# 2D

# My First Design with OpenSCAD

Let's look at a simple example using OpenSCAD. A rectangular plate with screw holes in the corners:

File:2d-design-with-openscad.png

So how do we produce that?

# Step 1. Install OpenSCAD.

Visit http://www.openscad.org/downloads.html and follow the instructions to download and install.

## Step 2. Designing.

Once OpenSCAD is installed and you've started it up, you'll be presented with a set of blank panes. Let's whack the following into the left hand pane:

```
difference() {
    // Plate size overall
    square([75,100]);
    // Screw holes
    translate([6,6,0]) { circle(2, $fn=50); }
    translate([6,94,0]) { circle(2, $fn=50); }
    translate([69,6,0]) { circle(2, $fn=50); }
    translate([69,94,0]) { circle(2, $fn=50); }
}
```

What does all this mean?

#### Difference

http://en.wikibooks.org/wiki/OpenSCAD\_User\_Manual/CSG\_Modelling#difference

difference() {

Subtract the intersections from each other.

## Square

http://en.wikibooks.org/wiki/OpenSCAD\_User\_Manual/Using\_the\_2D\_Subsystem#square

square([75,100]);

Draw a square that's X=75mm and Y=100mm.

#### Translate

http://en.wikibooks.org/wiki/OpenSCAD\_User\_Manual/Transformations#translate

translate([6,94,0]) { circle(2, \$fn=50); }

The object in the perenthesis will have it's origin at X,Y,Z. As we're drawing in 2D we don't care about Z so this will always be zero for this case. The origin for our circle is X=6mm and Y=94mm.

#### Circle

http://en.wikibooks.org/wiki/OpenSCAD\_User\_Manual/Using\_the\_2D\_Subsystem#circle

circle(2, \$fn=50);

The first parameter is the radius, not the diameter of the circle. The second parameter provides a higher resolution when drawing small circles.

### Step 3. Tweak and update

Make changes to your design, and then hit F5 to refresh the rendered view of your design.

## Step 4. Save the damn design!

Once you've got a design you're happy with, save it to disk before moving on.

#### Step 5. Compile, render and export.

Now it's time to compile that so that we can export the DXF. You can do this by:

- Hitting F6
- Using the menu Design → Compile and Render (GCAL)

Now we can select Design  $\rightarrow$  Export as DXF. Select a sensible file name in a location you'll remember.

That's our OpenSCAD work done. At this point, we can make changes using Inkscape or go directly to sending it to the cutter.

# 3D

# Nut Traps / Hexagons

cylinder(r=5.5 / 2 / cos(180 / 6) + 0.05, fn=6; Makes a perfectly snug M3 nut trap, loose enough that the nut can be placed by hand, but then snug enough that I can bang the object on my table and the nut stays in place. 5.5mm edge-to-edge is the size of a m3 nut.

Category:Guides Category:OpenSCAD

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3/3