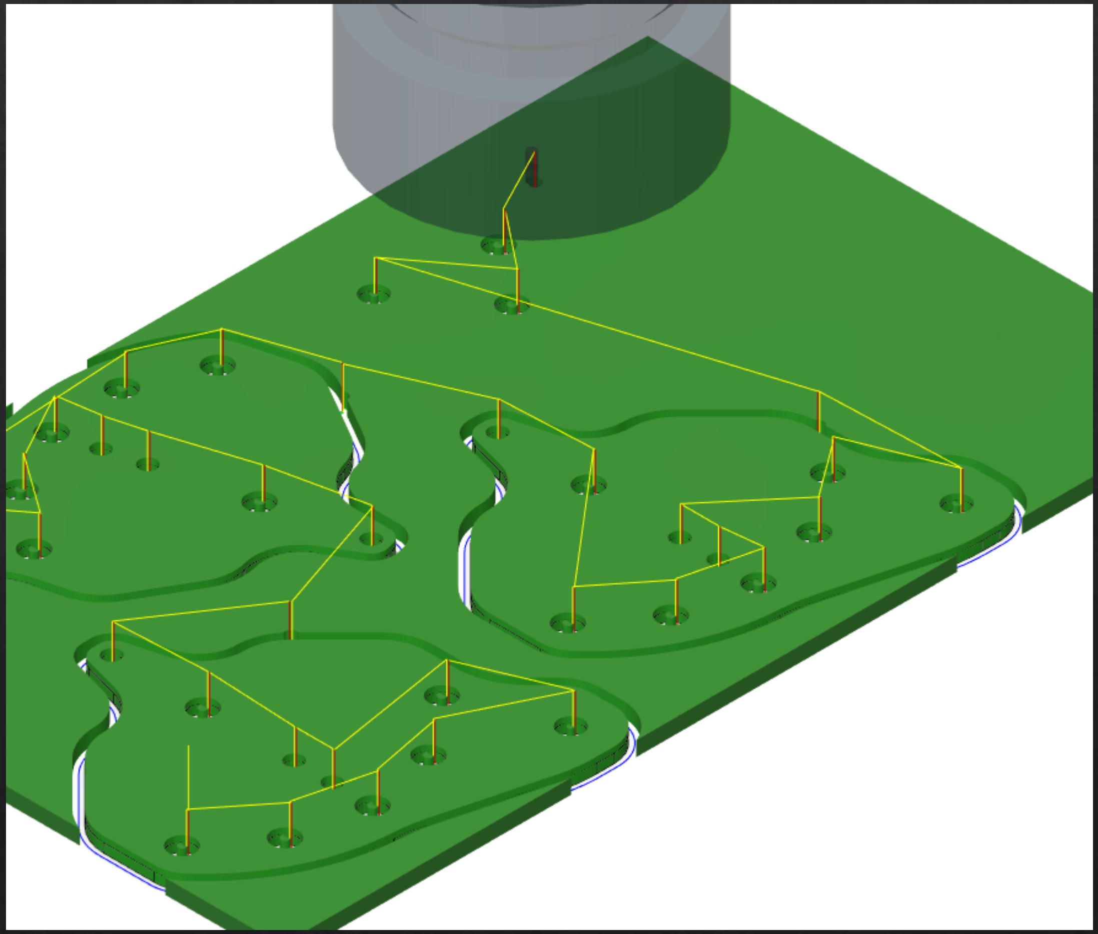
Step 4: Contours

- Select the 2D Contour button
- Select the mill bit you will be using
- Select the bottom edge of your contours
- Add a bottom offset to your bed level tolerance
- Add tabs to keep your parts in place
- For difficult to reach places, deselect lead-in and lead-out
- Set feeds and speeds accordingly to your material and machine standards
- If the material is thick you may want to run multiple passes



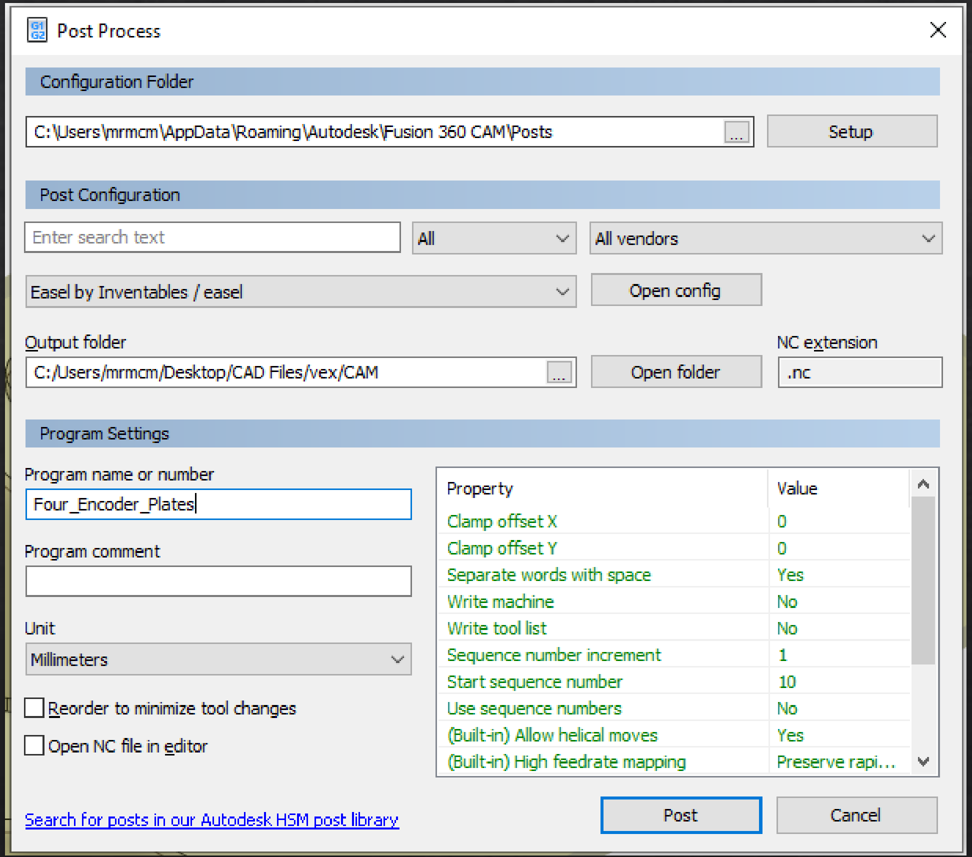
Step 5: Simulate Operations

- Select the simulate tool
- Press play and watch the operation run
- Tweak anything in the previous steps if there are issues in the simulation
- Always run simulations to catch bugs and save time and money!



Step 6: Post Processes

- Right click on setups and select post process
- If you have tool changes you should select post process on the separate operations
- Make sure your configuration file is set according to your CNC
- Select your output folder (i.e. flash drive)
- Set a descriptive program name
- Select the correct unit the CNC uses (usually mm)
- Review all CAM with an experienced user



Assessment:

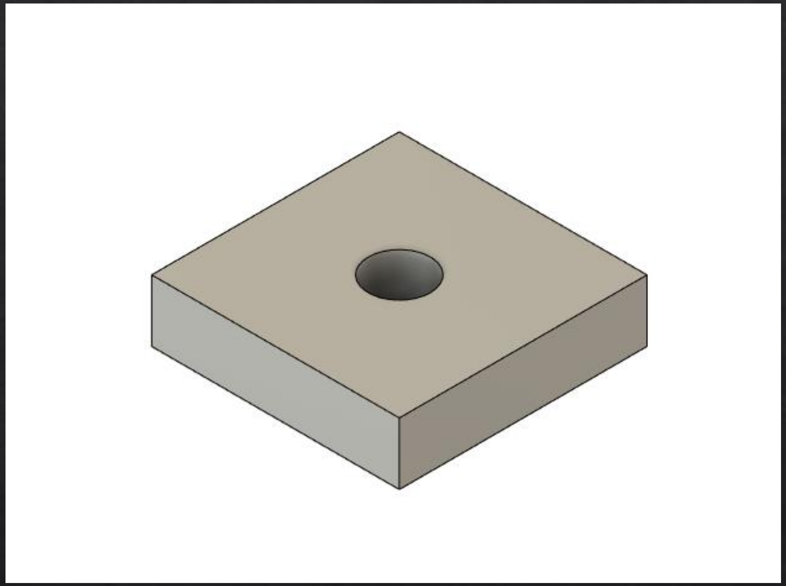
1. Where should you place your origin?
2. How do you select a bit to use in an operation?
3. When should you use a drill operation?
4. What will determine your feeds and speeds for operations?
5. When should you run a simulation?

Answers:

1. Where should you place your origin?
   A top corner
2. How do you select a bit to use in an operation?
   Through the tool library
3. When should you use a drill operation?
   To plunge into materials quickly and safely
4. What will determine your feeds and speeds for operations?
   Your CNC's capabilities, your material, and its thickness
5. When should you run a simulation?
   Always…

Try it!

- Open fusion 360 and create a block with a hole in it
- Generate the CAM and post process



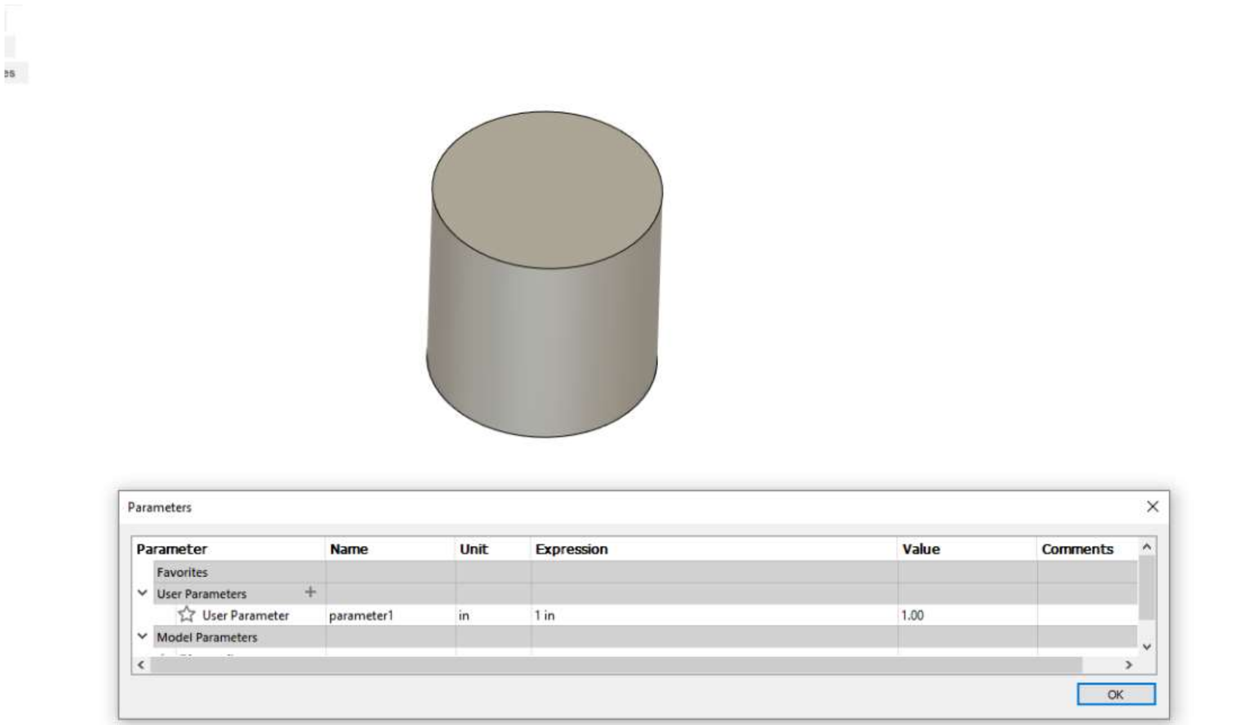
Recommended Further Learning

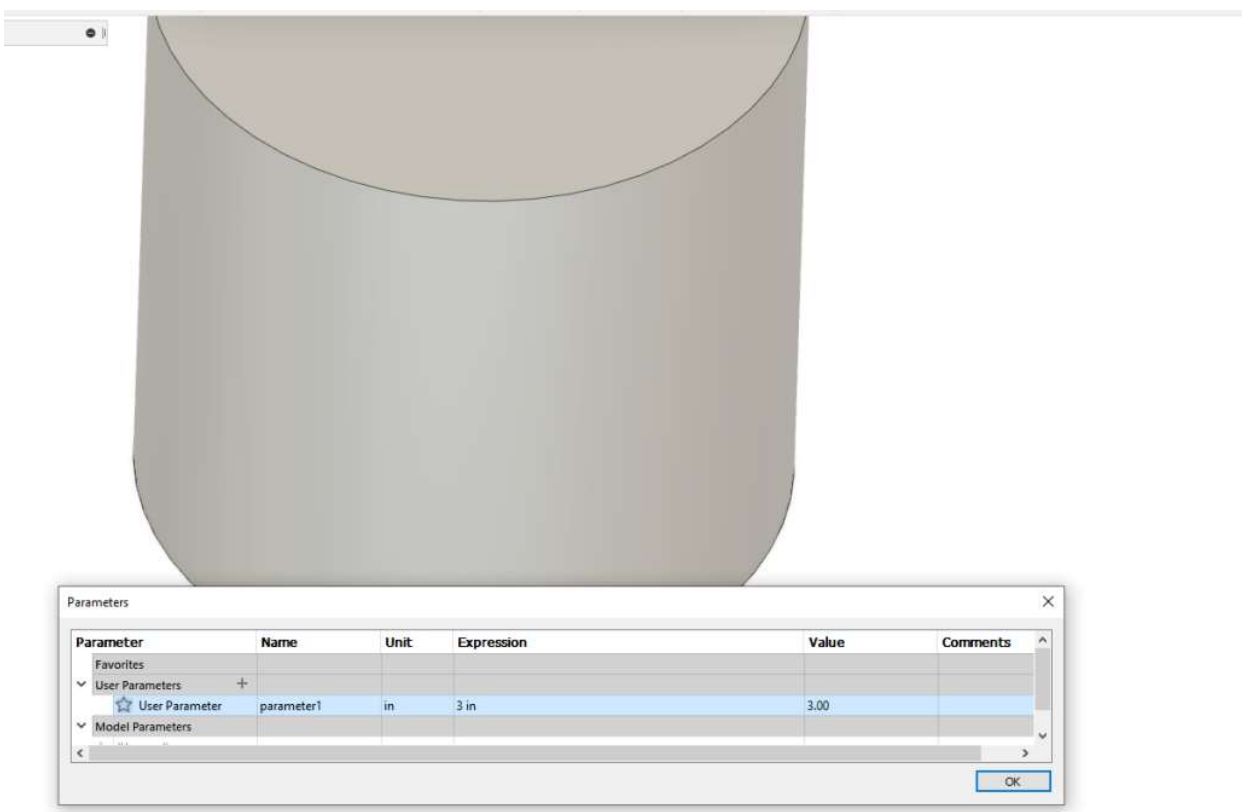
- Introduction to CAD, CAM, and Practical CNC Machining (autodesk.com)

Reflection on Parametric Design lesson by Grant Kobes

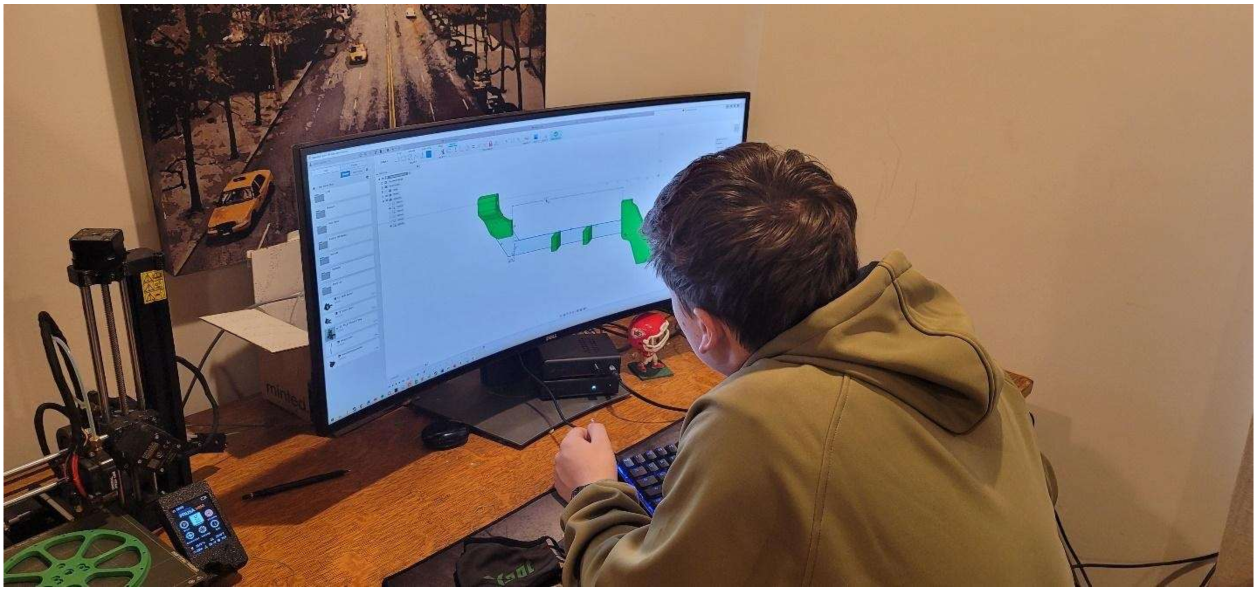
Mr. Patrick instructed our team on the use of parametric modeling using Fusion 360. While most of us have used various CAD problems in the past, we were not familiar with Fusion 360. Parametric modeling has helped us collaborate virtually this season while designing our machine. In the past, we have experienced issues with consistency of design when multiple team members try to CAD at the same time. For instance, while created custom 3D printed parts, we had issues keeping hole size consistent. While one member worked on a part in his sketch, the printed hole size was created and saved in his design only. Another member working on a separate sketch might have an entirely different hole size incorporated into his design. These plans made it difficult to fuse the work together into the final assembled design. The parametric design feature of Fusion 360 enables us the create standard hole sizes on our main variables page so that each individual sketch made after would automatically use the standard hole size. The beauty of Fusion 360's parameters is that now every part sketched "fuses" seamless together into the final design.

Fusion 360 is also invaluable as we train new students on the CAD process. Our engineering notebook contains standard parameter for each custom hole size that we routinely use. New students can now simply integrate these parameters from the onset of their sketch so that they know their design will print and integrate onto our existing work.





Grant's first practice with parameters

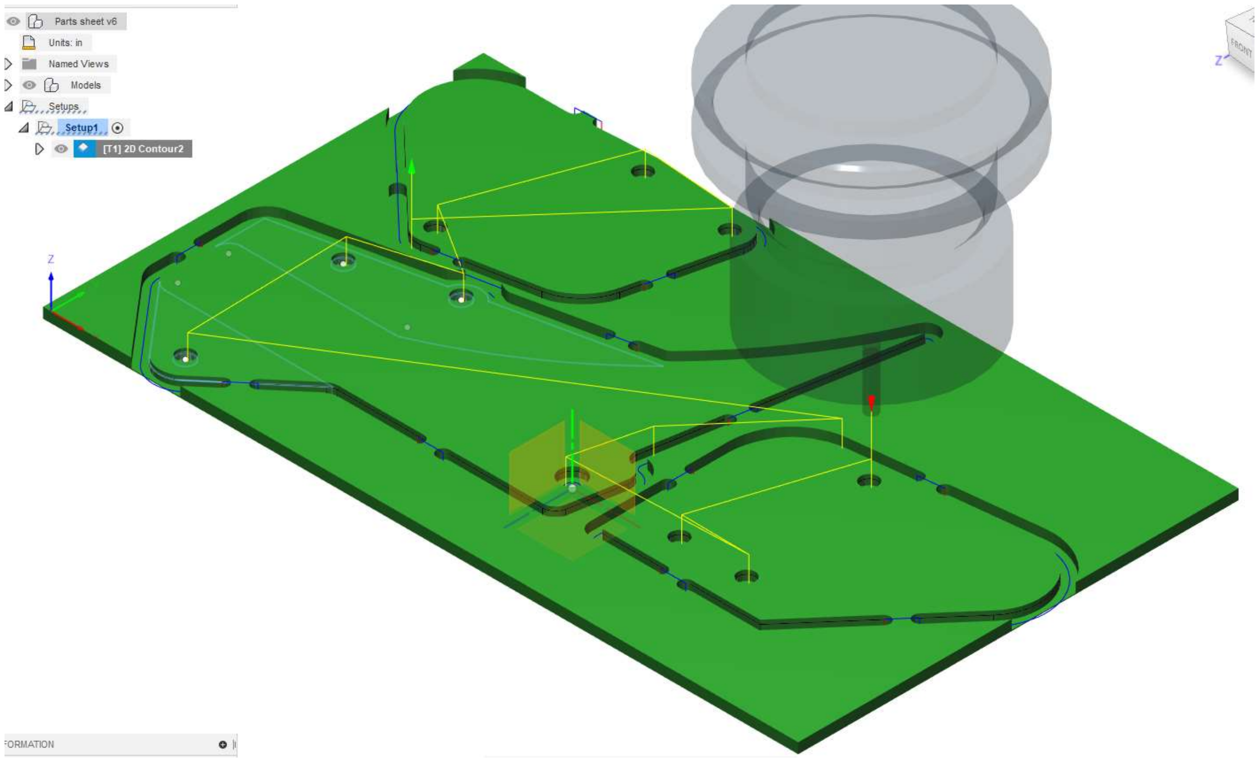


Grant working on a 3D Printed Part in Fusion using the new hole size library

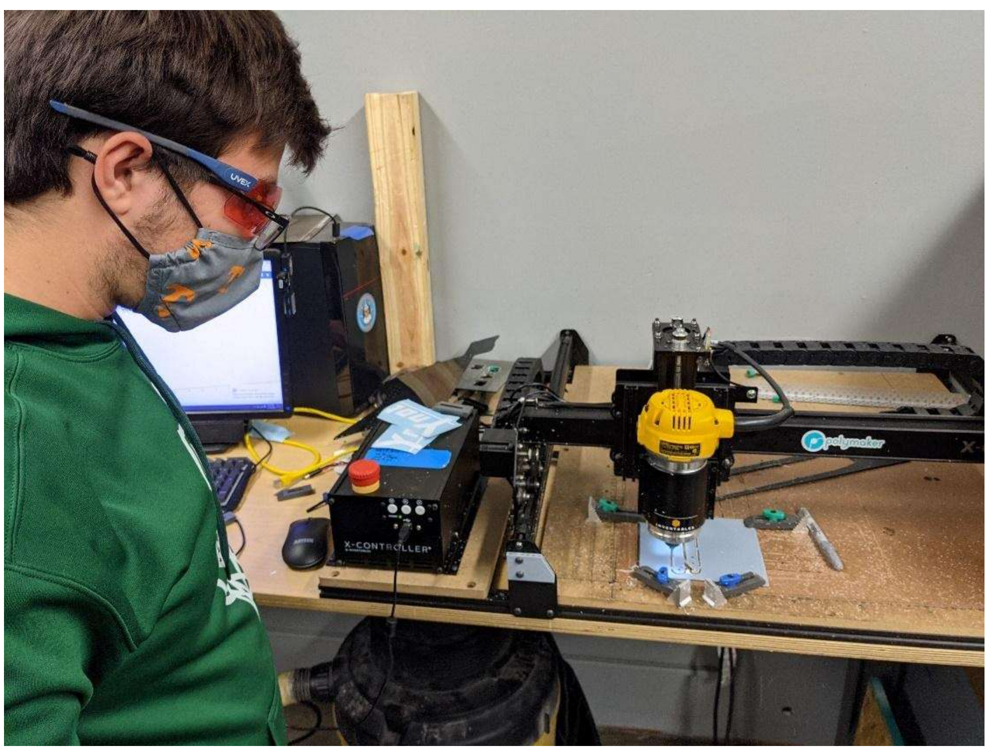
Reflection on CAM lesson by Tony Spezia

Mr. Patrick instructed on team on the use of Fusion 360 for computer aided manufacturing (CAM). Because team YNOT focuses on pushing students to design and manufacture multiple custom parts during the build season as a means to learn these processes through hands-on experience, it is essential that we have the most efficient production design and manufacturing capabilities. Last year, we bought a X-Carve CNC router in order to expand our manufacturing capacity. We wanted to be able to create more detailed polycarbonate parts as well as expand in to producing our own sheet metal parts. The problem was that the CAM software that came with our router (Easel) is not efficient at creating parts made using other software. With Fusion's CAM software, we use a specific profile for our CNC which allows to use create the most detailed parts in Fusion and easily cut them.

The beauty of this CAM lesson is that we now have a document on hand that gives any member the precise instructions for manufacturing parts using Fusion 360 CAM. In past months, only two students were familiar with the process which made it difficult to make progress on the build if they were not present at the meeting. Now every student on our team is capable of machining his or her design independently.



Tony's CAM he generated from the lesson



Tony cutting out a part he made using the CAM program he wrote.