

Learn



AUTODESK® FUSION 360™



Tutorial: Personalised car key fob, suitable for 3D printing. For the absolute beginner!

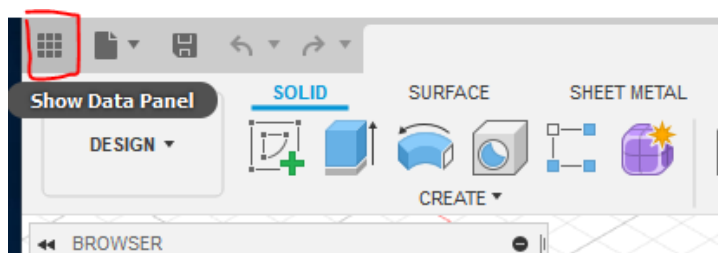
Eva Hakansson, KiwiFil and Green Envy Racing Team. May 2021.

Prepared for the EVelocity build contest. Rev. 2.0.

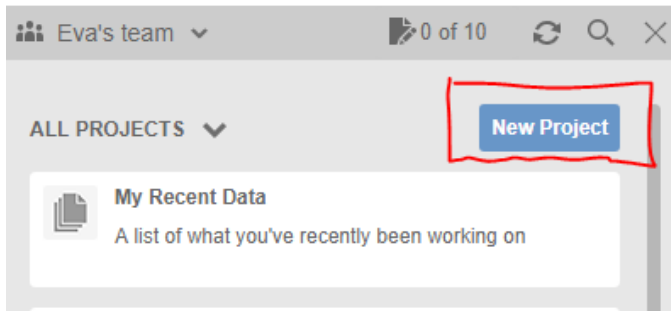
This is a step-by-step tutorial how to create a personalised, car-shaped key fob in Autodesk Fusion 360, suitable for 3D printing. Fusion 360 is a cloud-based, parametric CAD modeling software. It can be run both on PC and Mac. It is free for students and hobbyists and can be downloaded from <https://www.autodesk.com/products/fusion-360/free-trial>. You will need a free Autodesk user account to use Fusion 360. Follow the instructions at Autodesk.com to create an account, download the software, install Fusion 360 and verify your email address. Make sure to select “Educational use” if you are a student.



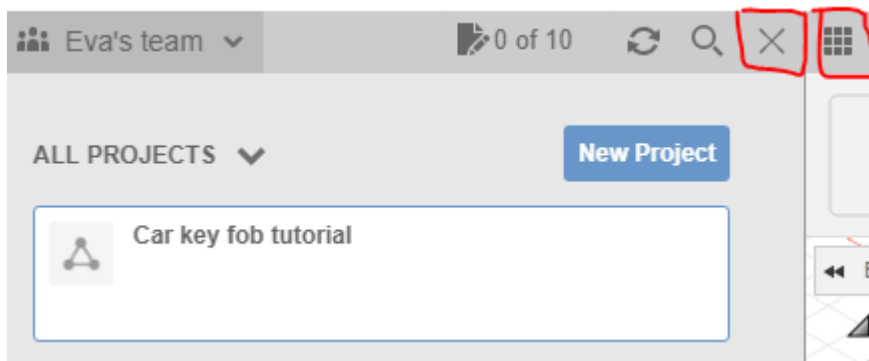
- 1) Open Fusion 360. The first time you open the program you might be asked if you want to create a team. Follow the instructions and create a single-person team.
- 2) If the data panel on the left-hand side of side of the screen isn't visible, click on the squares to make it visible.



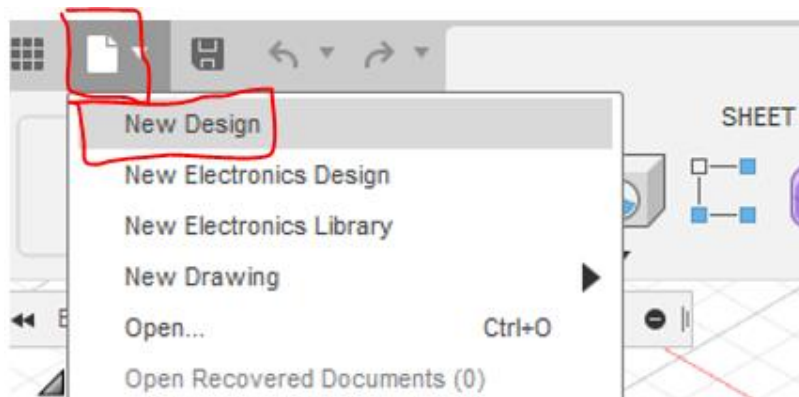
- 3) Click **New Project** to create a project for your design. If you already have created projects, you can use one of the existing projects.



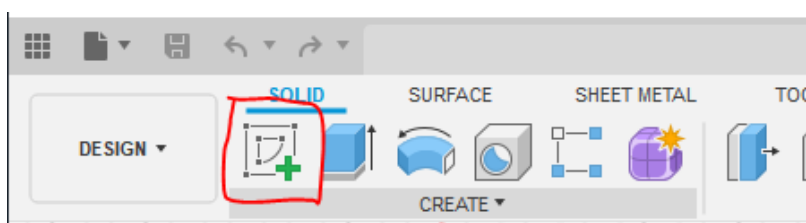
- 4) Give the project a *descriptive* name (this makes it much easier to find your files in the future). If you want to change the name of the project, you can right-click on the project and select **Rename**.
- 5) You can close the project menu and get more room for your design on the screen by clicking the **X** in the corner. To get it back, click the square symbol that is to the right of the X below.



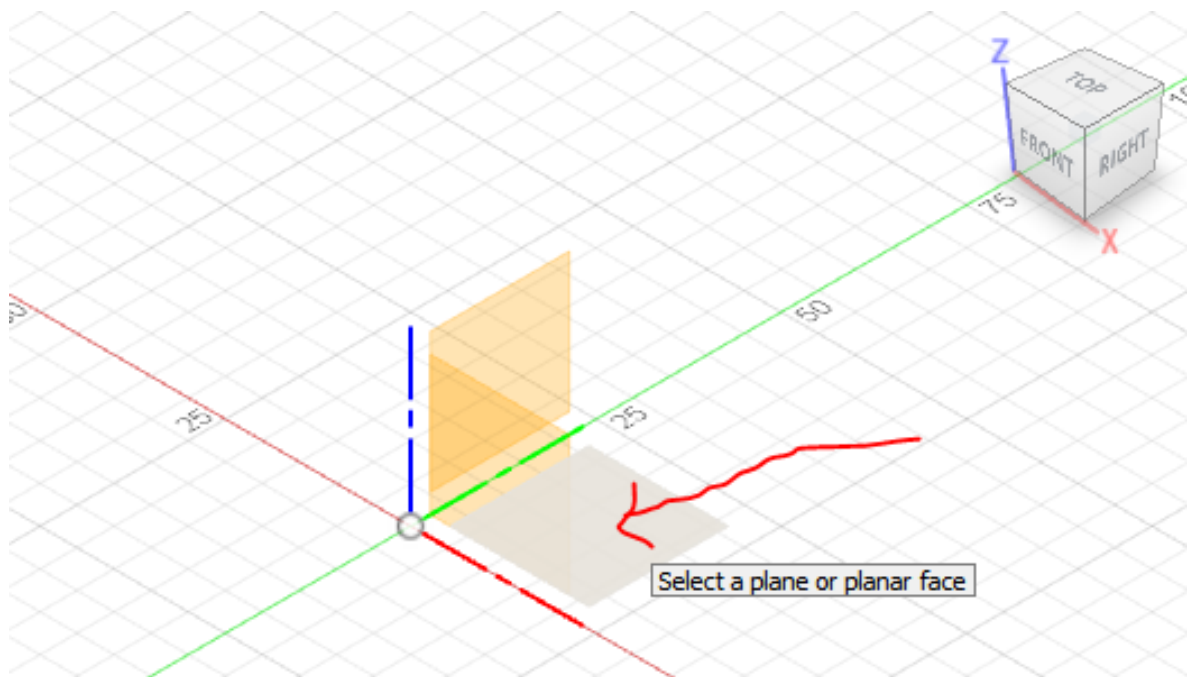
- 6) In the upper left-hand corner, click the symbol for **File** and select **New Design**.



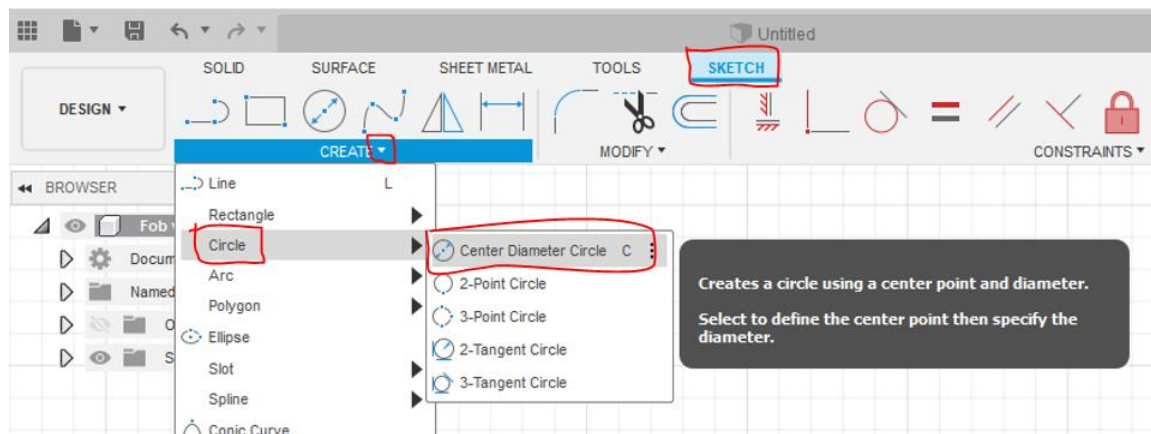
- 7) Click the **File** symbol again and select **Save**. Give your design a *descriptive* name so you can easily find it again later.
- 8) Click the **Sketch** button in the **CREATE** group.



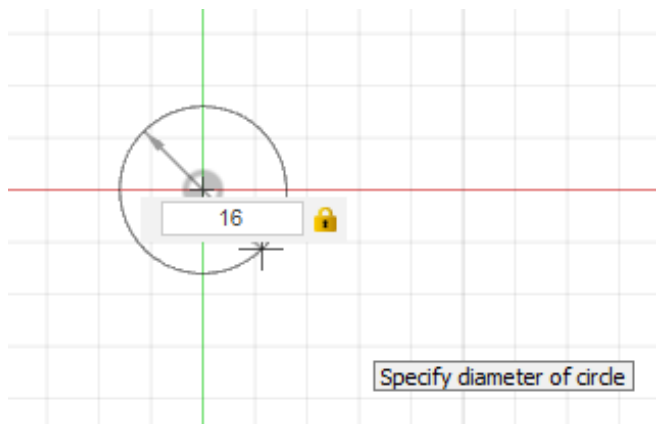
- 9) Click on the X-Y-plane to start a new sketch. You can start a sketch on any plane, but by selecting this particular plane, your finished key fob will be orientated correctly when we later import it into the 3D printer software.



- 10) In the Sketch tab, click on the chevron next to CREATE to expand that menu. Select **Circle** and then **Center Diameter Circle**. You can also press the **C** key on your keyboard to get directly to this circle tool (as illustrated with the “C” after the name in the menu).



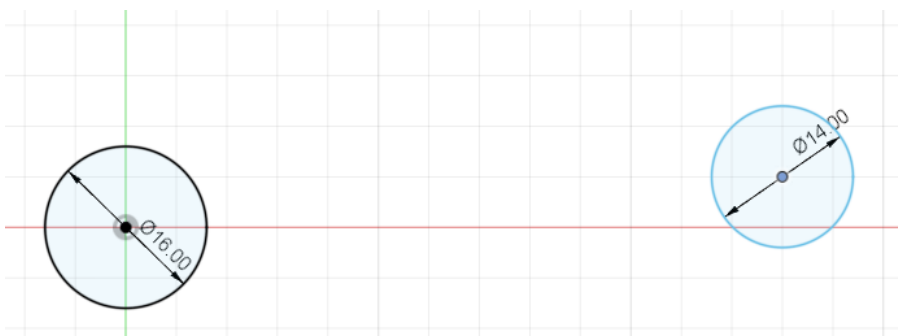
- 11) Left-click once on the **origin** (that's what the centre of the drawing plane is called). Drag out a circle (without clicking). Type 16 on your keyboard, and they press enter or left-click again. You have now created a circle with 16 mm diameter.



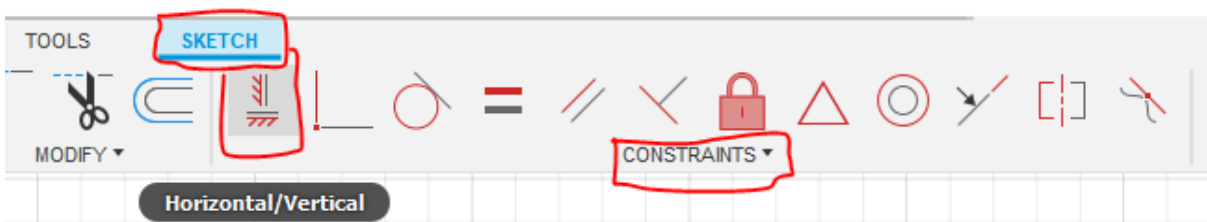
- 12) Use the same circle tool, and place another circle to the right of the first circle. Make it 14 mm in diameter. If you hover over the red line, you can see how the marker snaps to the line. You want it to snap to the line; however, if it doesn't, we can add this constraint later. If you give a feature an incorrect dimension, just click on the dimension again to change it.



If the second circle didn't snap to the red line (it may look like the image below).

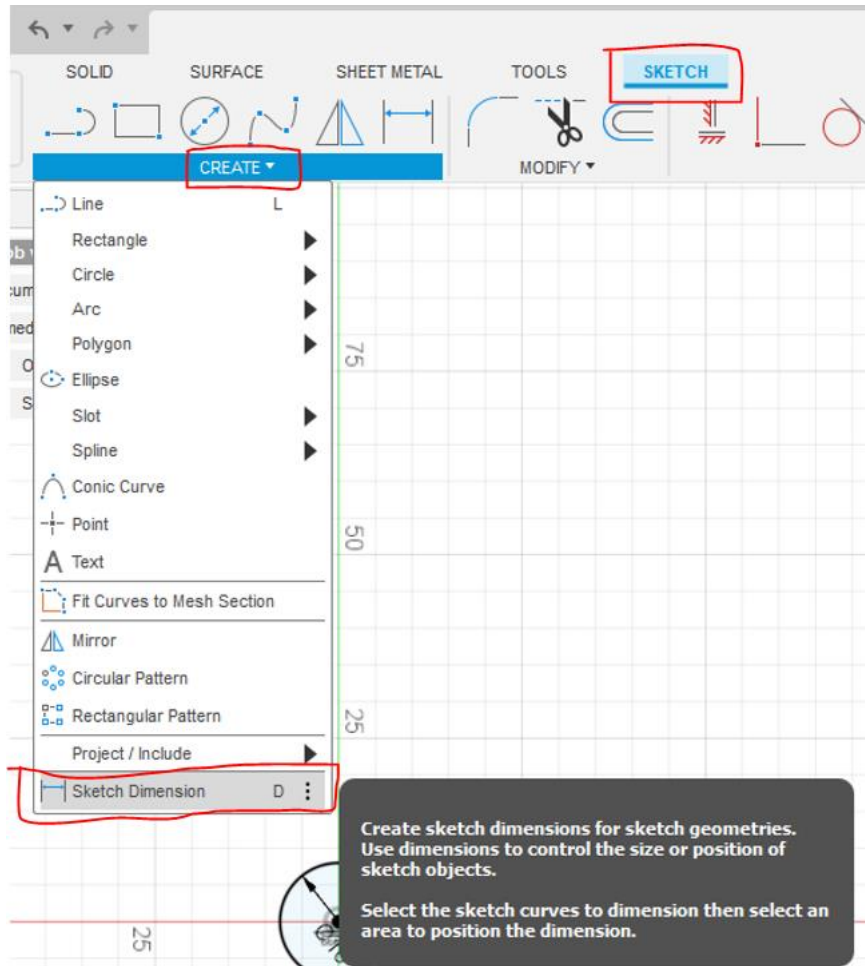


If this is the case, click the **Horizontal/Vertical** constraint in the group CONSTRAINTS (also under the Sketch tab).

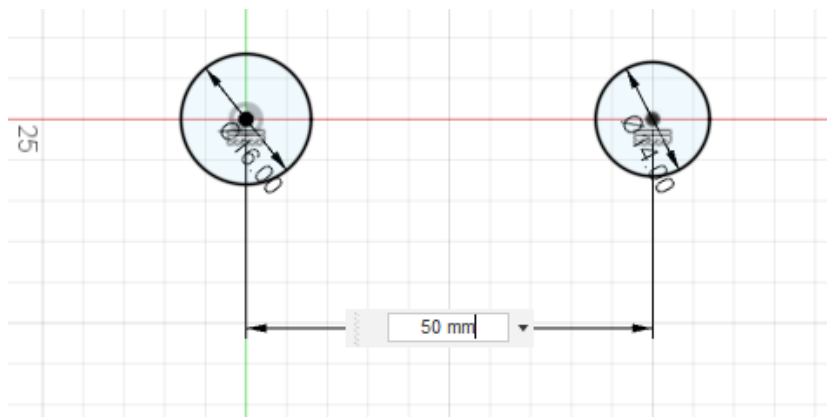


Click on the 14 mm circle, then on the 16 mm circle. Because the 16 mm circle was constrained to the origin when we made it, the 14 mm circle will move to be on the same horizontal line as the 16 mm circle, which in this case is the red line.

- 13) Remain in the **Sketch** tab, click the chevron next to CREATE. At the bottom, select **Sketch Dimension**. You can also access this command directly by pressing the **D** key on your keyboard.



- 14) Click on the centre of the 16 mm circle then on the 14 mm circle. Then move the mouse marker downwards and left-click once. Type 50 into the box. Then press enter or left-click outside the box. You have now set the dimension between the two circles to 50 mm.



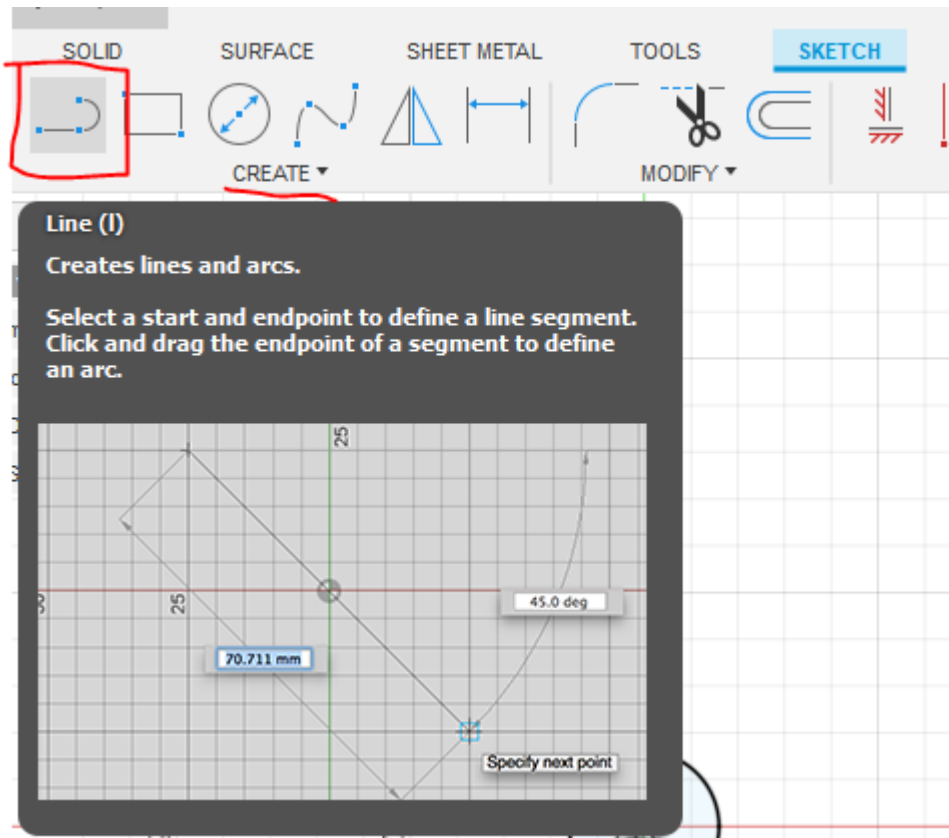
Press the **Esc** key on your keyboard to turn off the dimensioning command.

Tips & Tricks! The Esc key is your best friend when working with CAD! That gets you out of the current tool.

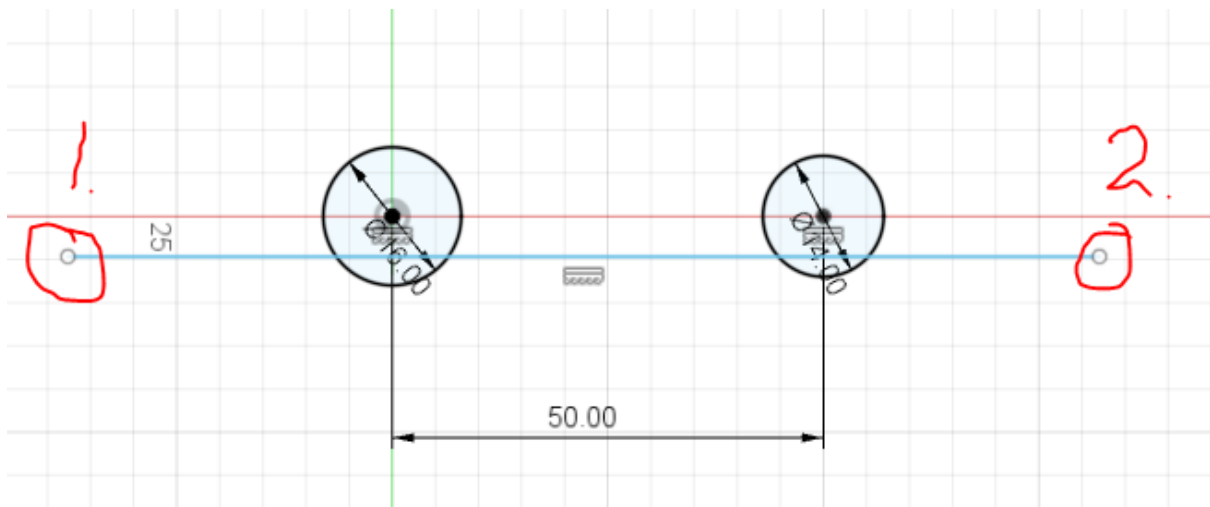
Another great friend is undo. You can use **Ctrl-Z**, or use the two arrows in the upper left-hand corner to undo and redo.



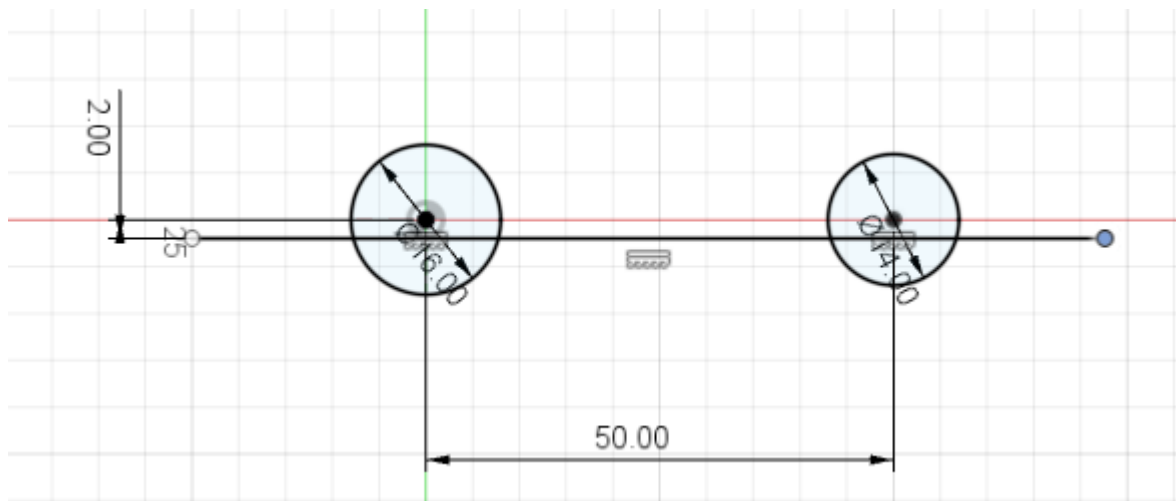
- 15) In the CREATE group (under the **Sketch** tab), select **Line**. You can also get to this command directly by pressing the **L** key on your keyboard. You can also find the command in the list if you click the chevron next to CREATE. If you hover your mouse marker over the line button, a box will appear with an explanation of the tool.



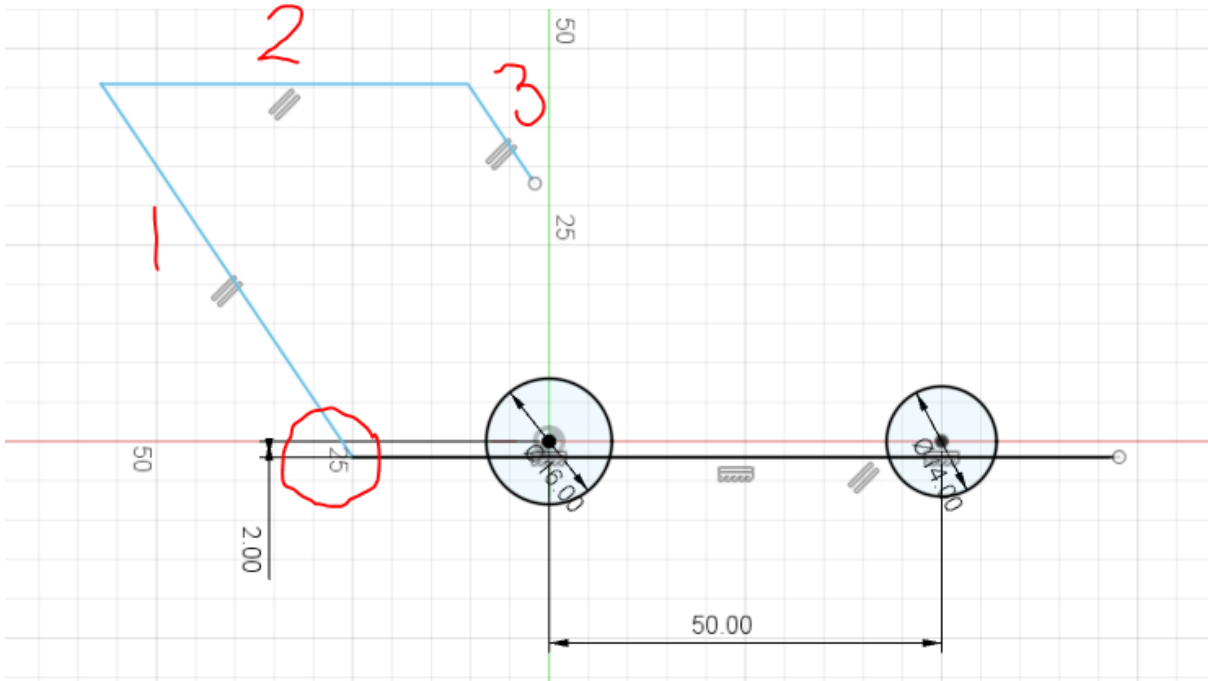
- 16) Left-click once to the left of the left circle, approximately in the location labelled with "1". Move the mouse marker to location "2", and left-click again. Press **Esc** to turn off the line tool.



- 17) Start the dimension tool that you used before. Click on the **centre of the left circle** and then click on the new line you just drew. Move the mouse marker to the left and notice how the dimension lines follow. Left click once and then enter 2 in the box. Press **Enter**. The new line is now located 2 mm below the circle centres.



18) Start the line tool again, and draw three lines roughly like below. The first line should connect to the line we drew earlier, but the exact dimensions are not important at this point in time. You will notice that you can draw multiple lines that are connected without restarting the line tool.



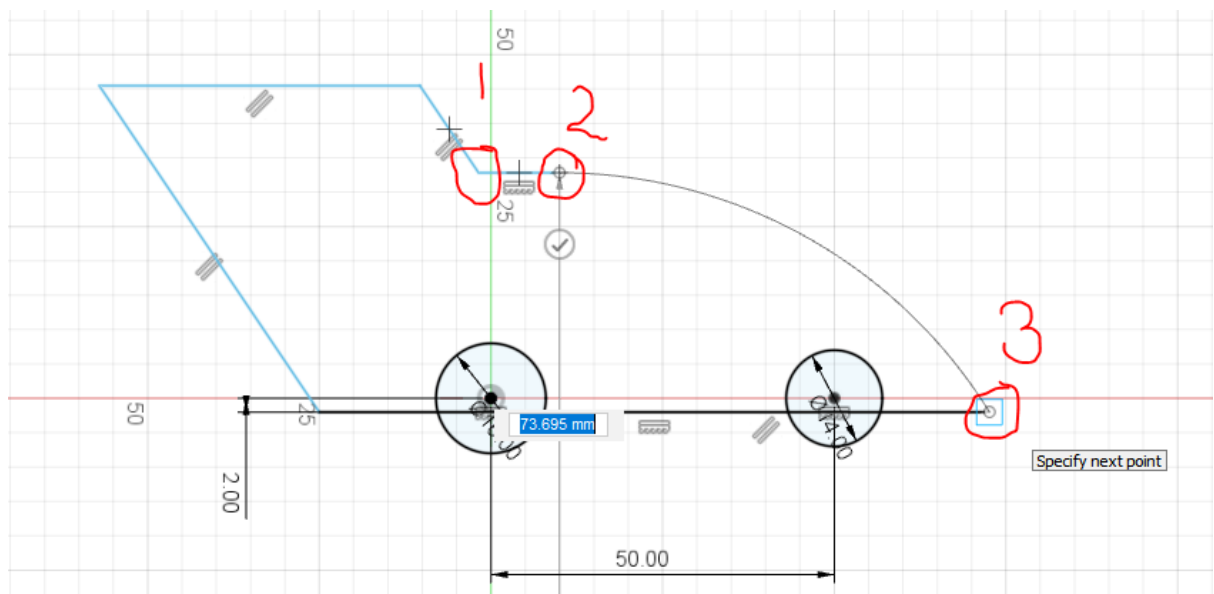
19) This step is a bit tricky, so you have to follow the instructions exactly to be successful.

If you exited the line tool, start it again.

a) Left-click on the end of the last line you created (location 1).

b) Left-click at a location roughly at 2., but **keep holding the left mouse button**.

c) Drag the arc to location 3 where it snaps to the end of the horizontal line (location 3). Let go of the left mouse button.

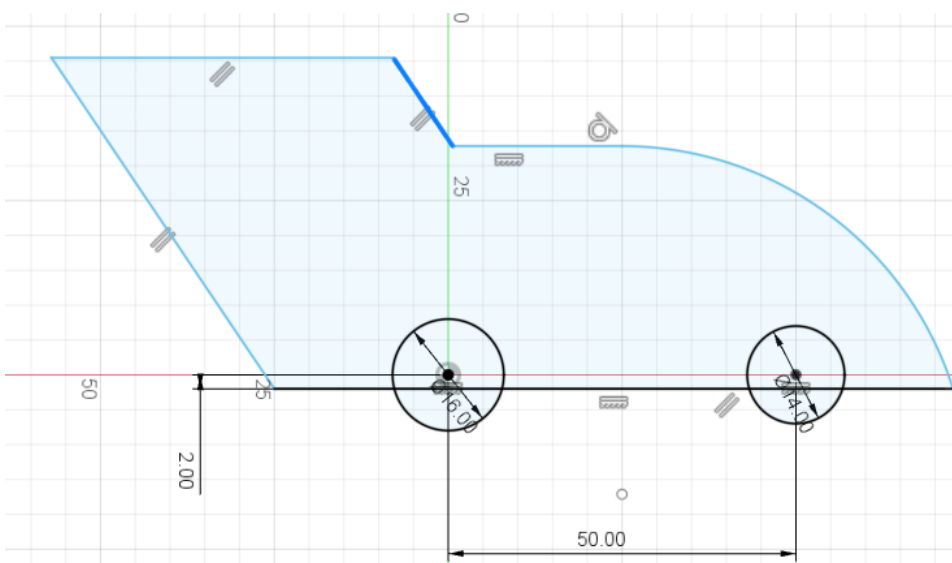
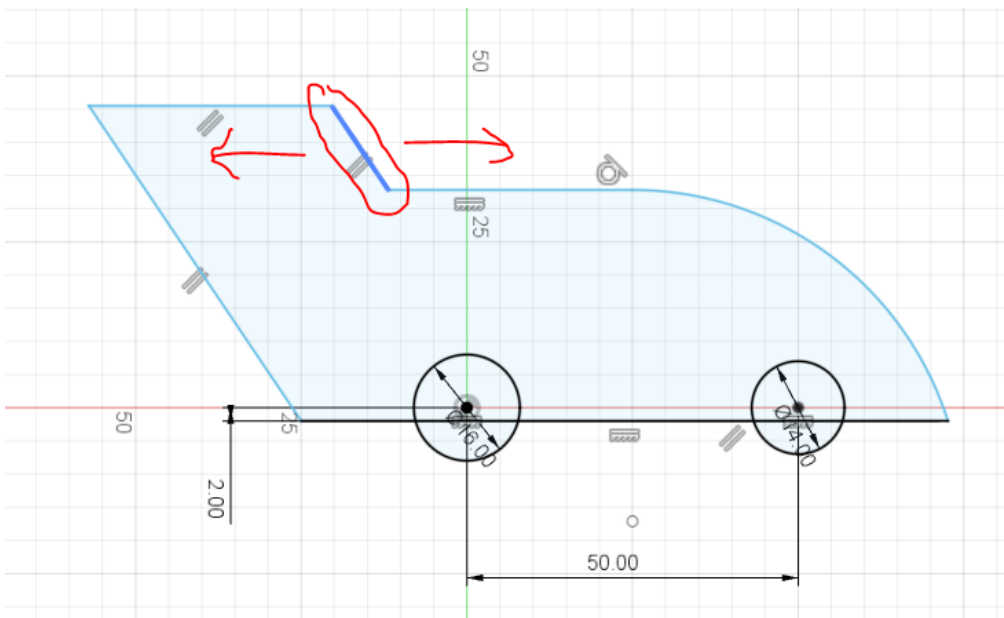


If it didn't work correctly, use the **Undo** button, or use the Ctrl-Z command on your keyboard to undo. Then try again. `



20) Now is a good time to save your work. Click the Save symbol in the upper left-hand corner.

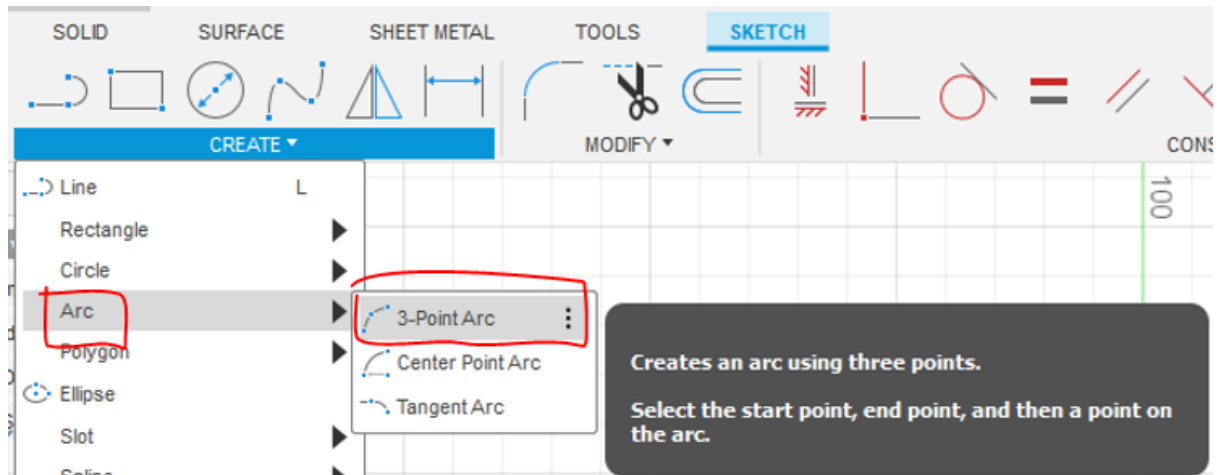
21) Left-click on the line highlighted below, and drag it to the left or the right so the line is approximately above the centre of the left circle.



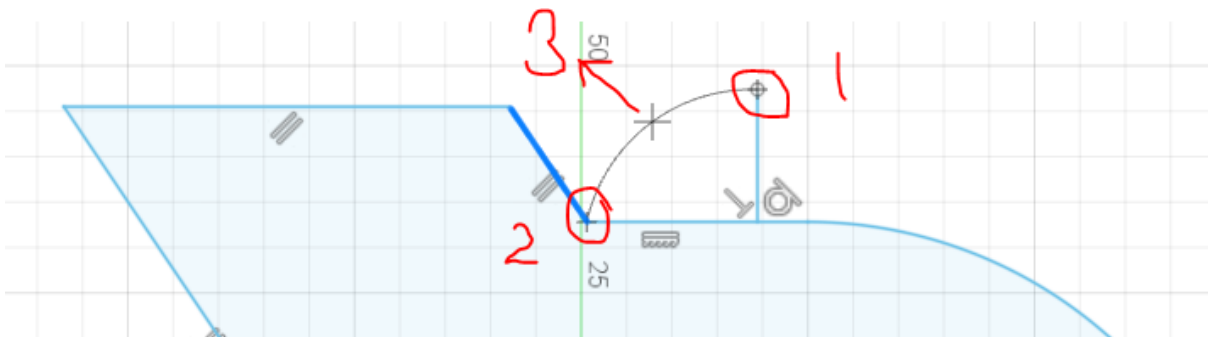
- 22) Start the **Line** tool. Draw a vertical line as shown below (circled in red). Make sure the line snaps to the horizontal line. Also make sure the new vertical line is to the *left* of the point where the horizontal line connects to the arc (circled in blue).



- 23) Click the chevron next to CREATE, and select **Arc** and then **3-Point Arc**.

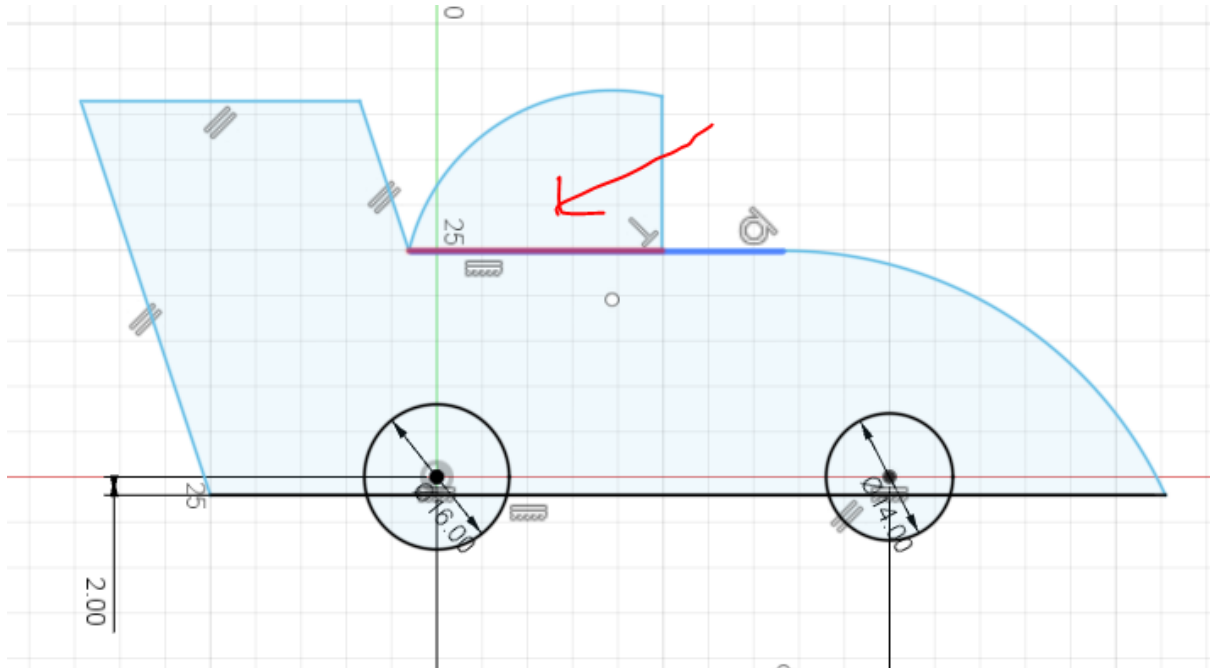


- 24) Left-click on the upper end of the vertical line (1), then on the corner (2). Move the mouse marker upwards to create an arc. When you like the shape of the arc, then left-click again.



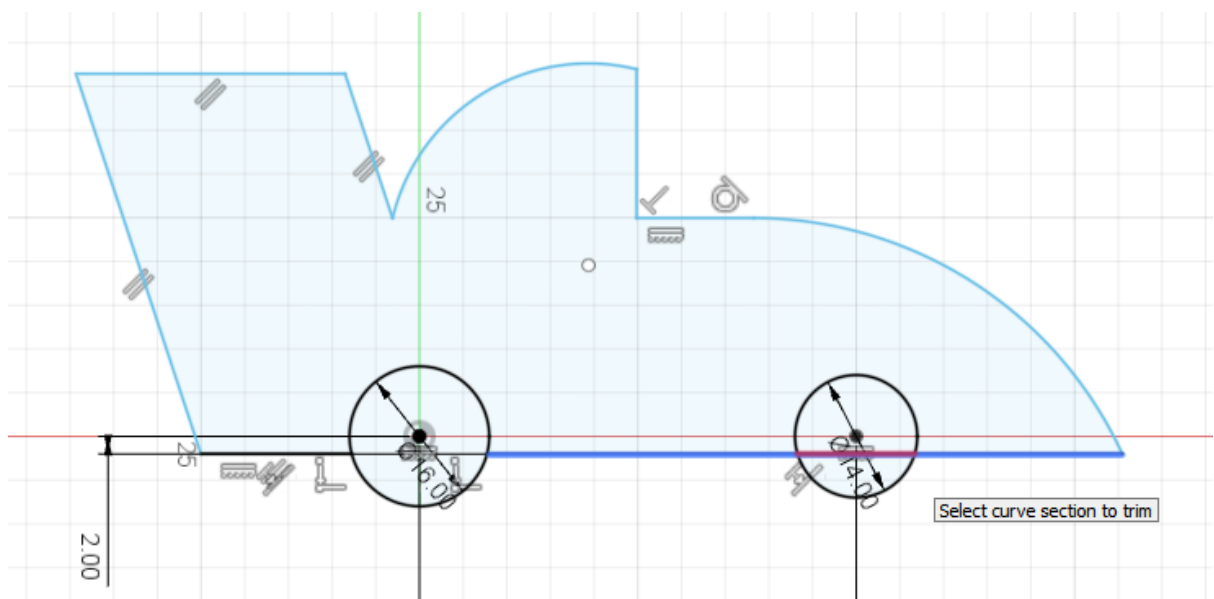
25) Under the **Sketch** tab, click the chevron next to MODIFY and select **Trim**. You can also get to this command directly by pressing **T** on your keyboard.

26) Hover over the horizontal line below the arc. When it turns red, click to trim it. Don't worry if you get a warning saying that constraints were removed.

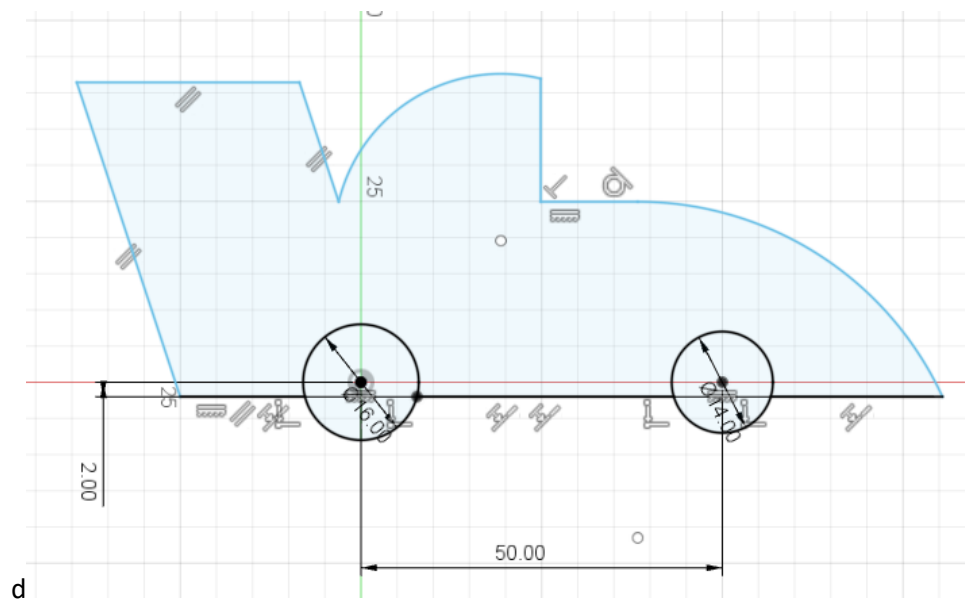


27) Now trim the horizontal lines that go through the “wheels”.

Note, depending on where you placed the 2 mm dimension, there may be dimensioning lines going through the wheels. If you are unsure, you can delete the dimension and add it again. You only need to trim the thick sketch lines; thin dimensioning lines (if there are any) should remain.



28) Your design should now look roughly like below.

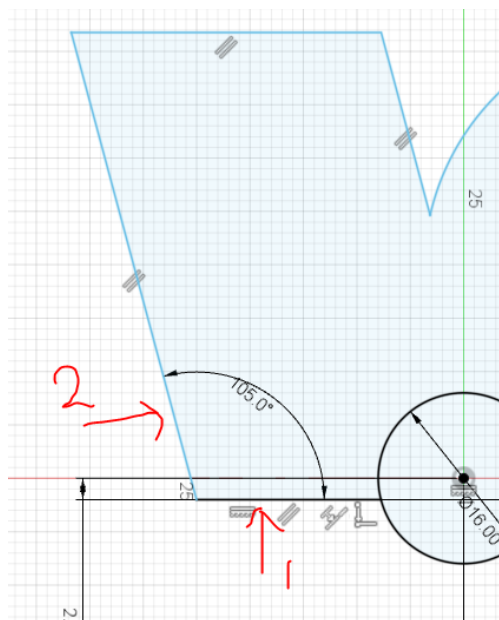


29) You will need to add quite a few more dimensions to fully define the design. Let's start with the angle in the lower left-hand corner.

To add an angle, first start the **Sketch Dimension** tool by pressing **D** on the keyboard, or select it from the list under CREATE.

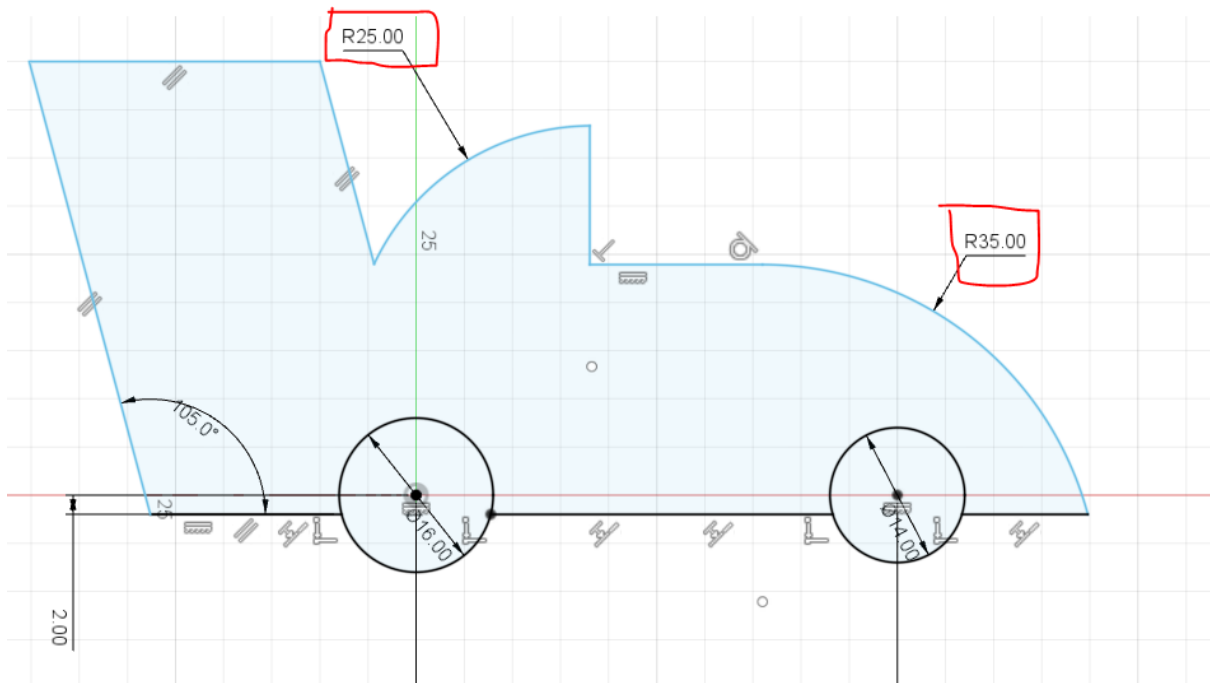
Click on each of the two lines that you want to define the angle between.

Move the mouse marker to see the dimension lines, left-click again, and type in the dimension. In this case we want it to be 105 degrees.

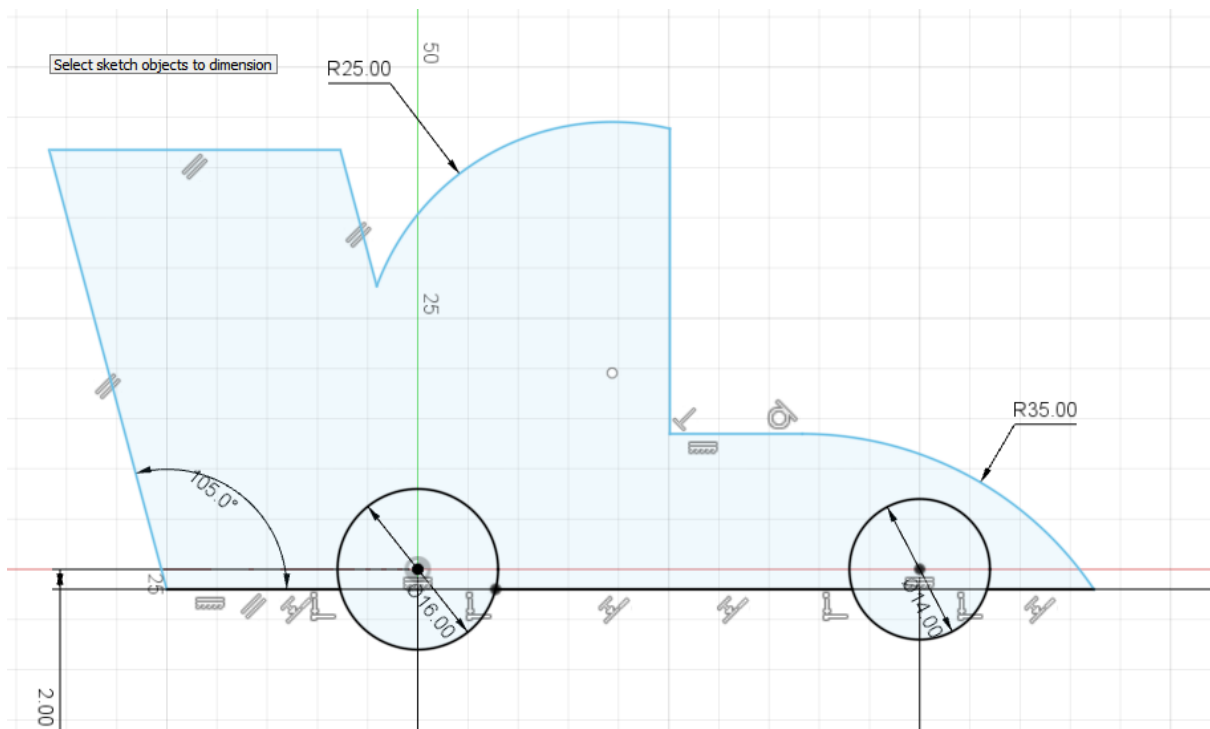


If you find it difficult to see, roll on the scroll wheel on the mouse to zoom and out.

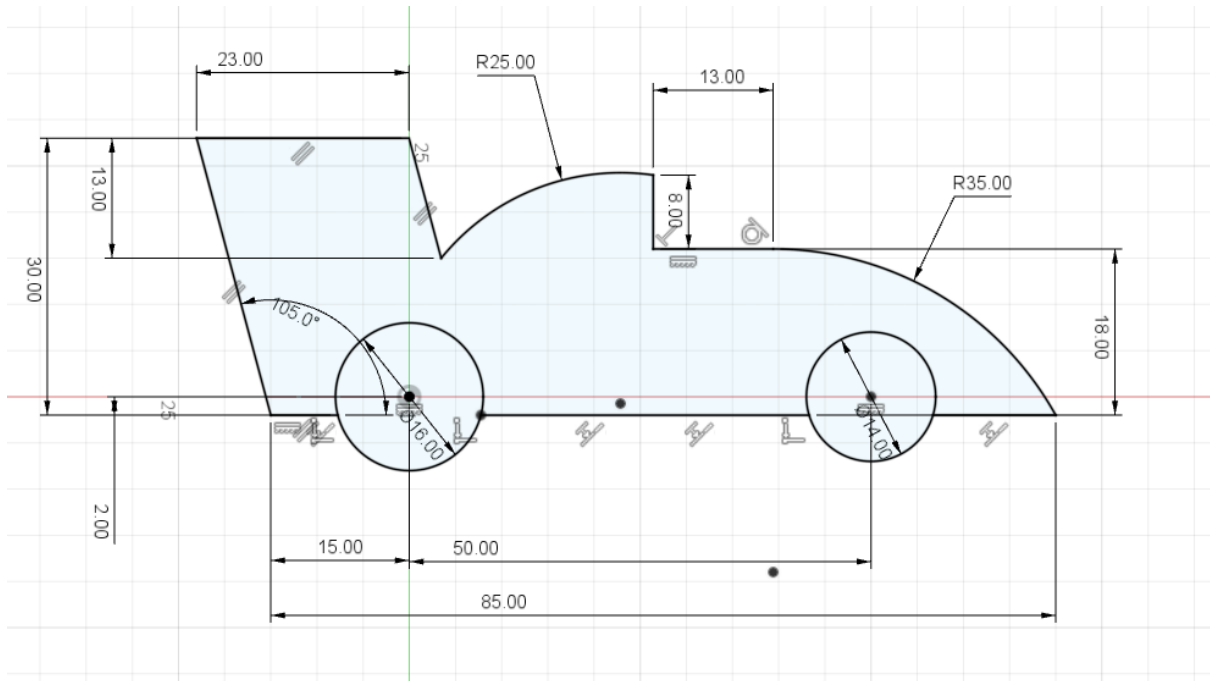
30) To set the radius of the two arcs, use the dimension tool again. Click on the arc, drag out the dimension lines by moving the mouse, left click, enter the desired dimension, and then press enter. Dimension the two arcs as shown below.



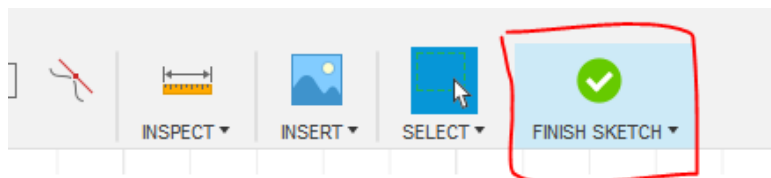
Don't worry if your car looks like below with weird proportions. We will fix it as we add dimensions. However, if you want it to look more like the race car above, press **Esc** to get out of the dimensioning tool, then try to grab lines or corners to drag it to the desired shape. The lines that are still blue are not fully defined and can be moved. The lines that have turned black have limited or no movement.



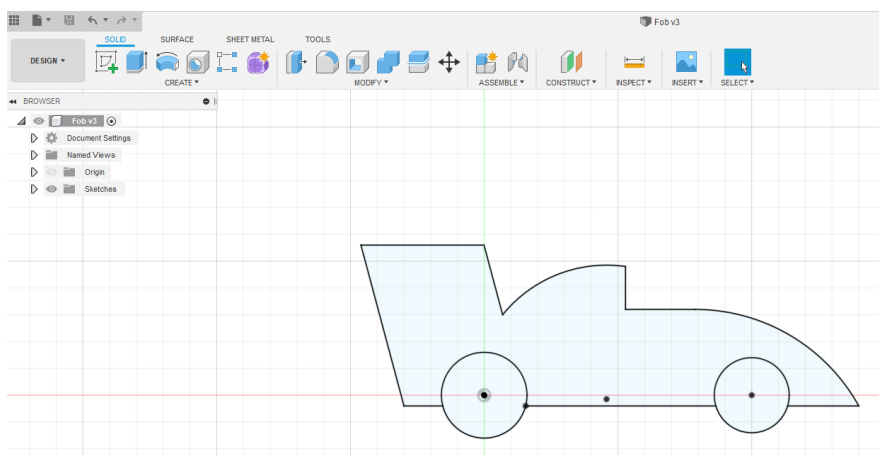
- 31) Continue using the dimension tool and add all the dimensions shown below. When the design is fully defined, all the lines will be black.
- You should also notice that the car is light blue; this means that the area is forming an enclosed geometry. This is important in the next steps.



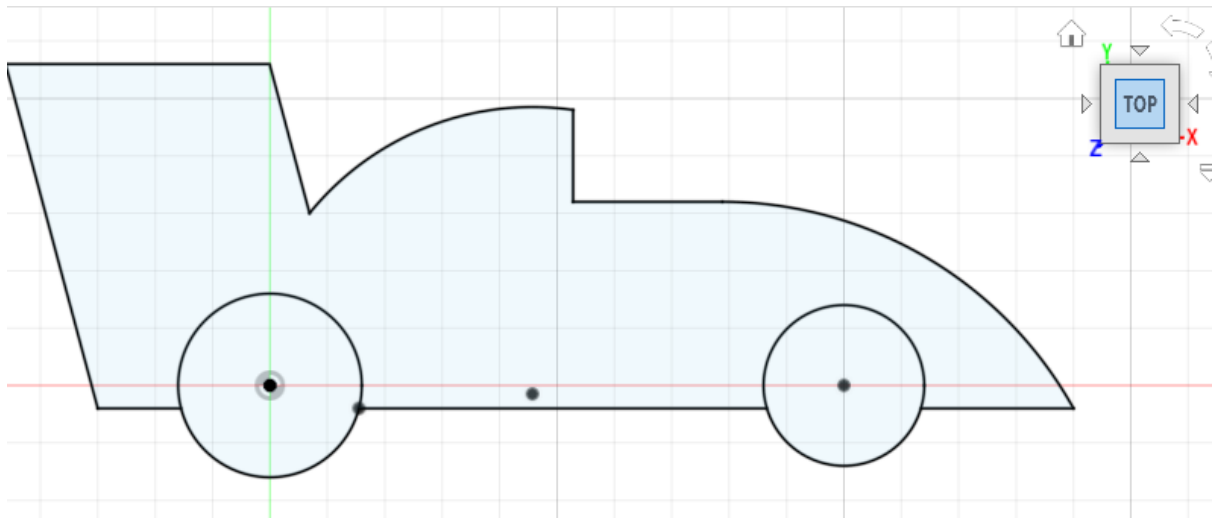
- 32) Save your design.
- 33) Click **Finish Sketch** towards the right in the ribbon.



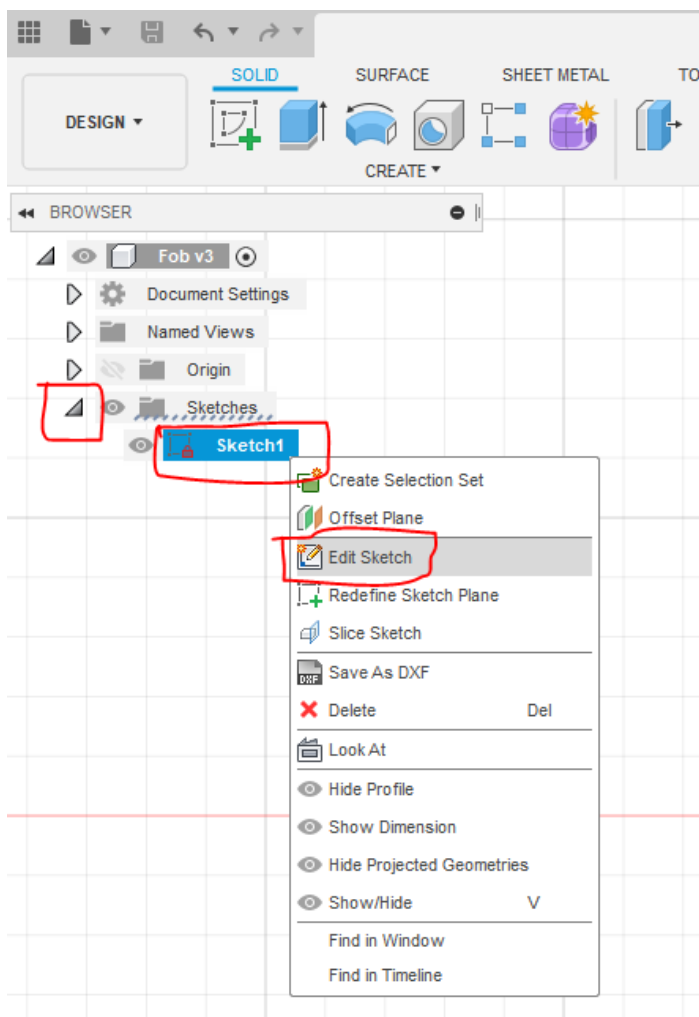
- 34) Your design will now look like this. Note that the dimensions are still there, they are just not visible anymore.
- If you hold down the centre mouse wheel and the shift key, and move the mouse, you **Free Orbit** (rotate) the design. Since it only a sketch, you will see that it has no thickness. We will change that in a moment.



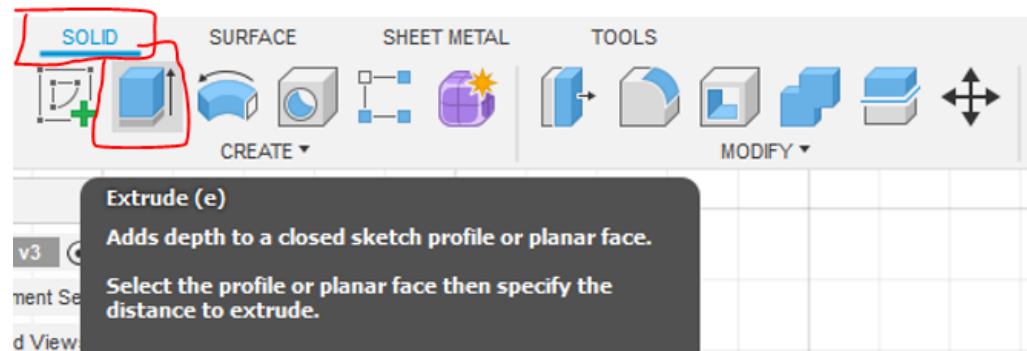
If you lose track of your design (free orbit is a bit finnick), use the view cube in the right-hand upper corner to get back to the original view. If you hold down the scroll wheel, you can drag your design (pan) on the screen.



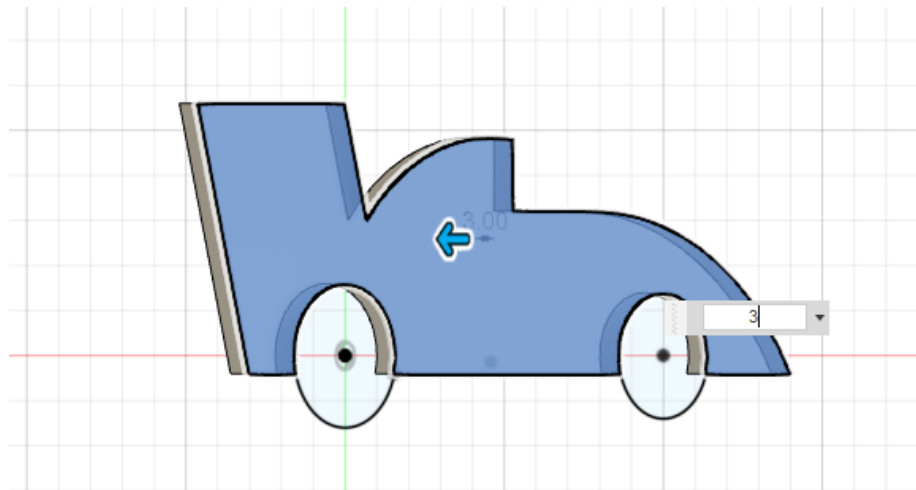
35) If you want to change something in the sketch, right-click on **Sketch1** in the model browser and select **Edit Sketch**. All your dimensions will come back and you can change them. If you are happy with your sketch, skip this step and continue to the next.



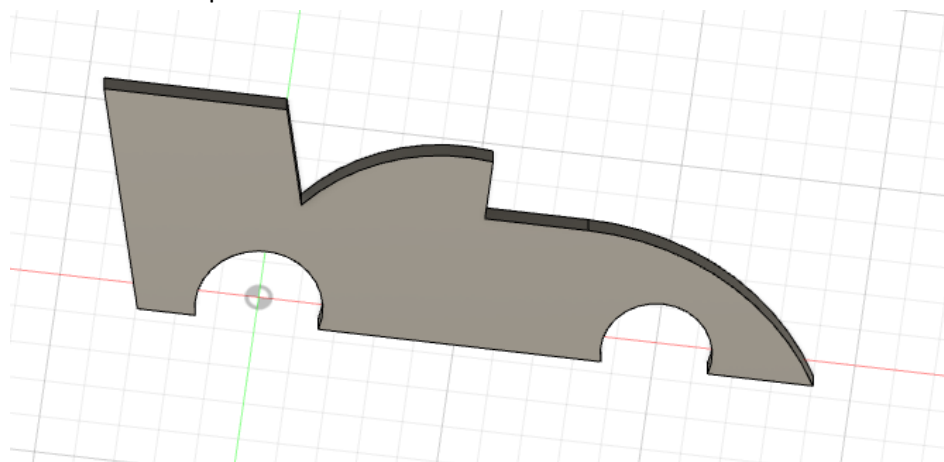
- 36) Under the **Solid** tab, click **Extrude**. Or press the **E** key to get directly to the extrude command. You can also access the command by clicking the chevron next to CREATE and select **Extrude** from the list.



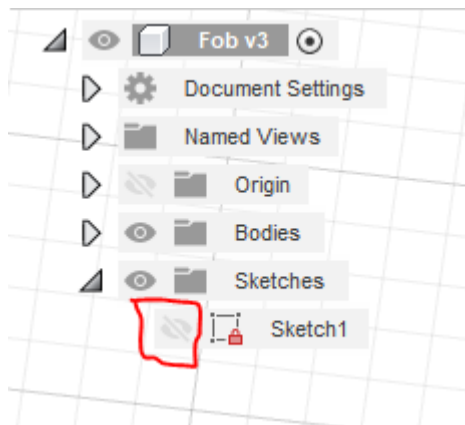
- 37) Click on the body of the car. Type in 3 in the box, and press enter.
An alternative way is to pull the blue arrow and make it thicker and thinner (feel free to play with the different commands, that's how you learn. You can always just undo if you don't like the result).



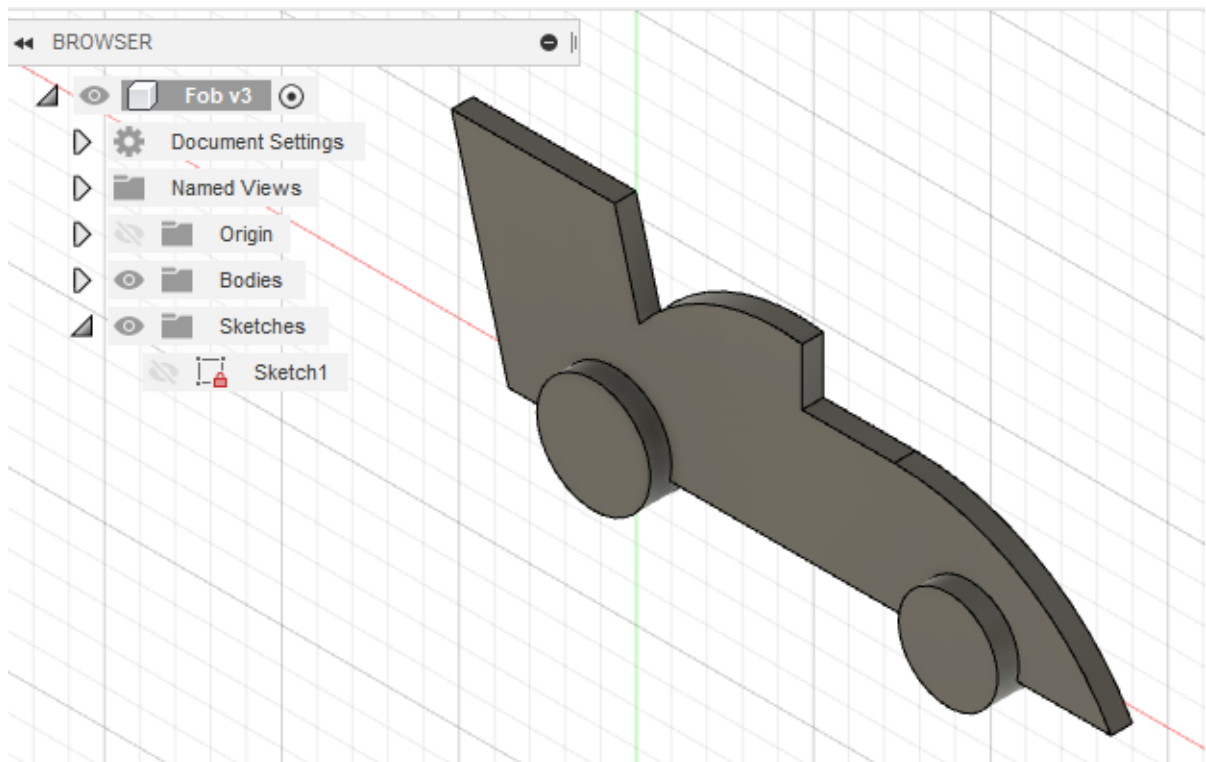
- 38) Use the Free Orbit command again by holding down the mouse wheel and shift key, and you can see that the part now has thickness.



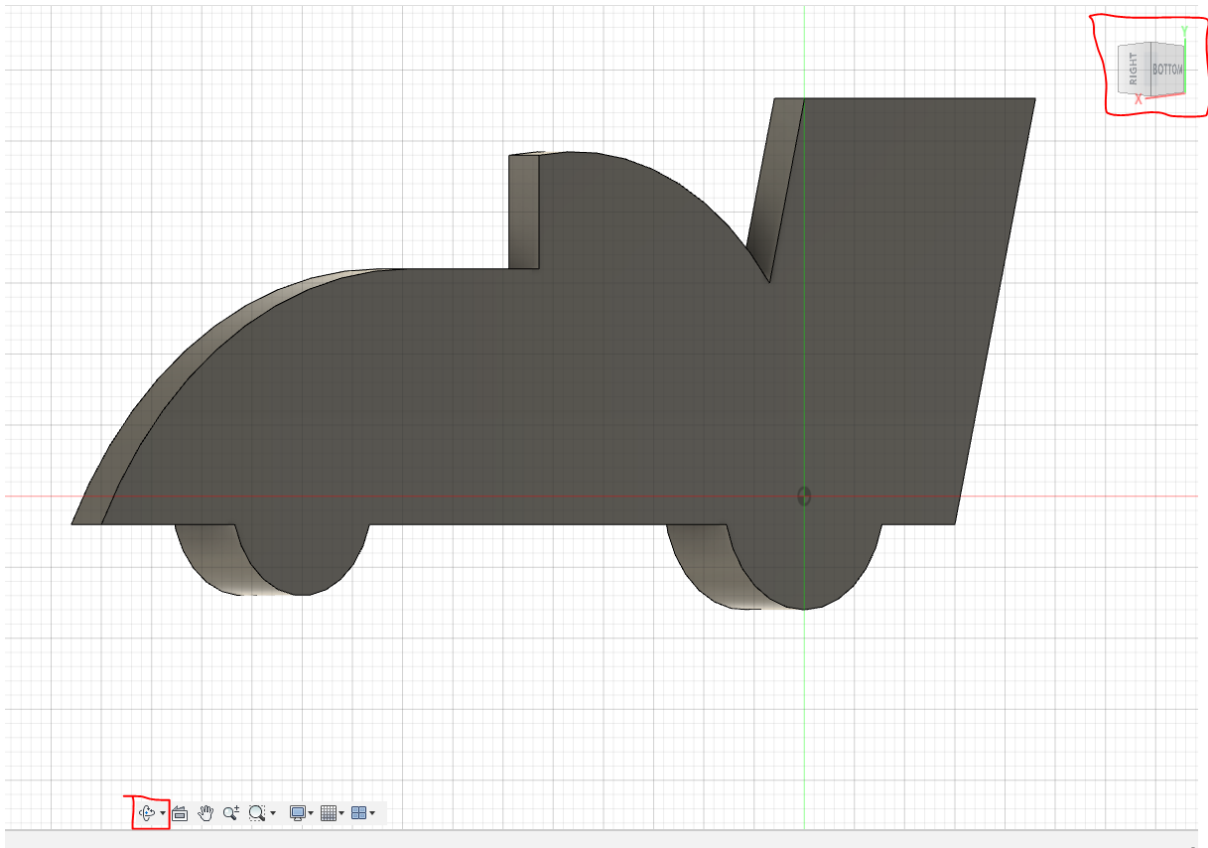
- 39) As you noticed, the sketches for the “wheels” disappeared when we extruded the body. Click on the eye next to Sketch1 in the model browser to make the sketch visible again.



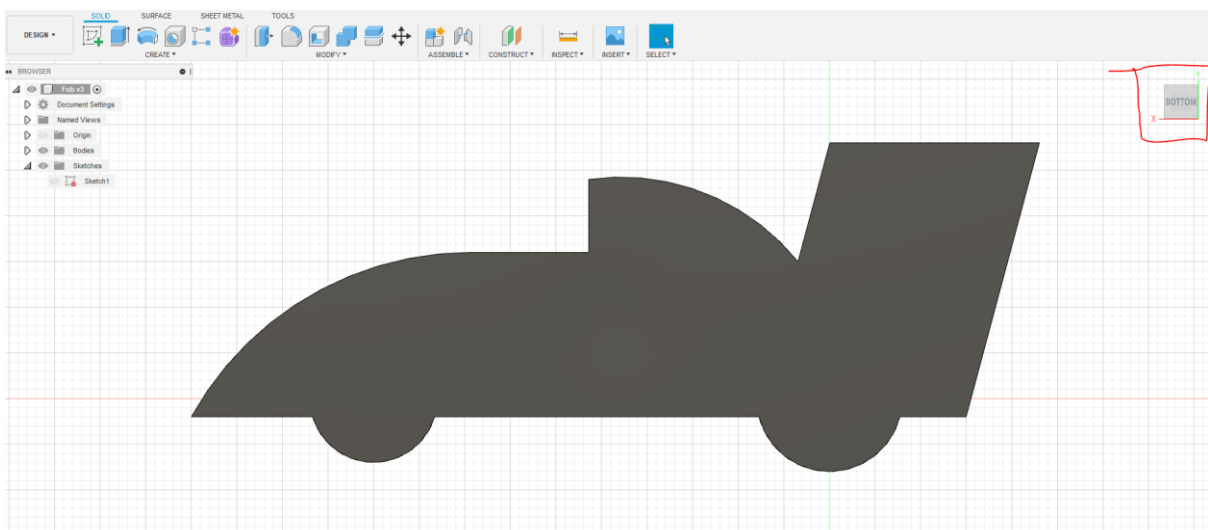
- 40) Select the Extrude command again, and extrude both wheels 6 mm. Then turn off the visibility of Sketch1 again.



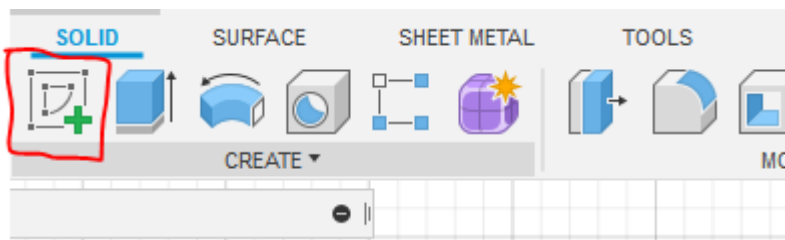
41) Turn the design over so you can see the back side of the car. There are several ways to do this. You can use the Free Orbit command as described above. You can also access Free Orbit as well as Constrained Orbit through the button at the bottom of the screen. You can also rotate the model by clicking on the different sides on the cube in the corner.



42) Click on the side labeled BOTTOM on the cube. Scroll the mouse wheel to zoom in or out until the entire car is visible in the window.



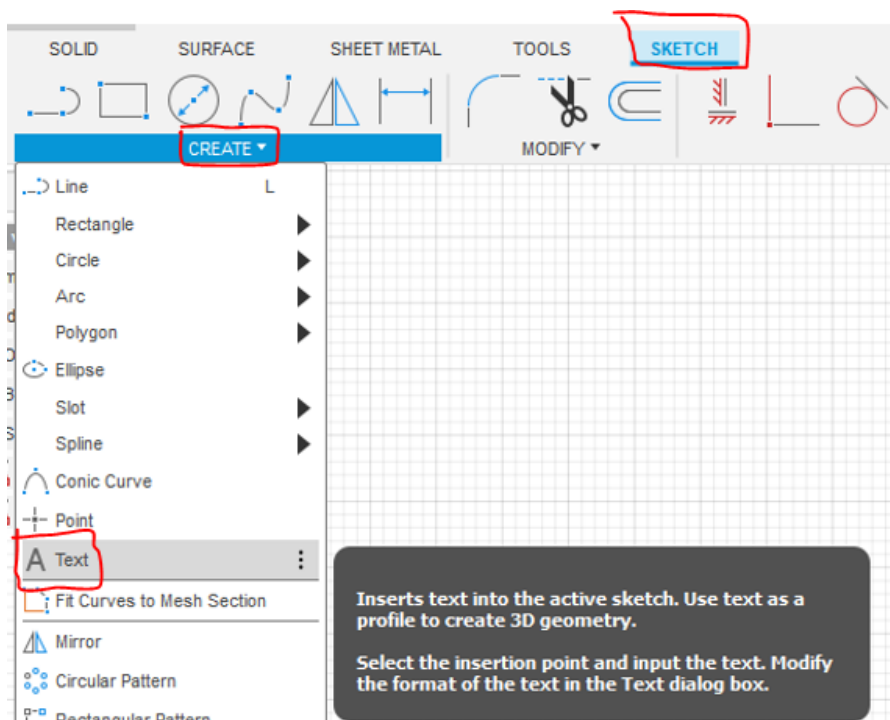
43) Create a **New Sketch**, and click on the back of the car.



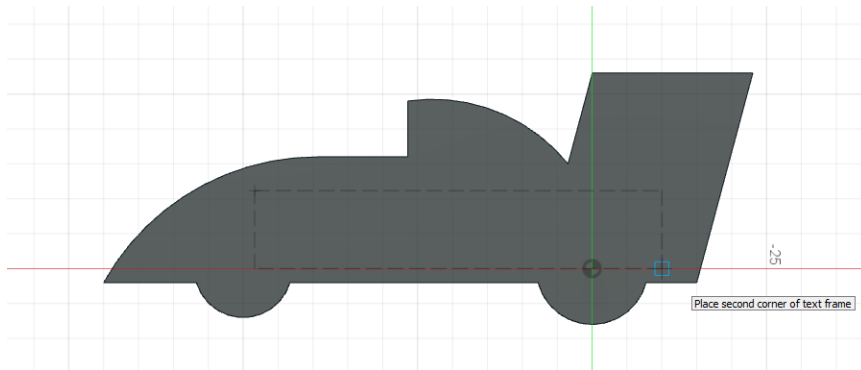
If the car ends up upside-down, use the arrows next to the view cube to turn it back.



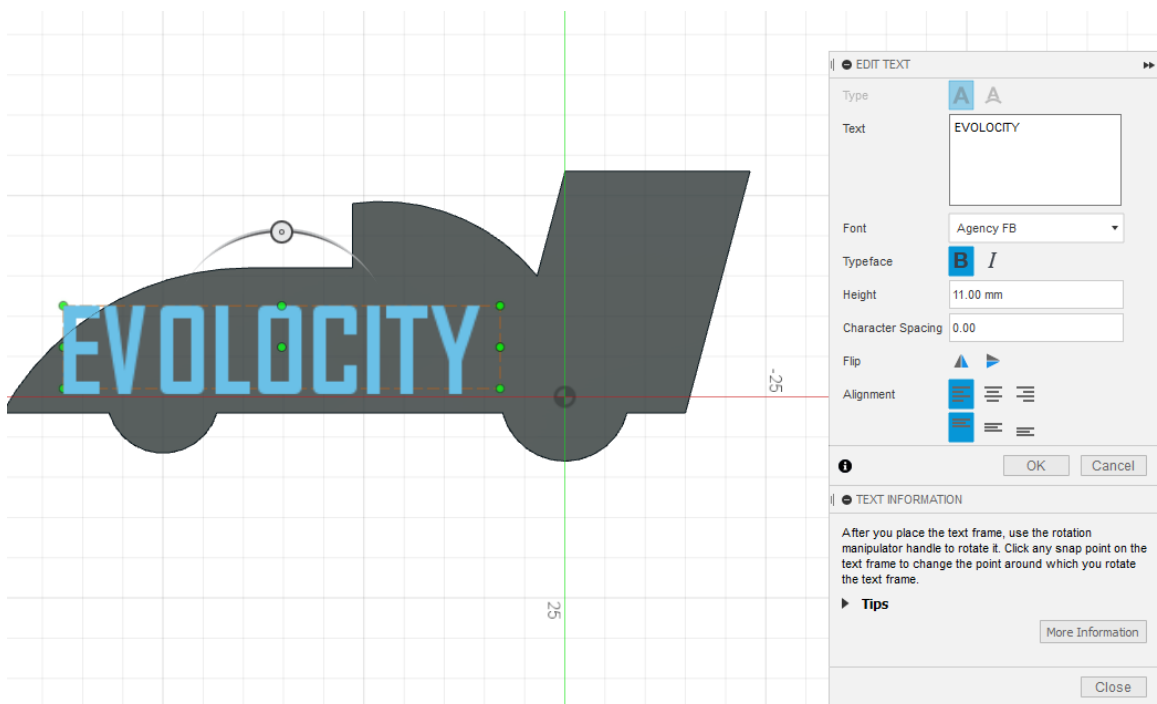
44) Expand the list under CREATE, and select **Text**.



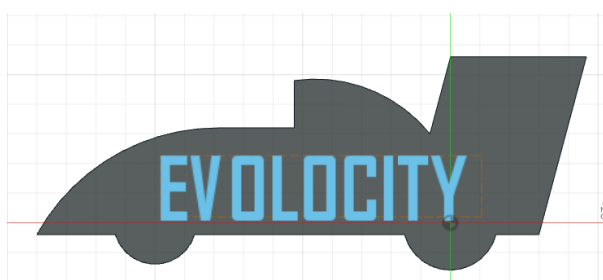
45) Click on the back of the car. Drag out a rectangle approximately as shown below. The exact size and location aren't important – we will adjust that later.



46) Enter **EVOLOCITY** in the box labeled Text. Select **Agency FB** as **Font** (if this font is not available, select another font).
Set the **Height** to 11 mm.
If your text is upside down, set the **Angle** to 180 deg.
Select bold font by clicking the **B**.
Click OK, even if the text isn't perfectly located. We will fix this next.



47) Click on the text and drag it into a good location in the centre of the car. If you want to change the size or font, double-click on the text and the dialogue box will open again.

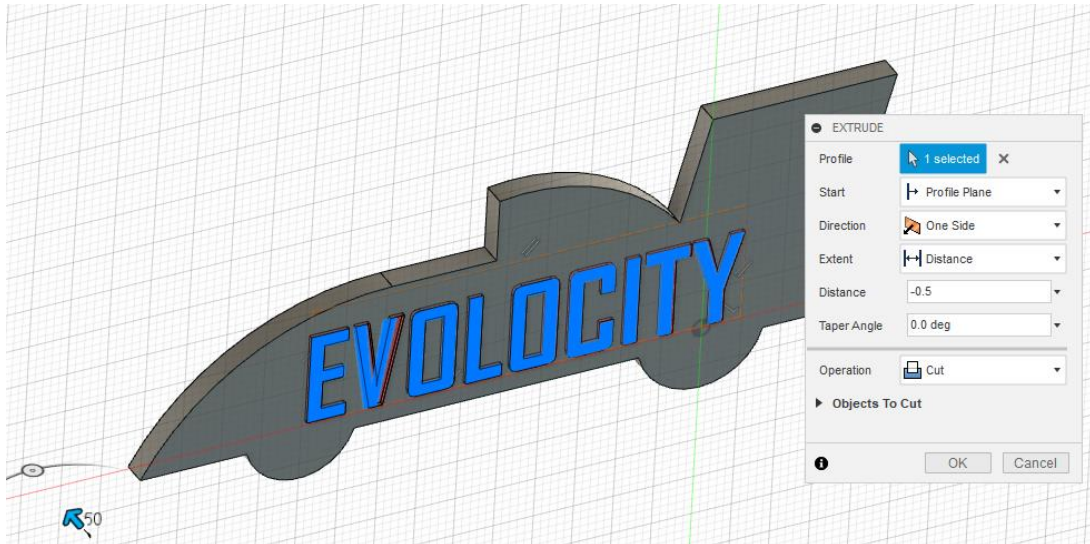


48) Click on the EVOLOCITY text on the car.

Then click on the **SOLID** tab, and start the **Extrude** command.

Drag the blue arrow and see how the extrusion grows and shrinks. You might have to rotate the car to see better.

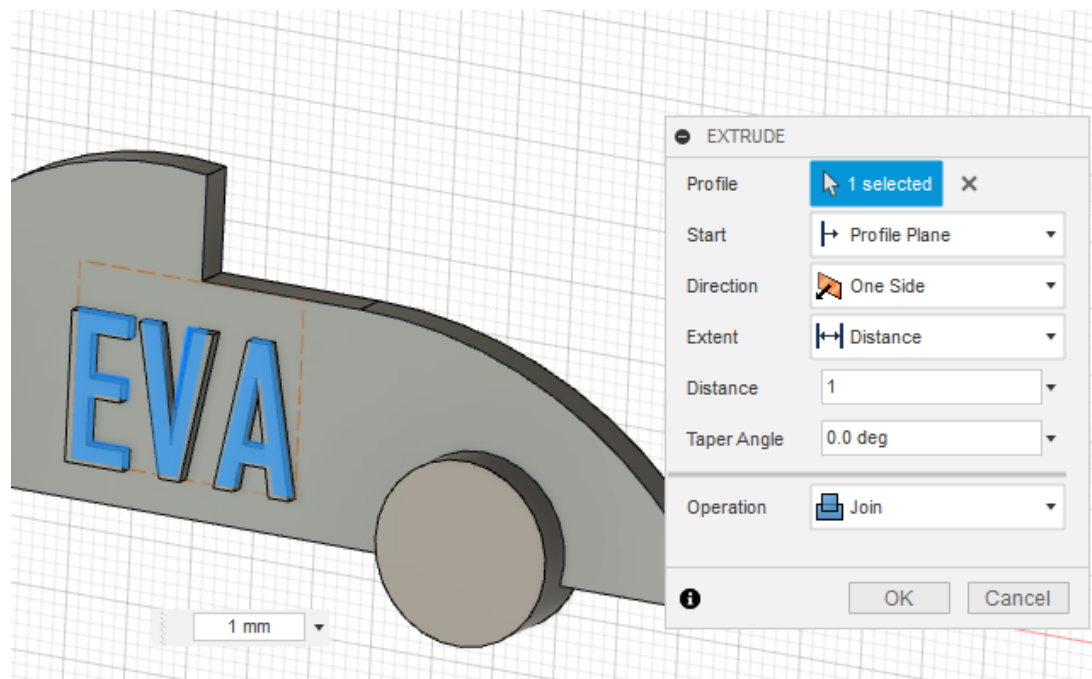
Set the Distance to -0.5 mm, and the Operation to Cut. If you rotate the car a little bit, you can see that the text is cutting *into* the car. Click OK.



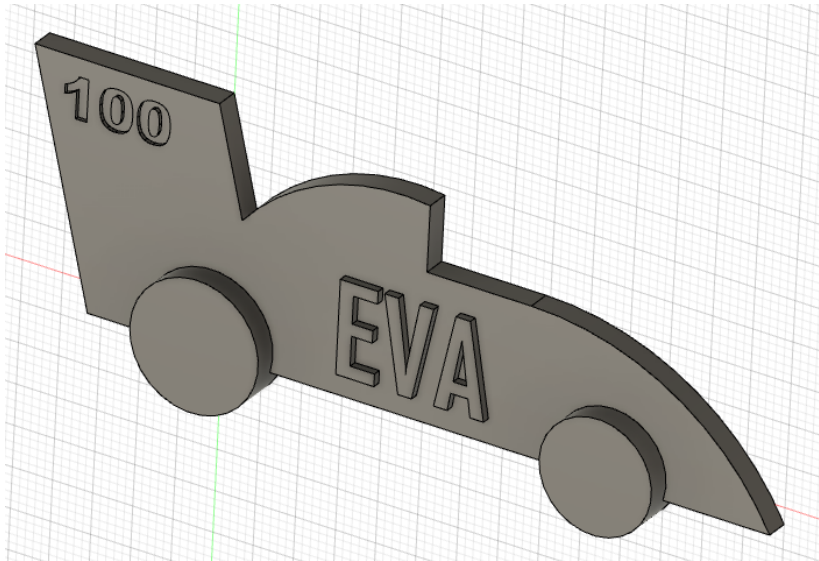
49) Turn the car over so you can see the front side.

Add your own name to the front side the same way as you added EVOLOCITY. Adjust the text size so your name will fit between the wheels.

Click the text, and then extrude the text by setting the **Distance** to 1 mm, and select the **Operation** to be **Join**. Remember that pressing **E** on your keyboard takes you directly to the extrusion command.

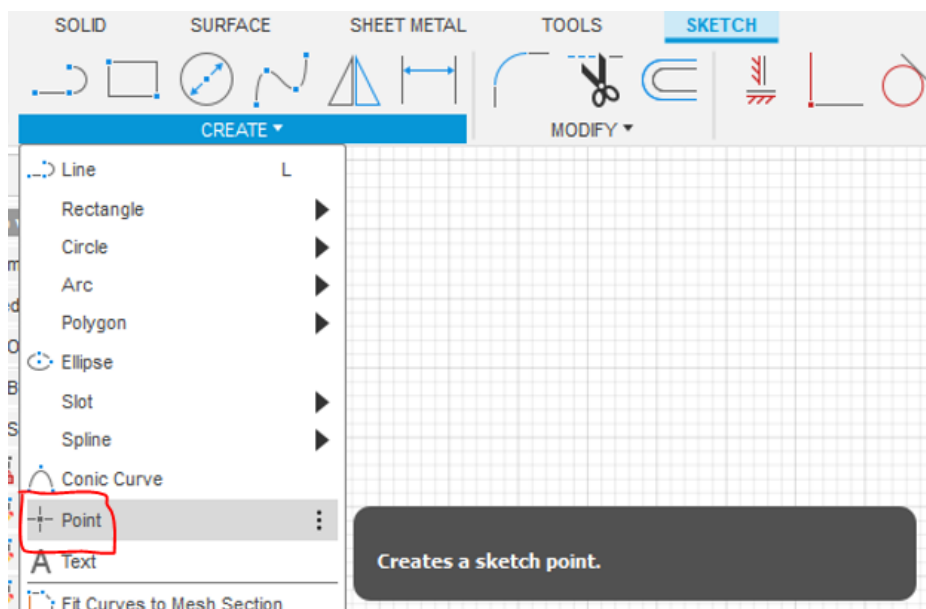


- 50) Add your racing number or name to the tail using the same method.
Set the **Font** to Arial Black and the size to 7 mm.
Extrude -0.5 mm and select the operation to be **Cut**.

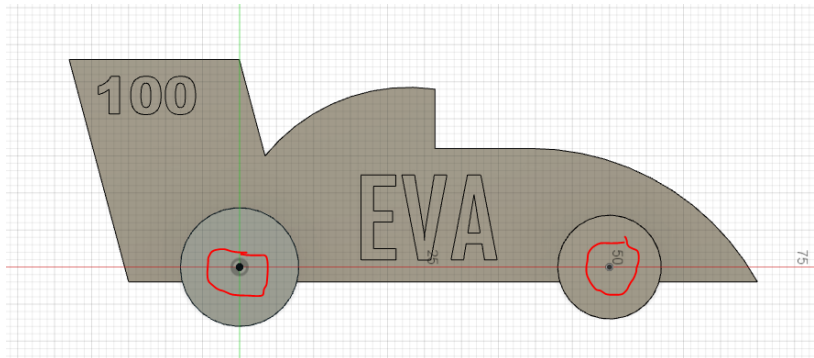


- 51) Start a sketch on the rear wheel by clicking the **Sketch** button and then clicking on the rear wheel.
Because we extruded both wheels the same amount, the plane of both wheels will now be in the plane of the new sketch.

- 52) Under the SKETCH tab, expand the list under CREATE and select **Point**.



53) Place a sketch **Point** in the centre of each wheel. Click **Finish Sketch**. The points are found in Sketch5, which is still visible as illustrated with the eye next to Sketch5 in the model browser.



54) Next, we will use the **Hole** tool. You could go to the SOLID tab and select **Hole** by pressing the button on selecting it in the list, but we will access it a different way. Press the **S** key on the keyboard. Start typing “Hole”, and the Hole tool will appear in the dropdown list. Click on it. The Hole tool is now active. You can press S and search for any command in Fusion 360.

55) Click on the point you placed in the centre of the *rear* wheel. Use the following settings in the Hole dialogue box:

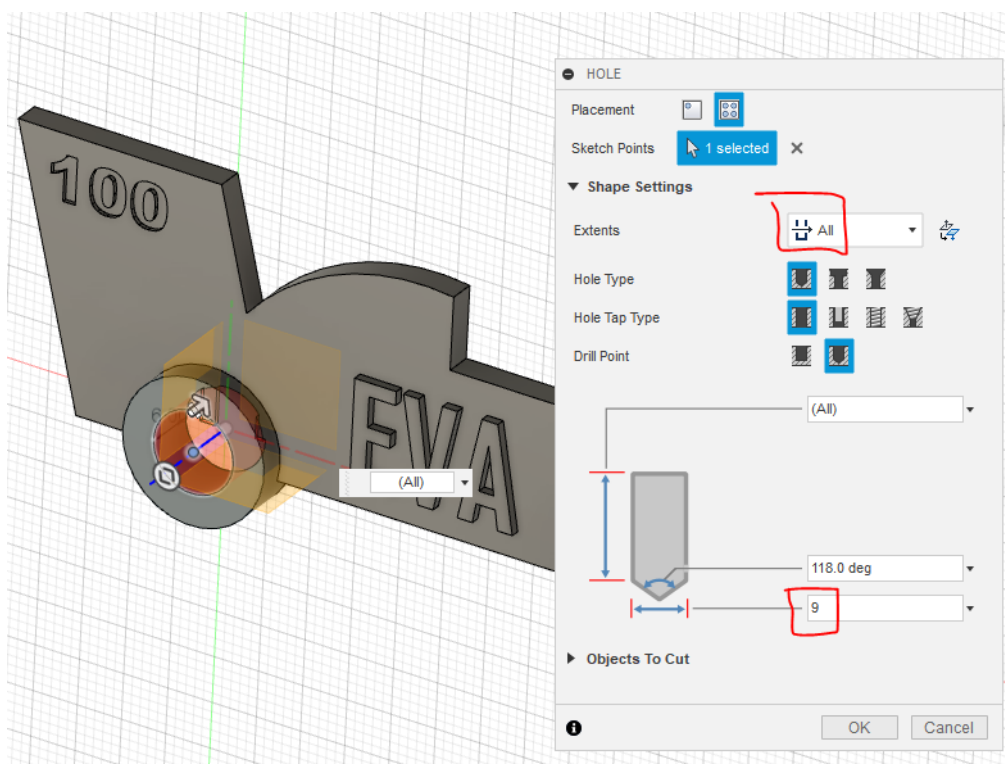
Extents – All

Hole type – Simple (default)

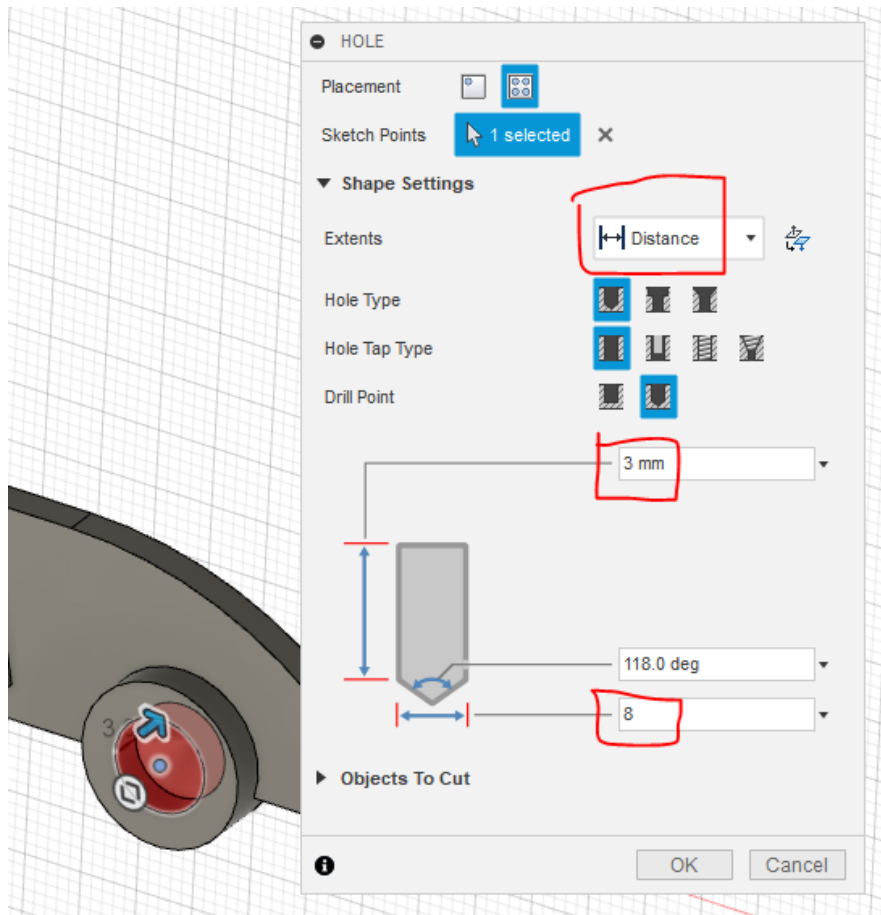
Hole Tap Type – Simple (default)

Diameter – 9 mm

Click **OK**.

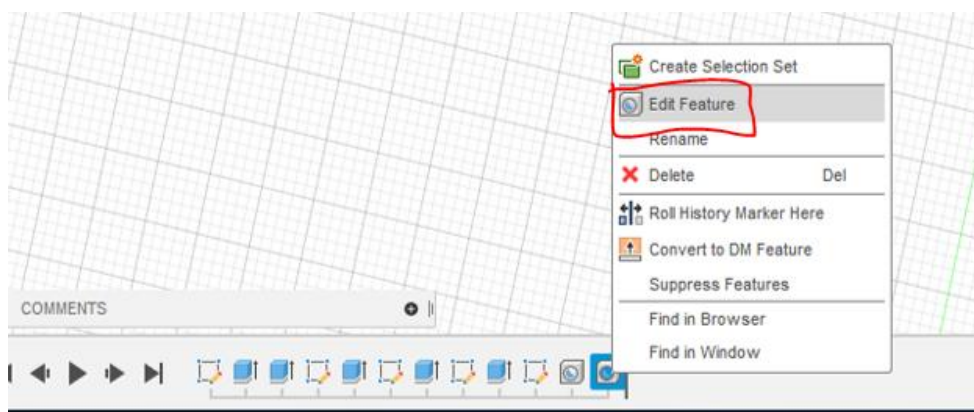


- 56) Make Sketch5 visible again by clicking on the eye next to it in the model browser. Start the Hole tool again. Click on the point in the centre of the *front* wheel. Set the Extents to Distance, and enter 3 mm. Set the Diameter to 8 mm. Click OK

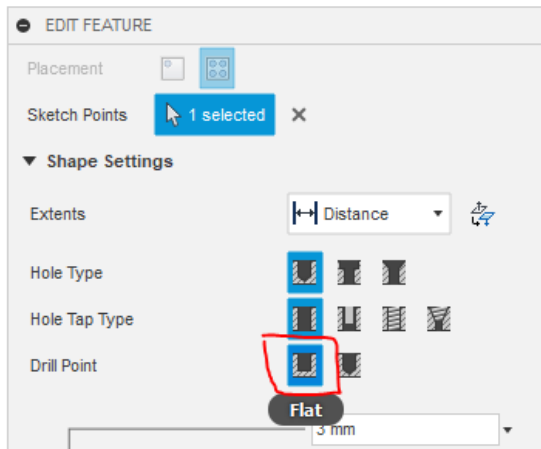


- 57) If you rotate the car you can see that the bottom of the hole is pointy. We want the bottom of the hole to be flat. You don't have to undo the hole, we can just edit it.

At the bottom of the page there is a bar showing the history of the actions. Right-click on the last hole symbol, which represents the hole we just added. Select **Edit Feature**.

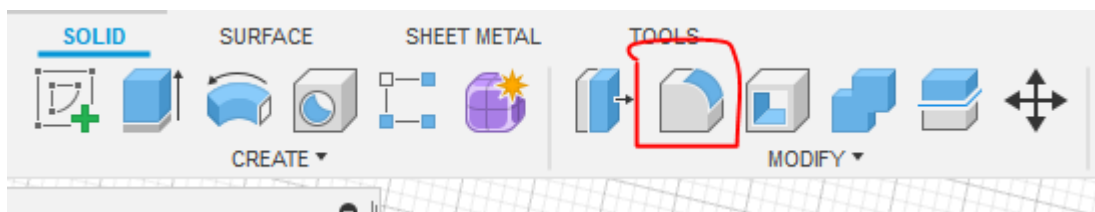


58) Set the **Drill Point** to **Flat**. Then click OK. You can use the same method to change all kinds of features, even long after they are created.

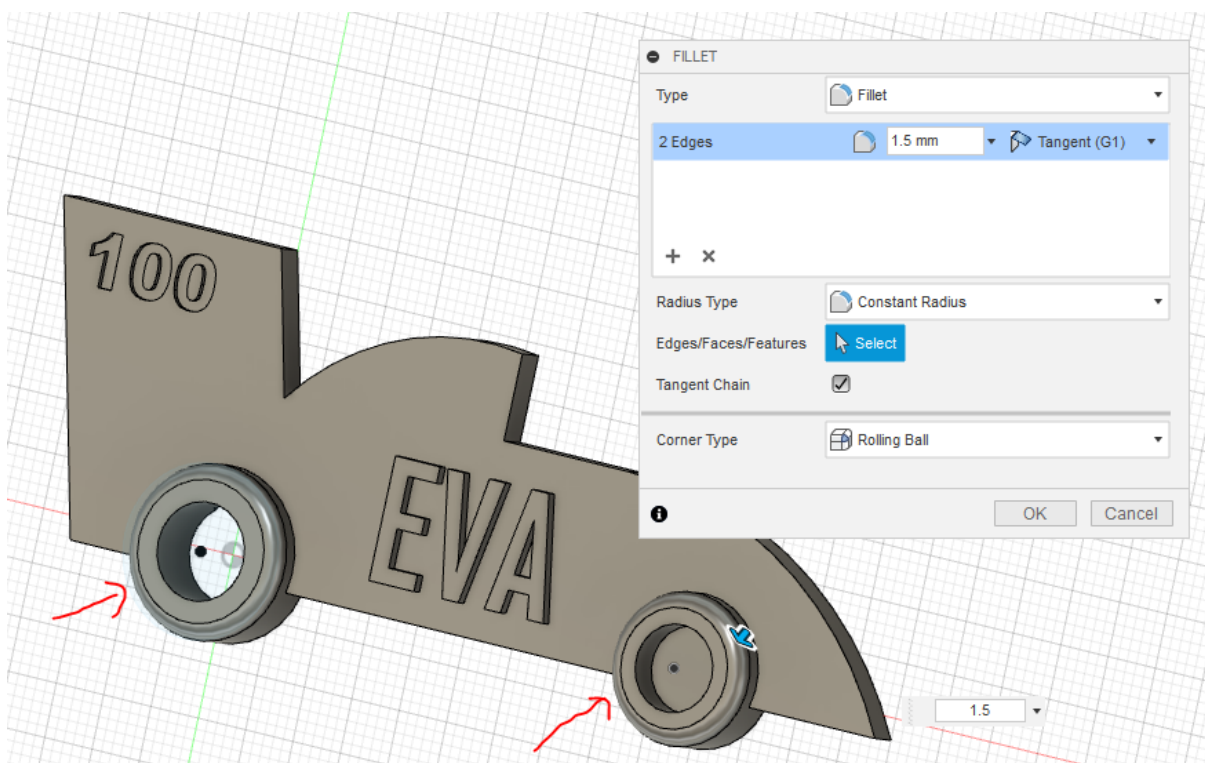


59) Save your design. Make it a habit to save your work frequently.

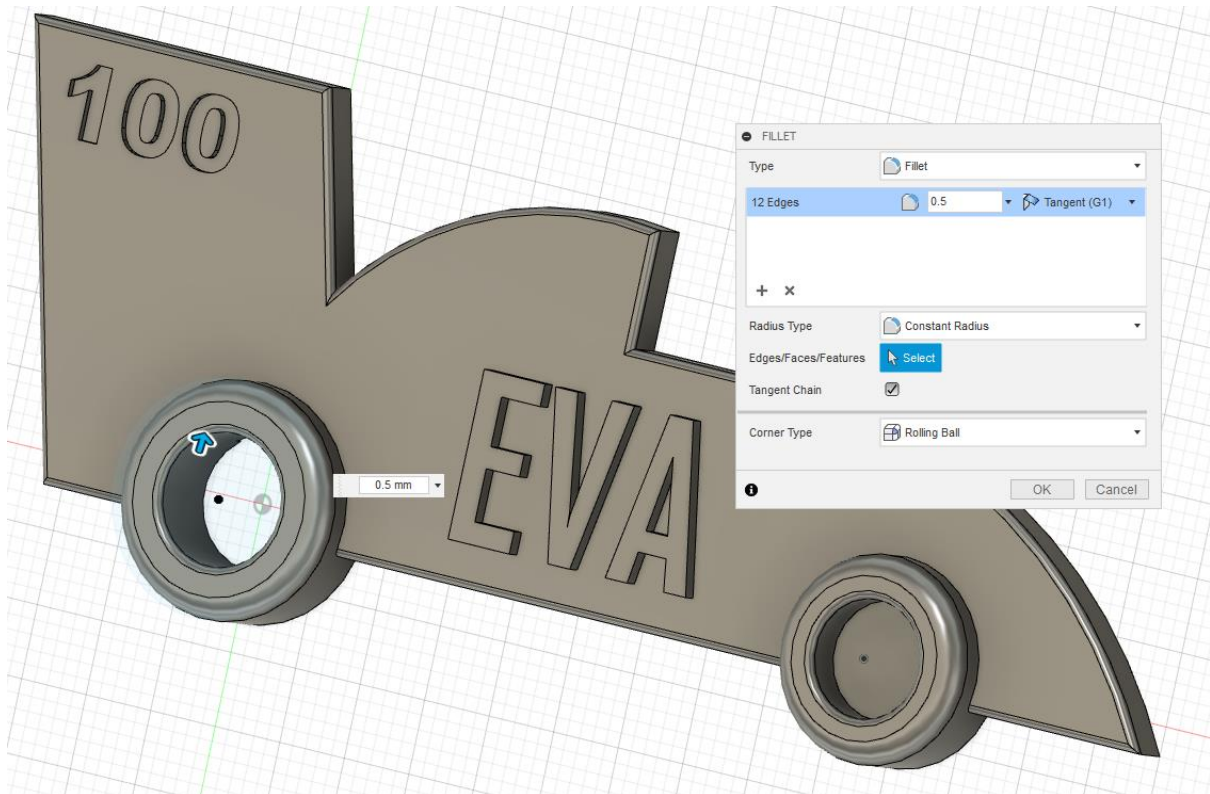
60) In the **Modify** group (under the SOLID tab) select **Fillet**.



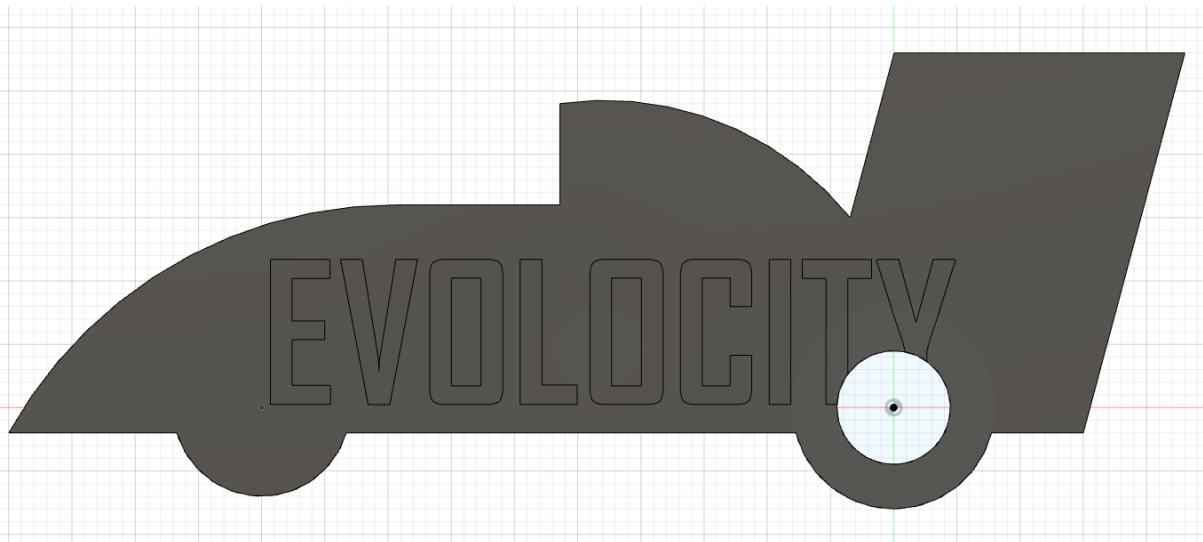
61) Click on the outer edges of the wheels, and set the radius to 1.5 mm. Click OK.



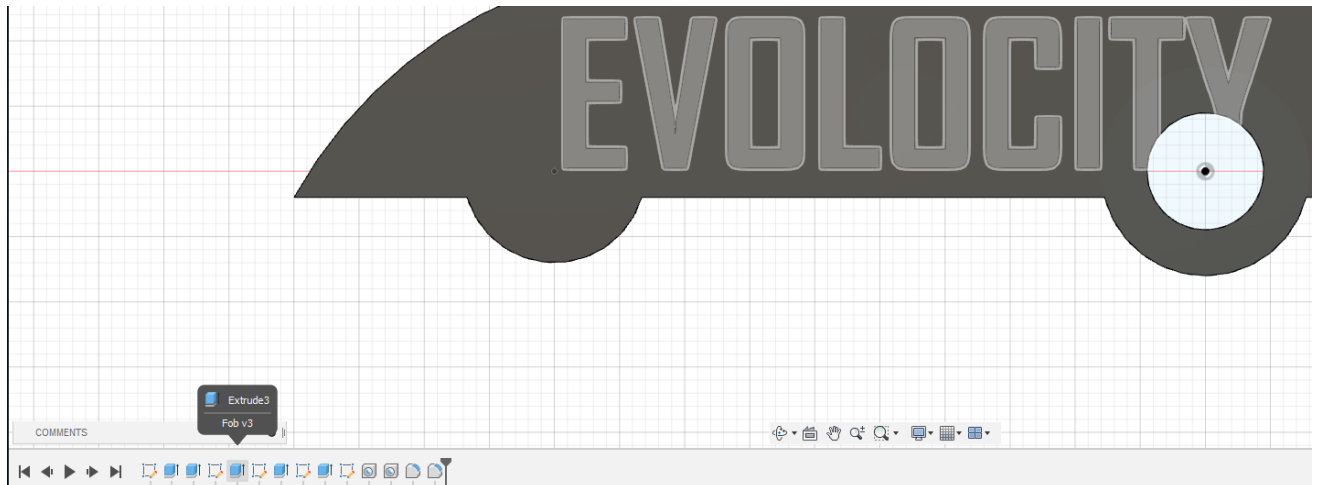
62) Select the **Fillet** tool again. Select all the edges on the front of the car including the inside of the wheels and set the radius to 0.5 mm. Click OK.



63) Turn the car over. You might now discover that the rear wheel has cut right through the EVOLOCITY name. No need to panic, we will fix this!



64) At the bottom of the window, hover over the symbols in the history line until the EVOLOCITY logo lights up. It looks like the extruded text is named Extrude3.

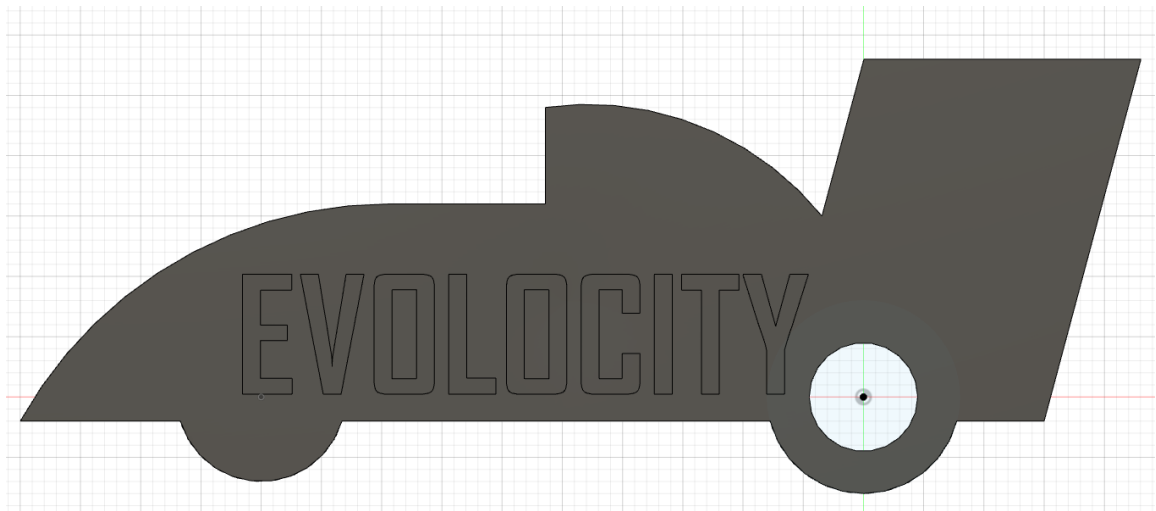


65) Because we created the extruded featured directly after we had added the text, the sketch containing the EVOLOCITY text should be the sketch directly to the left of Extrude3. This is Sketch2, and right-click on it and select **Edit Sketch**.

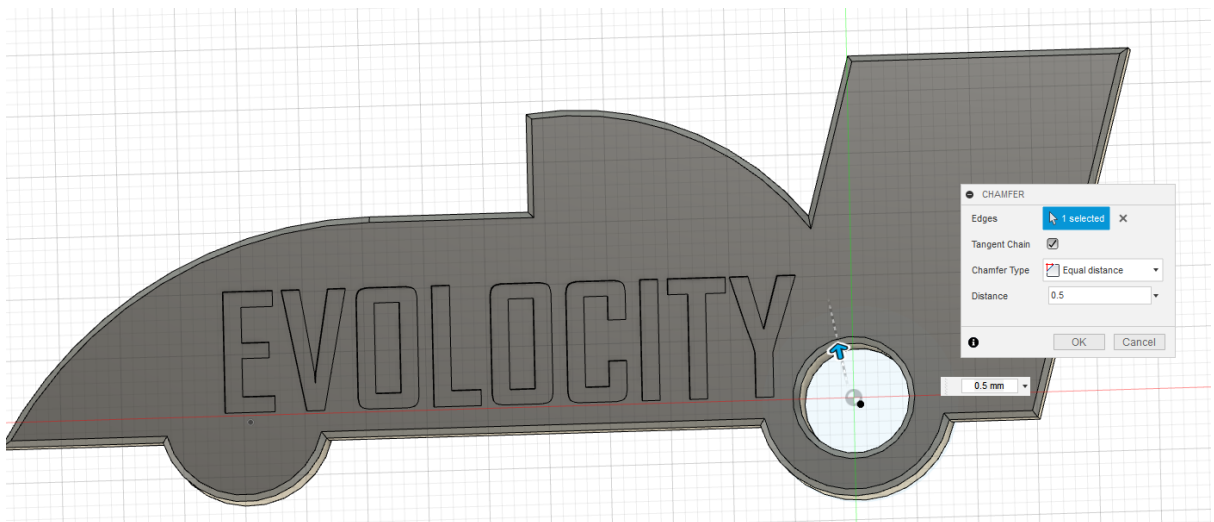


66) You can now drag the text to relocate it. You can also double-click to change the font size or font. You will notice that there is suddenly no hole through the rear wheel. This is because we have essentially gone back in time and we are editing things that happened before the hole was added. However, you can easily estimate where the hole will be, and place the text so it doesn't get cut. Click OK and then click Finish Sketch.

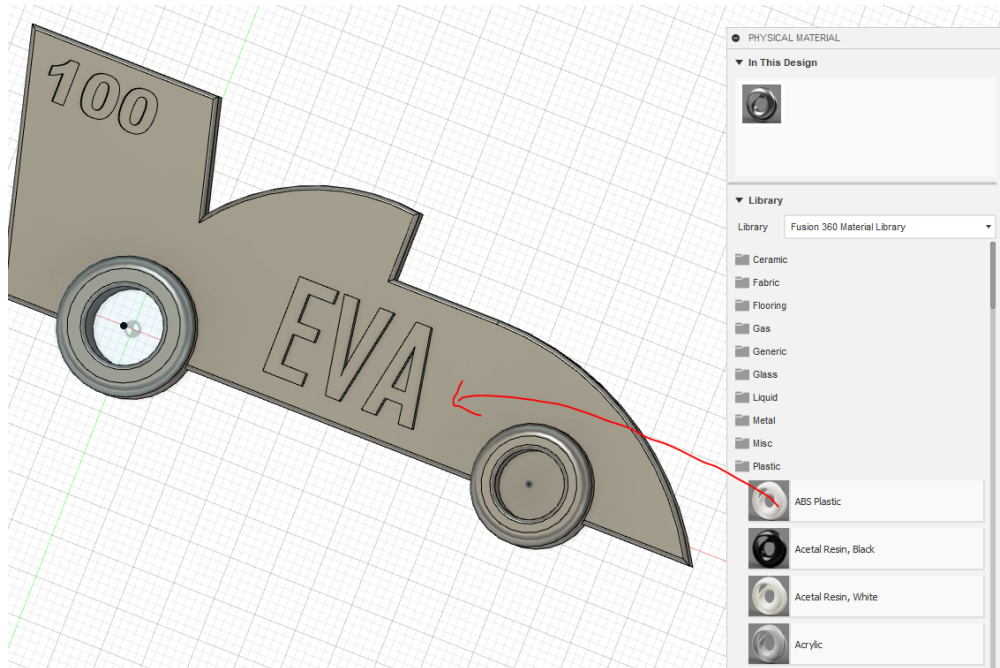
67) Your car should now look similar to this:



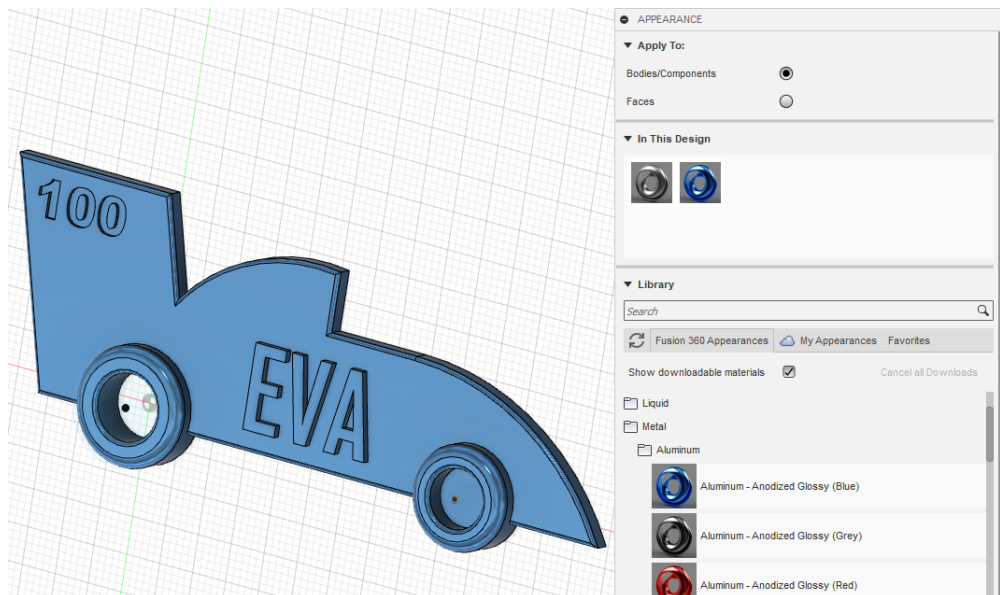
68) Expand the list under the **MODIFY** group (in the SOLID tab) by clicking the chevron. Select the **Chamfer** tool. This is similar to the Fillet tool, but it cuts the edge instead of rounding it. Select all edges of the back of the car include the inside of the rear wheel, and set the Distance to 0.5 mm. The chamfer will make it much easier to remove the printed part from the 3D printer bed. If you miss an edge, just start the tool again.



- 69) Expand the list under the **MODIFY** group and select **Physical Material** in the list. Navigate to **ABS Plastic**. Left-click on ABS Plastic and hold down the mouse button as you drag the material to the car. Release the mouse button. The material is now set to ABS Plastic. Close the dialogue box.



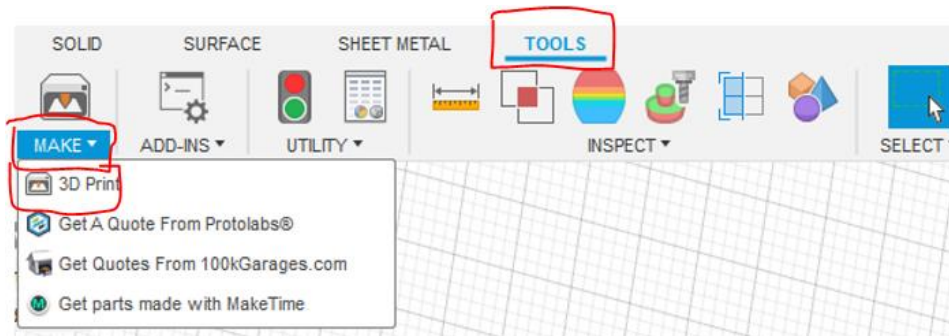
- 70) Expand the list under **MODIFY** again (under the SOLID tab) and select **Appearance**. Explore the appearances and select one you like. I chose Metal > Aluminum > Aluminum Anodized Glossy (Blue). Drag it to the car and release the mouse button.



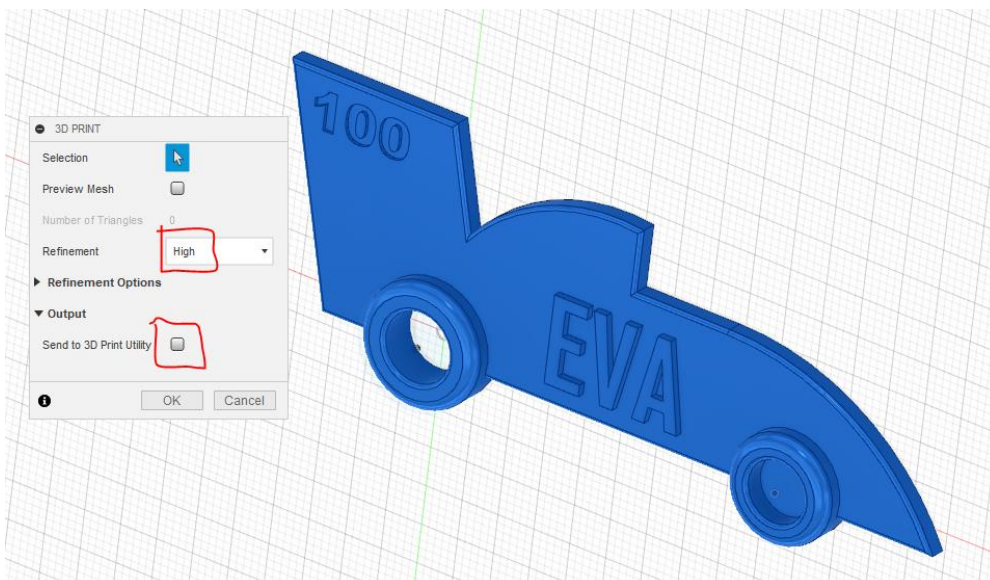
- 71) Your design is now done! Make sure to save the file.
- 72) Rotate it to admire your own work! That wasn't hard, was it? 😊

Export for 3D printing:

73) Click on the tab **TOOLS**. Click on the chevron next to **Make** and select **3D Print**.



74) Set the **Refinement** to **High**, and unclick the box **Send to Print Utility**. Click on the car to select it, then click OK.



75) Give the .STL file a descriptive name, and save it! (STL is the universal file format used for 3D printing. STL stands for “stereolithography”, which is a type of 3D printing).

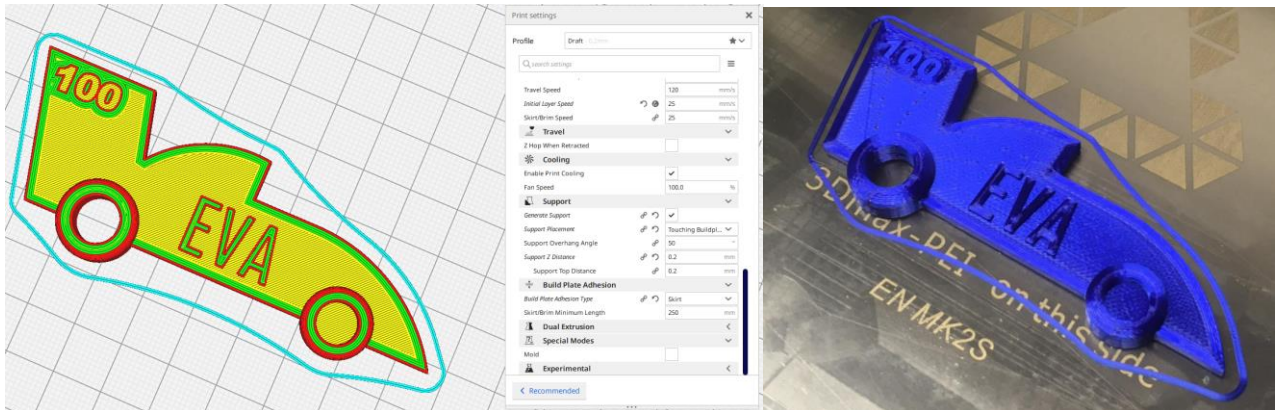
You are now done! Yay! 😊

3D print your design

76) The exact settings for 3D printing depend on your specific printer, but print it with the backside towards the bed. The key fob should take about 30-40 minutes to print on a typical consumer grade 3D printer. I printed mine on a Prusa MK2S in silk PLA filament. The Prusa is a nice printer, but the Creality Ender 3 Pro or X is the most affordable printer right now!

77) Thread a keyring through hole in the rear wheel to show off your work on your key chain!

Here is the key fob “sliced” for 3D printing using the free software Ultimaker Cura (left) and the printed key fob before removal from the bed (right).



Because the EVOLOCITY text on the back is cut *into* the design rather than extruded outwards, it prints nice and flat.



Happy CADing! ©