Fusion 360 Tutorial

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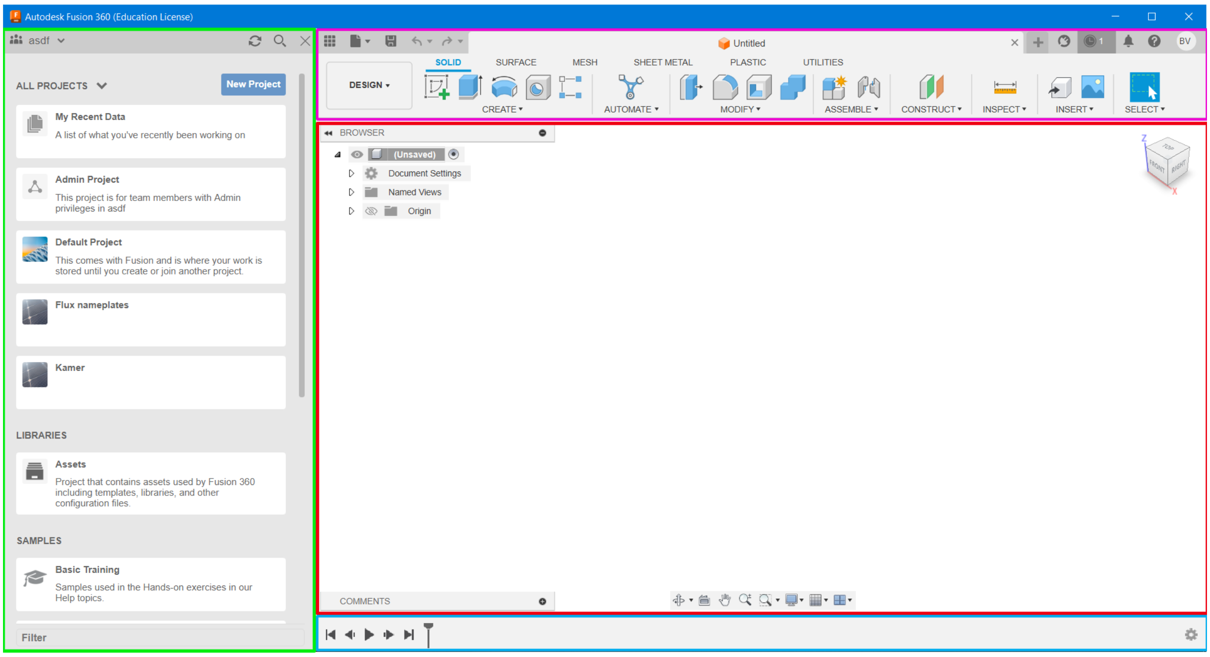


Figure 1: The starting interface

1 Introduction

Fusion 360 is a CAD program. CAD is short for Computer Aided Design. This is because it is used to design 2D or 3D objects, instead of having to draw them on paper.
There are a lot of different CAD programs, each having its own unique features. Fusion 360 is software that is also used in the industry a lot, and freely accessible for students through an educational license.
By now you should have downloaded Fusion and be ready to start designing!

In this workshop we will use a design of a fidget toy with 2 rotating gears, which you will reproduce. You can find the design at: https://www.thingiverse.com/thing:2173552

2 Basic topics

2.1 Interface

Before we start modeling complex gear systems, it's important to know what we're looking at. When starting up Fusion 360 you will be greeted by a window similar to Figure 1, which we divided in 4 parts: data panel, modeling tools, timeline and model window.

Data panel

Here you can manage all of your projects, libraries and find a collection of samples provided by AutoDesk. These files are conveniently synced to the cloud and accessible from the online version of Fusion. For now, only two buttons here will be important for creating your first project. To start, click on "New Project" on the top right of the

Figure 2: Solid workspace toolbar

green box to create your project, and call it "Volundr workshop". Double-click your newly created project and hit "CTRL-s" on your keyboard to save your first file. A new window will pop up where you can name this file and choose a location. Call it "Gear" and save it to the project you created earlier. Congratulations, you saved your first (albeit empty) model!

Toolbar

In Figure 2 you will find all tools to create your models. Going from left to right, in the orange box are buttons to hide and show the data panel, edit the file, save the file, undo, redo and file tabs. Just like tabs in your web browser, you can have multiple models open at the same time. We only have one open right now with the name of "Gear V1". Fusion 360 automatically adds version control to your files, more about that in subsection 2.5. Lastly then, you can open a new window with the plus button. Remember to save it though, as this doesn't automatically make a new file!

In the pink box you can find different workspaces. We will only be using the design workspace, which should be selected by default. When you become more comfortable with this software, you can start doing things like create attractive product renders for customers, animations to assist in assembly or do simulations with your model.

Next in the blue box are all actual editing tools. These are organized in 6 tabs: solid, surface, mesh, sheet metal, plastic and utilities. We will only be using the solid tab, but feel free to experiment with tools in other tabs after you have completed the workshop. To get a detailed explanation on each tool, you can hover over them. You don't need many tools to start creating things however, and this guide will teach you a few of the

Figure 3: Modelling window

fundamental tools to get started. Once you're done looking around in this menu, you can move on.

Model window

Next we have the model window, as shown in Figure 3. This is where the actual modelling happens. For now, it is mostly a large white space but it's time to change this. Start by taking a look at in the orange box. In this browser you can view all objects in your design. Your design starts at the origin, so where the coördinates are (0,0,0). In Figure 3 it's still invisible, if that's the case for you, you can make it visible by clicking the eye icon to the left of this folder. In the green box you can see the angle of your perspective. It can also be used to get straight angles on particular sides by clicking on the box. The pink box provides some more tools for moving around and changing display settings. Finally, if you ever notice something about your model you want to get back on later, there is the "Comment" window in the blue box, which will save the current position of your camera.

2.2 Workflow

Sketches

A simple way of creating 3D shapes in Fusion 360, is to start with 2D shapes called Sketches. Together with the Extrude tool, these will be your bread and butter. Create your first sketch by clicking the "Create sketch" button, on the left in blue box in Figure 2. Select the plane you want to start on, and you will be brought to the Sketch screen, as seen in Figure 8. Just like the other interfaces, you will find a bunch of different tools to create and modify shapes.

Let's start by creating a 1 by 4 centimeter rectangle. Press the "2-point Rectangle" button, seen in the blue rectangle in Figure 8. Click two times in your model window to set down the top-left and bottom-right corners of your rectangle to create a dimensionless rectangle. To add the specified dimensions, go to the Create dropdown menu and click "Sketch Dimension" all the way at the bottom, as seen in Figure 10, or use the shortcut "D". Once the tool has been activated, click either of the horizontal sides. You can then click on the location you want your dimension to be placed. You can place it anywhere, the location won't influence the dimension itself, but it can be nice to make sure your projects don't get to cluttered. Once you've done that, you will see a window as in Figure 11. All dimensions are in millimeters by default, so to make the rectangle 1 centimeter wide, type 10 and press enter. Similarly you can make it 4 centimeter high by doing the same with either of the vertical sides.

Constraints

A shape is defined by its constraints. For example, a rectangle has two horizontal and two vertical sides. Its corners can be anywhere in the model, but those two constraints cannot be broken to be a rectangle. In Figure 11 you can see the icons off these constraints on all of its sides, and you exactly what they mean by clicking them and looking at the bottom right of your model window.

To change the shape of your rectangle, click the horizontal constraint on the bottom side and delete it, as well as the vertical dimension. Now you can select the bottom-left corner of the rectangle and press "M" to move it. Your screen should look like Figure 12, where you can use the arrows to move the corner around. Note that it's only possible to move it up or down, and not left and right. This is because there is still the vertical constraint. To add the horizontal constraint back, use the "Horizontal/Vertical" tool, marked green in Figure 8. This tool will move the selected line to whichever alignment is closer.

These constraints can help you create more complex shapes, and help you maintain important shapes by throwing errors. The latter is especially important when your projects become bigger, and will be demonstrated later on.

Finally, you can finish your sketch by hitting "Finish Sketch" on the top right of your toolbar.

2.3 Extruding

It's time to create a 3D shape out of our sketch using the "Extrude" tool to the right of the "Create Sketch" tool. Activate it, and select your newly created shape (if it wasn't selected already).

You will see a window similar to Figure 4. To make it 1 centimeter thick, change the distance to 10 mm and press OK. Here you have created a new body, which you can see in the object browser under bodies. You'll also notice that your sketch has been made invisible. It is good practice to use a sketch for one single body, as this will make it easier to make adjustments later on. More on this in subsection 2.5.

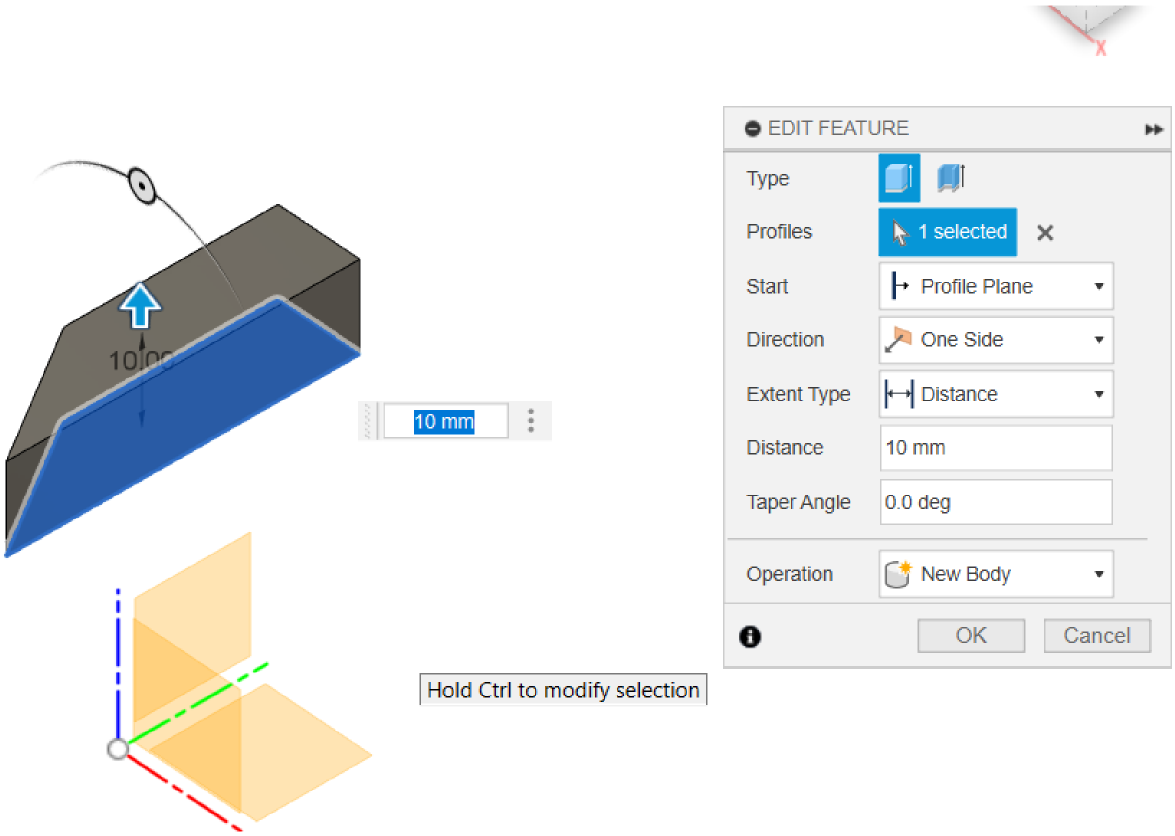


Figure 4: Sketch shape creation dropdown menu

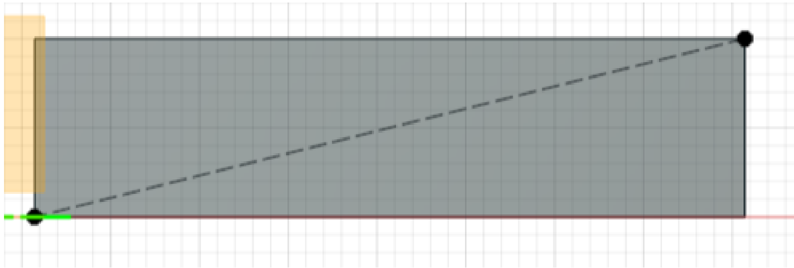


Figure 5: Square with a construction line through the middle

2.4 More complicated shapes

We just used one of the origin planes to build this body, but to create more complex shapes, you'll want to modify your existing body. You can do this by creating a sketch on one of the sides of the body. Simply do this by creating a sketch and selecting one of the planes of your body. You'll enter the sketch window again, but now you can also select the geometry of the plane you are working on.

Construction lines

Let's say we want to create a circle straight in the middle of this plane. You'll want to find the middle of this plane first, and we'll use construction lines for this. The difference between a regular line and a construction line is that a construction line will not affect the shape of a plane it's on.

In the sketch palette window on the right on the top you can select the linetype. You'll find it in Figure 6 in the blue square.

Activate it and create a line by using the "Create Line" tool on the left of your toolbar or by hitting L, and draw a line from two opposing corners. Your plane should look something like Figure 5, note the dashed line. You can change a regular line to a sketch line and back by selecting it and pressing the "Construction line" button again.

Now make sure that Construction mode is not active anymore, and in the "Create" dropdown menu, select a Point and put it anywhere on the construction line (except the outer points). Using the "MidPoint" constraint in the "Constraints" dropdown menu, you can put this point exactly in the middle of the line.

Using a combination of constraints, construction lines and some geometric reasoning, you can get very creative with the shapes you build. It's almost like high school mathematics! Now you can create a circle with the C shortcut, select the midpoint, and give it a radius that fits within the plane. Finish your sketch now.

Extrusion operations

Earlier you have used the "New body" operation with the extrustion tool, but it can do more than that. Activate the extrusion tool again and select the circle you just made. At the bottom of the extrusion window, you'll find a dropdown menu with different operations. Move the arrow in the model window outside of the body, and note how

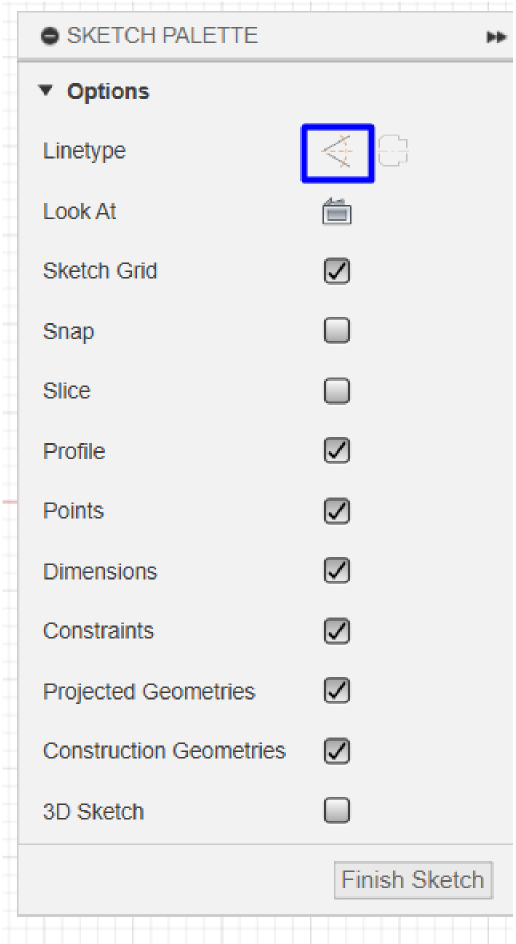


Figure 6: Sketch palette

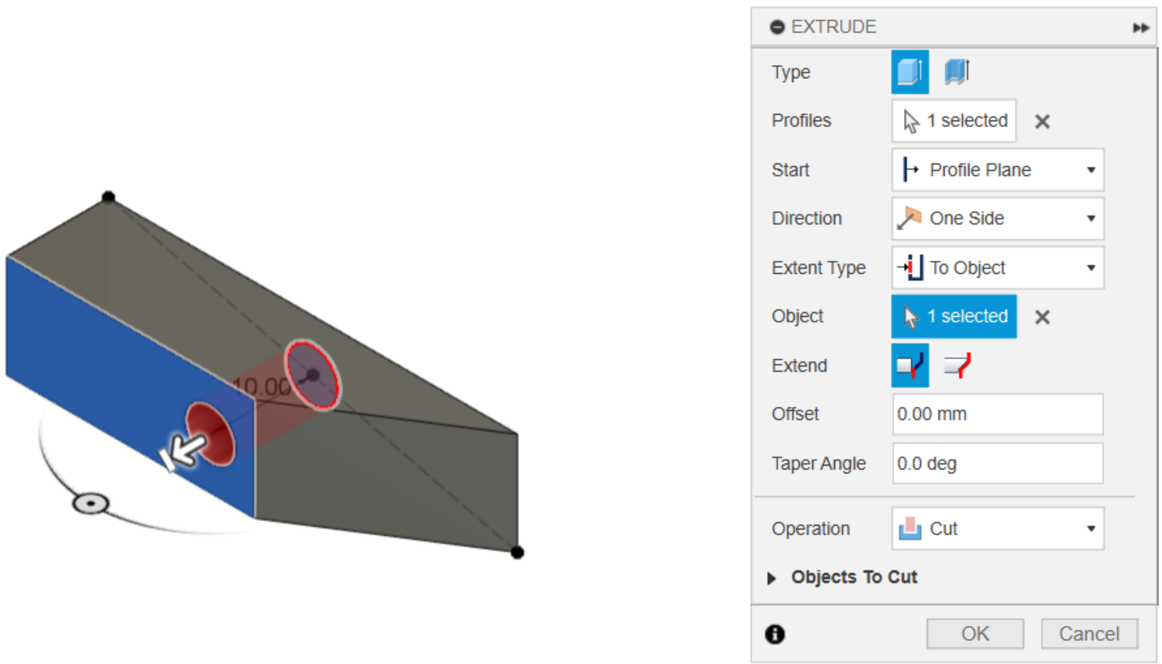


Figure 7: Extruding to object

you'll have started a *join* operation, meaning you'll add something to the original body. You can change the operation to "New body", which would simply create a new body. Now move the arrow inside of the body, and note how the operation changes to a *cut* operation. As expected, this will use the circle to cut into the body for whatever length specified. In the dropdown menu you can change it to a *intersect* operation, which will do the opposite of the *cut* operation, and only keep the intersection of the body and the circle you're extruding.

We're going to create a hole through the whole body. You could do this doing a very long cutting extrusion, but there's a cleaner method for this. In the extrusion window, change the "Extend type" to "To Object", and select the plane on the opposite side. This should look like Figure 7. Press OK now to finish the extrusion.

2.5 Timeline

By now you've seen how to create some basic and more complex shapes, but one of the powers of Fusion 360 (and similar software), is it's timeline feature. You can use this to go back in time and make changes, add or remove things. Fusion will then do its best to propagate these changes into the future.

If you've followed along, your timeline should look something like Figure 13. Move the marker around and watch your previous operations undo and redo themselves. Double click the first operation, which is where you made the first sketch. You'll see your original sketch here again. Try to change the shape somewhat, by making it slightly wider for example, and finish your sketch. Now the updated sketch is used for the body!

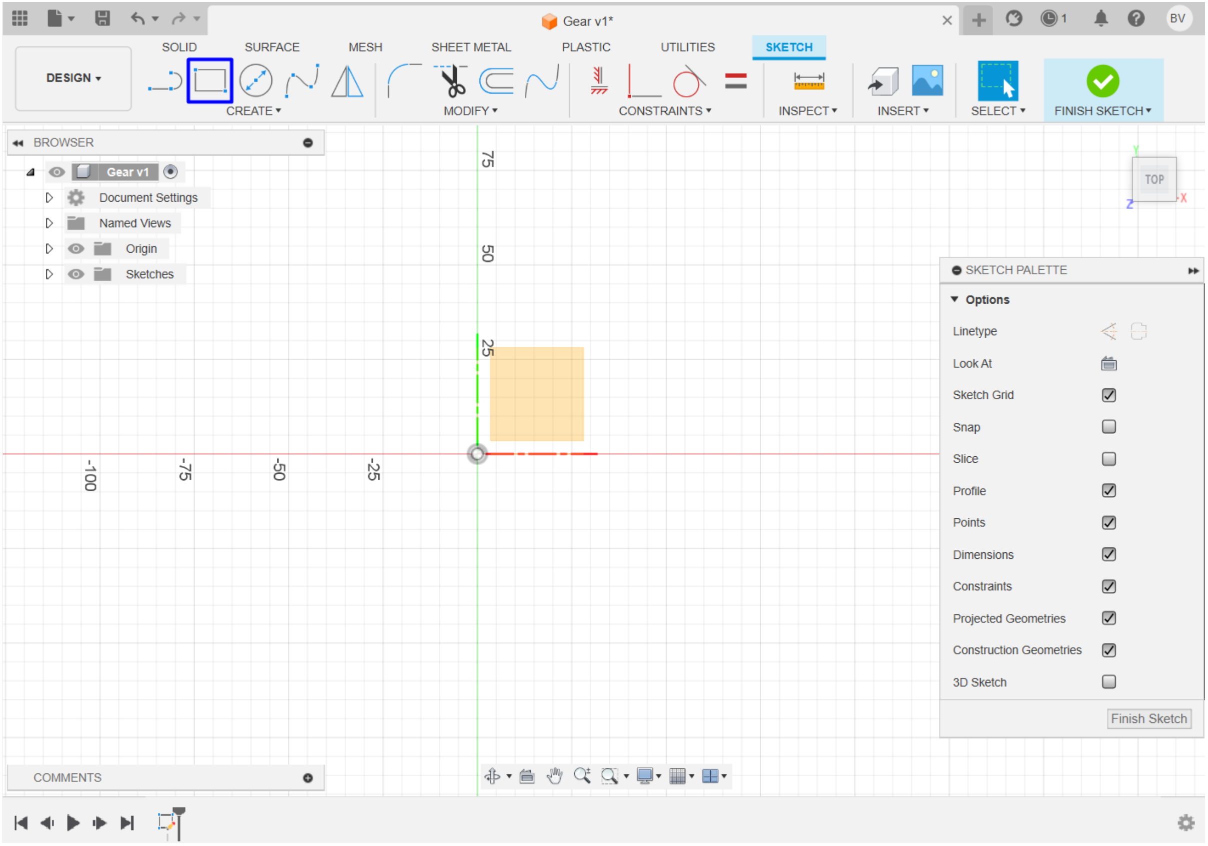


Figure 8: Sketch interface

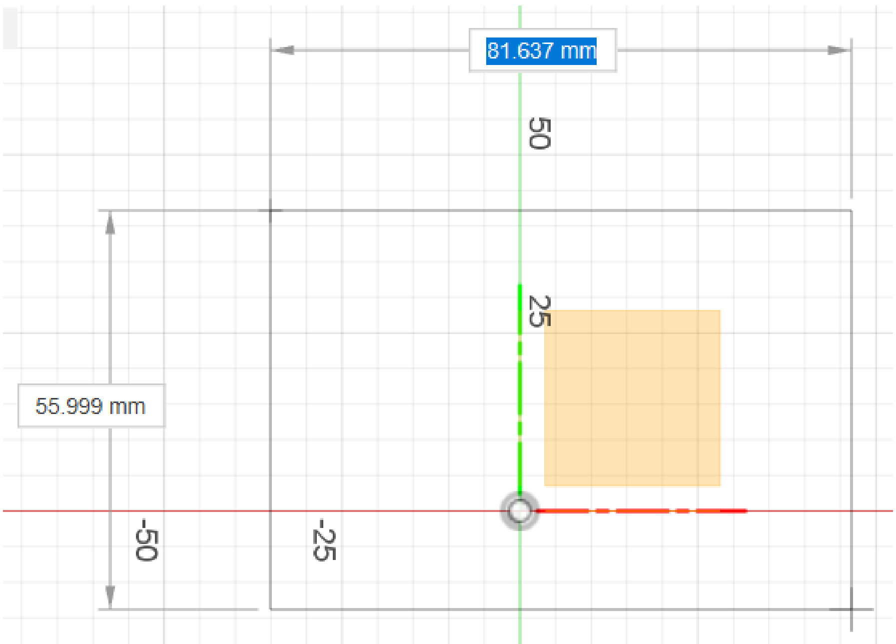


Figure 9: Creating a rectangle sketch with constrained size

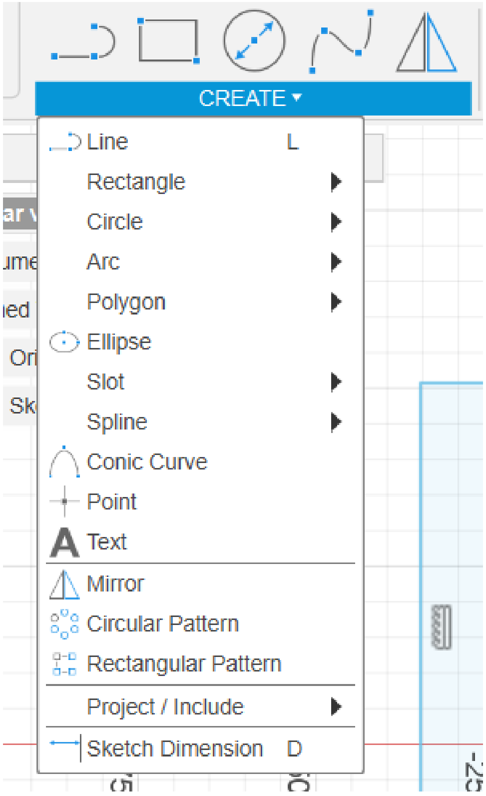


Figure 10: Sketch shape creation dropdown menu

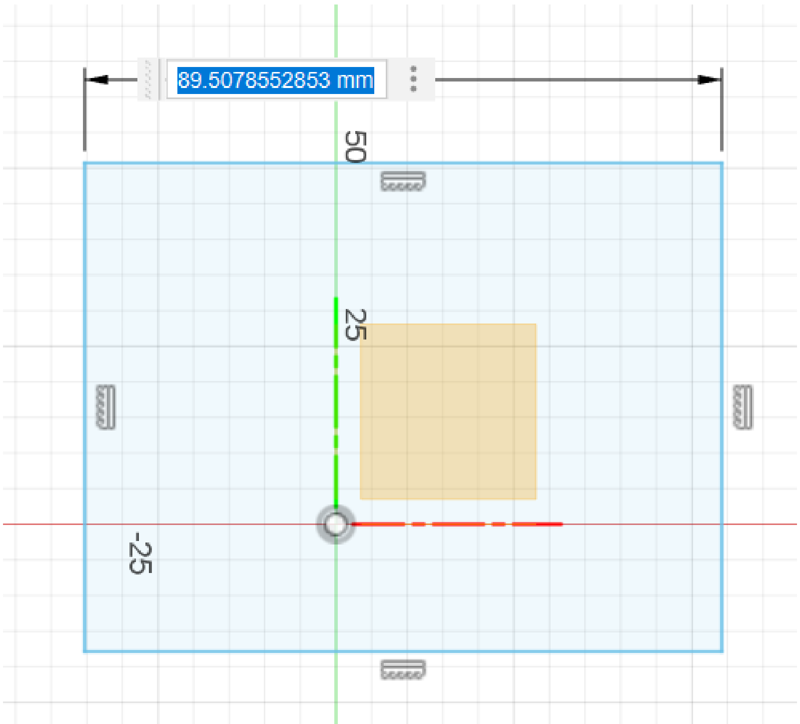


Figure 11: Sketch shape creation dropdown menu

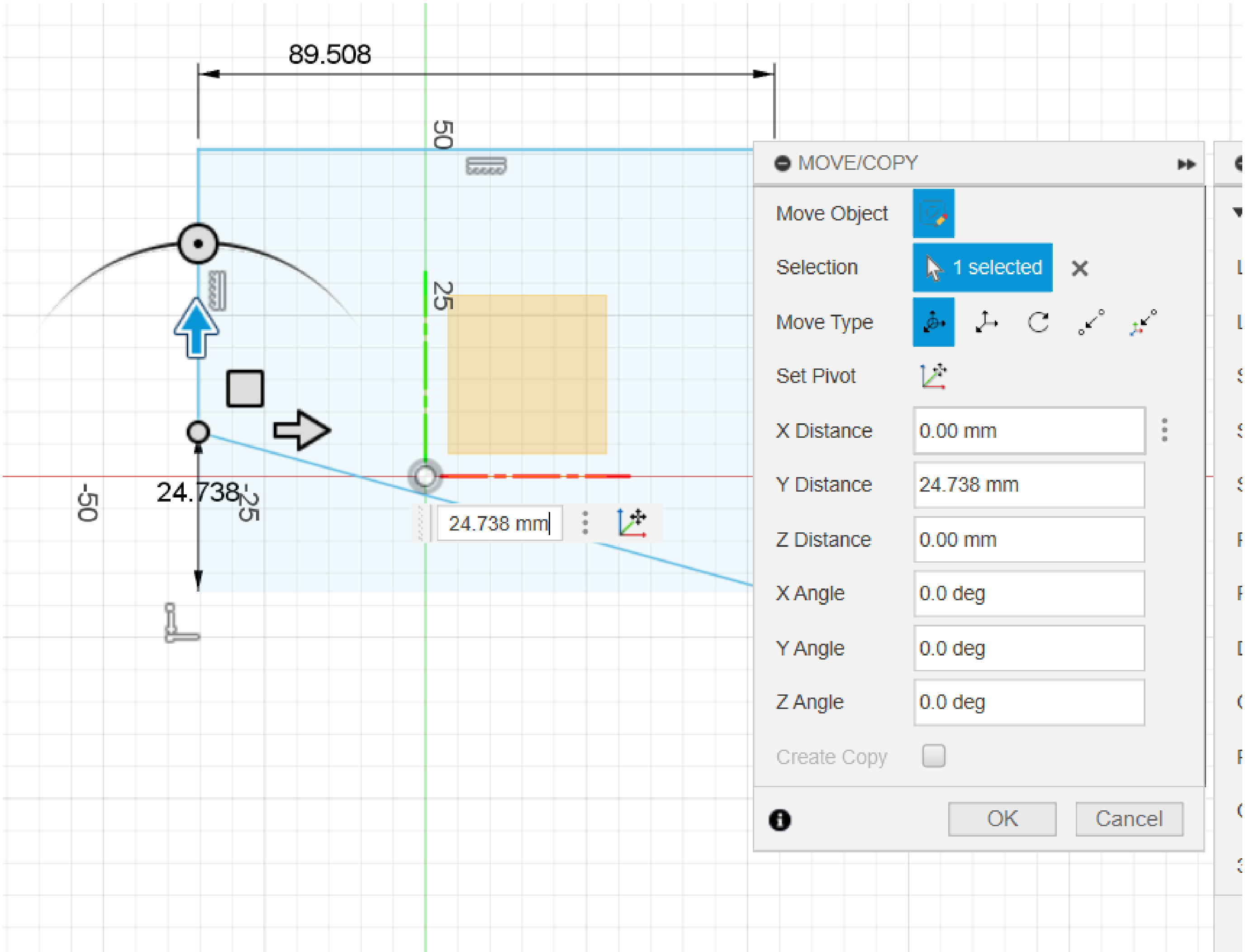


Figure 12: Sketch shape creation dropdown menu

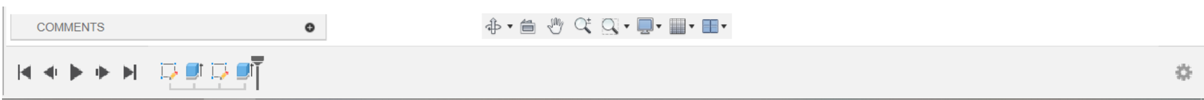
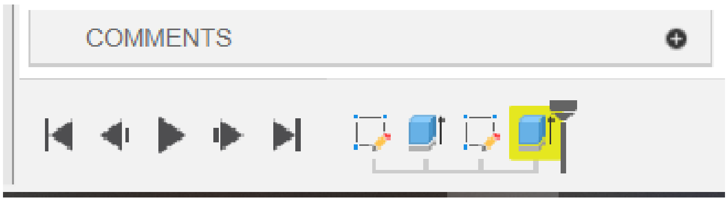
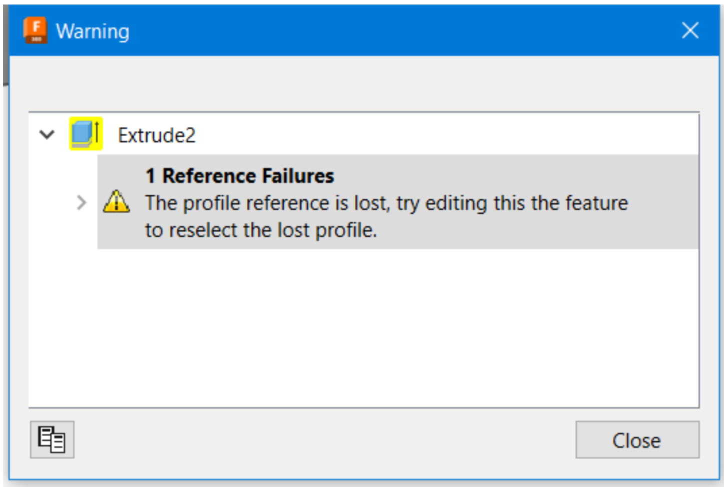


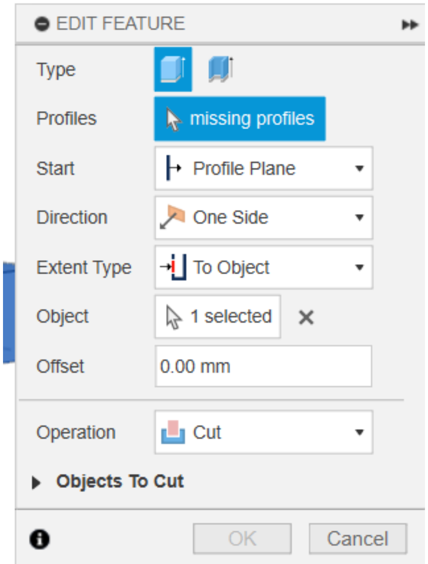
Figure 13: The timeline



(a) The yellow background in the timeline indicates a warning or error in that step



(b)



(c)

Figure 14

Preventing errors

Using the timeline is a very powerful tool, but you have to be careful to keep your geometry in-tact. For example, extrusions need a plane to extrude. Edit the sketch where you made a circle on your body, and delete the circle. This will generate a warning, as you can see in Figure 14a.

Right-click the problematic extrusion and click "Review warning". The window from Figure 14b should pop up with a *reference failure*.

In our case, the plane we extruded was deleted, thus it cannot reference this geometry anymore. Close the window, and double-click the same extrusion again. It should look like Figure 14c now, where you should note the "missing profiles" message.

Note that when you deleted the circle and created the error, the hole didn't disappear right away. Even though there wasn't a direct effect, it's important to fix errors and warning as soon as possible to prevent bigger issues later in your timeline.

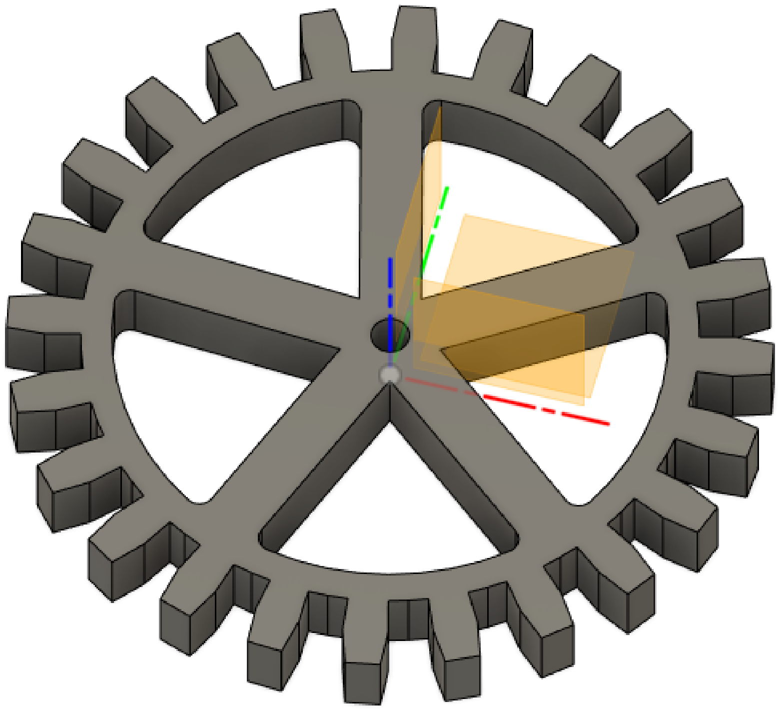


Figure 15: Final gear shape

Outer diameter	50 mm
Inner diameter	40 mm
Number of teeth	24
Number of center bars	5

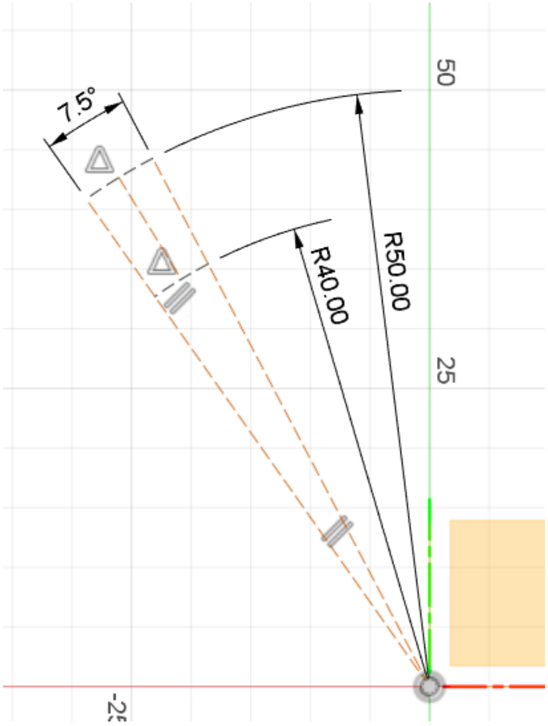
Table 1: Gear specifications

2.6 Gears!

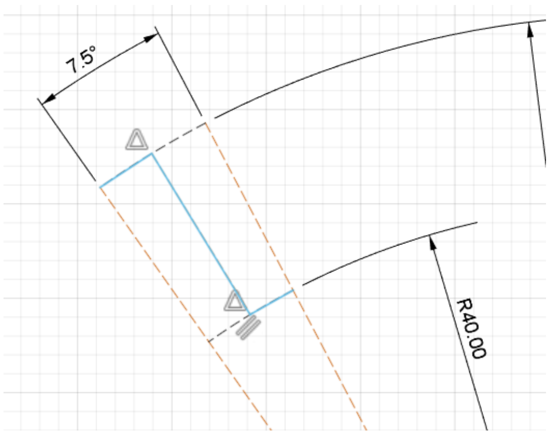
Now all the basics are out of the way, we are going to create a gear according to the specifications from Table 1. The final result should look like Figure 15.

Delete all your bodies and sketches to start out clean. There are many ways to approach this, but for this example, we will sketch a single tooth and circularly repeating that pattern. There are a lot of add-ons and other tools to making these in Fusion 360, but for the sake of learning, we'll only use basic tools we've touched on earlier.

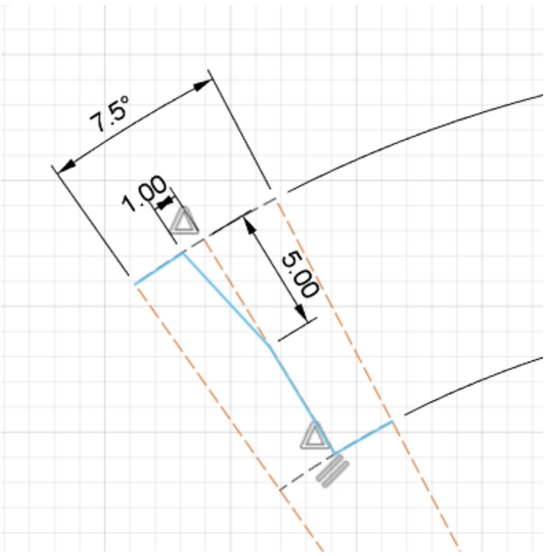
Start by drawing construction lines that will shape one half of a single tooth like in Figure 16a. This shape was made with two center point arcs, where the diameters are the inner and outer diameters and the angle $\frac{360}{24*2}$, since we want 24 teeth and we are drawing half a tooth now. With these construction lines, draw the rough shape of this half tooth and under the modify tab, use "Chamfer - Two Distance Chamfer" to create Figure 16a and Figure 16c.



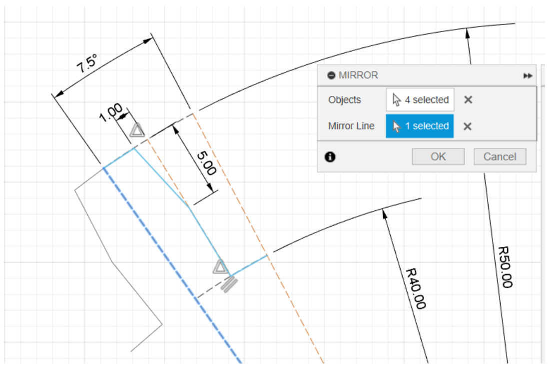
(a) Drawing construction lines



(b) Drawing broad outline of the gear



(c) Adding a Two Distance Chamfer



(d) Mirroring the shape

Figure 16

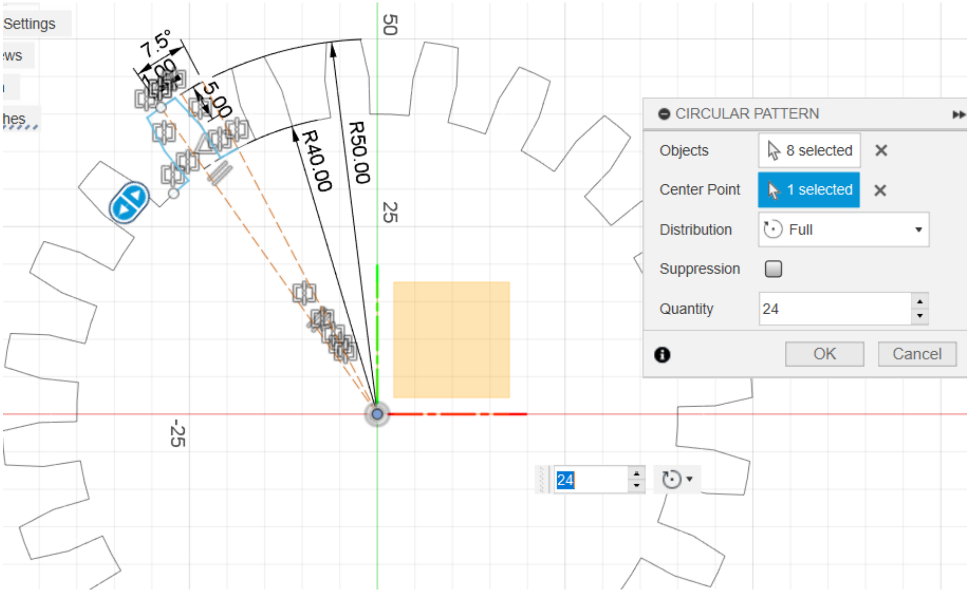


Figure 17: Rotating the tooth to create a whole gear shape

To complete the first whole tooth, use the mirror tool. This should look similar to Figure 16d. With this shape done, use the Circular pattern under the Create tab to complete the whole shape. This should look like Figure 17.

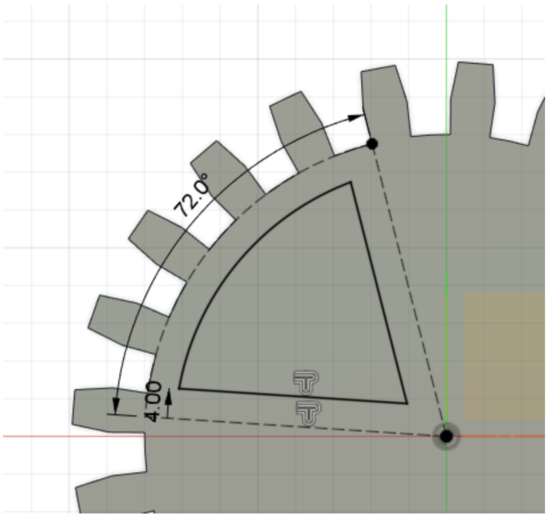
Make sure to set the quantity to the amount of teeth your gear should have, which is 24 in our case. If this worked, complete the sketch and extrude it by 10 mm. Here we our nice first gear! Now to add bars and a hole in the middle, create a sketch on the top your gear. In this example we're going to create 5 bars. Create the construction lines with the Center Point Arc tool again with a diameter from the center of the gear to the base of a tooth, and the angle of $\frac{360}{5}$. Select these construction lines, and use the Offset tool under the Modify tab to create the shape seen in Figure 18a with regular sketch lines.

You can round the corners of these with the Fillet tool under modify. Also create a small hole at the center of the gear with the circle tool to get the shape of Figure 18b. Use the Circular Pattern tool again to repeat the triangle shape that you made, like you did with the teeth earlier, and finish your sketch. Extrude your newly created shapes, and you should end up with a nice gear!

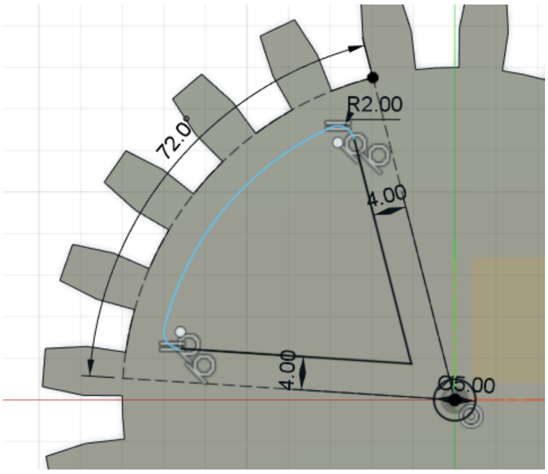
Back to the future

While it looks nice, the first sketch we made is very messy and large, making it hard to edit properly. Keeping sketches simple is key in keeping your designs modular, and making the timeline easy to use. So to fix this, we are going to go back in time and recreate our initial shape. You can do so by moving the timeline marker to just after the first sketch, and double click this sketch in the timeline to edit it. Clear all objects your made earlier year, and recreate the shape from Figure 16d. Close this shape with to lines to get the shape from Figure 19. Finish the sketch.

Move your timeline marker one step to the future again, right after the extrude. It should



(a) Using the Center Point Arc tool to create construction lines, and Offset tool to create the general bar outline



(b) Rounding the corners with the Fillet tool and create a hole in the center

Figure 18

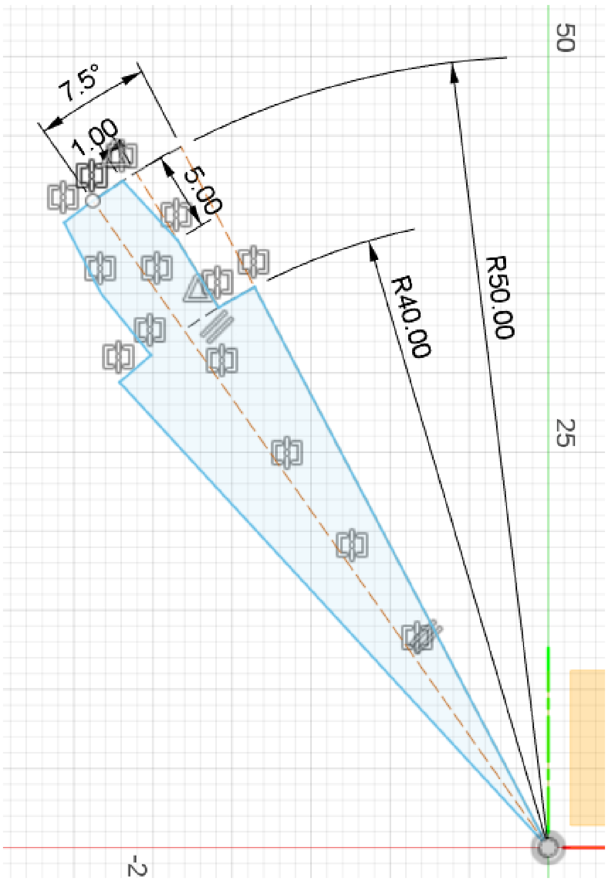


Figure 19: Sketch of a single tooth

come up with an error, because we deleted the sketch it initially extruded. Solve this error by double clicking the extrude in the timeline and selecting your newly created sketch of a single tooth. It may be invisible, in which case you can turn it visible again in the object tree. Instead of a whole gear, you'll now only have only a single tooth. Under the Create tab, go to Pattern -¿ Circular pattern and perform a similar operation as you did in the sketch before. You'll now have created 24 seperate bodies, which you can merge by selecting them all and using the Merge tool under the Modify tab. You'll now have a gear identical to the one your have created before! Move the timeline marker all the way to the front again, and the bars and hole your created earlier should appear again. The advantage of the second method of creating the gear, is that your initial sketch is less messy, making it easier to edit later on.

3 Assemblies

As you can imagine, a CAD model to design for instance a car can become quite complex. With thousands of components it will become very confusing. Therefore, most CAD software has some structure implemented, which divides the entire model into different parts: sub-assemblies. For example, a car (the complete assembly) might exist out of a sub-assembly for the motor, one for the wheels, one for the chairs etc. Below follows an explanation of how to create such a structure, by giving generic examples. You can follow along by quickly creating your own bodies or using the gear from the first section. After this explanation an exercise will follow to complete the fidget toy by creating the casing. With the tools provided in the explanation you should be able to create the casing yourself, although if you run into any problems you can of course ask any present Volundr member.

3.1 Components

You have already learned that every object in fusion consist out of bodies. Bodies are the blocks all components are made of, the most basic level in the assembly structure. If you want to make more complex structures, you can turn the body into a component. A component is a more independent part of an assembly. It has its own origin plane, and can be objected to constraints and joints (which we will see later). To create a component from a body, right click the body in the navigation pane and select "Create components from bodies": If you select multiple bodies, they will all be turned into components.

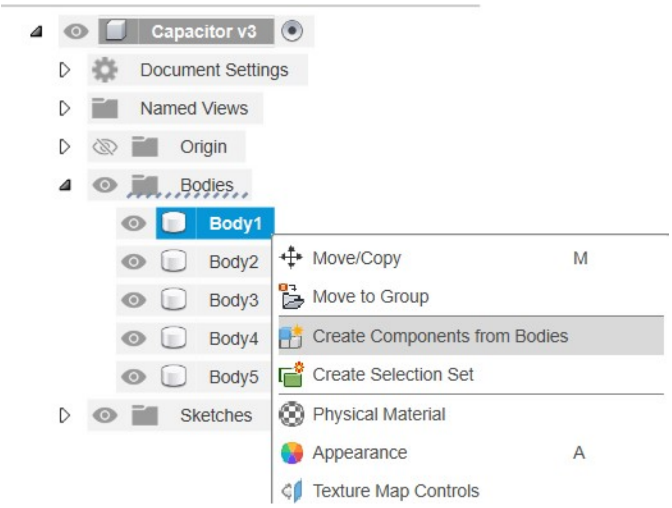


Figure 20: Create components from bodies

The components are no longer locked in place, so you can drag them around. If you accidentally messed up the position of your components and want to place them back where they were, you can press the "Revert Position" button in the top right corner. (see Figure 21) This will place them back: If you want to update the position that is saved for the components, you can do that with the button to the left of it: "Capture Position". Furthermore, if you want to lock a component in its place, right click the component again, and select "Ground".

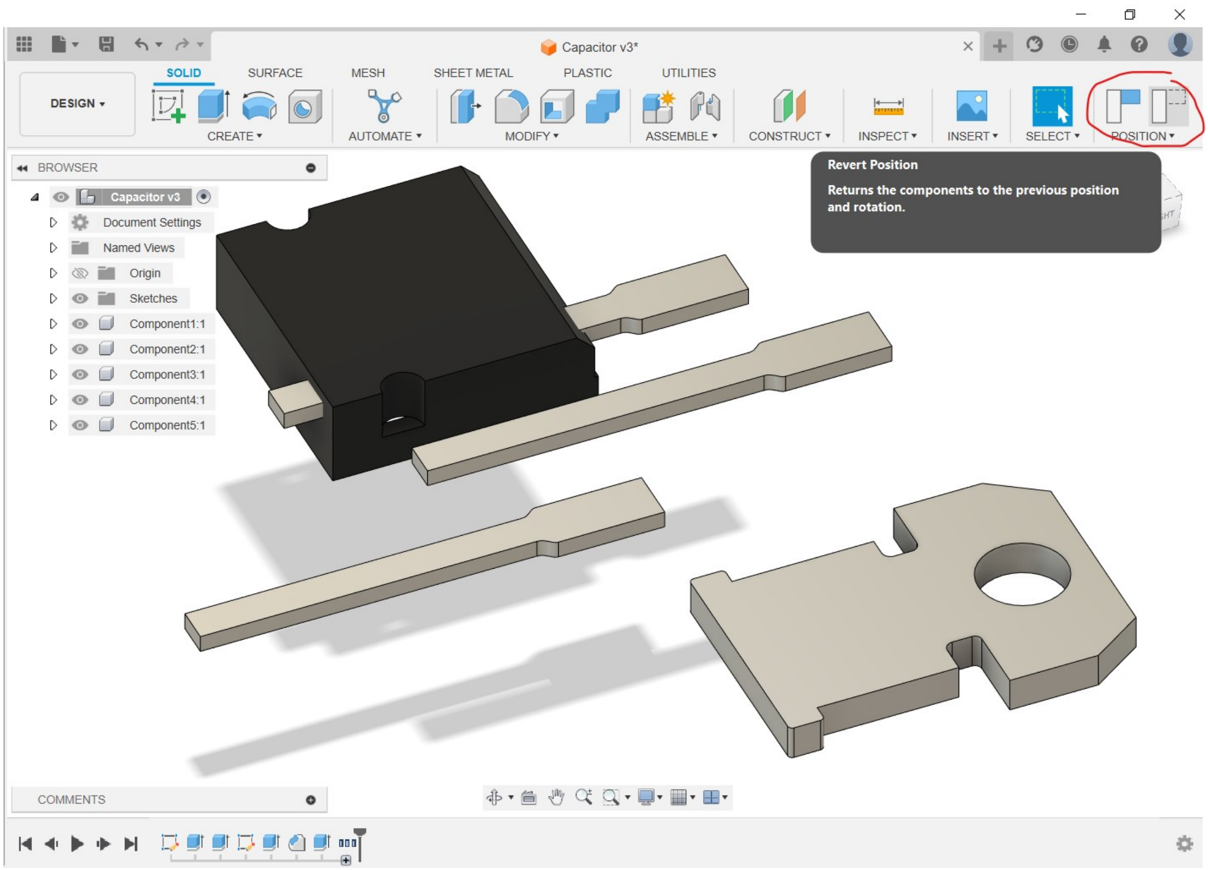


Figure 21: The Revert and Capture position buttons

When to create a component?

There are different strategies concerning the creation of components. Just now we first made a body, and later turned it into a component. This works fine for less complex projects, however, if you have a more complex project, and want to store everything in one file, it is smart to already start with the creation of components at the start of your project. The way to do this is when creating your file, is to create a file for your entire project, and create an empty component for every separate part you are going to make. For example, if you name your file "Big project", and right click it to add a "New Component" called "Component1" you will get the navigation pane in figure 22.

At this point, everything you will add to the project will be added to the main folder. To start working on the specific component, you need to 'Activate' it. This can be done by clicking the small round button on the right of the component that appears when hovering above it (figure 23). This way, all sketches and bodies you create will be stored in this component instead of the large project, making it a lot more clear.

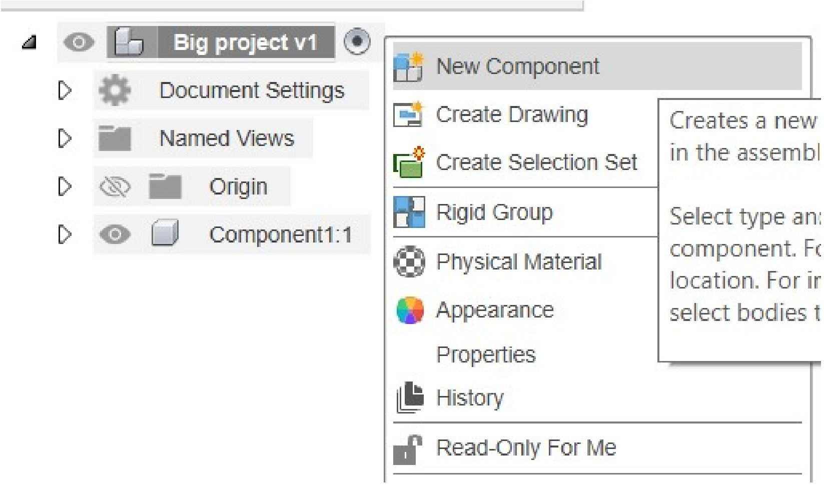


Figure 22: Adding a component to an assembly

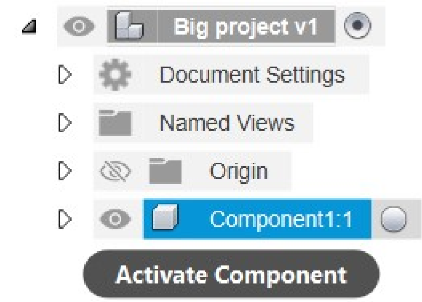


Figure 23: Activating a component to work in it

3.2 Combining Components

Alignment

Maybe you have already tried moving components around a bit, then you might imagine it can be quite a trouble to properly align a lot of components by just dragging. Fortunately it is possible to 'align' components to eachother, which means selecting certain geometry features that will be placed on the same line, surface, parallel etc. To do this, open the 'Align' (Figure 24) window from 'Modify'.

Here you can select two geometric features that will be aligned. This is not a permanent fix, so you can still drag the components around. We will see the fixing of components later in the 'Joints' part. You can use a variety of features to align to eachother, most commonly:

- Face to Face: Align two surfaces to the same face.
- Point to Point: Move two points of two different components to eachother.
- Axis to Axis: By selecting an axis (or a circular face) and another axis, they will be aligned to eachother.

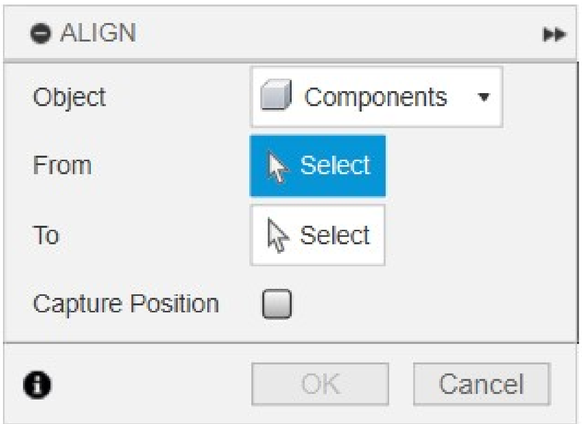


Figure 24: The Align window

In combination with the 'Move' function which allows you to move one (or more) bodies/components around you should be able to position all components where you would like them.

3.3 Joints

Placing components next to eachother is a great functionality. However, if you would like to for instance simulate the moving of a component, or test a feature of your design, you can include Joints in your CAD model. Joints are a way to (permanently) fix components to eachother, while still being albe to give them certain degrees of freedom. For example, if you are making a robot arm, and you would like to move it, you can align the two arm parts of the robot, and then add a joint to allow it to rotate. This way, you can simulate the movements of the robot arm.

Fixing components

The first step in adding joints is selecting your base. You will need to choose a component that will be fixed in its place. You can do this by right-clicking the component in the component overview and selecting 'Ground' (See Figure 25). In this case I grounded the base of the arm, since that will not need to move.

Adding the Joint

To add the joint, select the 'Joint' option in the taskbar (Figure 26. For this example I made a simple base and small arm, with both a hole of 10mm.) This will give you a few options. First of all, you will need to select where your components will be joined. This depends on the type of joint you want to add, and could be a point, edge or face. In the example we are adding a revolute joint, so we will select the axis of rotation on both components (Figure 27). Once you have selected both circles on both components, you need to select the type of joint you want to add. This can be one of several options, but the most important ones are the revolute, slider and fixed joint (Figure 28). In this case we select the revolute joint.

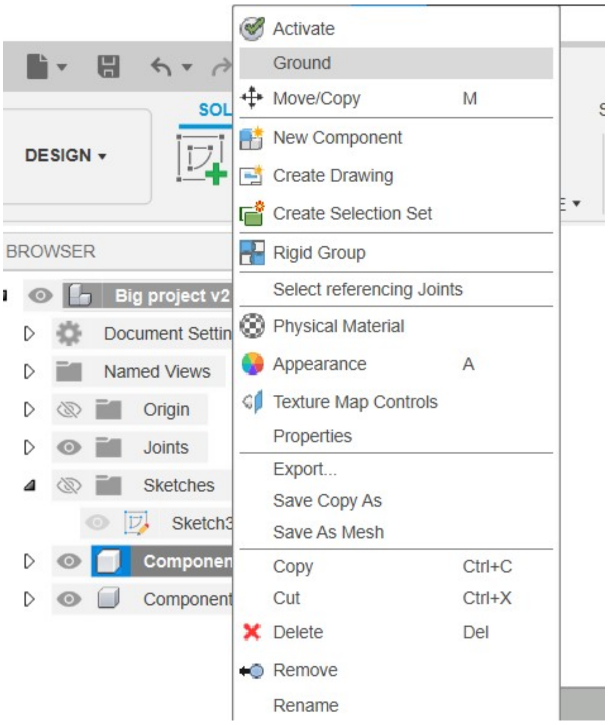


Figure 25: Grounding a component

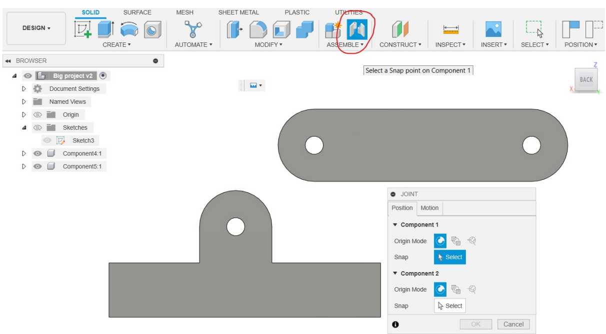


Figure 26: Adding a Joint

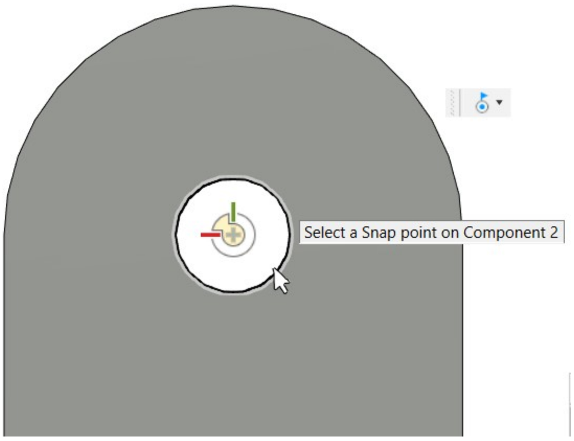


Figure 27: By selecting the circle you will align the axis to the other component

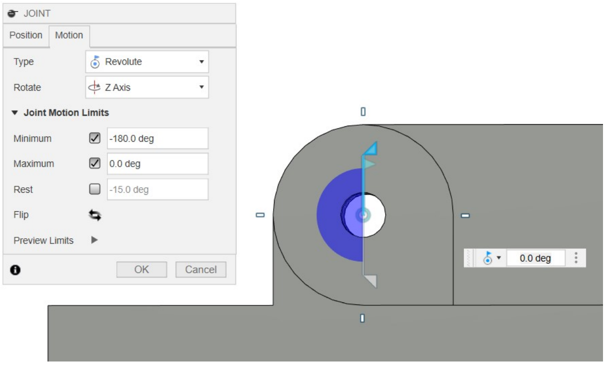


Figure 28: Selecting the type of joint and the limits

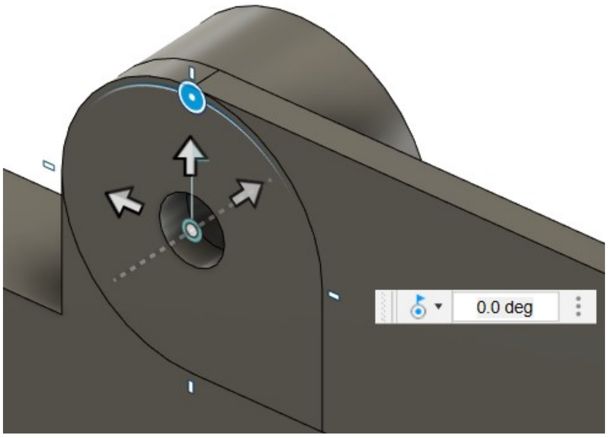


Figure 29: Joint added the wrong way

Limits

Once you added the joint, you will be able move the arm around the base infinately. If you need to limit the movements of the joint a bit, for instance a certain length for the slider joint, or a certain angle for the revolute joint, you can select the minimum and maximum option in the same menu as mentioned above. You can enter the values manually, or drag the handles around the joint to their maximum and minimum positions.

Flipping

Now that is finished, there is one more option that might be relevant. Because if you include 2 circles in the joint, Fusion 360 does not know on which side of your circle the rest of the component is. Therefore, it is possible that Fusion connects the component the wrong way around (see Figure 29). To fix this, you can click the 'Flip' button in the Joint menu, which will rotate the component (see Figure 30).

4 Final exercise

Now its time to bring all your freshly learned features together! in the Thingiverse link you have seen the gears and their casing, which you will now design and combine into one final design. Luckily you will not have to do this from scratch, since designing such a clicking mechanism can take quite some trial-and-error attempts. So what we will do is import the example and recreate that design ourselves.

4.1 Importing STL meshes

A lot of files on Thingiverse are STL files, which is a common format for 3D printing (see appendix). These files are hard to edit in software such as Fusion 360, but it is possible to import them and use them as an example. To import the file, first download the STL file from Thingiverse, and save it somewhere you can still find it. Next, in the top right corner select "Insert/Insert Mesh", as seen in figure 31. Now select the STL file to insert

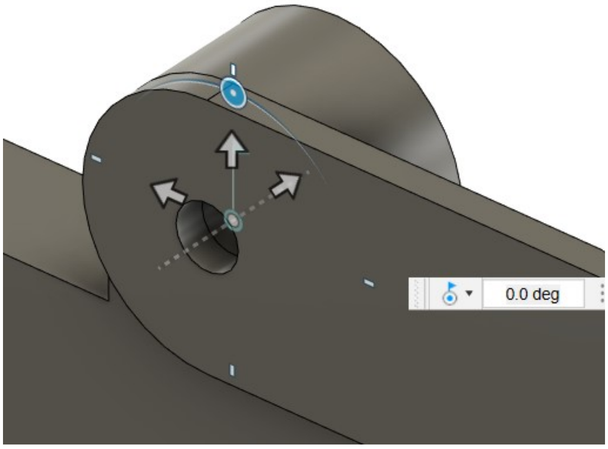


Figure 30: After selecting the Flip option

it in the design. If you want you can drag or move it to a new position, and when you are done moving click OK.

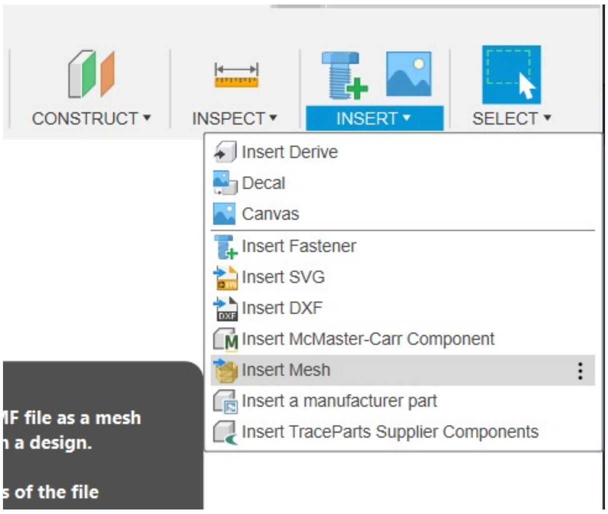


Figure 31: Insert Mesh option

You should now have something that looks like figure 32. Here you can nicely see what the component looks like, but it is not possible to create for instance a new sketch on a surface.

However, we can still use it as a reference for our own design. This can be done for instance by creating a sketch on a base plane, and 'drawing over' the mesh with a line tool. But another common option is to use the measuring tool to find the dimensions of a part of the design. You can select the measuring tool by clicking the ruler icon above "INSPECT". Then you can select 2 points, for instance the far ends of a circle/pin, and the tool will show you the distance between the 2 points, as shown in figure 33. There are

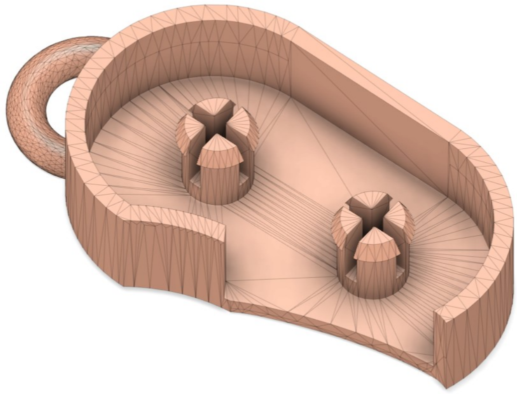


Figure 32: The inserted mesh

several ways to measure the distance, such as absolute distance or split into delta X,Y,Z lengths. You can play around with the tool to find the distances and diameters you need for your own design.

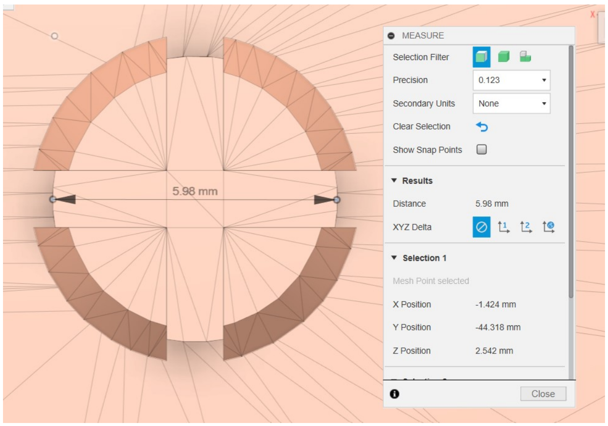


Figure 33: The measuring tool

4.2 Combining it all

Now it is your turn to combine everything and create the housing of the gears as well. For this, you can use the imported STL file as a reference. You might need some other features of Fusion, such as the chamfer, spline or arc, pipe or torus and finally fillets. Try to mess around with these features to get a better understanding of how they work, and of course if you have any questions feel free to ask!
Good luck!

5 Appendices

5.1 3D Printing

If you would like to use your CAD skills for beun projects and want to print your design, you can easily export your design to 3D printer software, also known as slicers. General examples are Cura and PrusaSlicer. This type of software can take your 3D model and convert it to gcode, which is what 3D printers use to navigate. The most common file format that for example Cura requires is a .stl file. Stl is an abbreviation of Stereolithography, and is basically used by all slicers. You can export (part of) your model in 2 ways, either by selecting one or multiple bodies in the component tree, or exporting the entire model. To export a single body right click the body you want to print in the navigation pane you can select "Save as Mesh". Here you can adjust a few settings to define for instance the amount of triangles, which is generally only interesting for more advanced or rounded models. For a basic model such as this the standard options should suffice. Press "Export" to save your model, and then you should be able to import that into e.g. Cura. The other option is to export the entire model, which can be found under file/export, and works very similar to exporting a single body.

5.2 Exporting for pcb design

CAD software can also be very useful for 2D drawings, for instance when designing pcb's. Say you would like to design a fancy pcb for a cool projects, designing the outline for the pcb in pcb software such as KiCAD (as used in the Volundr pcb-design workshop), could be quite a hassle. Especially since KiCAD is not very nice software to design such shapes. Luckily, it is possible to design something like that in Fusion 360 and export it to KiCAD. To do this, first create a sketch with the shapes you would like to export, and then right-click the sketch in the navigation pane and select "Save as DXF". DXF is a format used in more engineering applications and describes a 2D design or path that can be used by for example laser cutters. Once you saved your DXF file you can import this in e.g. KiCAD.