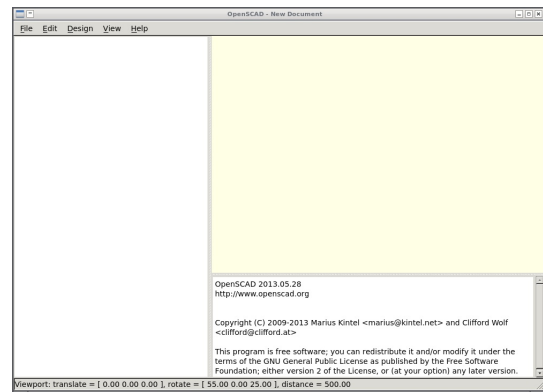
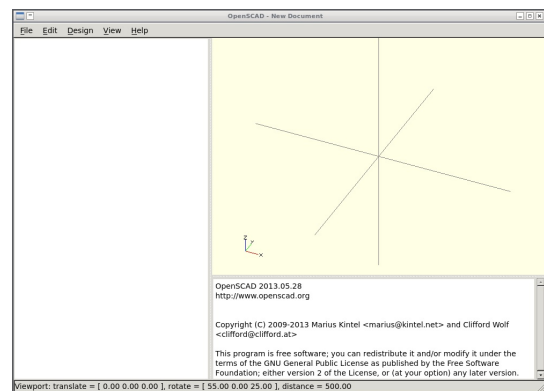


1. Open OpenSCAD - This is a design program which does not require drawing skills, it will require a bit of thinking in 3D though!

2. The OpenSCAD window has three areas:
Left, the code, a set of descriptions of what we want it to draw;
Top right, a 3D view of the model;
Bottom right, the status of the model creation - look here for errors!



3) Setup: To follow along with these instructions, visit the "View" menu and make sure the "Show Axes" option is ticked. To make sure the model picture always shows your current code, look in the "Design" menu and make sure the "Automatic Reload and Compile" option is ticked.

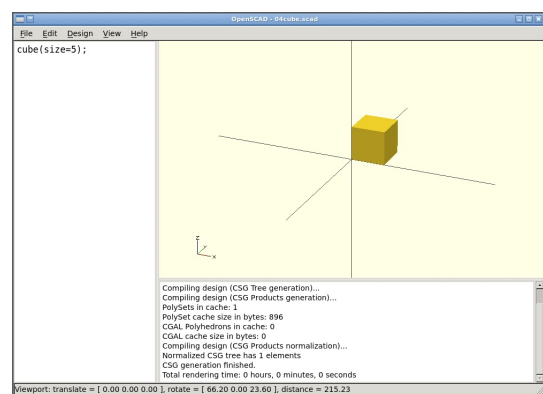


4) Get started! Lets draw something - to make a cube we type "cube" followed by parentheses to indicate parameters. Inside the parentheses we tell it the size (length of the edges), like this:

```
cube(size = 5);
```

The end of a statement has a semi-colon ";" to show where it finishes.

To show what this does, use CTRL+s (or File, Save) and save the file under your name, make sure it has a ".scad" file ending.



If OpenSCAD only displays a square, use your left mouse button to drag the image a bit the left so that you see the cube.