As we have seen, *GeoGebra* is a tool designed for mathematics. If we want to create objects for 3D printing we need to use `Computer Aided Design' software, known as CAD.  *GeoGebra* is free software designed specifically for educational use.  The CAD tool we will use know is called `*TinkerCAD*' and is also free software designed for educational use. The version of *GeoGebra* I used was software I installed in my laptop and can use without needing Internet access. TinkerCAD is designed to be used in an Internet `browser', such as Google's `*Chrome*'. It means you do not need to install any software, but you do need to have an Internet connection. That means you need to remember to save your work pretty frequently so it doesn't get lost if there's a loss of connection.

When you start up TinkerCAD from https://www.tinkercad.com you will need to register to use it. This means that each time you come back it will start up with your latest saved work. When you start a new design it will randomly generate some weird name for it. So I have changed my file's name to `*Basic wheel as a disc*'.  Down the right hand side is a menu of basic solid shapes, called `primitives'. As far as possible we will try to select objects from this menu which we can move about, rescale, and combine. I have dragged two Cylinders onto the main `Workplane'.  I have coloured the first one red, using the shape `Inspector' dialog. As I needed to change its dimensions I also dragged a `Ruler' onto the Workplane. You should be able to see 6 white areas, each with numbers in. Reading from left to right these show (a) 0.00, (b) 80.00, (c) 0.00, (d) 80.00, (e) 5.00 and (f) 0.00.   You should try changing each of these to see how they alter the layout. (a) and (c) change the position of the ruler relative to the object. (b) and (d) define the size of the cylinder's base – in this case a circle of diameter 80mm. If they are unequal you will get an ellipse instead of a circle. (e) defines the cylinder's depth i.e. 5mm. (f) defines its height above (or below) the Workplane.   The value of zero means it is resting on the Workplane. You can use the right-mouse button to change the 3D viewpoint. Clicking on the brown cylinder you can see it has a quite different set of dimensions
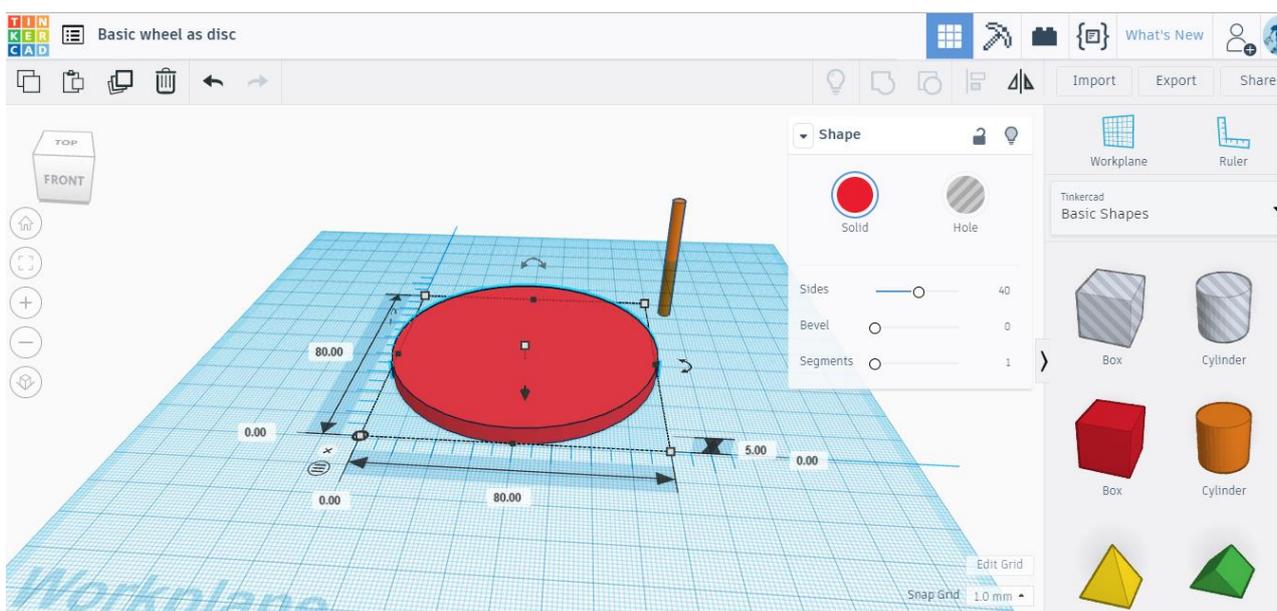
I have dragged the brown cylinder well away from the red one. Its cross section has a diameter of 4mm. Its height is 50mm and its base is 25mm below the Workplane. In order to position it at the centre of the red cylinder we just change the numbers 91.00 and 117.00 both to 40.00.
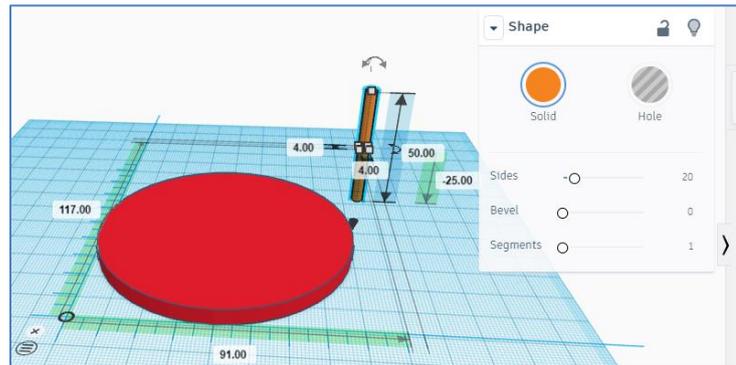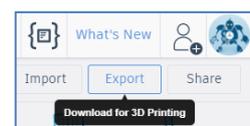
At the moment the thin brown cylinder is sticking out through the red one, like an axle. But I want it to drill a hole in the wheel to take an axle. So in the `Shape' dialog box I have clicked on the `Hole button'.
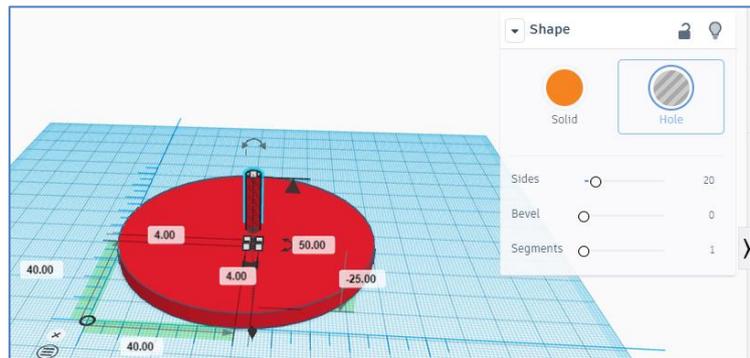
Now we just have to group the positive red cylinder and the negative brown one together. With the `Shift' key pressed, left click first on the red cylinder, then on the brown one. The icon representing the `Group' command should now turn bold. Click on it to combine the positive and negative shapes to drill a small hole through the centre of the `wheel'.
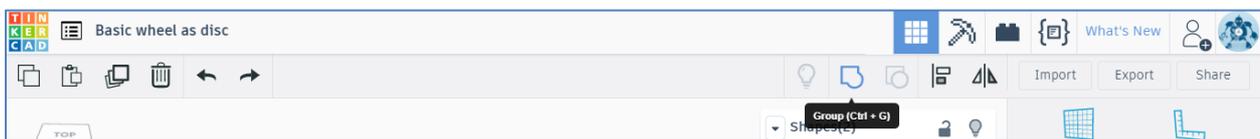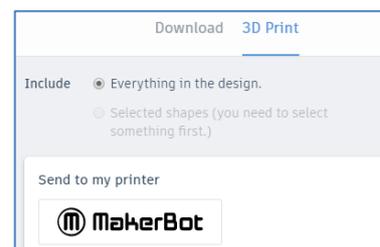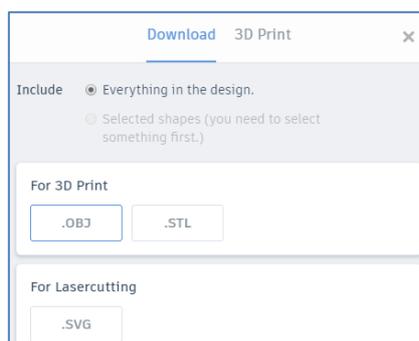
With the right mouse button held down you can adjust the view to see through the hole. Make sure you save your work. Now you can `Export' it as 3D printing file, or send the file directly to your 3D printer.

You can also use `Share' to publish your work so that other people can access your design.

I have uploaded my work to a website called `Thingiverse', so anyone can use and play with my creation. You can get many good ideas from other people's creations. If you want help on using TinkerCAD there is a very useful tutorial page here.

So here is the object published on Thingiverse ready for anyone to use.

We will now try to create a more sophisticated wheel using a hub, a tyre and 3 spokes, as we did in *GeoGebra*.
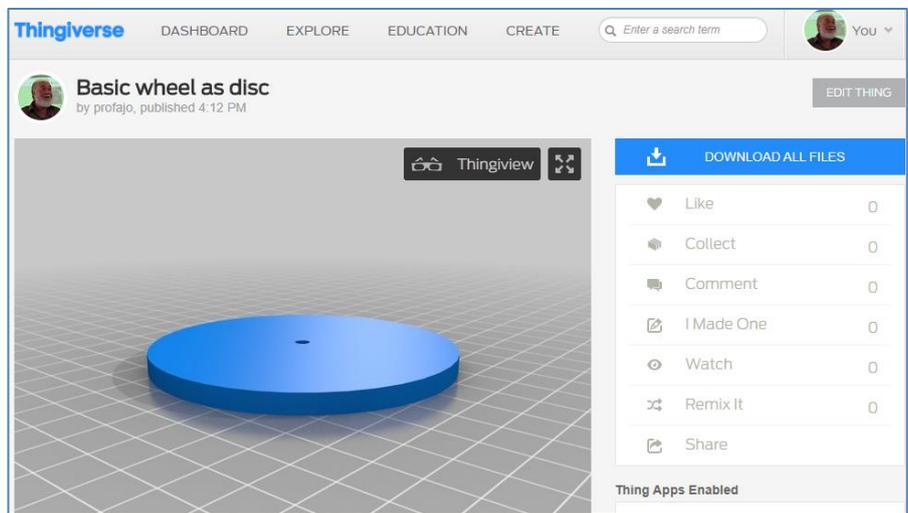
We have a couple of choices for the wheel itself, as either a Torus or a Tube. I'll make a design using a Tube, you can try using the Torus as a tyre instead. Actually we can use a ready-made Tube to create the hub as well.
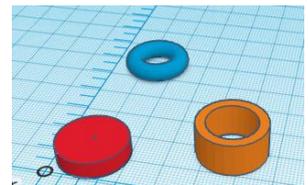
The outer tube is coloured magenta, and given an outer diameter of 80mm.

It is 5mm deep. The slider in the Shape Inspector is used to control the width of the tyre's wall. The inner tube is positioned to have the same centre as the outer one by adjusting the coordinates of the bottom left hand corner of its 20x20 box relative to the bottom left corner of the 80x80 box for the tyre.
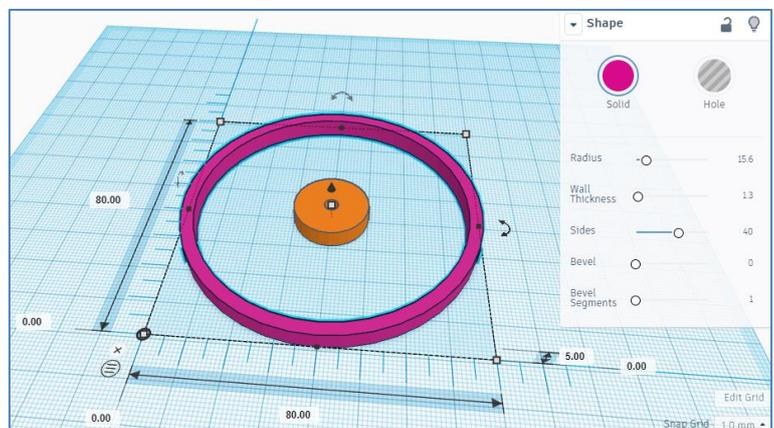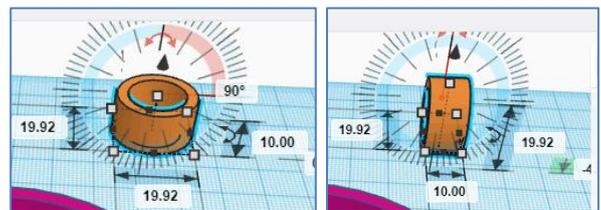
All we now have to do is to create the first spoke as a cylinder, and learn how to rotate it 120° to create the remaining spokes. Finally we can group all five objects together to create a single printable solid object.

It might be a good idea to use tubes rather than cylinders for the spokes, to save weight.
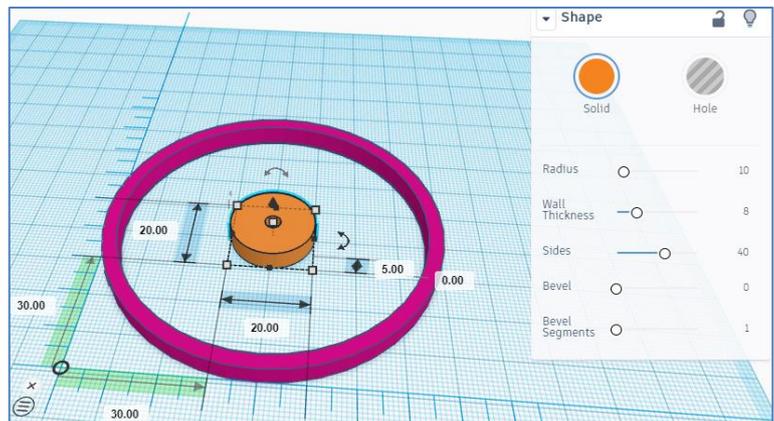
The trick to getting the tube in the right position is to learn how to rotate a shape around any of the 3 main axes. If you look carefully at the black arrow head pointing up from the tube you will see a couple of arrow heads above it linked by a circular arc. Click on the arc and a protractor will appear. Drag the red line through an angle, such as 90° and release. Now you know how to rotate! Now you can position the tube to join the hub and the tyre.

When this is made the correct length, and put in the right position we will have created the first spoke, shown in green. The other tool we need is called `Duplicate'. It is the third icon on the top left of the screen, after Cut and Paste and before Delete. With the green tube selected, click on Duplicate and drag the copied shape clear of the other components. Click on it and locate the arc with the two arrow heads for spinning in the Workplane. Rotate this through 60° - you will probably need to enter the

number 60 manually. Now drag this into position to form a second spoke and colour it blue. Finally Copy and Paste (or Duplicate) this spoke, and place it clear of the other components. Rotate it to point in the correct direction and place it back in the correct position between the hub and the tyre. So now we are ready to weld all 5 components by grouping them together. Hold down Shift and right-click on each of the 5 objects in turn. Click on the Group icon and wait a few seconds while TinkerCAD

makes them into a single object. Change the colour if you like. Rename the file to something sensible, and you have created you wheel. Now you can print and/or share it. Mine is here.