

OpenSCAD

Workshop 1: Making a mast holder

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Step 1: Download and install OpenSCAD.

Windows and Mac:

Download the installer from:

<http://www.openscad.org/downloads.html>

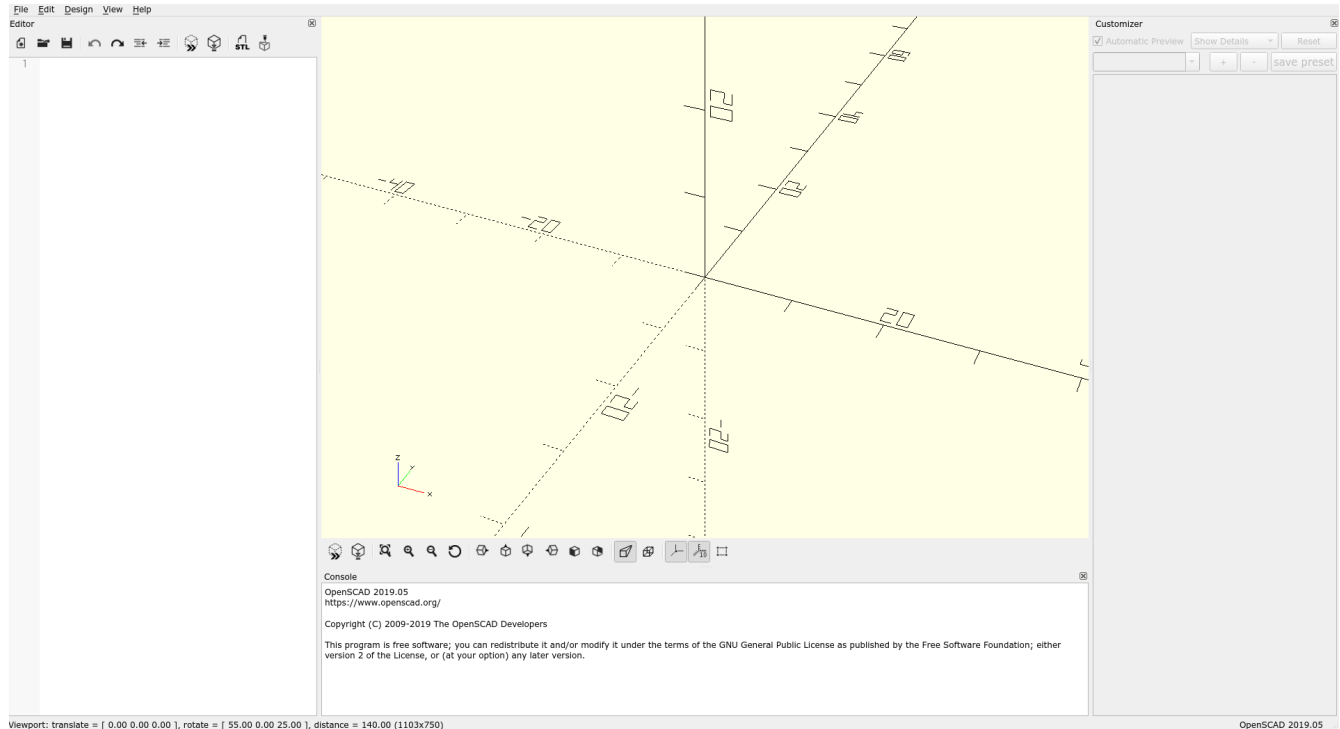
Linux:

There is probably a version in your distribution's repositories, if there is one, it's probably best to get it from there. On the page <http://www.openscad.org/downloads.html>, there are links to Snaps, Flatpaks, and the AppImage. If your distribution doesn't supply its own package, the AppImage is probably the easiest way to use the application unless you like

flooding your computer with random file mounts.

Step 2: Open OpenSCAD and click new.

It should take you to an empty project that looks something like this ↓



On the left is the editor. This is where you'll be writing your code. The panel in the top middle is the viewport. It will give a visual representation of what you are doing. The bottom middle panel is the console. Error messages will be highlighted in yellow. The panel on the right is the customizer. You can go ahead and close it using the x in its top right.

Take a moment to familiarize yourself with the viewport and its controls:

- Left-Click and drag is rotate.
- Right-Click and drag is pan.
- Scroll to zoom.
- You should also try out the buttons under the viewport.

Step 3: Make a cuboid

In OpenScad, the main way you make models is that you create shapes and then modify them. So in general the syntax looks like this:

```
...  
modification2()  
modification1()  
shape();  
...
```

The modifications are applied in reverse order, so in this example, “modification1” is applied before “modification2”.

Don’t worry if seems a little abstract for the moment, it will become clearer once you start using it.

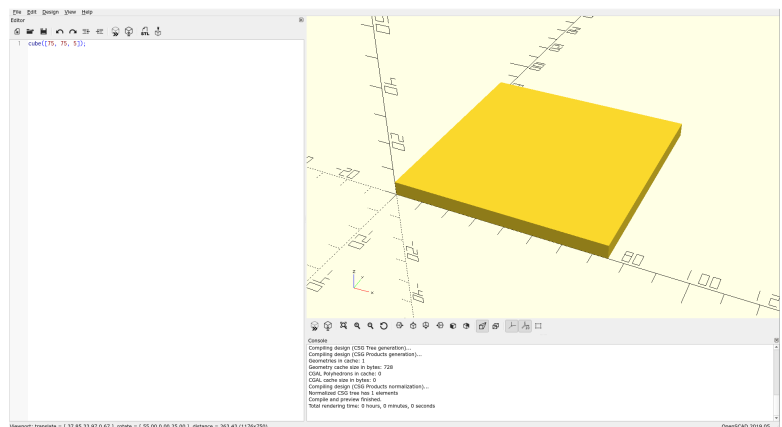
Now back onto the cuboid; To make a cuboid type this into the editor:

```
cube([75, 75, 5]);
```

The numbers in the array correspond to the x side-length, the y side-length, and z side-length (in that order) of the cuboid. The sizes are in millimeters (and could have been non-integers). Most things in OpenSCAD are measured in millimeters or degrees.

Next, you need to tell OpenSCAD to preview your code. Press F5 or the preview button, which is the box with two arrow-heads in-front of it (there are two preview buttons, one above the editor and one below the viewport).

You should end up with something that looks like this →



Step 4: Use variables.

You sometimes want to re-use a value multiple times. You could just type the value every time you need it. But that can quickly get messy if you want to change the value later. You will end up having to change every single instance of that value, and that's time-consuming and error-prone, so instead, we define variables.

Variables are defined like this:

```
my_variable_name = 34.5;
```

After you have done this you can now use the name you gave your variable instead of the value.

As we want to use the values for the size of the cuboid again later, we want to make variables out of them.

Make variables representing the values, and then replace the values with their variable names. You should end up with code that looks something like this:

```
base_width = 75;  
base_thickness = 5;  
cube([base_width, base_width, base_thickness]);
```

When you press preview **nothing should change**.

Step 5: Make the outside of the mast shaft.

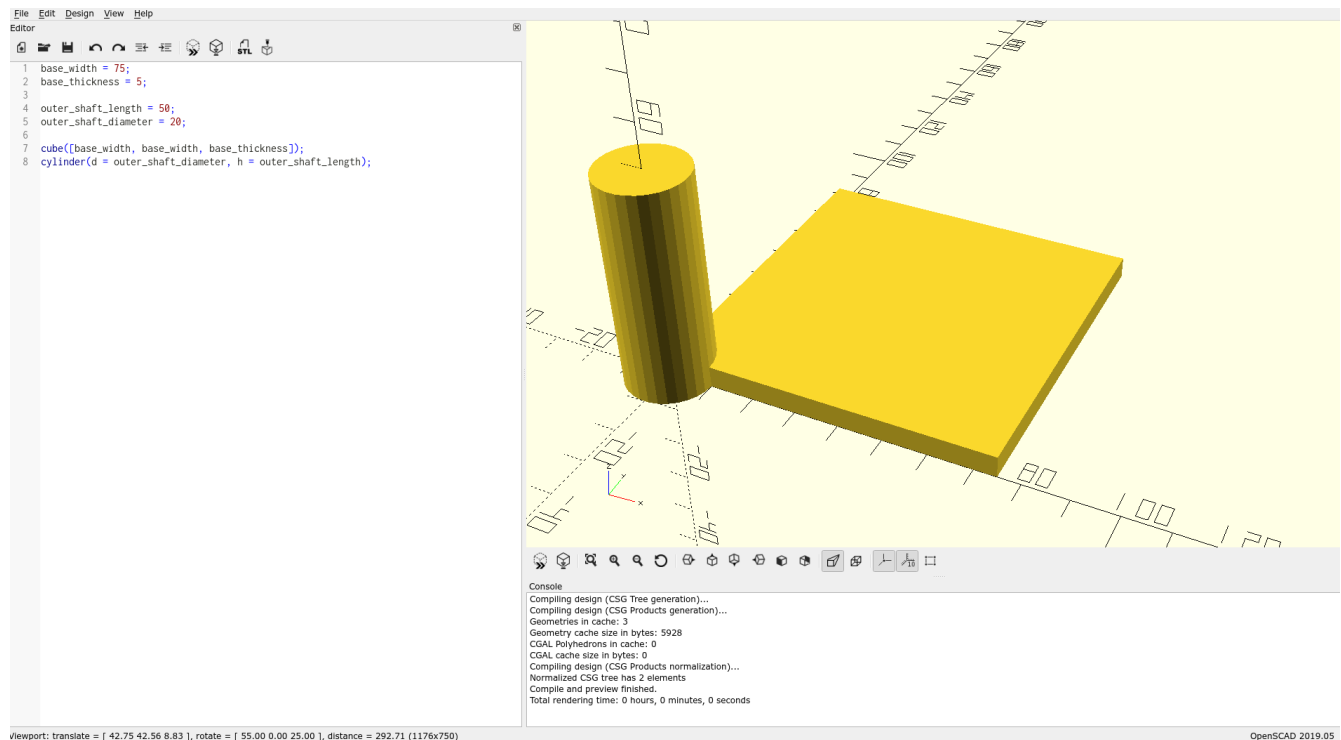
We are going to make the mast shaft out of a cylinder. We are going to need the values defining the cylinder multiple times so we need to turn them into variables.

```
outer_shaft_length = 50;  
outer_shaft_diameter = 20;
```

To create the cylinder itself the syntax is:

```
cylinder(d = outer_shaft_diameter, h = outer_shaft_length);
```

Once you have added these lines to the project, press the preview button and you should end up with something that looks like this ↓



Step 6: Move the cylinder into place.

We are now going to use one of those modifiers I talked about earlier. We are going to use the translate modifier.

The syntax for translate looks like this:

```
translate([x, y, z])
```

Where x is the amount to move the object in the x-direction, and y is the amount to move the object in the y-direction, and z is the amount to move the object in the z-direction.

If you remember from before the syntax for applying modifiers looks like this:

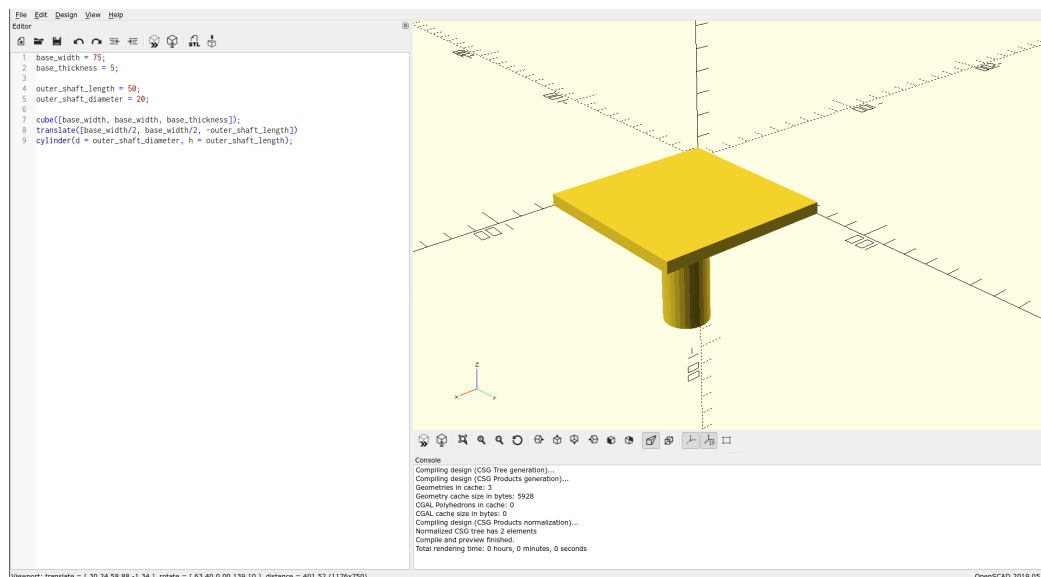
```
modification()  
shape();
```

So we are going to apply this pattern to the cylinder:

```
translate([base_width/2, base_width/2, -outer_shaft_length])  
cylinder(d = outer_shaft_diameter, h = outer_shaft_length);
```

Notice that the line with translate on it doesn't have ";" at the end.

After you press preview you should end up with something that looks like this ↓



Step 7: Make the cylinder smoother.

You have probably noticed that the cylinder is not very smooth. This is in-fact one of OpenSCAD's main limitations, it does not really deal with curves. For the most part, this can be avoided by increasing the number of line segments OpenSCAD will use. We can do this by changing one of the system variables.

Add this line to the beginning of the file:

```
$fn = 100;
```

Press preview again and the cylinder should become much smoother.

This system variable simply changes the **number of fractions** (lines) that circles are made up of. There are other system variables that instruct OpenSCAD to decide the fractional number for a given circle on the fly based on factors such as size of the circle. We won't be using these more advanced settings right now, but they can come in very handy when our models grow in size and complexity.

The performance effects can be best seen when you render the project. You can render the project by pressing F6 or the button with the cube with an hour-glass in-front (next to the preview buttons).

Try changing the value and then rendering to see the results and how much time it takes. "\$fn = 1000;" takes a few seconds for my laptop to render. Don't set the number too high (>5000ish) because this can cause OpenSCAD to hang.

Generally when editing, you want to use a relatively low number, something like "\$fn = 32;" as you'll want to be able to quickly view the changes you've made. But don't forget to change this to a higher number before you export the file (we'll come on to that later) or your final, physical model will have visible steps to all its curves!

Step 8: Cut a hole in the shaft.

When you want to cut a shape out of another shape you use a difference block. The syntax for this looks like this:

```
difference() {  
    shape();  
    shape2();  
    shape3();  
    ...  
}
```

When OpenSCAD encounters this, it will create “shape”, and then it will see where “shape” intersects with any of the shapes that come after it in the block. Anywhere they do intersect will be removed from “shape”.

But there is a problem for us. Our shape so far is made from two shapes. For it to be the first shape in a difference block it needs to be one shape. Otherwise, we would cut one out of the other. The way we make them one shape is by using a union block. The syntax for this is:

```
union() {  
    shape();  
    shape2();  
    shape3();  
    ...  
}
```

This tells OpenSCAD we want all of the shapes inside of the union to be treated as one shape.

Wrap the cuboid and cylinder in a union block and then difference block. Your code should end up looking something like this:

```
difference() {  
  union() {  
    cube([base_width, base_width, base_thickness]);  
    translate([base_width/2, base_width/2, -outer_shaft_length])  
    cylinder(d = outer_shaft_diameter, h = outer_shaft_length);  
  }  
}
```

When you press preview **nothing should change**.

Now we are going to cut a cylinder out of the shaft. We are going to be using the same values several times, so we need to make two more variables defining the shape of the cylinder we want to cut out:

```
inner_shaft_diameter = 15;  
inner_shaft_length = outer_shaft_length - 5;
```

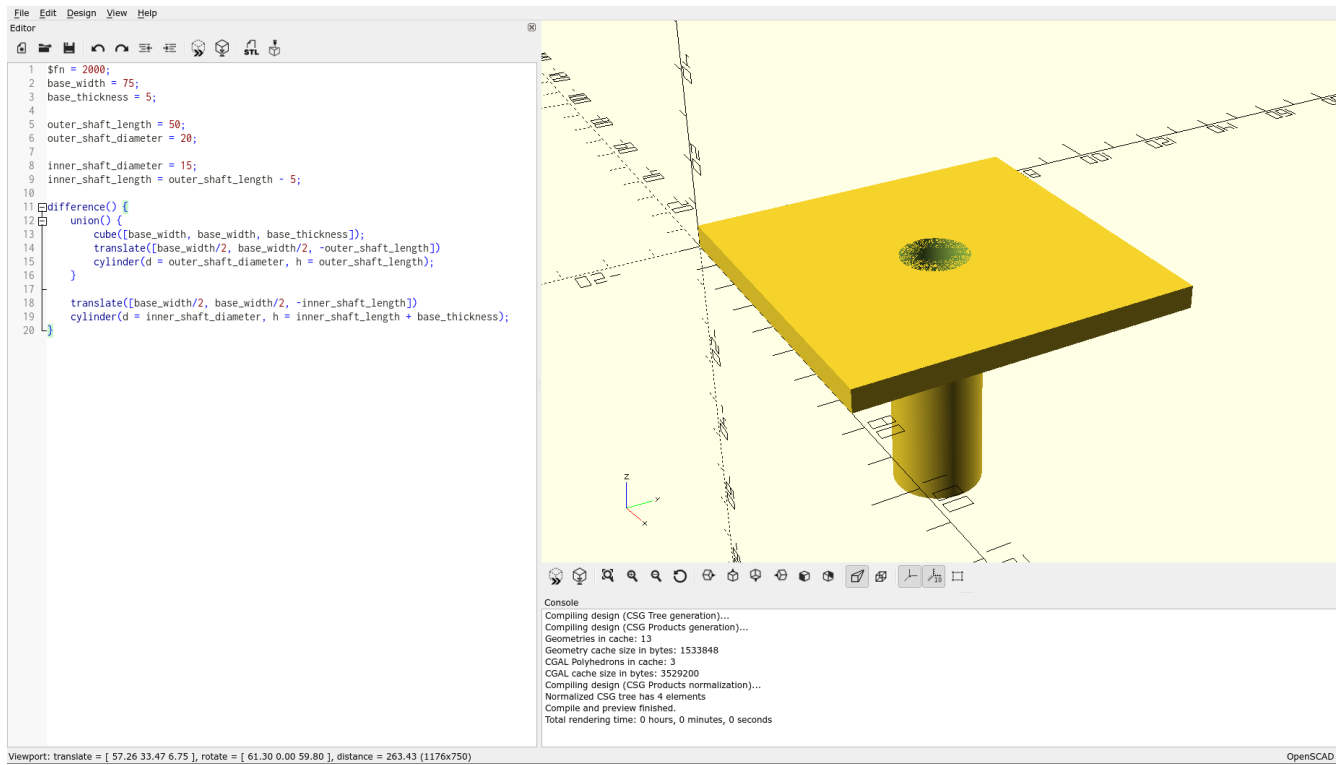
Notice how “inner_shaft_length” is defined in terms of “outer_shaft_length”.

To cut the cylinder out of the shaft we need to create it after the union but still inside of the difference. We are going to create the cylinder in the same way we did before. Firstly creating the correct size cylinder and then moving it to the correct location: The code should look something like this:

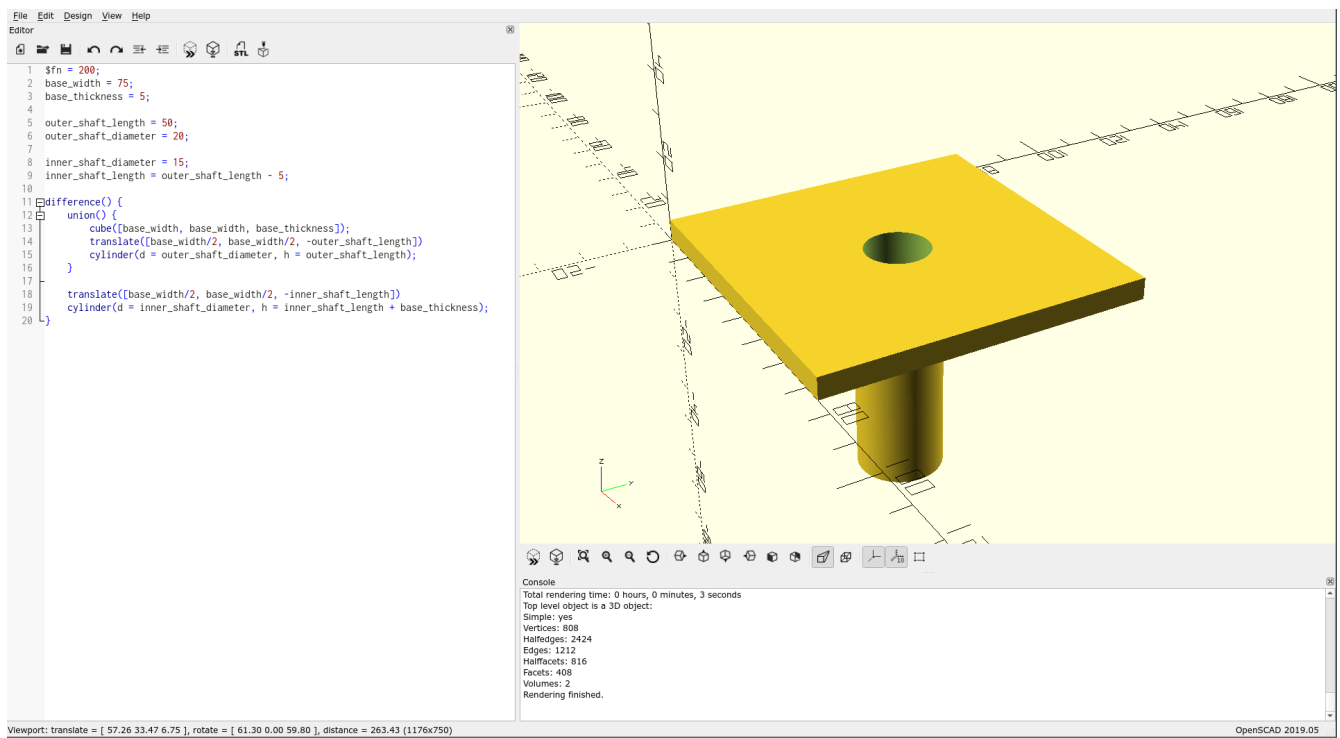
```
difference() {  
  union() {  
    ...  
  }  
  translate([base_width/2, base_width/2, -inner_shaft_length])  
  cylinder(d = inner_shaft_diameter, h = inner_shaft_length +  
    base_thickness);  
}
```

Press preview. If this has gone correctly you *may* see some visual artifacts on the top of the model. When you move the viewport around they may shimmer. This is normal. When you use difference and a face of the base shape and a face of the cutouts are exactly on top of each other this happens. You can get rid of this by rendering instead.

What it should look like after pressing preview ↓



What it should look like after pressing render ↓



Step 9: Add mounting holes

OpenSCAD has for loops that allow you to generate do the same thing multiple times but with changing values.

The syntax for a for loop looks like this:

```
for (i = [start:step:end]) {  
    ...  
}
```

What this will do is that it will first run the code in between the brackets with “i” set to the value of “start”. Then it will add “step” to “i”. And run the code between the brackets again. It will keep adding “step” to “i” and running the code between the brackets until “i” is greater than “end”.

While we are only going to use this to make four holes, and so this is a little overkill, this method is very useful for making lots of the same object so it's worth seeing.

First, we need to use some values several times so we are going to need to make some variables.

```
mounting_hole_radius = 2.5;  
distance_from_edge = 10;
```

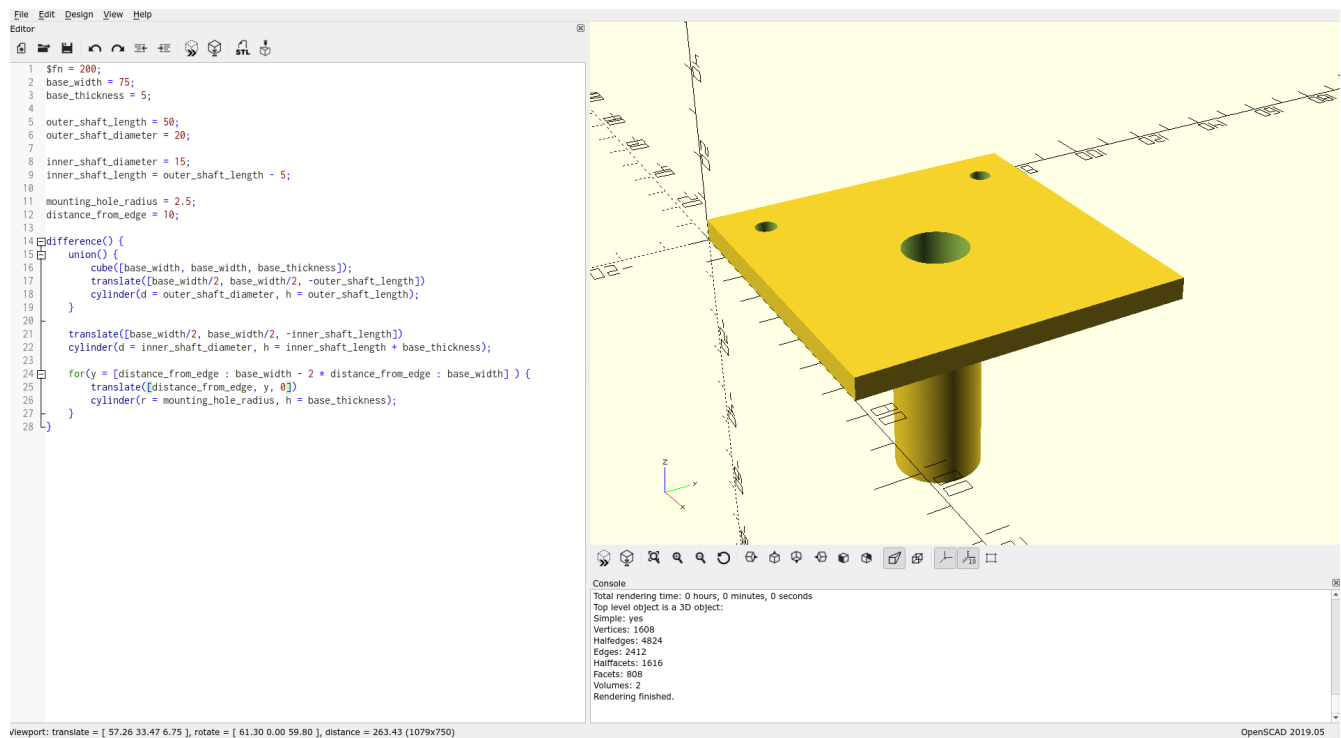
Right now, we are going to only make two of the holes, so it's a little easier to see what is happening.

Inside of the difference but after the union, we are going to make a for loop that creates cylinders of the correct size, then moves it to the correct place. The code should look like this:

```
difference() {
    union() {
        ...
    }
    ...
    for(y = [distance_from_edge : base_width - 2 * distance_from_edge :
base_width] ) {
        translate([distance_from_edge, y, 0])
        cylinder(r = mounting_hole_radius, h = base_thickness);
    }
}
```

This might look a little bit complicated, it's not helped by the fact that the for is going over two lines. You can expand your editor by dragging on the edge between it and the viewport to change its size.

Once you have pressed render you it should look something like this ↓

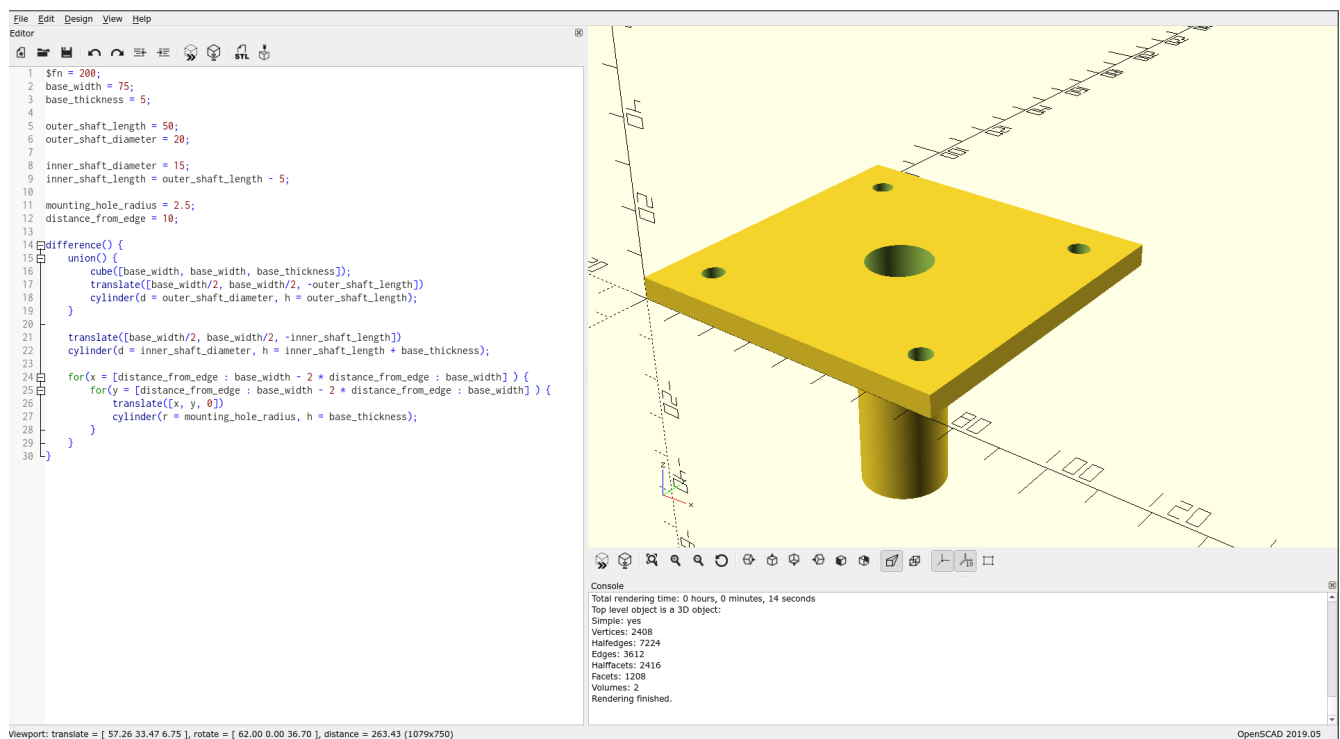


Now we're going to surround this is another for loop that is going to change the values in the x-direction in the same way:

```
difference() {
  union() {
    ...
  }
  ...
  for(x = [distance_from_edge : base_width - 2 * distance_from_edge :
base_width] ) {
    for(y = [distance_from_edge : base_width - 2 * distance_from_edge :
base_width] ) {
      translate([x, y, 0])
      cylinder(r = mounting_hole_radius, h = base_thickness);
    }
  }
}
```

Be careful not to miss that the “distance_from_edge” in the translate changes to an “x”.

Once you have rendered the project it should look like this ↓



Step 10: Export

To export your model to a format that other programs can use, you need to export it. First you need to render the project, then press F7 or the button in the GUI with a piece of paper that is folded over and has "STL" written in front of it (above the editor). You can then save the file where you like.

The STL will then be read by the 3D printing software to print your final model. We'll be covering this in a later workshop.