

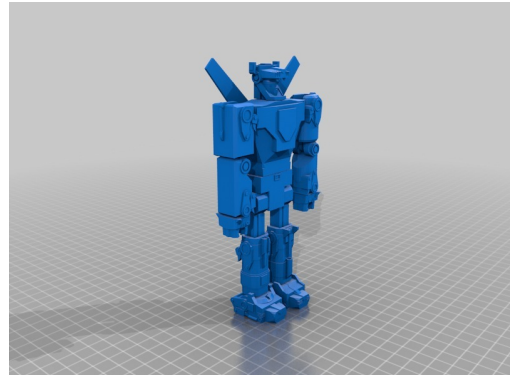
Computational Design and Fabrication Fall 2017

Programming Assignment 2: Parametric Design

Due October 13th at 11:59pm.

In this assignment you will design a parameterized object (or a set of objects). You will use OpenSCAD to create your objects. OpenSCAD is a software for creating solid 3D CAD objects. It is free software and available for GNU/Linux, MS Windows and Apple OS X.

OpenSCAD is not an interactive modeller. Instead it is something like a 3D-compiler that reads in a script file that describes the object and renders the 3D model from this script file. Your task in this assignment is to write a set of OpenSCAD scripts to create the objects. These objects will be printed in next week's lab.



In this assignment you will be responsible for using OpenSCAD to create a parameterized object with reasonably complexity. Your design should be a single OpenSCAD file with any number of modules. All the parts of your designs must be created by a *single high-level module*. The parameters of that module are the *exposed parameters* of your parametrized model. The requirements of your design are:

1. Use of CSG
2. At least two different parameters exposed.
3. Everything should be modular, e.g., computed using a DAG/tree (use auxiliary modules for subparts)
4. Sufficient complexity (e.g., not few spheres stitched together)
5. Present a few examples of the same design with different parameters
6. Explain the parameters of your high-level module and give the approximate range.

The remainder of this document is organized as follows:

1. Getting Started
2. Example designs and Implementation Notes
3. Extra Credit
4. Submission Instructions

1 Getting Started

1.1 Using the OpenSCAD Modeller

You will use OpenSCAD to create your 3D models. It is pretty straightforward to install OpenSCAD.

1. Download OpenSCAD installer from <http://www.openscad.org/downloads.html>
2. Create your designs with OpenSCAD language in the script panel
3. A list of OpenSCAD built-in functions: <http://www.openscad.org/cheatsheet/index.html>
4. To view your design from OpenSCAD, you need to *Compile* your code. It can be done by selecting "Compile" from the "Design" menu, or simply press F5.

1.2 Export for 3D printing

You will 3D print your designed model ultimately. In order to 3D print you will need to export your objects from OpenSCAD. A typical file format for 3D printing is the STL file format. OpenSCAD supports import and export of STL file format. To export your design, you need to:

1. *Compile and Render* your code. It can be done by selecting "Compile and Render" from the "Design" menu, or simply press F6.
2. Export to STL file. select "Export as STL..." from the "Design" menu, then enter a filename in the ensuing dialog box. Don't forget to add the ".stl" extension.

Notice that sometime you may be able to compile and render your design smoothly, but cannot output to the STL file. One of the typical reasons is that the object created is not a valid 2-manifold. If it happens, you may refer to the example in http://en.wikibooks.org/wiki/OpenSCAD_User_Manual/STL_Import_and_Export for further reading.

1.3 Online tutorials

There are some tutorials online:

1. OpenSCAD User Manual: http://en.wikibooks.org/wiki/OpenSCAD_User_Manual
2. An OpenSCAD Tutorial Series on the Makerbot blog: <http://www.makerbot.com/blog/2012/11/08/openscad-design-tips/>
3. OpenScad beginners tutorial: http://edutechwiki.unige.ch/en/OpenScad_beginners_tutorial

2 Example designs and Implementation Notes

There are some example designs built in OpenSCAD. You can see them by selecting "Examples" from the "File" menu. Below are some more examples on thingiverse that may be helpful for stimulating your design idea.

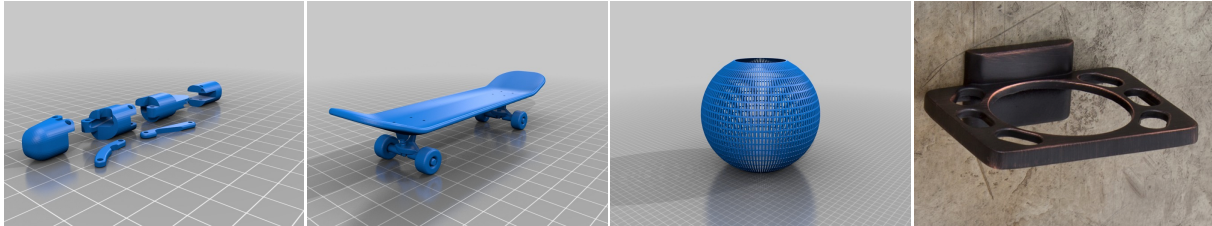


Figure 1: Some design examples.

1. Robotic finger with hinges. <http://www.thingiverse.com/thing:221560/#files>
2. Skateboard. <http://www.thingiverse.com/thing:172002/#files>
3. Lamp shade. <http://www.thingiverse.com/thing:11385/#files>
4. Gardenia Wall Mount Toothbrush Holder.

Some general guidelines for your design:

1. Imagine that you are a designer creating a customizable object for casual users. Try to expose parameters that make the most sense for customization: e.g. a container could have customizable volume, not radius and height, and the height of its lid is probably irrelevant.
2. Expose high-level parameters that control more than one low-level parameter value. For example, do not expose values of two parameters, where the value of one parameter depends on the value of another (such as the radius of the lid and radius of the pan).
3. Try to avoid global variables as much as possible.
4. Avoid copy-and-paste of code for the repeated components, e.g., create a module for the wheel of the car and reuse it.

3 Submission Instructions

Please provide a report with your submission (PDF). The report should include the following:

- Images of your designs.
- Were there any references (books, papers, websites, etc.) that you found particularly helpful for completing your assignment? Please provide a list.

- Are there any known problems with your design? If so, please provide a list and, if possible, describe what you think the cause is and how you might fix them if you had more time or motivation. This is very important, as we're much more likely to assign partial credit if you help us understand what's going on.
- Did you do any of the extra credit? If so, let us know how to use the additional features. If there was a substantial amount of work involved, describe what and how you did it. Provide at least one example for each extra feature implemented.
- Got any comments about this assignment that you'd like to share?

Remember, these assignments are to be done on your own. Please do not share code or implementation details with other students.

Submit your assignment via email (diwlevin@cs.toronto.edu, subject NAME_CSCCSC2521H_ASSIGNMENT.2) by **October 13th by 11:59pm**. Please submit a single archive (**.zip** or **.tar.gz**) containing:

- Your OpenSCAD source code saved as **.scad** file.
- Your designed 3D objects saved as **.stl** file.
- The PDF file.