Fusion 360 Tutorial – How to Make a Wheel Rim

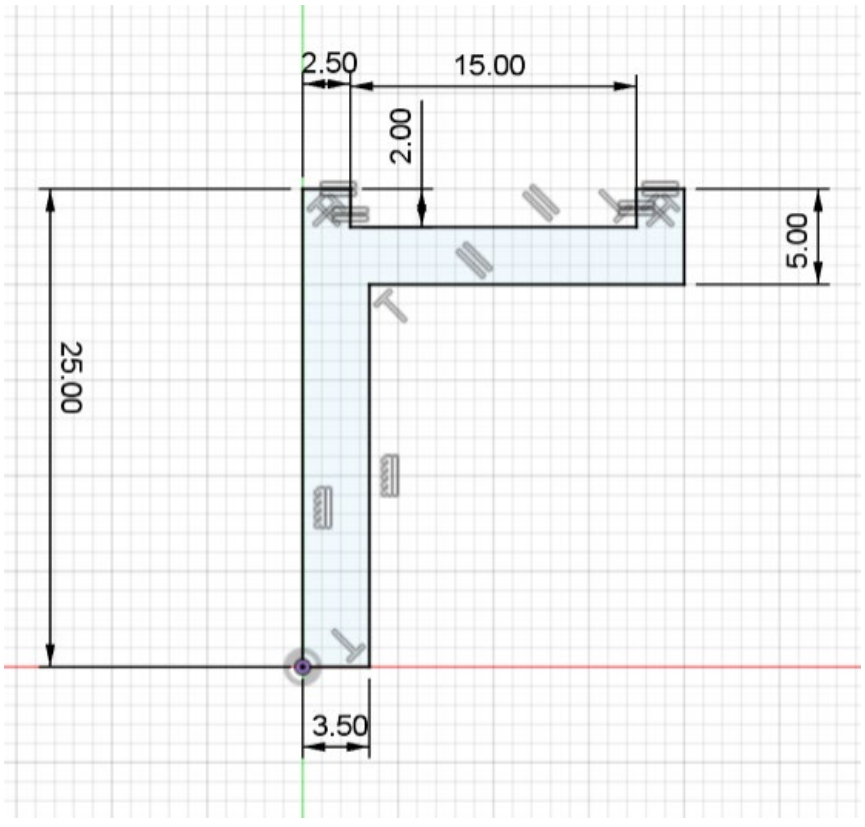
Step 1: Create a New Project and Start a Sketch

Begin by opening Fusion 360 and creating a new design.

Click on Create Sketch and select the right plane (Y-Z plane) as your sketch surface. This will be the side view of the wheel rim.

Now, we'll begin constructing the wheel profile.

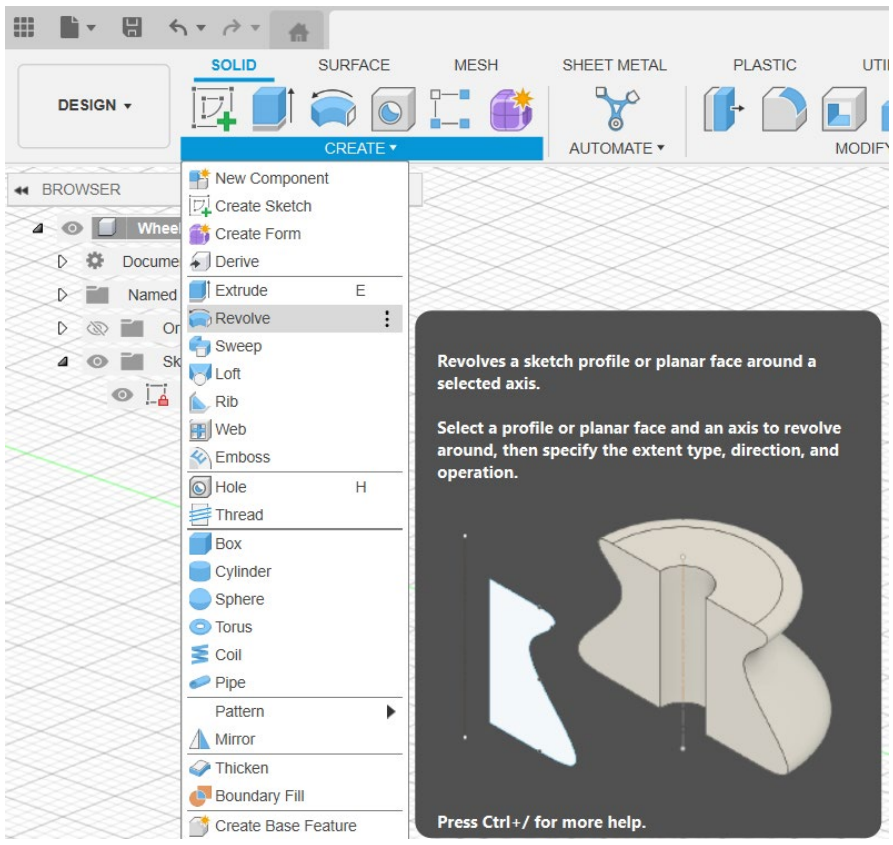
1. Start by drawing a vertical line from the origin going upward by 25 millimetres.
2. From the top of that line, draw a horizontal line to the right measuring 2.5 millimetres.
3. Next, draw a vertical line downward from the end of that line, measuring 2 millimetres. Then draw a horizontal line to the right measuring 15 millimetres.
4. Use the Equal constraint to ensure that the extra lines are the same length as what we just did. Then, use the Sketch Dimension tool to apply and double-check all measurements.



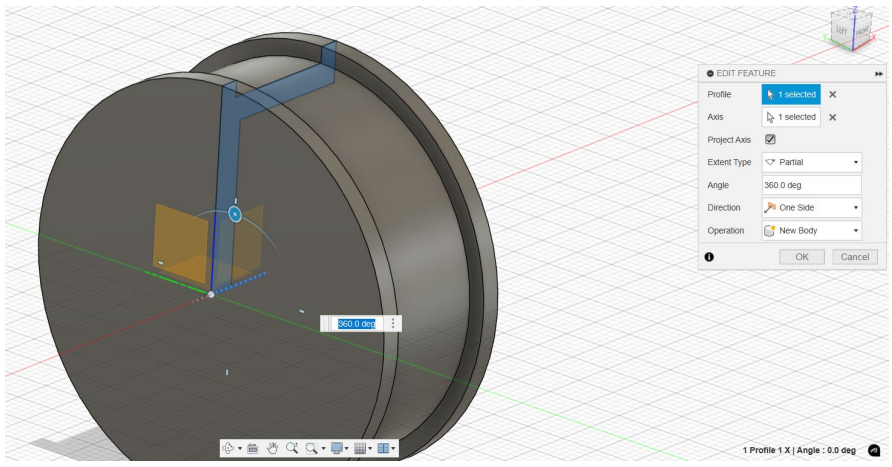
Once your profile is complete and all sketch lines turn black (indicating they are fully constrained), click Finish Sketch.

Step 2: Revolve the Sketch into a 3D Rim

To turn this profile into a 3D wheel rim, go to the Solid tab, then click Create > Revolve.



Fusion 360 will automatically detect the closed profile you just created. For the revolve axis, select the bottom horizontal line of your sketch.



Click OK, and you'll now see a solid 3D model of your wheel rim base.

Step 3: Sketch the Spoke Cutout

Next, we'll sketch the cutout that will form the shape of one of the wheel spokes.

1. Start a new sketch on the front face of the wheel rim (the circular face you're now looking at).
2. Press the P key to activate the Project tool, and then click the outer edge of the face to project its shape into the sketch. This will help us align our new geometry correctly.
3. Now, from the origin, draw a horizontal line to the right measuring 18 millimetres.
4. Then, draw another line from the origin measuring 18 millimetres at a 36-degree angle. This will define the angle of the spoke cutout.
5. Next, draw a Centre-Point Arc from the origin with a radius of 5 millimetres. You'll do this again—to connect each end of the two lines—so you end up with a curved "pizza slice" shape.
6. To keep the arcs aligned, use the constraints as needed.

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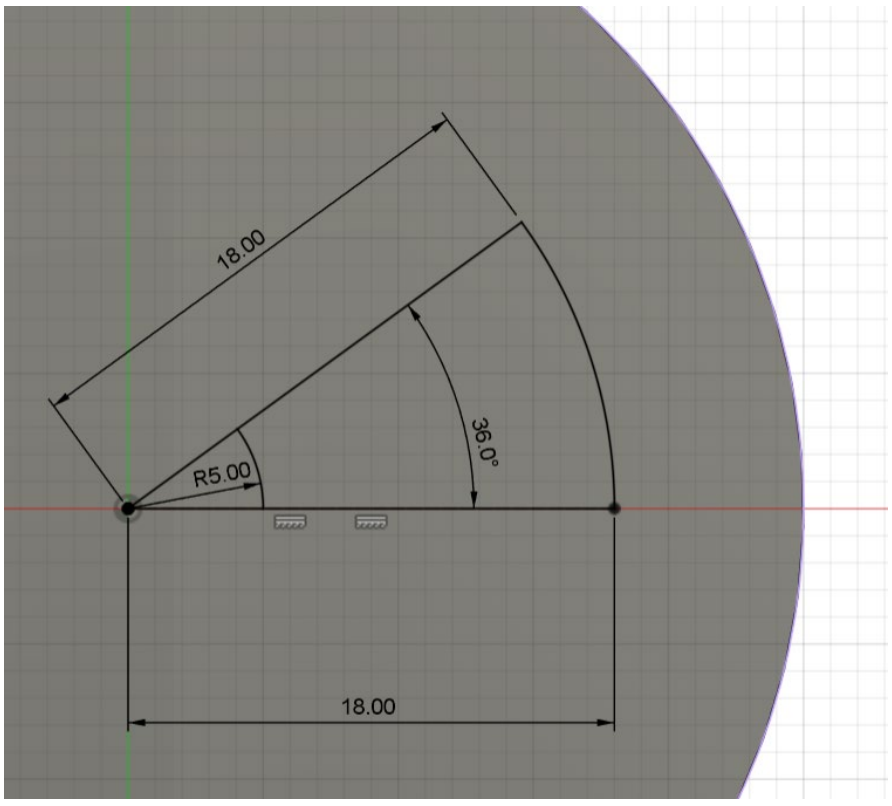
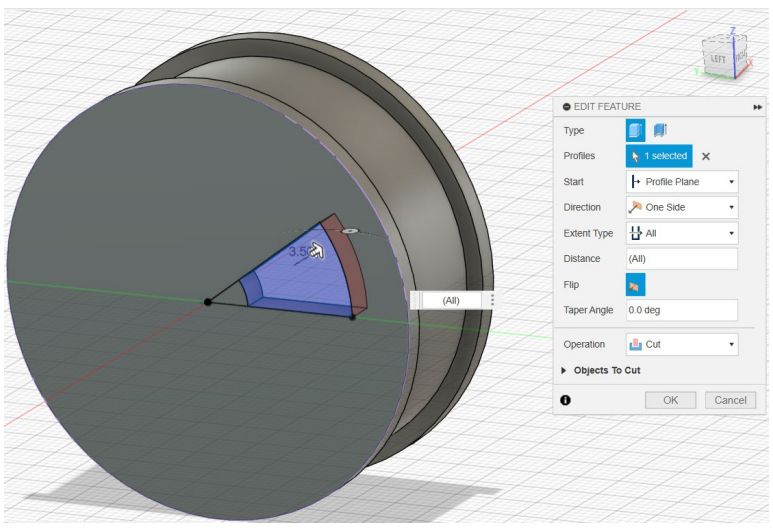


Once the shape is complete, click Finish Sketch.

Step 4: Cut the Spoke Shape Through the Rim

Select the pizza slice–shaped profile you just created.

Go to Create > Extrude, and in the dialog box, change the Direction to All so that it cuts through the entire rim.



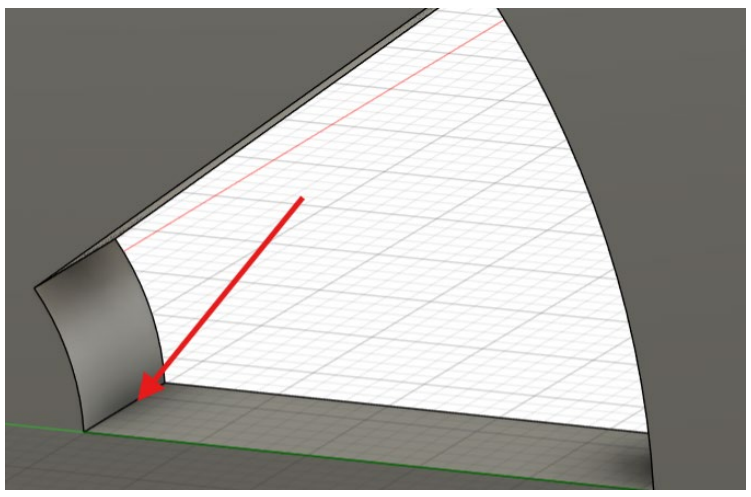
Set the Operation to Cut, then click OK.

You should now see a clean cutout through the rim.

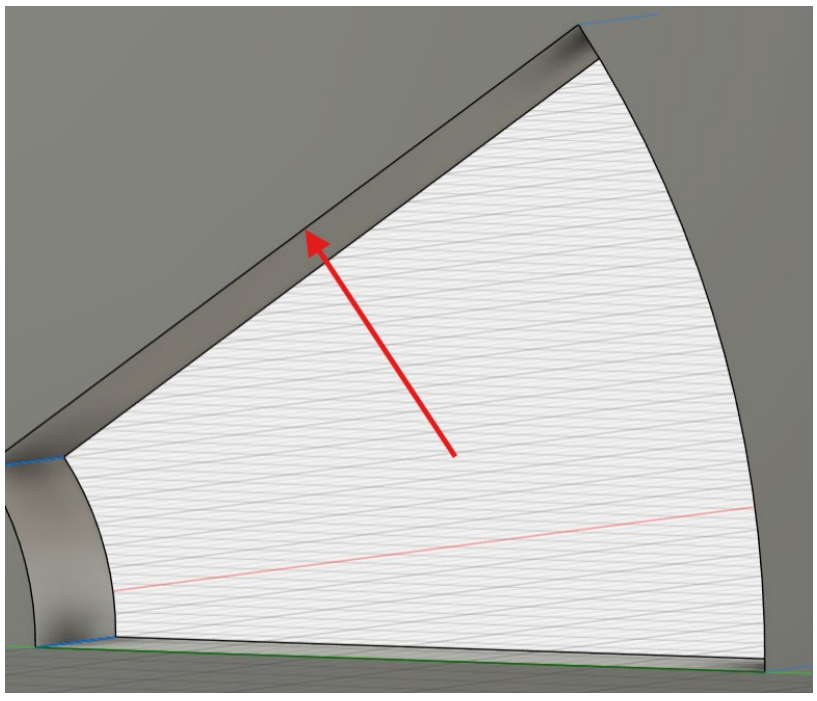
Step 5: Add Fillets for a Smooth Finish

Let’s round the edges of the cutout to give the rim a cleaner, more refined look.

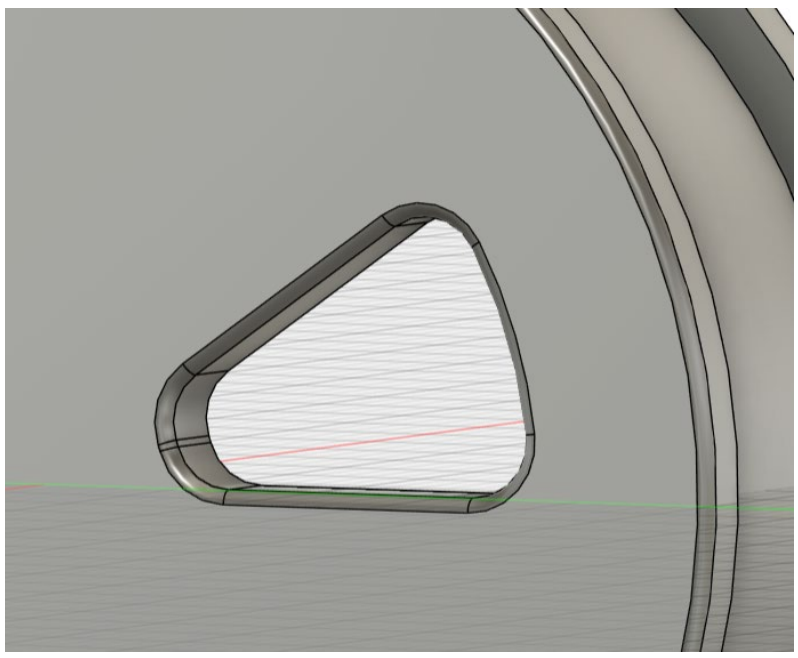
1. Go to Modify > Fillet.
2. Select the four inner edges of the cutout and apply a 2-millimetre fillet. This rounds off the inside corners of the spoke.



3. Next, select the four outer edges on the face of the rim where the cutout starts, and apply a 1-millimetre fillet.



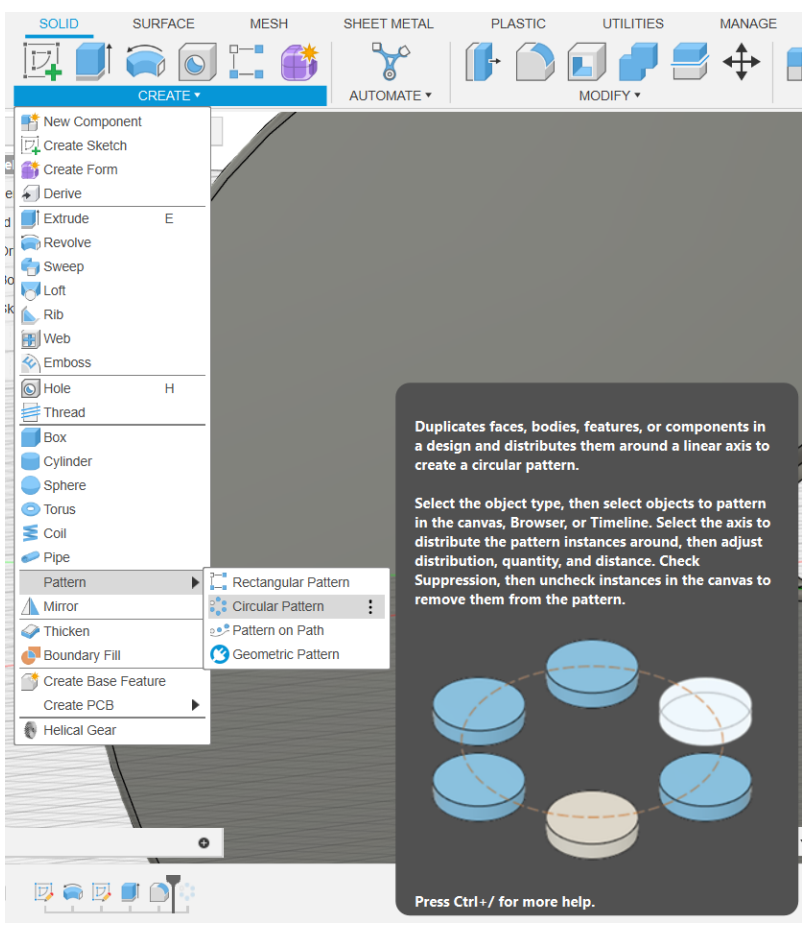
4. If needed, you can also apply smaller fillets like 0.5 millimetres to any remaining sharp corners.



Step 6: Use the Circular Pattern Tool

Now we’ll duplicate the spoke cutout around the wheel to create a full set of spokes.

1. Go to Create > Pattern > Circular Pattern.



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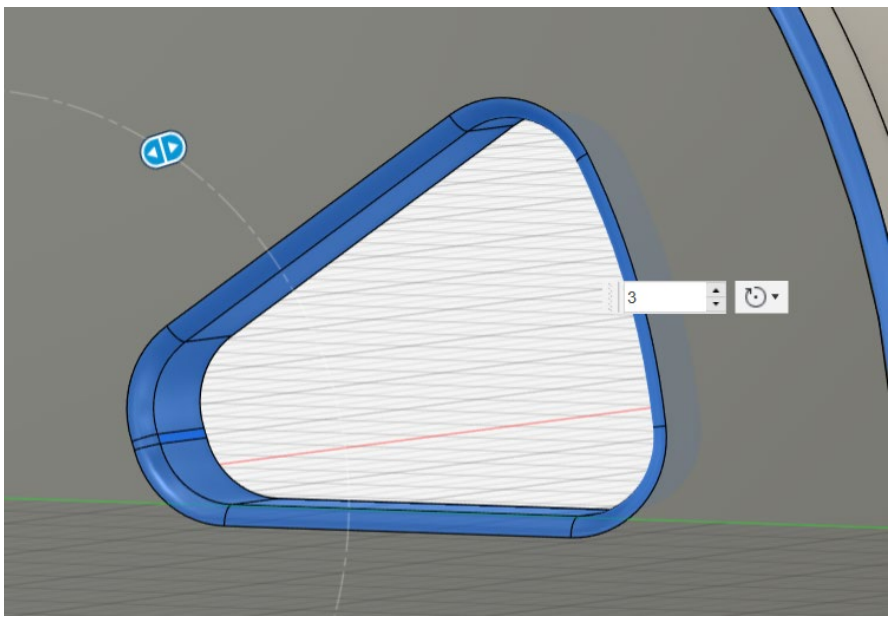
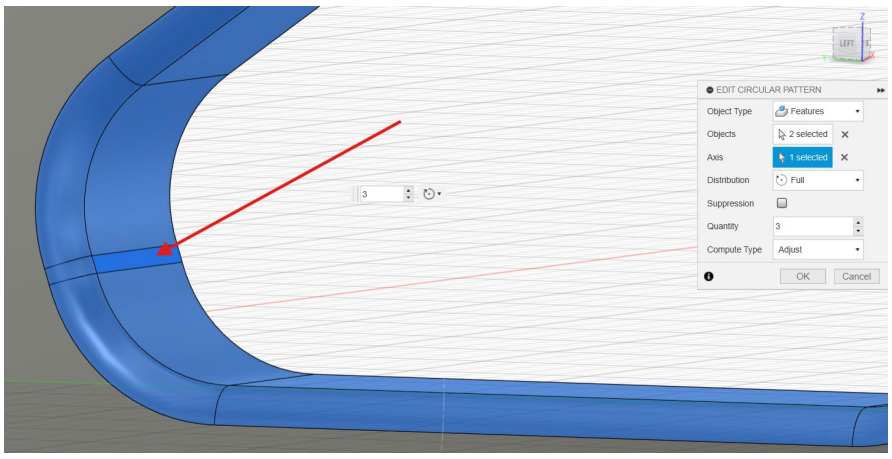
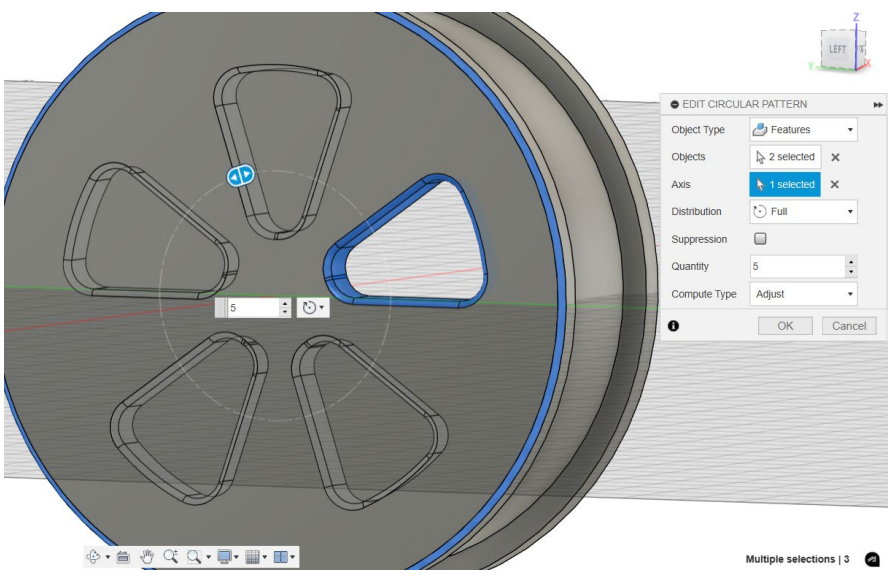
2. In the Pattern Type dropdown, select Features.
3. For Objects, select:
    - The cut feature (spoke shape)
    - Any fillets you applied to it



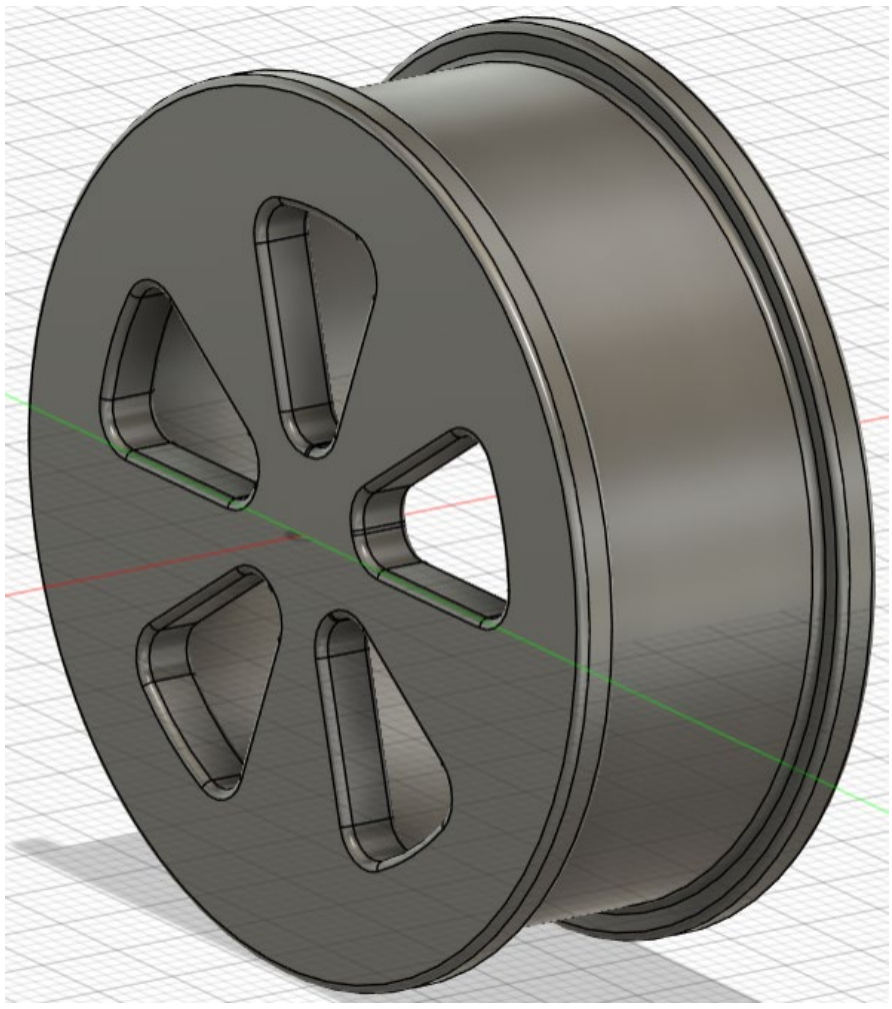
4. Then, for the Axis, select the circular edge of the rim's face (from the original revolve).



5. Set the quantity to 5 so you get five equally spaced cutouts around the wheel.



Click OK, and you'll now see all five spokes evenly arranged around the rim.



Step 7: Add Final Touches

To complete your design, go through and add any additional fillets to smooth out remaining sharp corners.

For example, use the Fillet tool and set values like 1 mm or 0.5 mm for minor edge rounding, depending on your design preference.

This helps make the wheel rim look more realistic and ready for fabrication or rendering.