

TinkerCad Wrench Tutorial

For this Tinkercad tutorial, we chose to design a wrench. It's a familiar and simple shape, and is useful in daily life. If you haven't seen a wrench before, it is used by mechanics for tightening nuts and bolts. Here's what one looks like:



We'll use this real-life wrench as a template for this Tinkercad tutorial.

For the purpose of this Tinkercad tutorial and to better acquaint you with the site's interface, we've split the design of the wrench into five simple steps. These cover the basics of 3D modeling as we go. At the end of the Tinkercad tutorial, you will not only be able to design the wrench yourself but will also have the skills to design any simple 3D model that you wish.

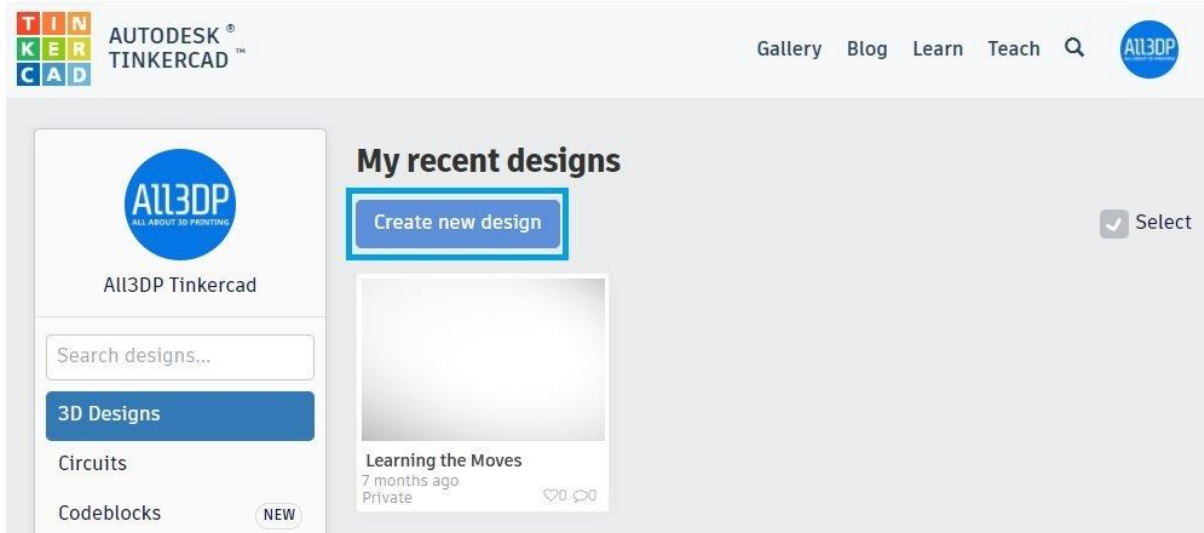
For those who prefer to watch their directions rather than read them, you can check out our YouTube version of this Tinkercad tutorial, embedded below.

1. Create a Tinkercad Account and Create a New Project

Using Tinkercad is easy because it runs in your browser — so you don't need to download or install any software. All you have to do to get started is sign up for a free Tinkercad account on their [website](#).

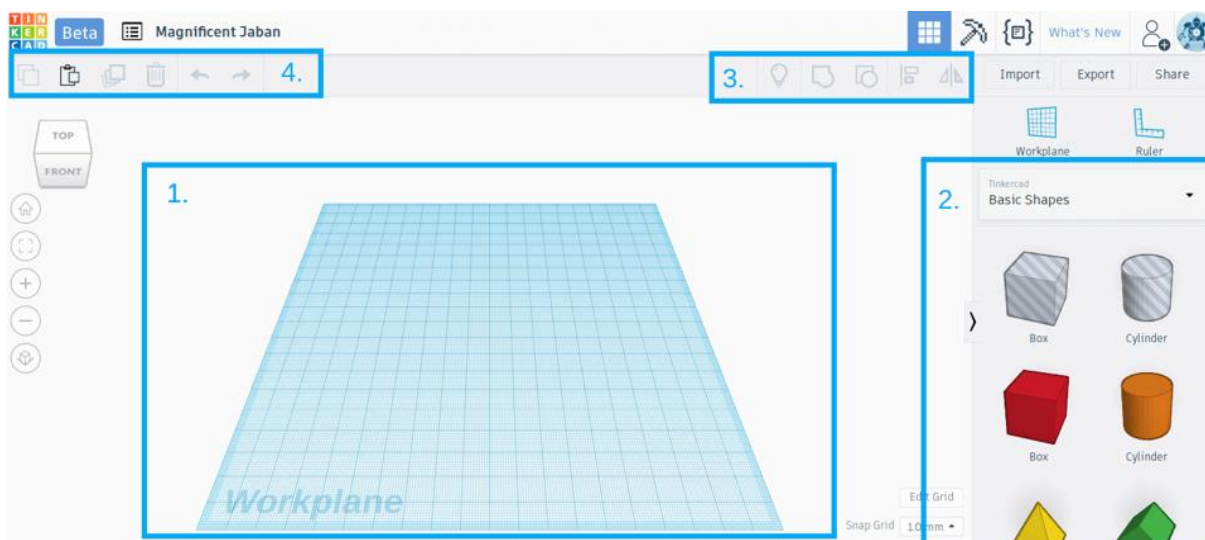
At the end of the signup process, you will be welcomed by the Tinkercad tutorial screen. You can skip the tutorial for now by closing it.

Instead, go to the home page (shown below), and click the "Create new design" button.



Creating a new project on Tinkercad.

This will open up a new project page, which looks as follows.



An empty Tinkercad workplane for you to tinker with.

This is the Tinkercad interface.

1. Notice the blue plane at the center of the page (marked 1). This plane is known as the *workplane*. This workplane is the 3D analog of a Photoshop or GIMP canvas. We will do all our designing on this workplane.
2. Notice the panel with geometric shapes on the right (marked 2). This panel contains the basic building blocks for everything we will design.
3. Notice the menu items on the top bar (marked 3 and 4). We will use these menu items to perform important operations like aligning, merging, etc.
4. Finally, right-click and drag your mouse to change the 3D perspective of the workplane. Scroll up and down to zoom in and out.

2. Design the Ring

Now that we have familiarized ourselves with the Tinkercad interface, we will start designing the wrench.

STEP 2.1: DIVIDE AND CONQUER

A key principle in 3D modeling is “divide and conquer.” It means that when modeling a complex real world object, we should divide it into smaller geometric shapes and then design these smaller units individually. In the final step, we should put these small units together. This makes modeling a complex object easy.

Applying this principle to the wrench, we divide it into three basic units:

1. The ring
2. The handle
3. The jaw



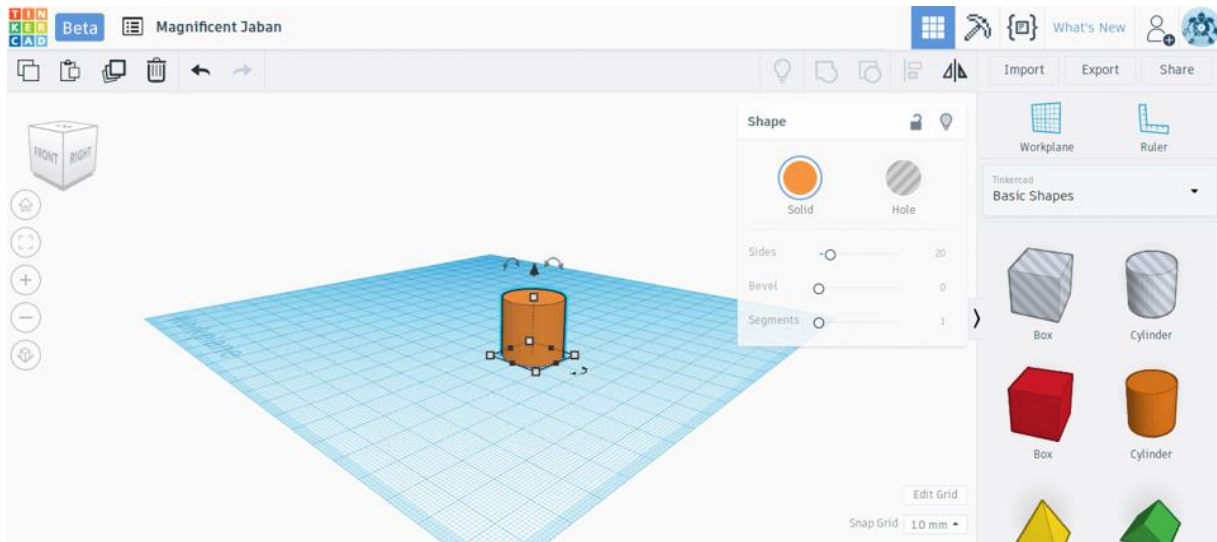
The different parts of a wrench – the ring, the handle, and the jaw.

We will start by designing the ring.

The ring of a wrench can be approximated by a solid circular disk with a hollow circular area in the middle.

STEP 2.2: CREATING THE SOLID CIRCULAR DISK

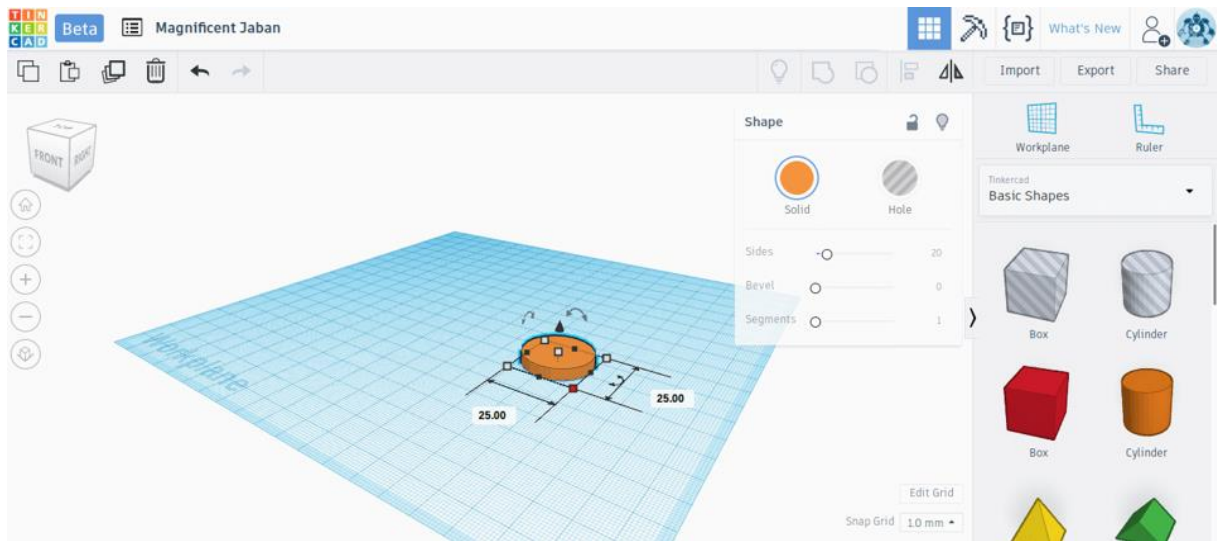
To create this shape, we will first drag a cylinder from the Basic Shapes panel to the workplane. You will notice that the cylinder has five white square handles, four of which are on the plane and one of which is above the cylinder.



The white handles let users change the dimension of a Tinkercad shape.

The four handles on the plane can be used for adjusting the cylinder's length and width. We want our ring to have a diameter of 25 mm. Therefore, drag the square handles till the cylinder has a length and width of 25 mm.

The one above the cylinder can be used for adjusting its height. We want our wrench to have a height of 4 mm. Therefore, click the handle with the left mouse button and drag it down till the cylinder has a height of 4 mm.



We changed the dimensions of the Tinkercad cylinder to make a disk with diameter 25 mm and height 4 mm.

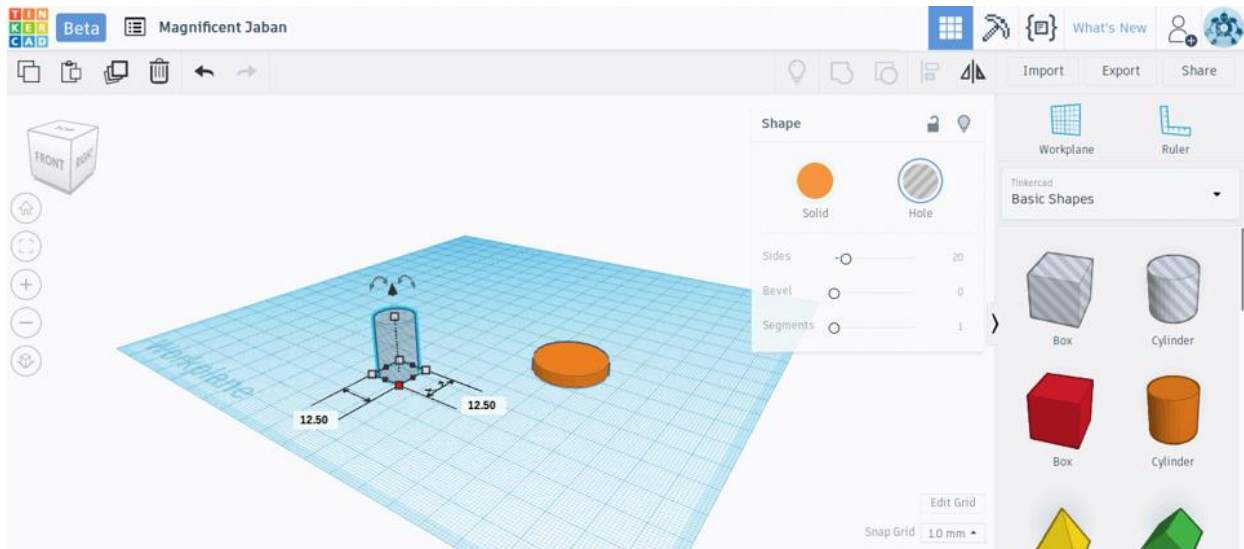
STEP 2.3: CREATING THE CIRCULAR HOLE

We are making progress: We now have the solid disk. But how do we make the circular hole in the middle?

Tinkercad operates on the principle of Boolean design, which means it allows you to *subtract* objects to create hollow areas or *add* objects to join two shapes. In this case, we will need to use subtraction to create the circular hole in the middle of our ring.

First, drag another cylinder to the workplane. Use the white handles on the plane to adjust the length and width of the cylinder to 12.5 mm, which is the diameter of the hole that we want to create.

Now comes the fun part. With this cylinder selected, choose “Hole” in the panel on the upper right corner (marked by the blue rectangle in the image). This prepares this shape to be subtracted out.

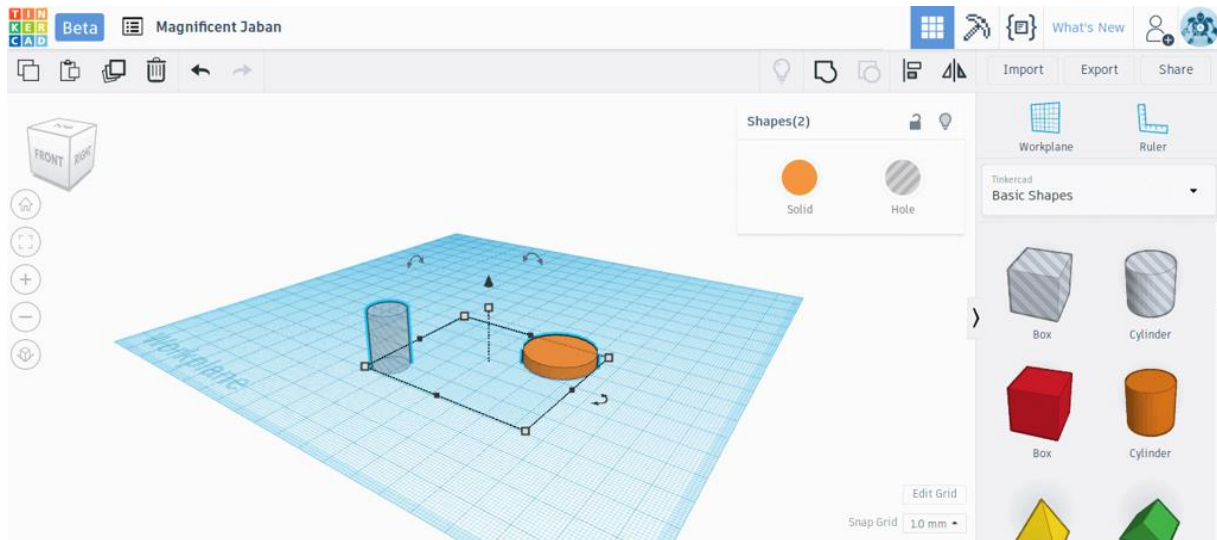


A cylinder-shaped hole.

STEP 2.4: ALIGNING THE HOLE AND THE DISK

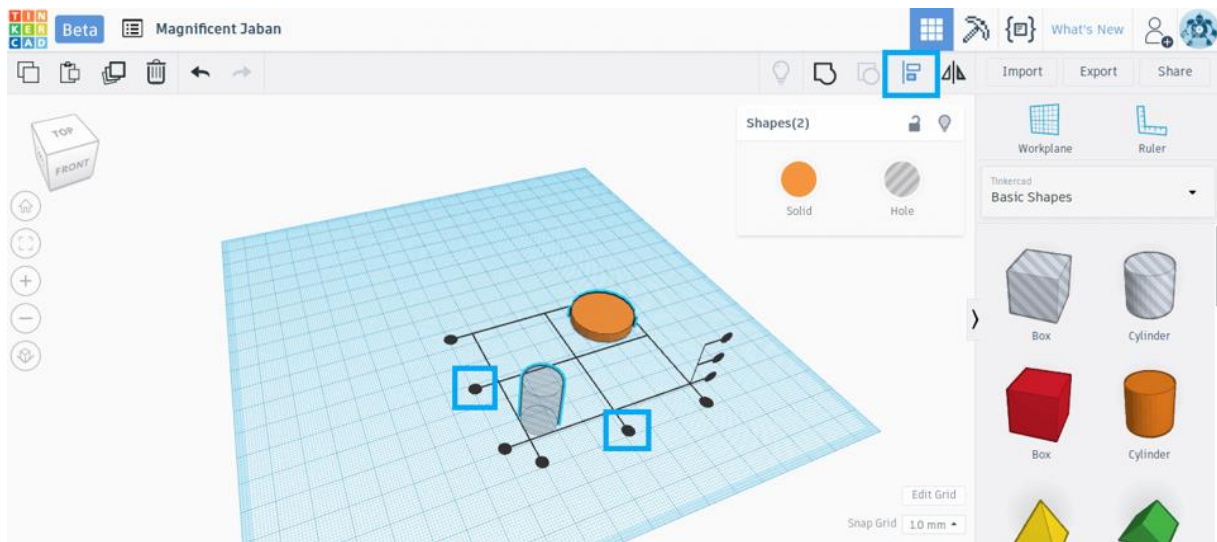
Next, we need to bring this cylinder-shaped “hole” in the middle of the solid disk. We will use the alignment tool in the top menu to achieve this.

To use the alignment tool, select the cylinder-shaped hole by left-clicking. Now press the shift button and select the solid disk. If you do this correctly, you will see that both objects are now simultaneously selected and highlighted.

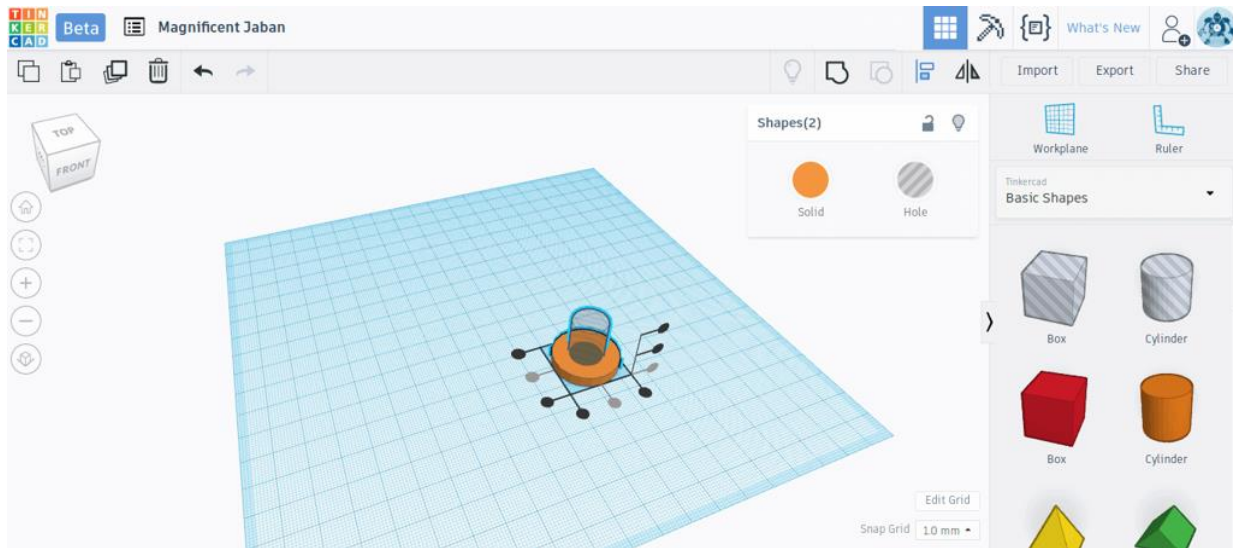


Aligning the objects.

Press the Align button on the top menu (marked with a blue rectangle). Black alignment handles will appear on the workplane. Click on the appropriate handles (also marked with blue rectangles) to bring the solid cylinder in the middle of the solid disk.



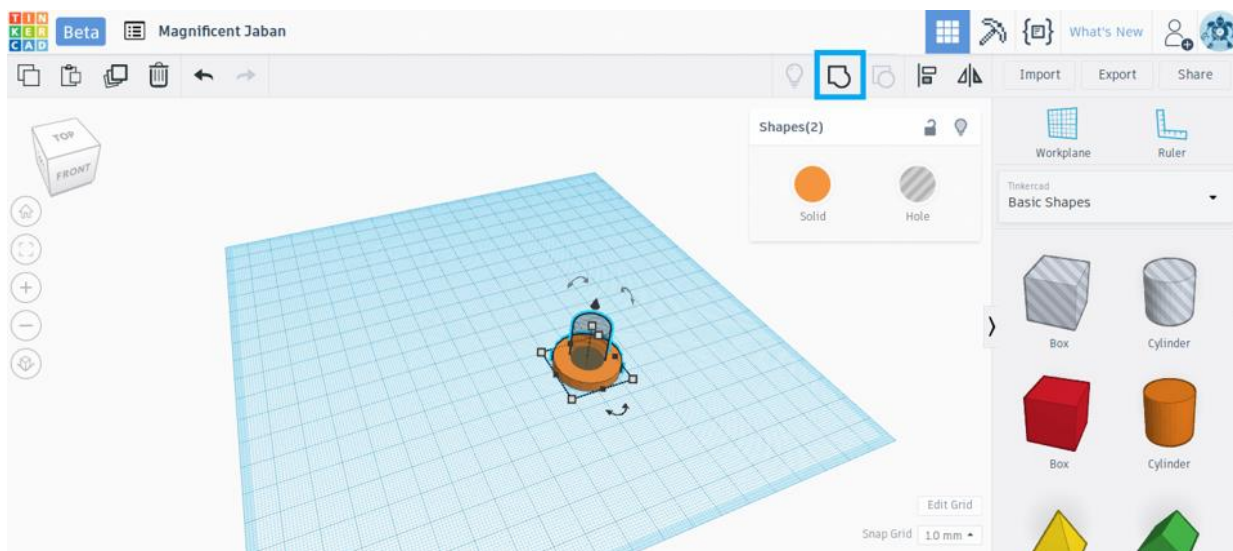
The black alignment handles help align two shapes in Tinkercad.



Voila!

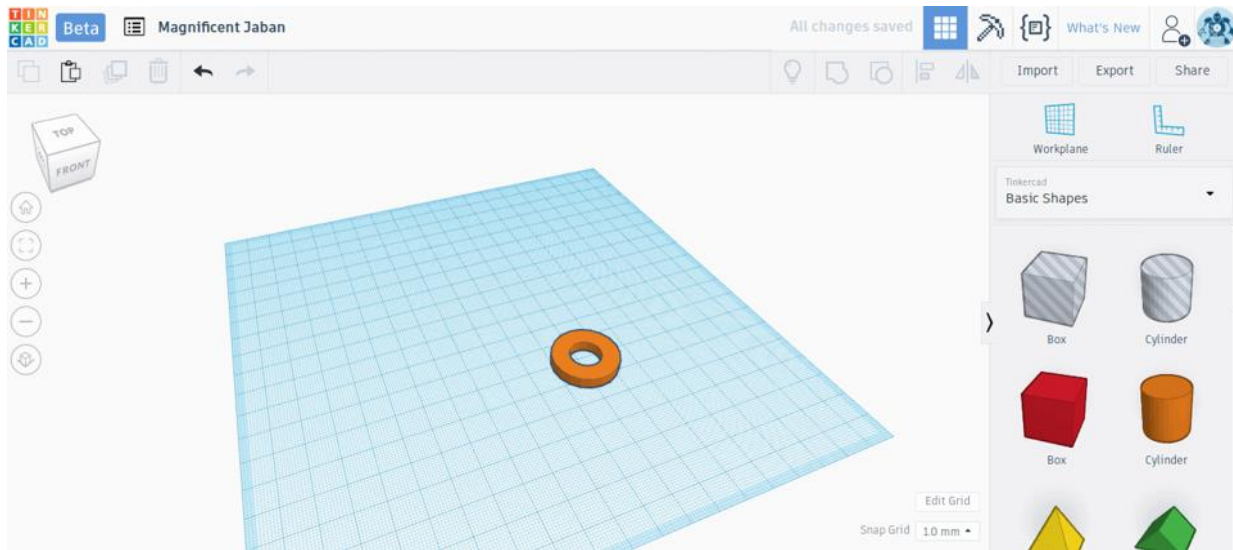
STEP 2.5: GROUPING THE CYLINDER SHAPED HOLE AND THE DISK TO CREATE THE RING

When the two shapes are aligned, press the “Group” button in the top menu (marked in the blue rectangle). The Group button essentially joins two shapes together. However, if one of the shapes is a hole, then it creates a hollow area instead.



The Group button merges two Tinkercad shapes.

Voila! We now have a beautiful ring and have finished the first step in creating our wrench.



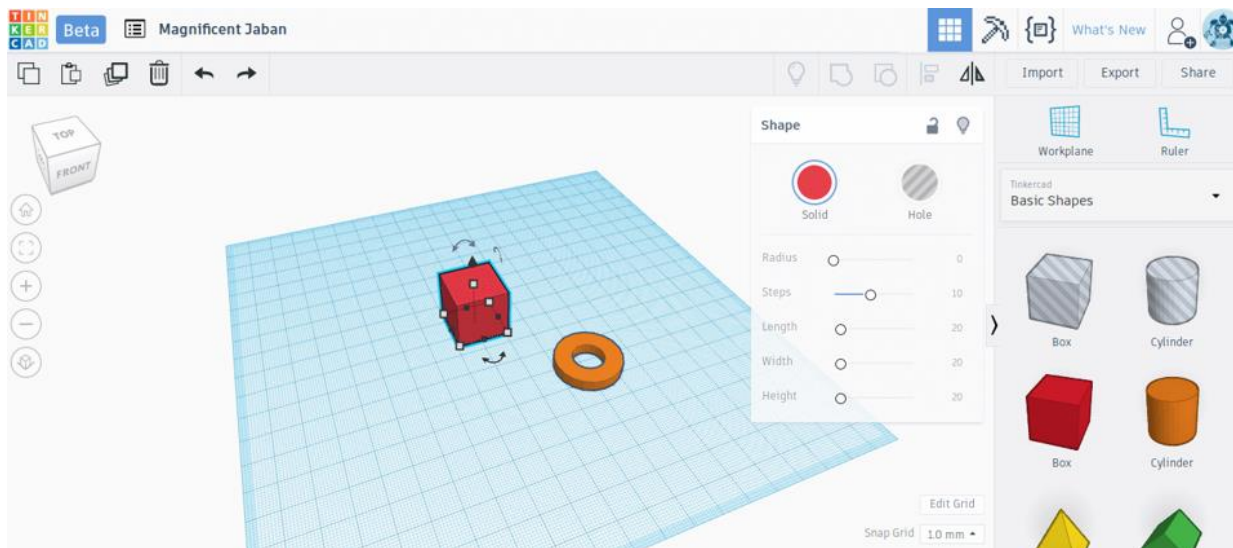
A beautiful ring on the workplane.

We learned quite a bit in this section. We learned how to change the dimensions of basic shapes, how to create holes, how to align shapes and finally how to group (or merge) two shapes together. The good thing is, these skills constitute 90 % of what you need to know to design 3D objects in Tinkercad. The next steps will involve the same skillset or small variations thereof.

3. Design the Handle

The handle is probably the easiest part of the wrench.

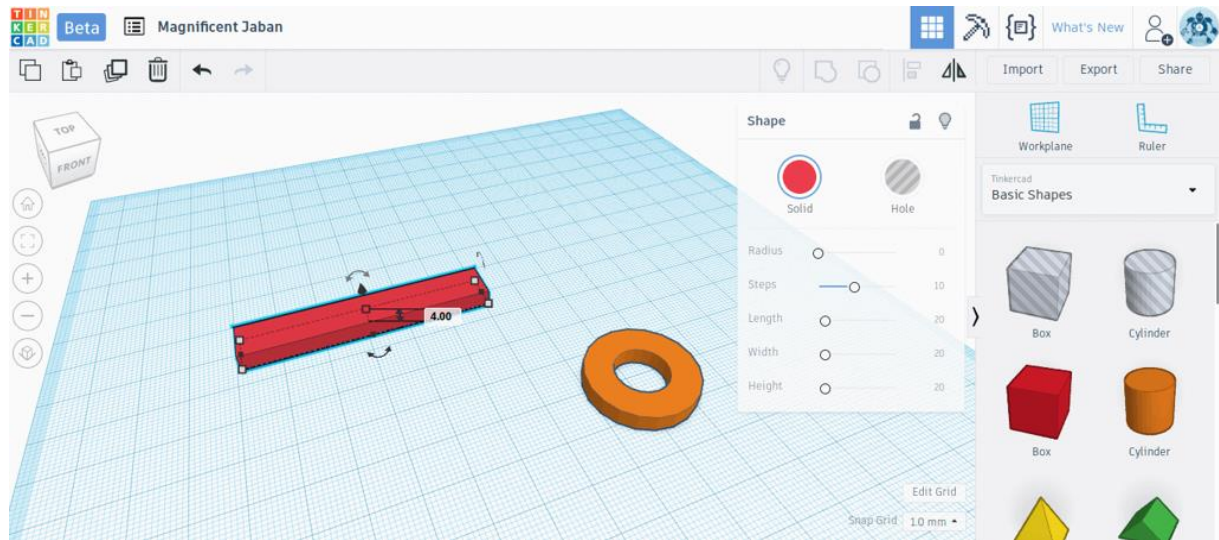
The handle is rectangular. So we simply drag a box from the basic shapes panel into the workplane.



We will use a box shape to create the handle of the wrench.

Using the white handles on the plane, we set the length of the box to 100 mm, the width of the box to 10 mm.

By dragging the white handle on the top of the box, we set the height to 4 mm so that it matches the height of the solid ring.

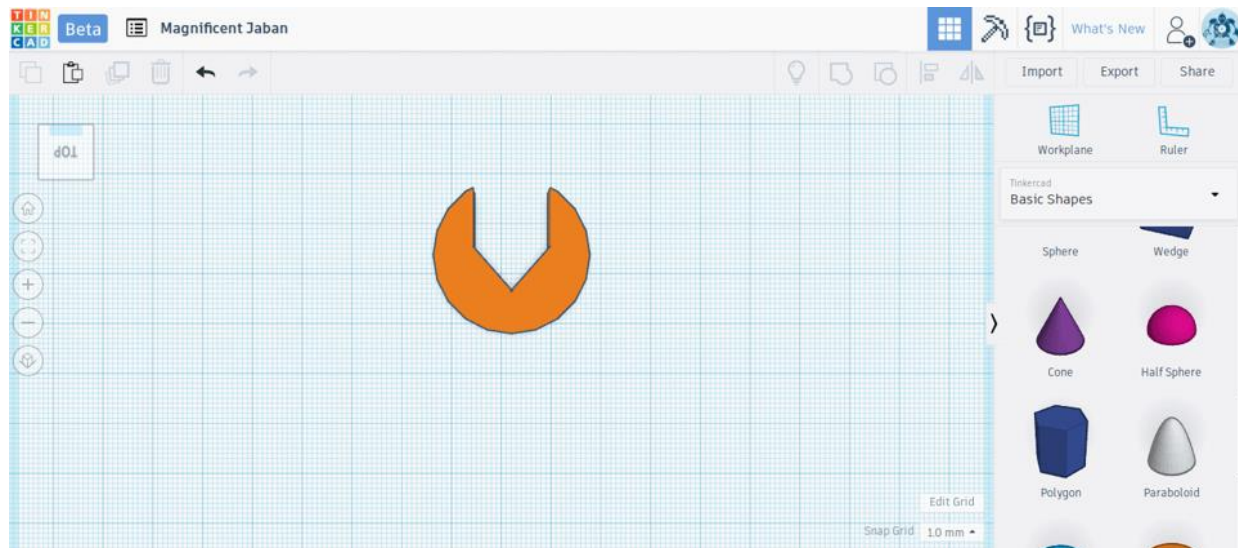


Consider the task “handled.”

That’s it. The handle is ready to go.

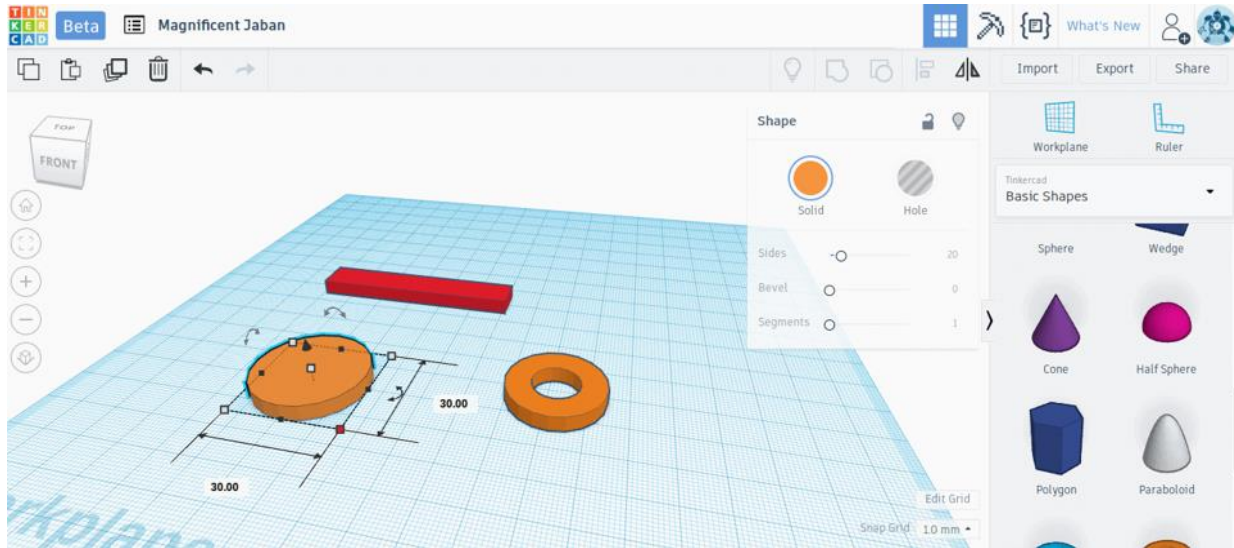
4. Design the Jaw

The design of the jaw is very similar to the ring that we designed in Step 2 of this Tinkercad tutorial. Basically, the jaw is also a circular disk, but with a polygonal hole carved out at an angle.



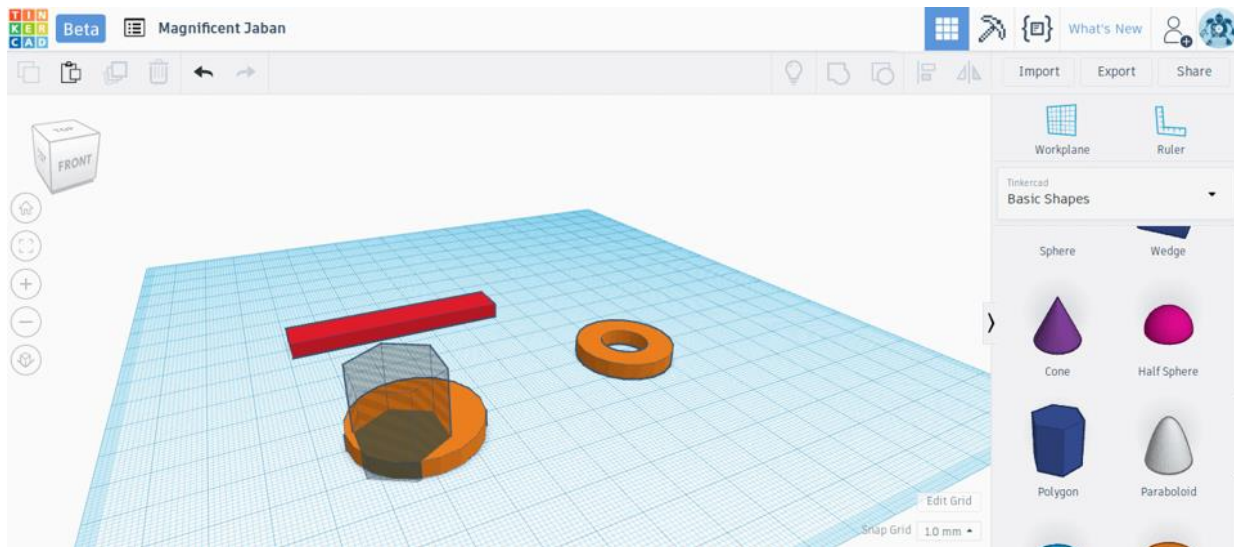
Top view of the jaw that we are going to design

As with the ring, we first need to create a circular disk of diameter 30 mm and height 4 mm. We have done this before, so you should be able to do this step without much problem.



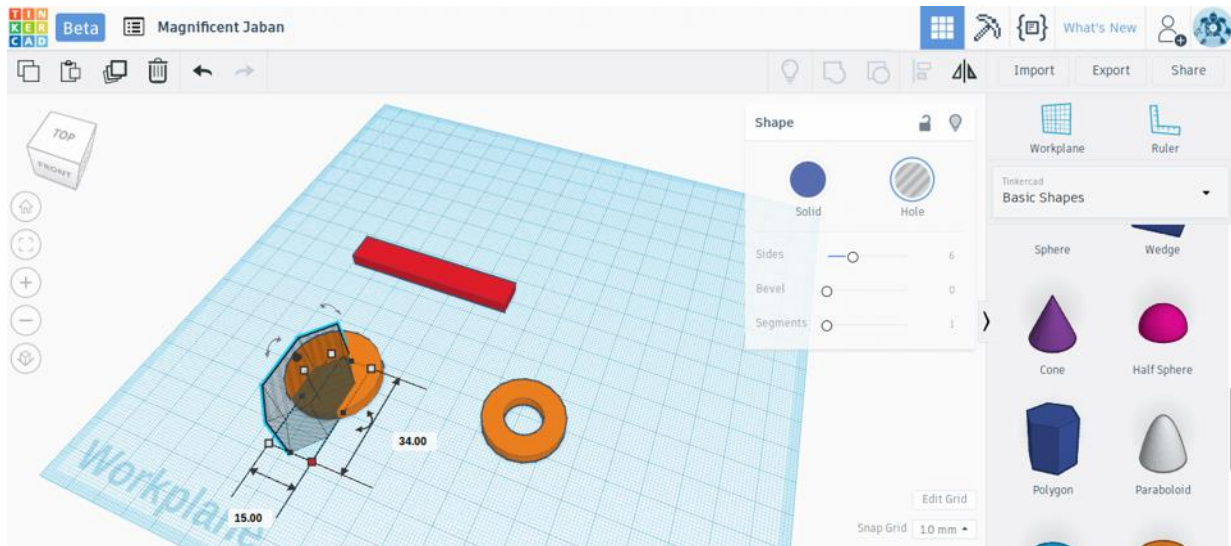
To create the jaw, we first create a disk with diameter 30 mm.

To create the jaw-like shape, we will use the basic shape called “Polygon”. Drag the Polygon to the workplane, select it, and turn it into a hole (as we did in Step 2). Now position the Polygon and adjust its dimensions.



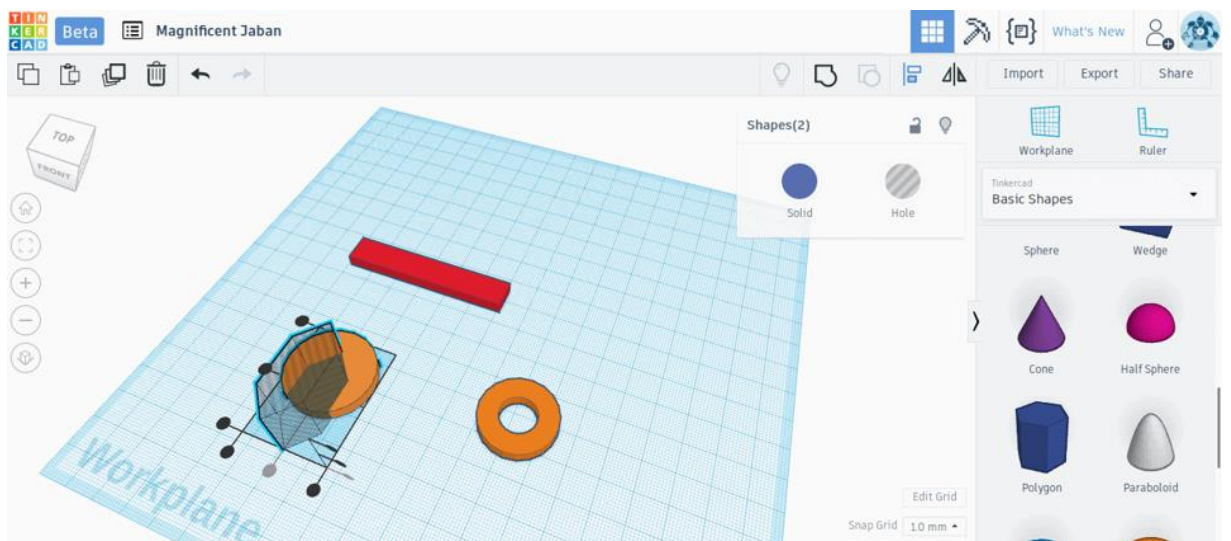
A polygon-shaped hole is used to create the jaw-like shape.

If you are planning to use the wrench from this Tinkercad tutorial for household repairs, you can choose the dimension of the corresponding nut and bolt. We will use a 15 mm jaw and adjust the width of the polygon to 15 mm accordingly.



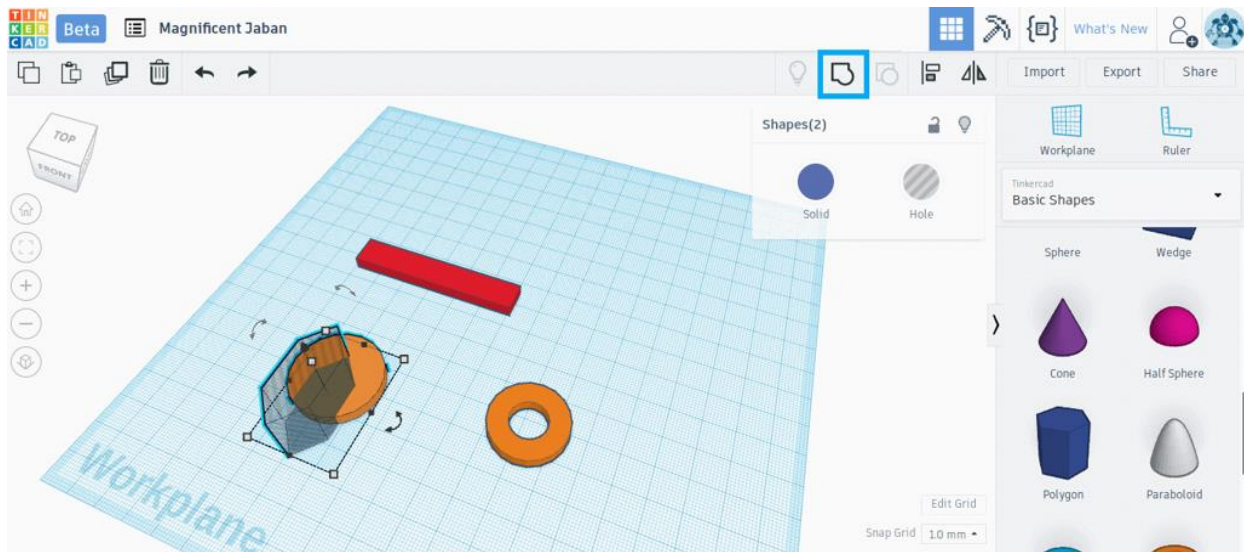
We set the width of the polygon to 15 mm.

Make sure the polygon is aligned with the center of the disk by using the alignment tool.



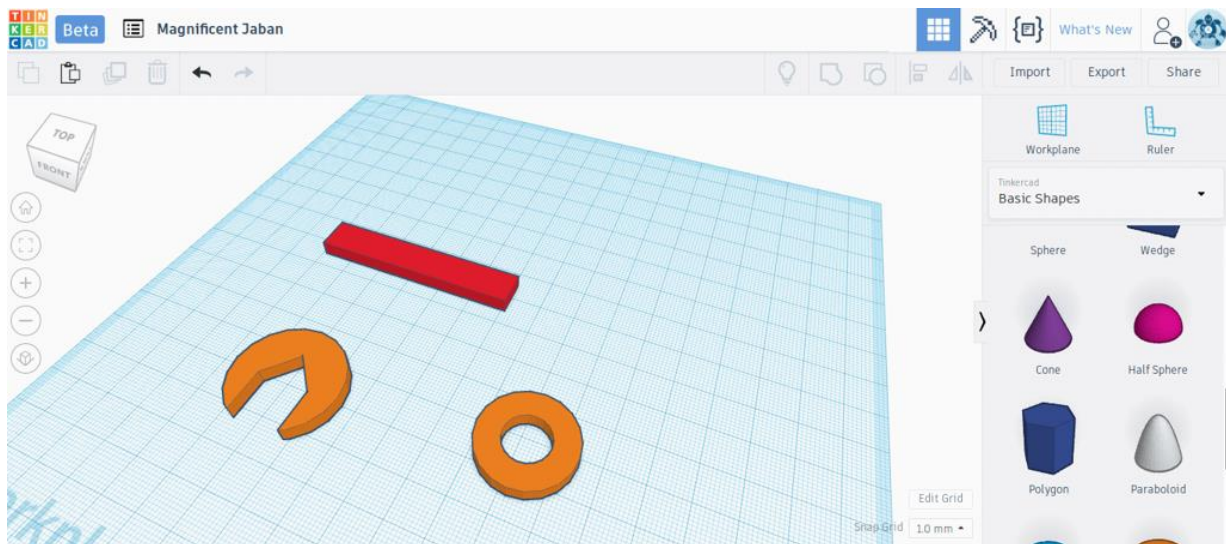
Align the polygon and the disk.

It's time to do some subtraction magic. Select both the polygonal hole and the disk by holding shift and selecting both. When they are selected simultaneously, press the Group button on the top menu (marked in the blue rectangle).



That's not so tricky, is it?

Congratulations! You have successfully created the jaw!



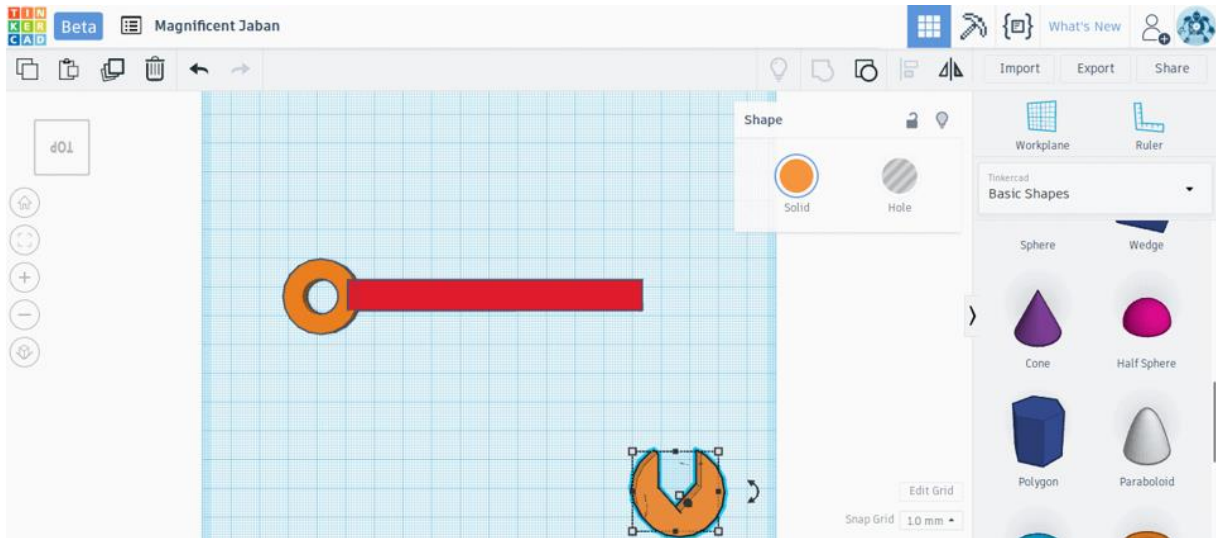
Almost done!

5. Putting it All Together

In this Tinkercad tutorial, we have so far designed the individual parts of the wrench. Now it is time to put it all together. Exciting, isn't it?

STEP 5.1: MERGING THE HANDLE AND THE RING

First, position the handle so that it goes a little bit into the body of the ring.

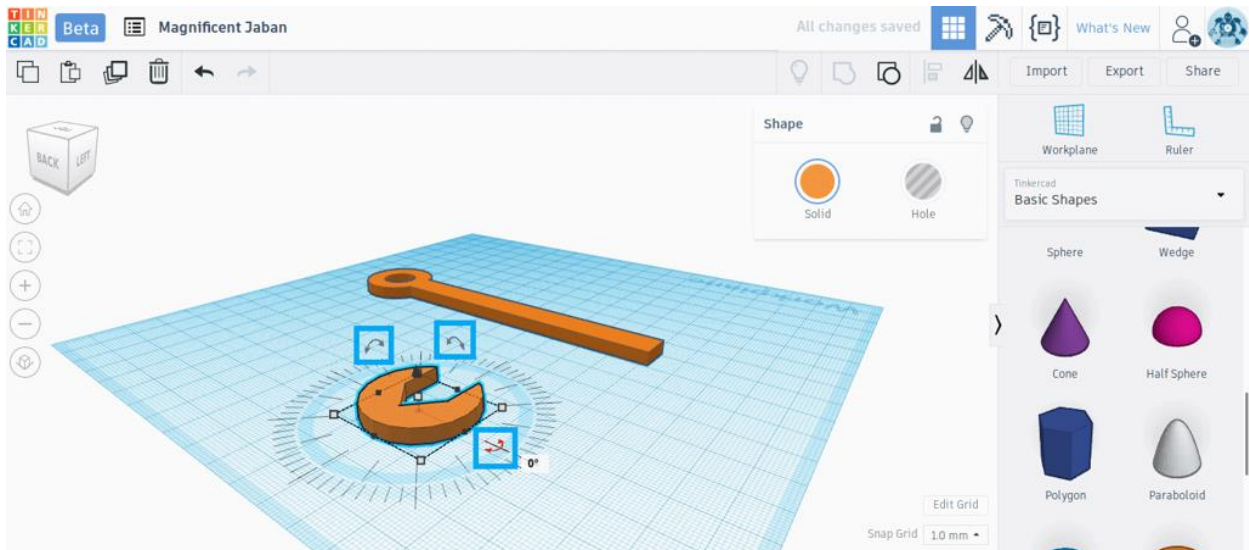


The handle should go just a little bit inside the ring.

Select the handle and the ring together and align it so that the handle aligns with the center of the ring. Now press the “Group” button. This should merge the handle and the ring into one continuous unit. Notice that the merged unit now has a single color instead of two colors.

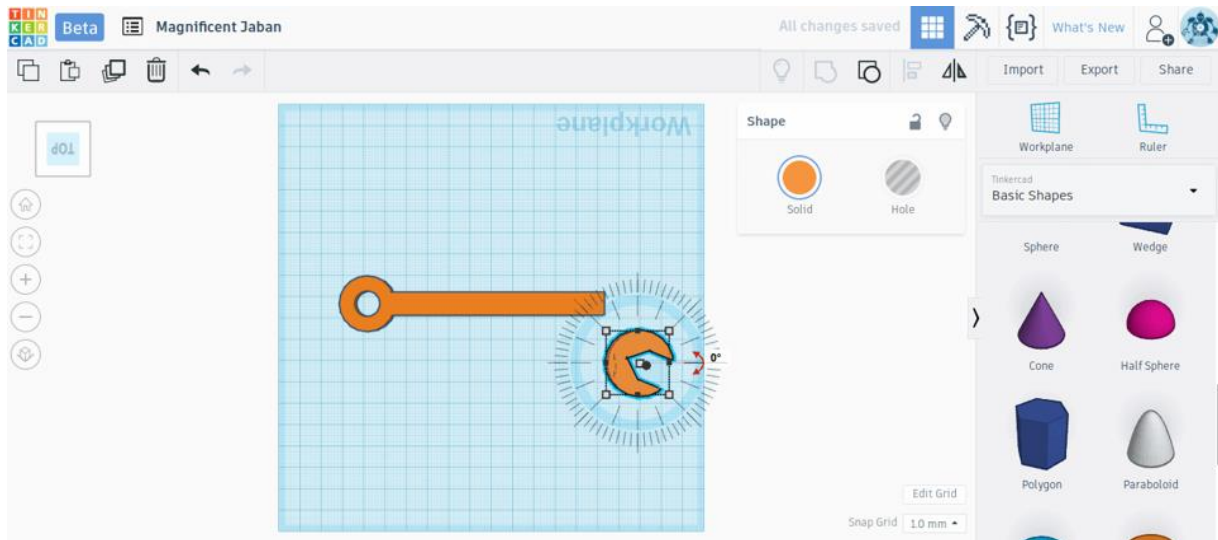
STEP 5.2: ROTATING THE JAW

The jaw is facing the wrong direction. We need to fix that by rotating it. To rotate the jaw, first select it. You will see that in addition to the white square handles from before, there are three curved handles with double arrows (marked in blue rectangles).



The rotation station.

These are the rotation handles. Use the X-Y plane rotation handle to rotate the jaw so that it faces the direction shown in the figure below.



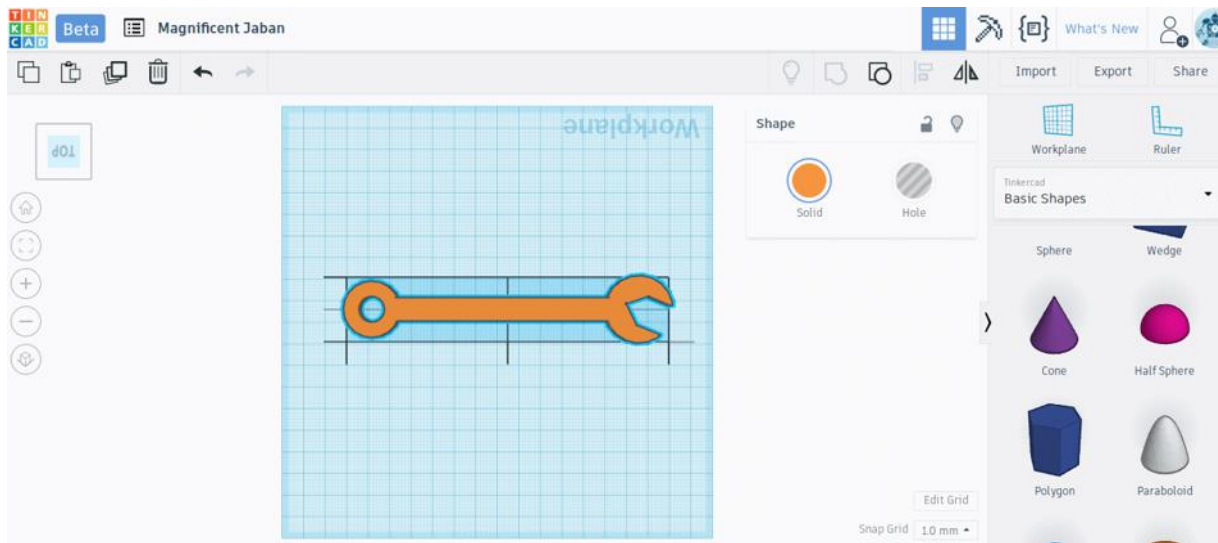
The jaw should make a small angle to the handle.

STEP 5.3: MERGING EVERYTHING TOGETHER

Now let's add the jaw to the previously merged unit. The procedure is more or less the same as what we used for adding the handle and the ring.

First, drag the jaw and position it so that the handle goes a little bit inside the body of the jaw. Select the jaw and the merged unit simultaneously and align them so that the handle aligns with the center of the jaw. Press the "Group" button to merge them.

A complete wrench on the workplane.



6. Add Your Name

Use the text tool on the right-hand tool window to write your name.

You will need to use the skills you've learned to size the text appropriately and then place it on top of the wrench.

You will need to set the height of your text in the shape window. You can set the height to 4.5.

7. Save Your Work

To save your project so that it can be printed, click on the Export button at the top right hand side of the screen.

Choose .OBJ

You can email this file to Mr. Neudorf along with your address. Your wrench should be emailed to you shortly!

You have just designed a beautiful gray 3D printable wrench. That's no small feat. Also, you now also know enough about Tinkercad to design any other 3D printable model that your heart desires.