Lab #3 – 3D Printing Designs

In this lab you will learn the basic steps for designing three-dimensional physical objects for 3D printing. This will focus on using the free program OpenSCAD.

"OpenSCAD User Manual"
`https://en.wikibooks.org/wiki/OpenSCAD_User_Manual/The_OpenSCAD_Language`

"Downloading OpenSCAD at home (not required)"
`http://www.openscad.org/downloads.html`

1) Run OpenSCAD. Identify the editor on the screen where you will be able to type in instructions, the viewing window where your designs will render, and the console where your compile errors will appear.

2) For your first design, enter "**cube([20,15,10]);**", then click on "Design" and "Preview". This will display a cube which is 20 mm in x, 15 mm in y, and 10 mm in z. Notice that one corner of the cube is at the "origin", corresponding to coordinate (0,0,0).

3) On the second line, create a sphere of radius 20 with "**sphere(r=20);**". Use preview as needed to display the result.

4) Your sphere is a bit blocky! Next, use an OpenSCAD trick to add a higher number of faces to your rendered sphere. Edit your sphere line to read "**sphere(r=20, $fn=80);**" and preview again.

5) It's hard to see your cube now, so go back and edit your cube so that it sticks out a total of 30 mm from the x axis, then preview again.

6) Use the left mouse button in the display window to rotate your design, and the right mouse button in the display window to translate, or slide, your design. Use the scrolling feature of the mouse, or the +/- magnifying glass buttons, to zoom in and out. Look around at all sides of it. When you are done, press the circular-arrow button to return to the original display perspective, then press the magnifying glass with the square around it to return to the original zoom.

7) Create a cylinder by typing "**cylinder(r=10, h=40);**". You can add in $fn=80 again to make it smoother.

8) The cylinder command can also be used to create a cone, or a section of a cone, by specifying two radii. Add a line "**cylinder(r1=22, r2=2, h=45);**". The first radius is the bottom of the cone, the second is for the top.

9) Notice that each thing you added so far is merged together as one object. In OpenSCAD this is called a "union" of the objects. Because the design is created dynamically each time you press preview, you can dynamically edit the object. Try "commenting out" your sphere by typing "//" at the far left side of the sphere line, then preview again.

10) Add a second cube, this time stretching farther in y, with "**cube([15, 50, 15]);**". Then we will rotate this cube, by adding on the line right BEFORE it, "**rotate([0,0,45])**". (Notice no semi-colon for this!) These are in units of degrees, and correspond to rotations about the x, y, and then z axis. The rotate command rotates its "child", which is the object immediately following it. In this case, the cube.

11) Edit your rotate command to also point upward at a 30° incline by adding a rotation around the x axis.

12) You can also move objects around by changing their position with the "translate" command. Shift your long cube 50 mm to the right by going before the rotate command, and entering "**translate([50,0,0])**". Again notice there is no semi-colon, because this operates on its "child", which in this case is a rotated cube. First the cube is drawn, then it is rotated, then it is translated, in the opposite order the lines appear.

13) You can also merge your objects into a single module with the module command. At the top of everything you typed so far, add a line "**module MyBlob() {**". At the bottom add the closing brace with "**}**".

14) Render again and everything will disappear. Don't panic! You now have a module, but have not rendered it. Below everything, add a line that says "**MyBlob();**". This will render one MyBlob module.

15) Now that your blob of stuff is a MyBlob, you can operate on the whole of it. Before your "MyBlob();" line enter "**difference() {**". After "MyBlob();" enter "**translate([0,-2,1]) cube([32,32,32]);**" Then add one more line with a closing brace, "**}**". Then preview again. This "difference" performs a subtraction. Your blob object now has a 32 mm cube subtracted out of it.

16) Save your .scad file in your user Documents folder.

17) When you are sure it has saved, go to File, New, and make a new design. This time, try to use what you learned to create a disk and put a smiley face on it. (With at least eyes and a mouth.)

18) Save your new .scad file to your user Documents folder, and upload both to Cabrini Learn.