OpenSCAD User Manual/Print version

Contents

Introduction

OpenSCAD is a software for creating solid 3D CAD objects.
It is free software (http://www.gnu.org/philosophy/free-sw.html) and available for GNU/Linux (http://www.gnu.org/), MS Windows and Apple OS X.

Unlike most free software for creating 3D models (such as the well-known application Blender), OpenSCAD does not focus on the artistic aspects of 3D modelling, but instead focuses on the CAD aspects. So it might be the application you are looking for when you are planning to create 3D models of machine parts, but probably is not what you are looking for when you are more interested in creating computer-animated movies or organic life-like models.

OpenSCAD, unlike many CAD products, is not an interactive modeler. Instead it is something like a 2D/3D-compiler that reads in a program file that describes the object and renders the model from this file. This gives you (the designer) full control over the modelling process. This enables you to easily change any step in the modelling process. This enables you to make designs that are defined by configurable parameters.

OpenSCAD has two main operating modes, Preview and Render. Preview is relatively fast using 3D graphics and the computer's GPU, but is an approximation of the model and can produce artifacts; Preview uses OpenCSG (http://opencsg.org/) and OpenGL. Render generates exact geometry and a fully tessellated mesh, it is not an approximation and as such it is often a lengthy process, taking minutes or hours for larger designs; Render uses CGAL as its geometry engine.

OpenSCAD provides two types of 3D modelling:

- Constructive Solid Geometry (CSG)
- extrusion of 2D primitives into 3D space.

Autocad DXF files are used as the data exchange format for 2D outlines. In addition to 2D paths for extrusion it is also possible to read design parameters from DXF files. Besides DXF files OpenSCAD can read and create 3D models in the STL and OFF file formats.

OpenSCAD can be downloaded from http://openscad.org/. You may find extra information in the mailing list (http://rocklinux.net/mailman/listinfo/openscad).

People who don't want to (or can't) install new software on their computer may be able to use OpenJSCAD (http://OpenJSCAD.org/), a port of OpenSCAD that runs in a web browser, if your browser supports WebGL

A pt_BR translation of this document is avaliable on GitHub repository (not completed/on development) [1] (http://www.github.com/ubb3rsith/OpenSCAD_doc_ptBR)

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OpenSCAD User Manual without The OpenSCAD Language Reference

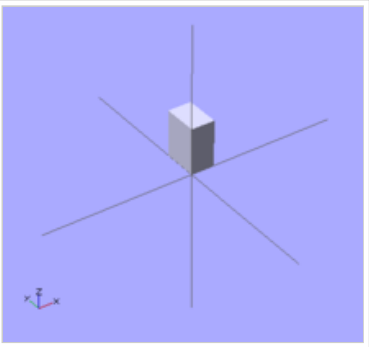
Chapter 1 -- First Steps

OpenSCAD User Manual

For our first model we will create a simple 2 x 3 x 4 cuboid. In the openSCAD editor, type the following one line command:

Usage example 1 - simple cuboid:

```
cube([2,3,4]);
```



OpenSCAD Simple Cuboid

Compiling and rendering our first model

The cuboid can now be compiled and rendered by pressing F5 or F6 while the OpenSCAD editor has focus.

See also

Positioning an object

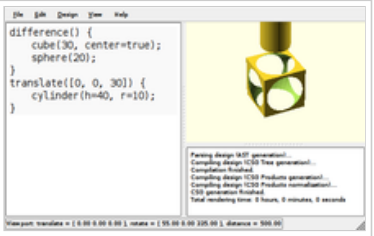
Open one of the many examples that come with OpenSCAD (*File*, *Examples*). Or you can copy and paste this simple example into the OpenSCAD window:

OpenSCAD after starting

Usage example 1

```
difference() {
    cube(30, center=true);
    sphere(20);
}
translate([0, 0, 30]) {
    cylinder(h=40, r=10);
}
```

Then **press F5** to get a graphical preview of what you typed (or **press F6** to get a graphical view).



OpenSCAD after pasting the example code and pressing F5

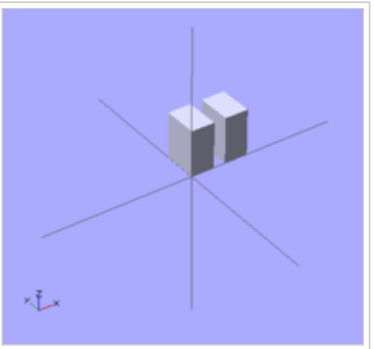
You get three types of movement in the preview frame:

1. Drag with left mouse button to rotate the view. The bottom line will change the rotate values.
2. Drag with an other mouse button (or control-drag under OSX) to translate (move) the view. The bottom line will change translate values.
3. Use the mouse scroll to zoom in and out. Alternatively you can use the + and - keys, or right-drag with the mouse while pressing a shift key (or control-shift-drag under OSX). The Viewport line at the bottom of the window will show a change in the distance value.

We have already seen how to create a simple cuboid. Our next task is to attempt to use the translate positioning command to place an identical cuboid next to the existing cuboid:

Usage example 1 - positioning an object:

```
cube([2,3,4]);
translate([3,0,0]) {
  cube([2,3,4]);
}
```



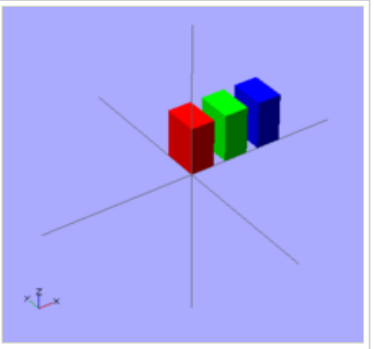
OpenSCAD positioning an object

There is no semicolon following the translate command

Notice that there is no semicolon following the translate command. This is because the translate command relates to the following object. If the semicolon was in place, then the effect of the position translation would end, and the second cuboid would be placed at the same position as the first cuboid. We can change the color of an object by giving it RGB values. Instead of the traditional RGB values from 0 to 255 floating point values are used from 0.0 to 1.0. Note! Changing the colors only works in Preview mode (F5). Render mode (F6) does not currently support color.

Usage example 1 - changing the color of an object:

```
color([1,0,0]) cube([2,3,4]);
translate([3,0,0])
color([0,1,0]) cube([2,3,4]);
translate([6,0,0])
color([0,0,1]) cube([2,3,4]);
```



OpenSCAD changing the color of an object

Color names can be used in the 2011.12 version (and newer). The names are the same used for Web colors (http://en.wikipedia.org/wiki/Web_colors). For example: `color("red") cube();`

If you think of the entire command as a sentence, then `color()` is an "adjective" that describes the "object" of the sentence (which is a "noun"). In this case, the object is the `cube()` to be created. The adjective is placed before the noun in the sentence, like so: `color() cube();`. In the same way, `translate()` can be thought of as a "verb" that acts upon the object, and is placed like this: `translate() color() cube();`. The following code will produce the same result:

```
translate([6,0,0])
{
   color([0,0,1])    // notice that there is NO semicolon
   cube([2,3,4]);    // notice the semicolon is at the end of all related commands
}
```

The "View" menu at the top of the OpenSCAD application window provides a variety of view options in the OpenSCAD model view window.

CGAL Surfaces

The surface view is the initial model view that appears when the model code is first rendered. You can get back to this view by choosing "View >> CGAL Surfaces".

CGAL Grid Only

Designers often choose "View >> CGAL Grid Only" when working with a particularly complex 3D model.

The Grid Only view presents only the "scaffolding" beneath the surface, also known as a wireframe. Think of the Eiffel Tower.

A wire frame is a visual presentation of a three dimensional or physical object. Using a wire frame model allows visualization of the underlying design structure of a 3D model. Since wireframe renderings are relatively simple and fast to calculate, they are often used in cases where a high screen frame rate is needed (for instance, when working with a particularly complex 3D model, or in real-time systems that model exterior phenomena). When greater graphical detail is desired, surface textures can be added automatically after completion of the initial rendering of the wireframe. This allows the designer to quickly review changes or rotate the object to new desired views without long delays associated with more realistic rendering. The wire frame format is also well suited and widely used in programming tool paths for DNC (Direct Numerical Control) machine tools. Wireframe models are also used as the input for CAM (computer-aided manufacturing). Wireframe is the most abstract and least realistic of the three main CAD models. This method of modelling consists only of lines, points and curves defining the edges of an object. (From Wikipedia: http://en.wikipedia.org/wiki/Wire-frame_model)

The OpenCSG View

Choosing "View >> OpenCSG" uses the open constructive solid geometry library to generate the model view utilizing OpenGL. If the OpenCSG library is not available or the video card or drivers do not support OpenGL, then this view will produce no visible output.

The thrown together view

Choosing "View >> Thrown Together" provides all the previous views, in the same screen.

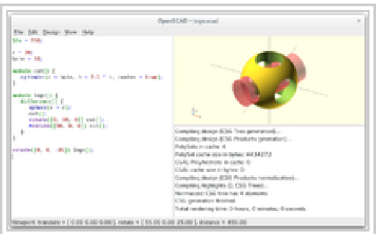
Chapter 2 -- The OpenSCAD User Interface

OpenSCAD User Manual

User Interface

The user interface of OpenSCAD has three parts

- The viewing area
- The console window
- The text editor



Main Window of OpenSCAD with a small program generating the OpenSCAD-Logo.

Viewing area

Preview and rendering output goes into the viewing area. Using the *Show Axes* menu entry an indicator for the coordinate axes can be enabled.

Console window

Status information, warnings and errors are displayed in the console window.

Text editor

The built-in text editor provides basic editing features like text search & replace and also supports syntax highlighting. There are predefined color schemes which can be selected in the Preferences dialog.

OpenSCAD Editor with Find / Replace functionality.

View navigation

The viewing area is navigated primarily using the mouse:

- Dragging with the left mouse button rotates the view along the axes of the viewing area. It preserves the vertical axis' direction.
- Dragging with the left mouse button when the shift key is pressed rotates the view along the vertical axis and the axis pointing towards the user.
- Dragging with the right mouse button moves the viewing area.
- For zooming, there are four ways:
  - using the scroll wheel
  - dragging with the middle mouse button
  - dragging with the right or middle mouse button and the shift key pressed
  - the keys + and -

Rotation can be reset using the shortcut Ctrl+0. Movement can be reset using the shortcut Ctrl+P.

View setup

The viewing area can be configured to use different rendering methods and other options using the View menu. Most of the options described here are available using shortcuts as well.

Render modes

OpenCSG (F9)

This method produces instantaneous results, but has low frame rates when working with highly nonconvex objects.

Note that selecting the OpenCSG mode using F9 will switch to the last generated OpenCSG view, but will not re-evaluate the source code. You may want to use the *Compile* function (F5, found in the *Design* menu) to re-evaluate the source code, build the OpenCSG objects and *then* switch to OpenCSG view.

Implementation Details

In OpenCSG mode, the OpenCSG library (http://opencsg.org/) is used for generating the visible model. This library uses advanced OpenGL features (2.0) like the Z buffer and does not require an explicit description of the resulting mesh – instead, it tracks how objects are to be combined. For example, when rendering a spherical dent in a cube, it will first render the cube on the graphics card and then render the sphere, but instead of using the Z buffer to **hide** the parts of the sphere that are covered by the cube, it will render **only** those parts of the sphere, visually resulting in a cube with a spherical dent.

CGAL (Surfaces and Grid, F10 and F11)

This method might need some time when first used with a new program, but will then have higher framerates.

As before with OpenCSG, F10 and F11 only enable CGAL display mode and don't update the underlying objects; for that, use the *Compile and Render* function (F6, found in the *Design* menu).

To combine the benefits of those two display methods, you can selectively wrap parts of your program in a render function and force them to be baken into a mesh even with OpenCSG mode enabled.

Implementation Details

The acronym CGAL refers to The Open Source Computational Geometry Algorithms Library.

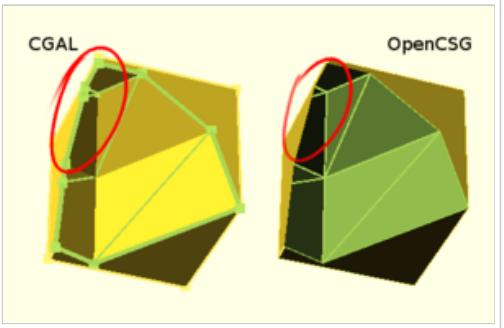
In CGAL mode, the CGAL library is used to compute the mesh of the root object, which is then displayed using simple OpenGL.

View options

Show Edges (Ctrl+1)

If *Show Edges* is enabled, both OpenCSG and CGAL mode will render edges as well as faces, CGAL will even show vertices. In CGAL grid mode, this option has no effect.

Enabling this option shows the difference between OpenCSG and CGAL quite clearly: While in CGAL mode you see an edge drawn everywhere it "belongs", OpenCSG will not show edges resulting from boolean operations – this is because they were never explicitly calculated but are just where one object's Z clipping begins or ends.



The difference between the CGAL and OpenCSG approaches can be seen at edges created by boolean operations.

Show Axes (Ctrl+2)

If *Show Axes* is enabled, the origin of the global coordinate system will be indicated by an orthogonal axes indicator. Additionally, a smaller axes indicator with axes names will be shown in the lower left corner of the viewing area. The smaller axes indicator is marked x, y, z and coloured red, green, blue respectively.

Show Crosshairs (Ctrl+3)

If *Show Crosshairs* is enabled, the center of the viewport will be indicated by four lines pointing in the room diagonal directions of the global coordinate system. This is useful when aligning the viewing area to a particular point in the model to keep it centered on screen during rotation.

Animation

The *Animate* option adds an animation bar to the lower edge of the screen. As soon as *FPS* and *Steps* are set (reasonable values to begin with are 10 and 100, respectively), the current *Time* is incremented by 1/*Steps*, *FPS* times per second, until it reaches 1, when it wraps back to 0.

Every time *Time* is changed, the program is re-evaluated with the variable `$t` set to the current time. Read more about how `$t` is used in section Other_Language_Features

View alignment

The menu items *Top*, *Bottom*, …, *Diagonal* and *Center* (Ctrl+4, Ctrl+5, …, Ctrl+0, Ctrl+P) align the view to the global coordinate system.

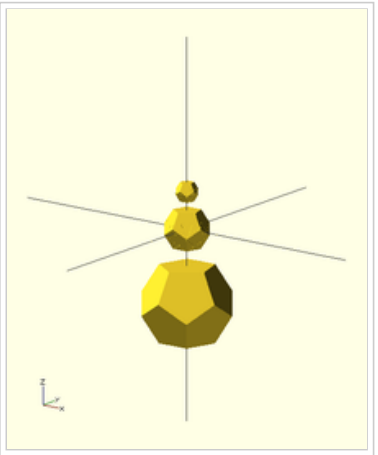
Top, *Bottom*, *Left*, *Right*, *Front* and *Back* align it in parallel to the axes, the *Diagonal* option aligns it diagonally as it is aligned when OpenSCAD starts.

The *Center* option will put the coordinate center in the middle of the screen (but not rotate the view).

By default, the view is in *Perspective* mode, meaning that distances far away from the viewer will look shorter, as it is common with eyes or cameras. When the view mode is changed to *Orthogonal*, visible distances will not depend on the camera distance (the view will simulate a camera in infinite distance with infinite focal length). This is especially useful in combination with the *Top* etc. options described above, as this will result in a 2D image similar to what one would see in an engineering drawing.

Dodecahedron

```
//create a dodecahedron by intersecting 6 boxes
module dodecahedron(height)
{
        scale([height,height,height]) //scale by height parameter
        {
                intersection(){
                        //make a cube
                        cube([2,2,1], center = true);
                        intersection_for(i=[0:4]) //loop i from 0 to 4, and intersect results
                        {
                                //make a cube, rotate it 116.565 degrees around the X axis,
                                //then 72*i around the Z axis
                                rotate([0,0,72*i])
                                        rotate([116.565,0,0])
                                        cube([2,2,1], center = true);
                        }
                }
        }
}
//create 3 stacked dodecahedra
//call the module with a height of 1 and move up 2
translate([0,0,2])dodecahedron(1);
//call the module with a height of 2
dodecahedron(2);
//call the module with a height of 4 and move down 4
translate([0,0,-4])dodecahedron(4);
```



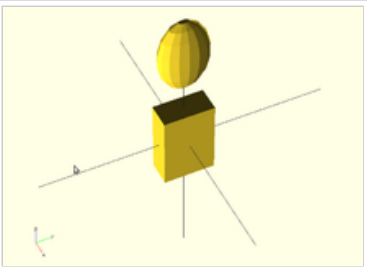
The Dodecahedron as rendered from the example.

Bounding Box

```
// Rather kludgy module for determining bounding box from intersecting projections
module BoundingBox()
{
        intersection()
        {
                translate([0,0,0])
                linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                projection(cut=false) intersection()
                {
                        rotate([0,90,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([0,-90,0])
                        children(0);

                        rotate([90,0,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([-90,0,0])
                        children(0);
                }
                rotate([90,0,0])
                linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                projection(cut=false)
                rotate([-90,0,0])
                intersection()
                {
                        rotate([0,90,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([0,-90,0])
                        children(0);

                        rotate([0,0,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
```



Bounding Box applied to an Ellipsoid

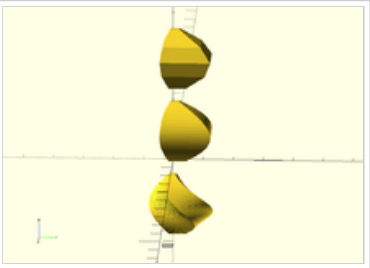
```
                        projection(cut=false)
                        rotate([0,0,0])
                        children(0);
                }
        }
}

// Test module on ellipsoid
translate([0,0,40]) scale([1,2,3]) sphere(r=5);
BoundingBox() scale([1,2,3]) sphere(r=5);
```

Linear Extrude extended use examples

Linear Extrude with Scale as an interpolated function

```
//Linear Extrude with Scale as an interpolated function
// This module does not need to be modified,
//  - unless default parameters want to be changed
//  - or additional parameters want to be forwarded (e.g. slices,...)
module linear_extrude_fs(height=1,isteps=20,twist=0){
  //union of piecewise generated extrudes
  union(){
      for(i = [ 0: 1: isteps-1]){
          //each new piece needs to be adjusted for height
          translate([0,0,i*height/isteps])
            linear_extrude(
              height=height/isteps,
              twist=twist/isteps,
              scale=f_lefs((i+1)/isteps)/f_lefs(i/isteps)
            )
              // if a twist constant is defined it is split into pieces
              rotate([0,0,-(i/isteps)*twist])
                // each new piece starts where the last ended
                scale(f_lefs(i/isteps))
                  obj2D_lefs();
      }
  }
}
// This function defines the scale function
//  - Function name must not be modified
//  - Modify the contents/return value to define the function
function f_lefs(x) =
  let(span=150,start=20,normpos=45)
  sin(x*span+start)/sin(normpos);
// This module defines the base 2D object to be extruded
//  - Function name must not be modified
//  - Modify the contents to define the base 2D object
module obj2D_lefs(){
  translate([-4,-3])
    square([9,12]);
}
```



Example Linear Extrude of a rectangle with scale following part of a sine curve function

```
//Top rendered object demonstrating the interpolation steps
translate([0,0,25])
linear_extrude_fs(height=20,isteps=4);
```

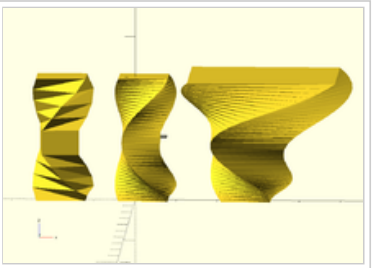
```
linear_extrude_fs(height=20);
```

```
//Bottom rendered object demonstrating the inclusion of a twist
translate([0,0,-25])
linear_extrude_fs(height=20,twist=90,isteps=30);
```

Linear Extrude with Twist as an interpolated function

```
//Linear Extrude with Twist as an interpolated function
// This module does not need to be modified,
//  - unless default parameters want to be changed
//  - or additional parameters want to be forwarded (e.g. slices,...)
module linear_extrude_ft(height=1,isteps=20,scale=1){
    //union of piecewise generated extrudes
    union(){
        for(i = [ 0: 1: isteps-1]){
            //each new piece needs to be adjusted for height
```

```
            translate([0,0,i*height/isteps])
              linear_extrude(
                height=height/isteps,
                twist=f_left((i+1)/isteps)-f_left((i)/isteps),
                scale=(1-(1-scale)*(i+1)/isteps)/(1-(1-scale)*i/isteps)
              )
                //Rotate to next start point
                rotate([0,0,-f_left(i/isteps)])
                  //Scale to end of last piece size
                  scale(1-(1-scale)*(i/isteps))
                    obj2D_left();
        }
    }
}
// This function defines the twist function
//  - Function name must not be modified
//  - Modify the contents/return value to define the function
function f_left(x) =
    let(twist=90,span=180,start=0)
    twist*sin(x*span+start);
// This module defines the base 2D object to be extruded
//  - Function name must not be modified
//  - Modify the contents to define the base 2D object
module obj2D_left(){
   translate([-4,-3])
     square([12,9]);
}
```



Example Linear Extrude of a rectangle with twist following part of a sine curve function

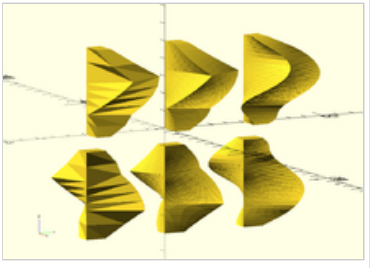
```
//Left rendered object demonstrating the interpolation steps
translate([-20,0])
linear_extrude_ft(height=30,isteps=5);
```

```
linear_extrude_ft(height=30);
```

```
//Right rendered object demonstrating the scale inclusion
translate([25,0])
linear_extrude_ft(height=30,scale=3);
```

Linear Extrude with Twist and Scale as interpolated functions

```
//Linear Extrude with Twist and Scale as interpolated functions
// This module does not need to be modified,
//  - unless default parameters want to be changed
//  - or additional parameters want to be forwarded
module linear_extrude_ftfs(height=1,isteps=20,slices=0){
    //union of piecewise generated extrudes
    union(){
      for(i=[0:1:isteps-1]){
        translate([0,0,i*height/isteps])
          linear_extrude(
            height=height/isteps,
            twist=leftfs_ftw((i+1)/isteps)-leftfs_ftw(i/isteps),
            scale=leftfs_fsc((i+1)/isteps)/leftfs_fsc(i/isteps),
            slices=slices
          )
            rotate([0,0,-leftfs_ftw(i/isteps)])
              scale(leftfs_fsc(i/isteps))
                obj2D_leftfs();
      }
    }
}
// This function defines the scale function
//  - Function name must not be modified
//  - Modify the contents/return value to define the function
function leftfs_fsc(x)=
    let(scale=3,span=140,start=20)
    scale*sin(x*span+start);
// This function defines the twist function
//  - Function name must not be modified
//  - Modify the contents/return value to define the function
function leftfs_ftw(x)=
    let(twist=30,span=360,start=0)
    twist*sin(x*span+start);
// This module defines the base 2D object to be extruded
//  - Function name must not be modified
//  - Modify the contents to define the base 2D object
module obj2D_leftfs(){
     square([12,9]);
}
```



Example Linear Extrude of a rectangle with twist and scale following part of a sine curve function

```
//Left rendered objects demonstrating the steps effect
translate([0,-50,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50,isteps=3);
```

```
translate([0,-50,0])
linear_extrude_ftfs(height=50,isteps=3);
```

```
//Center rendered objects demonstrating the slices effect
translate([0,0,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50,isteps=3,slices=20);

linear_extrude_ftfs(height=50,isteps=3,slices=20);
```

```
//Right rendered objects with default parameters
translate([0,50,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50);

translate([0,50,0])
linear_extrude_ftfs(height=50);
```

Command line usage

OpenSCAD can not only be used as a GUI, but also handles command line arguments. Its usage line says:

OpenSCAD 2015.03-1 has these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] \
             [ --help ] print this help message and exit \
             [ --version ] [ --info ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --autocenter ] \
             [ --viewall ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             [ --render | --preview[=throwntogether] ] \
             [ --colorscheme=[Cornfield|Sunset|Metallic|Starnight|BeforeDawn|Nature|DeepOcean] ] \
             [ --csglimit=num ]\
             filename
```

OpenSCAD 2014.03+ has these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] \
             [ --version ] [ --info ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             [ --render | --preview[=throwntogether] ] \
             [ --csglimit=num ] \
             filename
```

Openscad 2013.05 had these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] [ --render ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             filename
```

Earlier releases had only these:

```
openscad [ -o output_file [ -d deps_file ] ] \
         [ -m make_command ] [ -D var=val [..] ] filename
```

The usage on OpenSCAD version 2011.09.30 (now deprecated) was:

```
openscad [ { -s stl_file | -o off_file | -x dxf_file } [ -d deps_file ] ]\
         [ -m make_command ] [ -D var=val [..] ] filename
```

Export options

When called with the `-o` option, OpenSCAD will not start the GUI, but execute the given file and export the to the *output_file* in a format depending on the extension (`.stl` / `.off` / `.dxf`, `.csg`).

Some versions use -s/-d/-o to determine the output file format instead; check with "openscad --help".

If the option `-d` is given in addition to an export command, all files accessed while building the mesh are written in the argument of `-d` in the syntax of a Makefile.

For at least 2015.03-2+, specifying the extension `.echo` causes openscad to produce a text file containing error messages and the output of all `echo()` calls in `filename` as they would appear in the console window visible in the GUI. Multiple output files are not supported, so using this option you cannot also obtain the model that would have normally been generated.

Camera and image output

For 2013.05+, the option to output a `.png` image was added. There are two types of cameras available for the generation of images.

The first camera type is a 'gimbal' camera that uses Euler angles, translation, and a camera distance, like OpenSCAD's GUI viewport display at the bottom of the OpenSCAD window.

!!! There is a bug in the implementation of cmdline camera, where the rotations do not match the numbers in the GUI. This will be fixed in an upcoming release so that the GUI and cmdline camera variables will work identically.

The second camera type is a 'vector' camera, with an 'eye' camera location vector and a 'lookat' center vector.

--imgsize chooses the .png dimensions and --projection chooses orthogonal or perspective, as in the GUI.

By default, cmdline .png output uses Preview mode (f5) with OpenCSG. For some situations it will be desirable instead to use the full render, with CGAL. This is done by adding '--render' as an option.

Constants

In order to pre-define variables, use the `-D` option. It can be given repeatedly. Each occurrence of `-D` must be followed by an assignment. Unlike normal OpenSCAD assignments, these assignments don't define variables, but constants, which can not be changed inside the program, and can thus be used to overwrite values defined in the program at export time.

If you want to assign the -D variable to another variable, the -D variable MUST be initialised in the main .scad program

```
param1=0;   // must be initalised
len=param1; // param1 passed via -D on cmd-line
echo(len,param);
```

without the first line len wound be undefined.

The right hand sides can be arbitrary OpenSCAD expressions, including mathematical operations and strings. Be aware that strings have to be enclosed in quotes, which have to be escaped from the shell. To render a model that takes a quality parameter with the value "production", one has to run

```
openscad -o my_model_production.stl -D 'quality="production"' my_model.scad
```

On Windows you may need to escape the inner quotes instead:

```
openscad -o my_model_production.stl -D "quality=\"production\"" my_model.scad
```

Command to build required files

In a complex build process, some files required by an OpenSCAD file might be currently missing, but can be generated, for example if they are defined in a Makefile. If OpenSCAD is given the option `-m make`, it will start `make` *`file`* the first time it tries to access a missing *file*.

Processing all .scad files in a folder

Example to convert all the .scad in a folder into .stl:

In a folder with .scad files, make a .bat file with text:

```
  FOR %%f in (*.scad)  DO openscad -o "%%~nf.stl" "%%f"
```

If it closes without processing, check to set the PATH by adding openscad directory to:

```
    Start - Settings - Control Panel - System - Advanced tab - Environment Variables - System Variables, select Path, then click Edit.
```

Add the openscad directory to the list

Makefile example

The `-d` and `-m` options only make sense together. (`-m` without `-d` would not consider modified dependencies when building exports, `-d` without `-m` would require the files to be already built for the first run that generates the dependencies.)

Here is an example of a basic Makefile that creates an .stl file from an .scad file of the same name:

```
# explicit wildcard expansion suppresses errors when no files are found
include $(wildcard *.deps)

%.stl: %.scad
	openscad -m make -o $@ -d $@.deps $<
```

When `make my_example.stl` is run for the first time, it finds no .deps files, and will just depend on `my_example.scad`; since `my_example.stl` is not yet preset, it will be created unconditionally. If OpenSCAD finds missing files, it will call `make` to build them, and it will list all used files in `my_example.stl.deps`.

When `make my_example.stl` is called subsequently, it will find and include `my_example.stl.deps` and check if any of the files listed there, including `my_example.scad`, changed since `my_example.stl` was built, based on their time stamps. Only if that is the case, it will build `my_example.stl` again.

Automatic targets

When building similar .stl files from a single .scad file, there is a way to automate that too:

```
# match "module foobar() { // `make` me"
TARGETS=$(shell sed '/^module [a-z0-9_-]*().*make..\?me.*$$/!d;s/module //;s/().*/.stl/' base.scad)

all: ${TARGETS}

# auto-generated .scad files with .deps make make re-build always. keeping the
# scad files solves this problem. (explanations are welcome.)
.SECONDARY: $(shell echo "${TARGETS}" | sed 's/\.stl/.scad/g')

# explicit wildcard expansion suppresses errors when no files are found
include $(wildcard *.deps)

%.scad:
	echo -n 'use <base.scad>\n$*();' > $@

%.stl: %.scad
	openscad -m make -o $@ -d $@.deps $<
```

All objects that are supposed to be exported automatically have to be defined in `base.scad` in an own module with their future file name (without the ".stl"), and have a comment like "`// make me`" in the line of the module definition. The "`TARGETS=`" line picks these out of the base file and creates the file names. These will be built when `make all` (or `make`, for short) is called.

As the convention from the last example is to create the .stl files from .scad files of the same base name, for each of these files, an .scad file has to be generated. This is done in the "`%.scad:`" paragraph; `my_example.scad` will be a very simple OpenSCAD file:

```
use <base.scad>
my_example();
```

The "`.SECONDARY`" line is there to keep `make` from deleting the generated .scad files. If it deleted it, it would not be able to automatically determine which files need no rebuild any more; please post ideas about what exactly goes wrong there (or how to fix it better) on the talk page!

Windows notes

On Windows, openscad.com should be called from the command line as a wrapper for openscad.exe. This is because Openscad uses the 'devenv' solution to the Command-Line/GUI output issue. Typing 'openscad' at the cmd.exe prompt will, by default, call the .com program wrapper.

MacOS notes

On MacOS the binary is normally hidden inside the App folder. If OpenSCAD is installed in the global Applications folder, it can be called from command line like in the following example that just shows the OpenSCAD version:

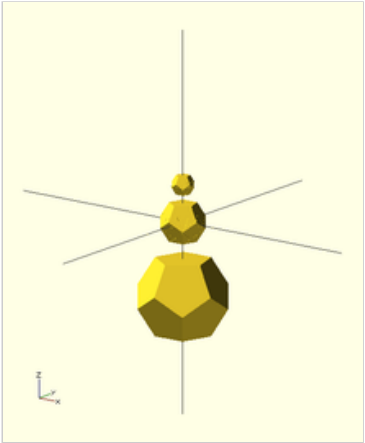
```
macbook:/$ /Applications/OpenSCAD.app/Contents/MacOS/OpenSCAD -v
OpenSCAD version 2013.06
```

Chapter 3 -- Commented Example Projects

OpenSCAD User Manual

Dodecahedron

```
//create a dodecahedron by intersecting 6 boxes
module dodecahedron(height)
{
        scale([height,height,height]) //scale by height parameter
        {
                intersection(){
                        //make a cube
                        cube([2,2,1], center = true);
                        intersection_for(i=[0:4]) //loop i from 0 to 4, and intersect results
                        {
                                //make a cube, rotate it 116.565 degrees around the X axis,
                                //then 72*i around the Z axis
                                rotate([0,0,72*i])
                                        rotate([116.565,0,0])
                                        cube([2,2,1], center = true);
                        }
                }
        }
}
//create 3 stacked dodecahedra
//call the module with a height of 1 and move up 2
translate([0,0,2])dodecahedron(1);
//call the module with a height of 2
dodecahedron(2);
//call the module with a height of 4 and move down 4
translate([0,0,-4])dodecahedron(4);
```



The Dodecahedron as rendered from the example.

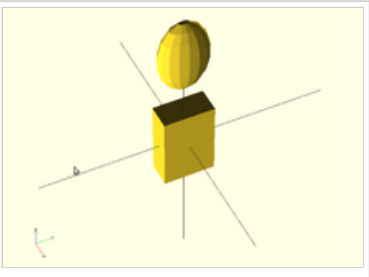
Bounding Box

```
// Rather kludgy module for determining bounding box from intersecting projections
module BoundingBox()
{
        intersection()
        {
                translate([0,0,0])
                linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                projection(cut=false) intersection()
                {
                        rotate([0,90,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([0,-90,0])
                        children(0);

                        rotate([90,0,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([-90,0,0])
                        children(0);
                }
                rotate([90,0,0])
                linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                projection(cut=false)
                rotate([-90,0,0])
                intersection()
                {
                        rotate([0,90,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([0,-90,0])
                        children(0);

                        rotate([0,0,0])
                        linear_extrude(height = 1000, center = true, convexity = 10, twist = 0)
                        projection(cut=false)
                        rotate([0,0,0])
                        children(0);
                }
        }
}

// Test module on ellipsoid
translate([0,0,40]) scale([1,2,3]) sphere(r=5);
BoundingBox() scale([1,2,3]) sphere(r=5);
```

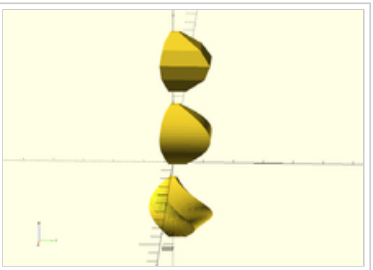


Bounding Box applied to an Ellipsoid

Linear Extrude extended use examples

Linear Extrude with Scale as an interpolated function

```
//Linear Extrude with Scale as an interpolated function
// This module does not need to be modified,
//  - unless default parameters want to be changed
//  - or additional parameters want to be forwarded (e.g. slices,...)
module linear_extrude_fs(height=1,isteps=20,twist=0){
  //union of piecewise generated extrudes
  union(){
      for(i = [ 0: 1: isteps-1]){
          //each new piece needs to be adjusted for height
          translate([0,0,i*height/isteps])
            linear_extrude(
              height=height/isteps,
              twist=twist/isteps,
              scale=f_lefs((i+1)/isteps)/f_lefs(i/isteps)
            )
              // if a twist constant is defined it is split into pieces
              rotate([0,0,-(i/isteps)*twist])
                // each new piece starts where the last ended
                scale(f_lefs(i/isteps))
                  obj2D_lefs();
      }
  }
}
// This function defines the scale function
//  - Function name must not be modified
//  - Modify the contents/return value to define the function
function f_lefs(x) =
  let(span=150,start=20,normpos=45)
  sin(x*span+start)/sin(normpos);
// This module defines the base 2D object to be extruded
//  - Function name must not be modified
//  - Modify the contents to define the base 2D object
module obj2D_lefs(){
  translate([-4,-3])
    square([9,12]);
}
```



Example Linear Extrude of a rectangle with scale following part of a sine curve function

```
//Top rendered object demonstrating the interpolation steps
translate([0,0,25])
linear_extrude_fs(height=20,isteps=4);
```

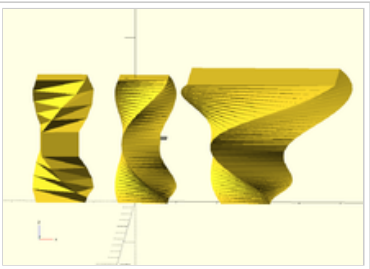
```
linear_extrude_fs(height=20);
```

```
//Bottom rendered object demonstrating the inclusion of a twist
translate([0,0,-25])
linear_extrude_fs(height=20,twist=90,isteps=30);
```

Linear Extrude with Twist as an interpolated function

```
//Linear Extrude with Twist as an interpolated function
// This module does not need to be modified,
//  - unless default parameters want to be changed
//  - or additional parameters want to be forwarded (e.g. slices,...)
module linear_extrude_ft(height=1,isteps=20,scale=1){
    //union of piecewise generated extrudes
    union(){
        for(i = [ 0: 1: isteps-1]){
            //each new piece needs to be adjusted for height
            translate([0,0,i*height/isteps])
              linear_extrude(
                height=height/isteps,
                twist=f_left((i+1)/isteps)-f_left((i)/isteps),
                scale=(1-(1-scale)*(i+1)/isteps)/(1-(1-scale)*i/isteps)
              )
                //Rotate to next start point
                rotate([0,0,-f_left(i/isteps)])
                  //Scale to end of last piece size
                  scale(1-(1-scale)*(i/isteps))
                    obj2D_left();
        }
```

```
    }
}
// This function defines the twist function
//   - Function name must not be modified
//   - Modify the contents/return value to define the function
function f_left(x) =
    let(twist=90,span=180,start=0)
    twist*sin(x*span+start);
// This module defines the base 2D object to be extruded
//   - Function name must not be modified
//   - Modify the contents to define the base 2D object
module obj2D_left(){
   translate([-4,-3])
     square([12,9]);
}
```



Example Linear Extrude of a rectangle with twist following part of a sine curve function

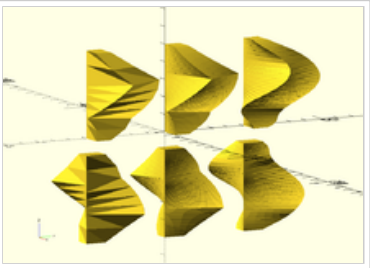
```
//Left rendered object demonstrating the interpolation steps
translate([-20,0])
linear_extrude_ft(height=30,isteps=5);
```

```
linear_extrude_ft(height=30);
```

```
//Right rendered object demonstrating the scale inclusion
translate([25,0])
linear_extrude_ft(height=30,scale=3);
```

Linear Extrude with Twist and Scale as interpolated functions

```
//Linear Extrude with Twist and Scale as interpolated functions
// This module does not need to be modified,
//   - unless default parameters want to be changed
//   - or additional parameters want to be forwarded
module linear_extrude_ftfs(height=1,isteps=20,slices=0){
    //union of piecewise generated extrudes
    union(){
      for(i=[0:1:isteps-1]){
        translate([0,0,i*height/isteps])
          linear_extrude(
            height=height/isteps,
            twist=leftfs_ftw((i+1)/isteps)-leftfs_ftw(i/isteps),
            scale=leftfs_fsc((i+1)/isteps)/leftfs_fsc(i/isteps),
            slices=slices
          )
            rotate([0,0,-leftfs_ftw(i/isteps)])
              scale(leftfs_fsc(i/isteps))
                obj2D_leftfs();
      }
    }
}
// This function defines the scale function
//   - Function name must not be modified
//   - Modify the contents/return value to define the function
function leftfs_fsc(x)=
    let(scale=3,span=140,start=20)
    scale*sin(x*span+start);
// This function defines the twist function
//   - Function name must not be modified
//   - Modify the contents/return value to define the function
function leftfs_ftw(x)=
    let(twist=30,span=360,start=0)
    twist*sin(x*span+start);
// This module defines the base 2D object to be extruded
//   - Function name must not be modified
//   - Modify the contents to define the base 2D object
module obj2D_leftfs(){
     square([12,9]);
}
```



Example Linear Extrude of a rectangle with twist and scale following part of a sine curve function

```
//Left rendered objects demonstrating the steps effect
translate([0,-50,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50,isteps=3);

translate([0,-50,0])
linear_extrude_ftfs(height=50,isteps=3);
```

```
//Center rendered objects demonstrating the slices effect
translate([0,0,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50,isteps=3,slices=20);
```

```
linear_extrude_ftfs(height=50,isteps=3,slices=20);
```

```
//Right rendered objects with default parameters
translate([0,50,-60])
rotate([0,0,90])
linear_extrude_ftfs(height=50);

translate([0,50,0])
linear_extrude_ftfs(height=50);
```

Chapter 4 -- Export

OpenSCAD User Manual

Export

After rendering with F6, the "File --> Export" menu can be used to export as STL, OFF, AMF, DXF, SVG, CSG OR PNG (Image).

Be sure to check the console window for err messages.

```
   STL, OFF and DXF are imported using import().
   CSG can be imported using include<> or loaded like an SCAD file
   PNG can be imported using surface()
   There are open pull requests for SVG and AMF, which require a bit more work/testing.
   The file suffix is used to determine type.
```

STL Export

To export your design, select "Export as STL..." from the "File --> Export" menu, then enter a filename in the ensuing dialog box. Don't forget to add the ".stl" extension.

Trouble shooting:

After *compile and render GCAL* (F6), you may see that your design is *simple: no*. That's bad news.

See line 8 in the following output from *OpenSCAD 2010.02*:

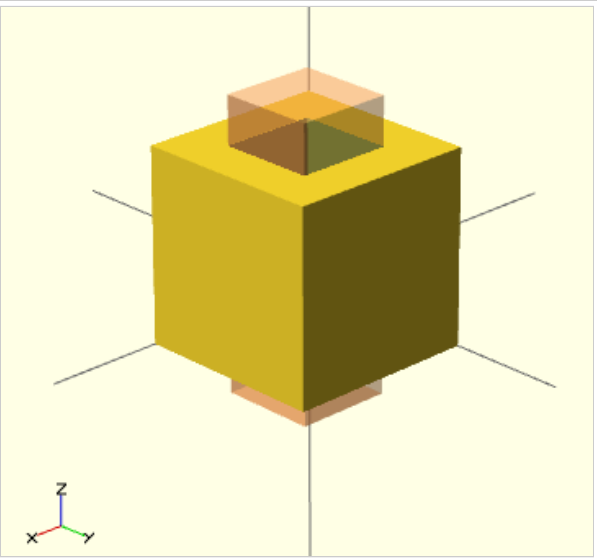
```
Parsing design (AST generation)...
Compiling design (CSG Tree generation)...
Compilation finished.
Rendering Polygon Mesh using CGAL...
Number of vertices currently in CGAL cache: 732
Number of objects currently in CGAL cache: 12
   Top level object is a 3D object:
   Simple:         no                 <*****************
   Valid:         yes
   Vertices:       22
   Halfedges:      70
   Edges:          35
   Halffacets:     32
   Facets:         16
   Volumes:         2
Total rendering time: 0 hours, 0 minutes, 0 seconds
Rendering finished.
```

When you try to export this to .STL you will get a message like:

```
Object isn't a valid 2-manifold! Modify your design..
```

"Manifold" means that it is "water tight" and that there are no holes in the geometry. In a valid 2-manifold each edge must connect exactly two facets. That means that the program must be able to connect a face with an object. E.g. if you use a cube of height 10 to carve out something from a wider cube of height 10, it is not clear to which cube the top or the bottom belongs. So make the small extracting cube a bit "longer" (or "shorter"):

```
difference() {
    // original
    cube (size = [2,2,2]);
    // object that carves out
    # translate ([0.5,0.5,-0.5]) {
        cube (size = [1,1,3]);
    }
}
```

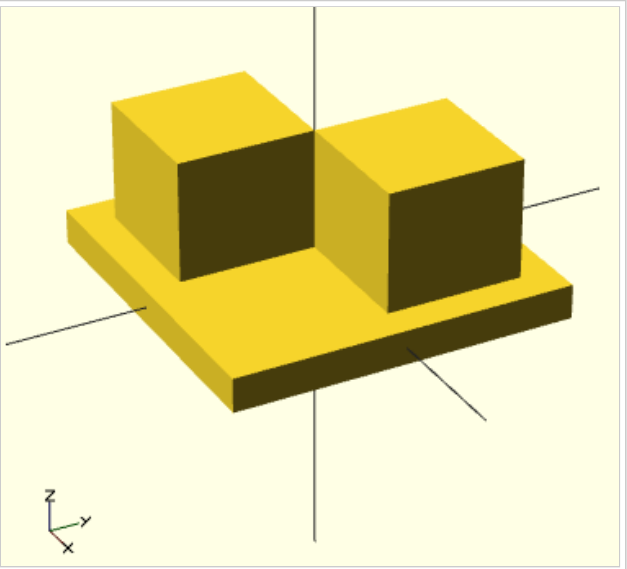


Correct use of difference

Here is a more tricky little example taken from the OpenSCAD (http://rocklinux.net/pipermail/openscad/2009-December/000018.html) Forum (retrieved 15:13, 22 March 2010 (UTC)):

```
module example1() {
        cube([20, 20, 20]);
        translate([-20, -20, 0]) cube([20, 20, 20]);
        cube([50, 50, 5], center = true);
    }
module example2() {
        cube([20.1, 20.1, 20]);
        translate([-20, -20, 0]) cube([20.1, 20.1, 20]);
        cube([50, 50, 5], center = true);
    }
```

Example1 would render like this:



A not valid 2-manifold cube (simple = no)

The **example1** module is not a valid 2-manifold because both cubes are sharing one edge. They touch each other but do not intersect.

Example2 is a valid 2-manifold because there is an intersection. Now the construct meets the 2-manifold constraint stipulating that *each edge* must connect exactly two facets.

Pieces you are subtracting must extend past the original part. (OpenSCAD Tip: Manifold Space and Time (http://www.iheartrobotics.com/2010/01/openscad-tip-manifold-space-and-time.html), retrieved 18:40, 22 March 2010 (UTC)).

For reference, another situation that causes the design to be non-exportable is when two faces that are each the result of a subtraction touch. Then the error message comes up.

```
difference () {
```

```
   cube ([20,10,10]);
   translate ([10,0,0]) cube (10);
}
difference () {
   cube ([20,10,10]);
   cube (10);
}
```

simply touching surfaces is correctly handled.

```
translate ([10,0,0]) cube (10);
cube (10);
```

- STL, OFF, AMF, DXF, SVG, CSG, PNG

With the import() and extrusion modules it is possible to convert 2D objects read from DXF files to 3D objects. See also 2D to 3D Extrusion.

Linear Extrude

Example of linear extrusion of a 2D object imported from a DXF file.

```
linear_extrude(height = fanwidth, center = true, convexity = 10)
   import (file = "example009.dxf", layer = "fan_top");
```

Rotate Extrude

Example of rotational extrusion of a 2D object imported from a DXF file.

```
rotate_extrude(convexity = 10)
   import (file = "example009.dxf", layer = "fan_side", origin = fan_side_center);
```

Getting Inkscape to work

Inkscape is an open source drawing program. Tutorials for transferring 2d DXF drawings from Inkscape to OpenSCAD are available here:

- http://repraprip.blogspot.com/2011/05/inkscape-to-openscad-dxf-tutorial.html (Very simple, needs path segments to be straight lines)
- http://tonybuser.com/?tag=inkscape (More complicated, involves conversion to Postscript)
- http://bobcookdev.com/inkscape/inkscape-dxf.html (Better DXF Export, native support for bezier curves)
- http://www.bigbluesaw.com/saw/big-blue-saw-blog/general-updates/big-blue-saws-dxf-export-for-inkscape.html (even better support, works as of 10/29/2014, see link below registration window. Note: As of 6/17/15 only works with version 0.48.5 or earlier of inkscape, due to a breaking change made in 0.91.)
- http://www.instructables.com/id/Convert-any-2D-image-to-a-3D-object-using-OpenSCAD/ (Convert any 2D image to a 3D object using OpenSCAD)

Currently, OpenSCAD only supports DXF as a graphics format for 2D graphics. Other common formats are PS/EPS, SVG and AI.

PS/EPS

The pstoedit (http://www.pstoedit.net/) program can convert between various vector graphics formats. OpenSCAD needs the `-polyaslines` option passed to the dxf output plugin to understand the file. The `-mm` option sets one mm to be one unit in the dxf; include this if you use one unit in OpenSCAD as equal to one millimeter. The `-dt` options instructs pstoedit to render texts, which is usually what you want if you include text. (If the rendered text's resolution in terms of polygon count is too low, the easiest solution is to scape up the eps before converting; if you know a more elegant solution, please add it to the example.)

```
pstoedit -dt -f "dxf: -polyaslines -mm" infile.eps outfile.dxf
```

SVG

Inkscape (http://inkscape.org) can convert SVG to EPS. Then pstoedit can convert the EPS to DXF.

```
inkscape -E intermediate.eps infile.svg
```

```
pstoedit -dt -f dxf:-polyaslines\ -mm intermediate.eps outfile.dxf
```

Makefile automation

The conversion can be automated using the make system; put the following lines in your `Makefile`:

```
all: my_first_file.dxf my_second_file.dxf another_file.dxf

%.eps: %.svg
   inkscape -E $@ $<

%.dxf: %.eps
   pstoedit -dt -f dxf:-polyaslines\ -mm $< $@
```

The first line specifies which dxf files are to be generated when `make` is called in the current directory. The second paragraph specifies how to convert a file ending in .svg to a file ending in .eps, and the third from .eps to .dxf.

A more complete makefile could autogenerate dxf files from the any svg in the folder. In which case, put the following lines into your `Makefile`:

```
SVG := $(wildcard *.svg)
DXF := $(SVG:%.svg=%.dxf)
EPS := $(SVG:%.svg=%.eps)

.PHONY: all clean clean-eps clean-dxf

all: $(DXF)

%.eps: %.svg
   inkscape -E $*.eps $*.svg

%.dxf: %.eps
   pstoedit -dt -f "dxf: -polyaslines -mm" $*.eps $*.dxf

clean: clean-dxf clean-eps

clean-dxf:
   rm -f $(DXF)

clean-eps:
   rm -f $(EPS)
```

It's still possible to call `make filename.dxf` to build a particular file, but this code also allows for (re)building of all dxf files in a folder just by calling `make` or `make all`.

This code is also universal enough that it's possible to put the code in a single file and symlink every makefile in any directory that has svg files for dxf conversion by running:

```
ls -s /path/to/this/svg_to_dxf_makefile makefile
```

in each respective directory.

AI (Adobe Illustrator)

Although Adobe Illustrator CC/CC.2014 allows you to export illustrations as DXF (and select DXF format versions as early as 12), it will use DXF entities that are not supported by OpenSCAD, such as `POLYLINE` and `SPLINE`.

Since pstoedit does not natively support Adobe Illustrator files, one alternative is to use EXDXF (http://www.baby-universe.co.jp/en/plug-in/products/exdxf-pro/) which is an Adobe Illustrator plug-in (30 free trial exports and then you have to pay $90 to register the plugin).

Before exporting, it is recommended that you ensure that your Artboard is the same dimensions as the component you are exporting. Although EXDXF provides you with numerous options when exporting to DXF the most important option for OpenSCAD compliance is to set `Line Conversion` to `Line and Arc`.

OpenSCAD doesn't always provide information about the issues it encountered with a DXF import. If this happens, select `Design | Flush Caches` and then `Design | Reload and Compile`.

Chapter 5 -- Using an external Editor with OpenSCAD

OpenSCAD User Manual

Why use an external editor

OpenSCAD session using emacs as an external editor

Many people prefer to use a certain editor. They are used to the feature set and know the keybindings. OpenSCADs editor is functional and simplistic but might lack features people know from other editors.

How to use an external editor

OpenScad is able to check for changes of files and automatically recompile if a file change occurs. To use this feature enable *"Design->Automatic Reload and Compile"*

Once the feature is activated, just load the scad file within OpenSCAD as usual (*"File->Open.."*). After that, open the scad file in your favorite editor too. Edit and work on the scad file within the external editor. Whenever the file is saved to disk (from within the external editor), OpenSCAD will recognize the file change and automatically recompiles accordingly.

The internal editor can be hidden by minimizing the frame with the mouse or by selecting *"View->Hide editor"*.

Support of external editors

In principle all editors can be used. For some exist extensions/modes to provide features for OpenSCAD.

- **Emacs**: OpenSCAD provides an emacs mode (https://github.com/openscad/openscad/blob/master/contrib/scad-mode.el) for OpenSCAD files. Use the link or install via emacs package management (ELPA) with the MELPA (https://melpa.org) repository.
- **Kate**: nerd256 (http://www.thingiverse.com/nerd256/overview) provides a kate syntax file (http://www.thingiverse.com/thing:29505) for OpenSCAD. See *Instructions* tab in Thingiverse to install it. You could create also a kate *External tool* to open OpenSCAD with the current file with script `openscad %directory/%filename`
- **Notepad++**: TheHeadlessSourceMan (http://www.thingiverse.com/TheHeadlessSourceMan/overview) provides a Notepad++ syntax file (http://www.thingiverse.com/thing:280319) for OpenSCAD. See *Instructions* tab in Thingiverse to install it.
- **Geany**: cobra18t (http://www.thingiverse.com/cobra18t/overview) provides a Geany syntax file (http://www.thingiverse.com/thing:263620) for OpenSCAD. See *Instructions* tab in Thingiverse to install it.
- **VIM**: vim.org provides a VIM syntax file (http://www.vim.org/scripts/script.php?script_id=3556) for OpenSCAD.
- **Gedit**: Andy Turner (https://github.com/AndrewJamesTurner) provides a Gedit syntax file (https://github.com/AndrewJamesTurner/openSCAD-lang-file) for OpenSCAD.
- **Atom**: There is a Language OpenSCAD package (https://atom.io/packages/language-openscad) for Atom (https://atom.io) that provides highlighting and snippets.
- **Sublime**: Syntax highlighting and Customizer support (http://www.thingiverse.com/thing:67566)
- **Textmate**: Syntax highlighting and Customizer support (http://www.thingiverse.com/thing:67566)
- **OpenSCADitor**: OpenSCAD-dedicated editor (http://rcs-34.net//software/openscaditor/index.php)
- **Visual Studio Code**: Free, open-source code editor (https://code.visualstudio.com/) Install the scad extension for syntax highlighting

Additional benefits

Besides using your editor of choice, this solutions enables the flexible usage of multi-monitor set-ups. One can have one monitor set-up to depict the 3D object on the entire screen and a second monitor for the editor and other tools.

Chapter 6 -- Using OpenSCAD in a command line environment

OpenSCAD User Manual

Command line usage

OpenSCAD can not only be used as a GUI, but also handles command line arguments. Its usage line says:

OpenSCAD 2015.03-1 has these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] \
             [ --help ] print this help message and exit \
             [ --version ] [ --info ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --autocenter ] \
             [ --viewall ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             [ --render | --preview[=throwntogether] ] \
             [ --colorscheme=[Cornfield|Sunset|Metallic|Starnight|BeforeDawn|Nature|DeepOcean] ] \
             [ --csglimit=num ]\
             filename
```

OpenSCAD 2014.03+ has these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] \
             [ --version ] [ --info ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             [ --render | --preview[=throwntogether] ] \
             [ --csglimit=num ] \
             filename
```

Openscad 2013.05 had these options:

```
openscad     [ -o output_file [ -d deps_file ] ]\
             [ -m make_command ] [ -D var=val [..] ] [ --render ] \
             [ --camera=translatex,y,z,rotx,y,z,dist | \
               --camera=eyex,y,z,centerx,y,z ] \
             [ --imgsize=width,height ] [ --projection=(o)rtho|(p)ersp] \
             filename
```

Earlier releases had only these:

```
openscad [ -o output_file [ -d deps_file ] ] \
         [ -m make_command ] [ -D var=val [..] ] filename
```

The usage on OpenSCAD version 2011.09.30 (now deprecated) was:

```
openscad [ { -s stl_file | -o off_file | -x dxf_file } [ -d deps_file ] ]\
         [ -m make_command ] [ -D var=val [..] ] filename
```

Export options

When called with the `-o` option, OpenSCAD will not start the GUI, but execute the given file and export the to the *output_file* in a format depending on the extension (`.stl` / `.off` / `.dxf`, `.csg`).

Some versions use -s/-d/-o to determine the output file format instead; check with "openscad --help".

If the option `-d` is given in addition to an export command, all files accessed while building the mesh are written in the argument of `-d` in the syntax of a Makefile.

For at least 2015.03-2+, specifying the extension `.echo` causes openscad to produce a text file containing error messages and the output of all `echo()` calls in `filename` as they would appear in the console window visible in the GUI. Multiple output files are not supported, so using this option you cannot also obtain the model that would have normally been generated.

Camera and image output

For 2013.05+, the option to output a `.png` image was added. There are two types of cameras available for the generation of images.

The first camera type is a 'gimbal' camera that uses Euler angles, translation, and a camera distance, like OpenSCAD's GUI viewport display at the bottom of the OpenSCAD window.

!!! There is a bug in the implementation of cmdline camera, where the rotations do not match the numbers in the GUI. This will be fixed in an upcoming release so that the GUI and cmdline camera variables will work identically.

The second camera type is a 'vector' camera, with an 'eye' camera location vector and a 'lookat' center vector.

--imgsize chooses the .png dimensions and --projection chooses orthogonal or perspective, as in the GUI.

By default, cmdline .png output uses Preview mode (f5) with OpenCSG. For some situations it will be desirable instead to use the full render, with CGAL. This is done by adding '--render' as an option.

Constants

In order to pre-define variables, use the `-D` option. It can be given repeatedly. Each occurrence of `-D` must be followed by an assignment. Unlike normal OpenSCAD assignments, these assignments don't define variables, but constants, which can not be changed inside the program, and can thus be used to overwrite values defined in the program at export time.

If you want to assign the -D variable to another variable, the -D variable MUST be initialised in the main .scad program

```
param1=0;   // must be initalised
len=param1; // param1 passed via -D on cmd-line
echo(len,param);
```

without the first line len wound be undefined.

The right hand sides can be arbitrary OpenSCAD expressions, including mathematical operations and strings. Be aware that strings have to be enclosed in quotes, which have to be escaped from the shell. To render a model that takes a quality parameter with the value "production", one has to run

```
openscad -o my_model_production.stl -D 'quality="production"' my_model.scad
```

On Windows you may need to escape the inner quotes instead:

```
openscad -o my_model_production.stl -D "quality=\"production\"" my_model.scad
```

Command to build required files

In a complex build process, some files required by an OpenSCAD file might be currently missing, but can be generated, for example if they are defined in a Makefile. If OpenSCAD is given the option `-m make`, it will start `make` *`file`* the first time it tries to access a missing *file*.

Processing all .scad files in a folder

Example to convert all the .scad in a folder into .stl:

In a folder with .scad files, make a .bat file with text:

```
  FOR %%f in (*.scad)  DO openscad -o "%%~nf.stl" "%%f"
```

If it closes without processing, check to set the PATH by adding openscad directory to:

```
    Start - Settings - Control Panel - System - Advanced tab - Environment Variables - System Variables, select Path, then click Edit.
```

Add the openscad directory to the list

Makefile example

The `-d` and `-m` options only make sense together. (`-m` without `-d` would not consider modified dependencies when building exports, `-d` without `-m` would require the files to be already built for the first run that generates the dependencies.)

Here is an example of a basic Makefile that creates an .stl file from an .scad file of the same name:

```
# explicit wildcard expansion suppresses errors when no files are found
include $(wildcard *.deps)

%.stl: %.scad
	openscad -m make -o $@ -d $@.deps $<
```

When `make my_example.stl` is run for the first time, it finds no .deps files, and will just depend on `my_example.scad`; since `my_example.stl` is not yet preset, it will be created unconditionally. If OpenSCAD finds missing files, it will call `make` to build them, and it will list all used files in `my_example.stl.deps`.

When `make my_example.stl` is called subsequently, it will find and include `my_example.stl.deps` and check if any of the files listed there, including `my_example.scad`, changed since `my_example.stl` was built, based on their time stamps. Only if that is the case, it will build `my_example.stl` again.

Automatic targets

When building similar .stl files from a single .scad file, there is a way to automate that too:

```
# match "module foobar() { // `make` me"
TARGETS=$(shell sed '/^module [a-z0-9_-]*().*make..\?me.*$$/!d;s/module //;s/().*/.stl/' base.scad)

all: ${TARGETS}

# auto-generated .scad files with .deps make make re-build always. keeping the
# scad files solves this problem. (explanations are welcome.)
.SECONDARY: $(shell echo "${TARGETS}" | sed 's/\.stl/.scad/g')

# explicit wildcard expansion suppresses errors when no files are found
include $(wildcard *.deps)

%.scad:
	echo -n 'use <base.scad>\n$*();' > $@

%.stl: %.scad
	openscad -m make -o $@ -d $@.deps $<
```

All objects that are supposed to be exported automatically have to be defined in `base.scad` in an own module with their future file name (without the ".stl"), and have a comment like "`// make me`" in the line of the module definition. The "`TARGETS=`" line picks these out of the base file and creates the file names. These will be built when `make all` (or `make`, for short) is called.

As the convention from the last example is to create the .stl files from .scad files of the same base name, for each of these files, an .scad file has to be generated. This is done in the "`%.scad:`" paragraph; `my_example.scad` will be a very simple OpenSCAD file:

```
use <base.scad>
my_example();
```

The "`.SECONDARY`" line is there to keep `make` from deleting the generated .scad files. If it deleted it, it would not be able to automatically determine which files need no rebuild any more; please post ideas about what exactly goes wrong there (or how to fix it better) on the talk page!

Windows notes

On Windows, openscad.com should be called from the command line as a wrapper for openscad.exe. This is because Openscad uses the 'devenv' solution to the Command-Line/GUI output issue. Typing 'openscad' at the cmd.exe prompt will, by default, call the .com program wrapper.

MacOS notes

On MacOS the binary is normally hidden inside the App folder. If OpenSCAD is installed in the global Applications folder, it can be called from command line like in the following example that just shows the OpenSCAD version:

```
macbook:/$ /Applications/OpenSCAD.app/Contents/MacOS/OpenSCAD -v
OpenSCAD version 2013.06
```

Chapter 7 -- Building OpenSCAD from Sources

OpenSCAD User Manual

Prebuilt binary packages

If you are lucky, you won't have to build it. Many Linux and BSD systems have pre-built OpenSCAD packages including Debian, Ubuntu, Fedora, Arch, NetBSD and OpenBSD. Check your system's package manager for details.

For Ubuntu systems you can also try chrysn's Ubuntu packages at his launchpad PPA (https://launchpad.net/~chrysn/+archive/openscad), or you can just copy/paste the following onto the command line:

```
sudo add-apt-repository ppa:chrysn/openscad
sudo apt-get update
sudo apt-get install openscad
```

His repositories for OpenSCAD and OpenCSG are here (http://archive.amsuess.com/pool/contrib/o/openscad/) and here (http://archive.amsuess.com/pool/main/o/opencsg/).

There is also a generic linux binary package at http://www.openscad.org that can be unpacked and run from within most linux systems. It is self contained and includes the required libraries.

Building OpenSCAD yourself

If you wish to build OpenSCAD for yourself, start by installing ***git*** on your system using your package manager. Git is sometimes packaged under the name 'scmgit' or 'git-core'. Then, use git to download the OpenSCAD source code

```
cd ~/
git clone https://github.com/openscad/openscad.git
cd openscad
```

Then get the MCAD library, which is now included with OpenSCAD binary distributions

```
git submodule init
git submodule update
```

Installing dependencies

OpenSCAD uses a large number of third-party libraries and tools. These are called dependencies. An up to date list of dependencies can usually be found in the README.md in openscad's main directory, here: https://github.com/openscad/openscad/ A brief list follows:

Eigen, GCC or Clang, Bison, Flex, CGAL, Qt, GMP, MPFR, boost, cmake, OpenCSG, GLEW, QScintilla, glib2, harfbuzz, freetype2, pkg-config, fontconfig

Prepackaged dependencies

Most systems are set up to install pre-built dependencies using a 'package manager', such as ***apt*** on ubuntu or ***pacman*** on Arch Linux. OpenSCAD comes with a 'helper script' that will attempt to automatically run your package manager for you and download and install these pre-built packages if they exist. Note you must be running as root and/or using *sudo* to try this. Note that these scripts will likely fail on Sun, Solaris, AIX, IRIX, etc (skip to the 'building dependencies' section below).

```
./scripts/uni-get-dependencies.sh
```

Verifying dependencies

After attempting to install dependencies, you should double check them. Exit any shells and perhaps reboot.

Now verify that the version numbers are up to those listed in openscad/README.md file. Also verify that no packages were accidentally missed. For example open a shell and run 'flex --version' or 'gcc --version'. These are good sanity checks to make sure your environment is proper.

OpenSCAD comes with another helper script that attempts to automate this process on many Linux and BSD systems (Again, it won't work on Sun/Solaris/Irix/AIX/etc).

```
./scripts/check-dependencies.sh
```

If you cannot verify that your dependencies are installed properly and of a sufficient version, then you may have to install some 'by hand' (see the section below on building your own dependencies).

If your system has all the proper versions of dependencies, then continue to the 'Building OpenSCAD' section.

Building the dependencies yourself

On systems that lack updated dependency libraries or tools, you will need to download each and build it and install it by hand. You can do this by downloading and following installation instructions for each package separately. However OpenSCAD comes with scripts that attempt to automate this process on Linux and BSD systems, by installing everything into a folder created under $HOME/openscad_deps. This script will not build typical development dependencies like X11, Qt4, gcc, bash etc. But it will attempt things like OpenCSG, CGAL, boost, etc.

To run the automated script, first set up the environment variables (if you don't use bash, replace "source" with a single ".")

```
source scripts/setenv-unibuild.sh
```

Then, run a second script to download and build.

```
./scripts/uni-build-dependencies.sh
```

(If you only need CGAL or OpenCSG, you can just run ' ./scripts/uni-build-dependencies.sh cgal' or opencsg and it will only build a single library.)

The complete download and build process can take several hours, depending on your network connection speed and system speed. It is recommended to have at least 2 Gigabyte of free disk space to do the full dependency build. Each time you log into a new shell and wish to re-compile OpenSCAD you need to re-run the 'source scripts/setenv-unibuild.sh' script.

After completion, return to the section above on 'verifying dependencies' to see if they installed correctly.

Build the OpenSCAD binary

Once you have installed your dependencies, you can build OpenSCAD.

```
qmake       # or qmake-qt4, depending on your distribution
make
```

You can also install OpenSCAD to /usr/local/ if you wish. The 'openscad' binary will be put under /usr/local/bin, the libraries and examples will be under something like /usr/local/share/openscad possibly depending on your system. Note that if you have previously installed a binary linux package of openscad, you should take care to delete /usr/local/lib/openscad and /usr/local/share/openscad because they are not the same paths as what the standard qmake-built 'install' target uses.

```
sudo make install
```

> **Note:** on Debian-based systems create a package and install OpenSCAD using:
>
> ```
> sudo checkinstall -D make install
> ```

If you prefer not to install you can run "`./openscad`" directly whilst still in the ~/openscad directory.

Compiling the test suite

OpenSCAD comes with over 740 regression tests. To build and run them, it is recommended to first build the GUI version of OpenSCAD by following the steps above, including the downloading of MCAD. Then, from the same login, run these commands:

```
cd tests
mkdir build && cd build
cmake ..
make
ctest -C All
```

The file 'openscad/doc/testing.txt' has more information. Full test logs are under `tests/build/Testing/Temporary`. A pretty-printed index.html web view of the tests can be found under a machine-specific subdirectory thereof and opened with a browser.

Troubleshooting

If you encounter any errors when building, please file an issue report at https://github.com/openscad/openscad/issues/ .

Errors about incompatible library versions

This may be caused by old libraries living in /usr/local/lib like boost, CGAL, OpenCSG, etc, (often left over from previous experiments with OpenSCAD). You are advised to remove them. To remove, for example, CGAL, run rm -rf /usr/local/include/CGAL && rm -rf /usr/local/lib/*CGAL*. Then erase $HOME/openscad_deps, remove your openscad source tree, and restart fresh. As of 2013 OpenSCAD's build process does not advise nor require anything to be installed in /usr/local/lib nor /usr/local/include.

Note that CGAL depends on Boost and OpenCSG depends on GLEW - interdependencies like this can really cause issues if there are stray libraries in unusual places.

Another source of confusion can come from running from within an 'unclean shell'. Make sure that you don't have LD_LIBRARY_PATH set to point to any old libraries in any strange places. Also don't mix a Mingw windows cross build with your linux build process - they use different environment variables and may conflict.

OpenCSG didn't automatically build

If for some reason the recommended build process above fails to work with OpenCSG, please file an issue on the OpenSCAD github. In the meantime, you can try building it yourself.

```
wget http://www.opencsg.org/OpenCSG-1.3.2.tar.gz
sudo apt-get purge libopencsg-dev libopencsg1 # or your system's equivalent
tar -xvf OpenCSG-1.3.2.tar.gz
cd OpenCSG-1.3.2
# edit the Makefile and remove 'example'
make
sudo cp -d lib/lib* $HOME/openscad_deps/lib/
sudo cp include/opencsg.h $HOME/openscad_deps/include/
```

> **Note:** on Debian-based systems (such as Ubuntu), you can add the 'install' target to the OpenCSG Makefile, and then use checkinstall to create a clean .deb package for install/removal/upgrade. Add this target to Makefile:
>
> ```
> install:
>    # !! THESE LINES PREFIXED WITH ONE TAB, NOT SPACES !!
>    cp -d lib/lib* /usr/local/lib/
>    cp include/opencsg.h /usr/local/include/
>    ldconfig
> ```
>
> Then:
>
> ```
> sudo checkinstall -D make install
> ```
>
> .. to create and install a clean package.

CGAL didn't automatically build

If this happens, you can try to compile CGAL yourself. It is recommended to install to $HOME/openscad_deps and otherwise follow the build process as outlined above.

Compiling fails with an Internal compiler error from GCC or GAS

This can happen if you run out of virtual memory, which means all of physical RAM as well as virtual swap space from the disk. See below under "horribly slow" for reasons. If you are non-root, there are a few things you can try. The first is to use the 'clang' compiler, as it uses much less RAM than gcc. The second thing is to edit the Makefile and remove the '-g' and '-pipe' flags from the compiler flags section.

If, on the other hand, you are root, then you can expand your swap space. On Linux this is pretty standard procedure and easily found in a web search. Basically you do these steps (after verifying you have no file named /swapfile already):

```
sudo dd if=/dev/zero of=/swapfile bs=1M count=2000  # create a roughly 2 gig swapfile
sudo chmod 0600 /swapfile # set proper permissions for security
sudo mkswap /swapfile  # format as a swapfile
sudo swapon /swapfile  # turn on swap
```

For permanent swap setup in /etc/fstab, instructions are easily found through web search. If you are building on an SSD (solid state drive) machine the speed of a swapfile will allow a very reasonable build time.

Compiling is horribly slow and/or grinds the disk

It is recommended to have at least 1.5 Gbyte of RAM to compile OpenSCAD. There are a few workarounds in case you don't. The first is to use the experimental support for the Clang Compiler (described below) as Clang uses much less RAM than GCC. Another workaround is to edit the Makefile generated by qmake and search/replace the optimization flags (-O2) with -O1 or blank, and to remove any '-g' debug flags from the compiler line, as well as '-pipe'.

If you have plenty of RAM and just want to speed up the build, you can try a paralell multicore build with

```
make -jx
```

Where 'x' is the number of cores you want to use. Remember you need x times the amount of RAM to avoid possible disk thrashing.

The reason the build is slow is because OpenSCAD uses template libraries like CGAL, Boost, and Eigen, which use large amounts of RAM to compile - especially CGAL. GCC may take up 1.5 Gigabytes of RAM on some systems during the build of certain CGAL modules. There is more information at StackOverflow.com (http://stackoverflow.com/questions/3634203/why-are-templates-so-slow-to-compile).

BSD issues

The build instructions above are designed to work unchanged on FreeBSD and NetBSD. However the BSDs typically require special environment variables set up to build any QT project - you can set them up automatically by running

```
source ./scripts/setenv-unibuild.sh
```

NetBSD 5.x, requires a patched version of CGAL. It is recommended to upgrade to NetBSD 6 instead as it has all dependencies available from pkgin. NetBSD also requires the X Sets to be installed when the system was created (or added later (http://ghantoos.org/2009/05/12/my-first-shot-of-netbsd/)).

On OpenBSD it may fail to build after running out of RAM. OpenSCAD requires at least 1 Gigabyte to build with GCC. You may have need to be a user with 'staff' level access or otherwise alter required system parameters. The 'dependency build' sequence has also not been ported to OpenBSD so you will have to rely on the standard OpenBSD system package tools (in other words you have to have root).

Sun / Solaris / IllumOS / AIX / IRIX / Minix / etc

The OpenSCAD dependency builds have been mainly focused on Linux and BSD systems like Debian or FreeBSD. The 'helper scripts' likely will fail on other types of Un*x. Furthermore the OpenSCAD build system files (qmake .pro files for the GUI, cmake CMakeFiles.txt for the test suite) have not been tested thoroughly on non-Linux non-BSD systems. Extensive work may be required to get a working build on such systems.

Test suite problems

Headless server

The test suite will try to automatically detect if you have an X11 DISPLAY environment variable set. If not, it will try to automatically start Xvfb or Xvnc (virtual X framebuffers) if they are available.

If you want to run these servers manually, you can attempt the following:

```
$ Xvfb :5 -screen 0 800x600x24 &
$ DISPLAY=:5 ctest
```

Alternatively:

```
$ xvfb-run --server-args='-screen 0 800x600x24' ctest
```

There are some cases where Xvfb/Xvnc won't work. Some older versions of Xvfb may fail and crash without warning. Sometimes Xvfb/Xvnc have been built without GLX (OpenGL) support and OpenSCAD won't be able to generate any images.

Image-based tests takes a long time, they fail, and the log says 'return -11'

Imagemagick may have crashed while comparing the expected images to the test-run generated (actual) images. You can try using the alternate ImageMagick comparison method by by erasing CMakeCache, and re-running cmake with `-DCOMPARATOR=ncc`. This will enable the Normalized Cross Comparison method which is more stable, but possibly less accurate and may give false positives or negatives.

Testing images fails with 'morphology not found" for ImageMagick in the log

Your version of imagemagick is old. Upgrade imagemagick, or pass -DCOMPARATOR=old to cmake. The comparison will be of lowered reliability.

I moved the dependencies I built and now openscad won't run

It isn't advised to move them because the build is using RPATH hard coded into the openscad binary. You may try to workaround by setting the LD_LIBRARY_PATH environment variable to place yourpath/lib first in the list of paths it searches. If all else fails, you can re-run the entire dependency build process but export the BASEDIR environment variable to your desired location, before you run the script to set environment variables.

Tricks and tips

Reduce space of dependency build

After you have built the dependencies you can free up space by removing the $BASEDIR/src directory - where $BASEDIR defaults to $HOME/openscad_deps.

Preferences

OpenSCAD's config file is kept in ~/.config/OpenSCAD/OpenSCAD.conf.

Setup environment to start developing OpenSCAD in Ubuntu 11.04

The following paragraph describes an easy way to setup a development environment for OpenSCAD in Ubuntu 11.04. After executing the following steps QT Creator can be used to graphically start developing/debugging OpenSCAD.

- Add required PPA repositories:

```
# sudo add-apt-repository ppa:chrysn/openscad
```

- Update and install required packages:

```
# sudo apt-get update
# sudo apt-get install git build-essential qtcreator libglew1.5-dev libopencsg-dev libcgal-dev libeigen2-dev bison flex
```

- Get the OpenSCAD sources:

```
# mkdir ~/src
# cd ~/src
# git clone https://github.com/openscad/openscad.git
```

- Build OpenSCAD using the command line:

```
# cd ~/src/openscad
# qmake
# make
```

- Build OpenSCAD using QT Creator:

Just open the project file openscad.pro (CTRL+O) in QT Creator and hit the build all (CTRL+SHIFT+B) and run button (CTRL+R).

The Clang Compiler

There is experimental support for building with the Clang compiler under linux. Clang is faster, uses less RAM, and has different error messages than GCC. To use it, first of all you will need CGAL of at least version 4.0.2, as prior versions have a bug that makes clang unusable. Then, run this script before you build OpenSCAD.

```
source scripts/setenv-unibuild.sh clang
```

Clang support depends on your system's QT installation having a clang enabled qmake.conf file. For example, on Ubuntu, this is under /usr/share/qt4/mkspecs/unsupported/linux-clang/qmake.conf. BSD clang-building may require a good deal of fiddling and is untested, although eventually it is planned to move in this direction as the BSDs (not to mention OSX) are moving towards favoring clang as their main compiler. OpenSCAD includes convenience scripts to cross-build Windows installer binaries using the MXE system (http://mxe.cc). If you wish to use them, you can first install the MXE Requirements such as cmake, perl, scons, using your system's package manager (click to view a complete list of requirements) (http://mxe.cc/#requirements). Then you can perform the following commands to download OpenSCAD source and build a windows installer:

```
  git clone https://github.com/openscad/openscad.git
  cd openscad
  source ./scripts/setenv-mingw-xbuild.sh
  ./scripts/mingw-x-build-dependencies.sh
  ./scripts/release-common.sh mingw32
```

The x-build-dependencies process takes several hours, mostly to cross-build QT. It also requires several gigabytes of disk space. If you have multiple CPUs you can speed up things by running **export NUMCPU=x** before running the dependency build script. By default it builds the dependencies in $HOME/openscad_deps/mxe. You can override the mxe installation path by setting the BASEDIR environment variable before running the scripts. The OpenSCAD binaries are built into a separate build path, openscad/mingw32.

Note that if you want to then build linux binaries, you should log out of your shell, and log back in. The 'setenv' scripts, as of early 2013, required a 'clean' shell environment to work.

If you wish to cross-build manually, please follow the steps below and/or consult the release-common.sh source code.

Setup

The easiest way to cross-compile OpenSCAD for Windows on Linux or Mac is to use mxe (M cross environment). You will need to install git to get it. Once you have git, navigate to where you want to keep the mxe files in a terminal window and run:

```
git clone git://github.com/mxe/mxe.git
```

Add the following line to your `~/.bashrc` file:

```
export PATH=/<where mxe is installed>/usr/bin:$PATH
```

replacing `<where mxe is installed>` with the appropriate path.

Requirements

The requirements to cross-compile for Windows are just the requirements of mxe. They are listed, along with a command for installing them here (http://mxe.cc/#requirements). You don't need to type 'make'; this will make everything and take up >10 GB of diskspace. You can instead follow the next step to compile only what's needed for openscad.

Now that you have the requirements for mxe installed, you can build OpenSCAD's dependencies (CGAL, Opencsg, MPFR, and Eigen2). Just open a terminal window, navigate to your mxe installation and run:

```
make mpfr eigen opencsg cgal qt
```

This will take a few hours, because it has to build things like gcc, qt, and boost. Just go calibrate your printer or something while you wait. To speed things up, you might want do something like "make -j 4 JOBS=2" for parallel building. See the mxe tutorial (http://mxe.c

c/#tutorial) for more details.

Optional: If you want to build an installer, you need to install the nullsoft installer system. It should be in your package manager, called "nsis".

Build OpenSCAD

Now that all the requirements have been met, all that remains is to build OpenSCAD itself. Open a terminal window and enter:

```
git clone git://github.com/openscad/openscad.git
cd openscad
```

Then get MCAD:

```
git submodule init
git submodule update
```

You need to create a symbolic link here for the build system to find the libraries:

```
ln -s /<where mxe is installed>/usr/i686-pc-mingw32/ mingw-cross-env
```

again replacing `<where mxe is installed>` with the appropriate path

Now to build OpenSCAD run:

```
i686-pc-mingw32-qmake CONFIG+=mingw-cross-env openscad.pro
make
```

When that is finished, you will have openscad.exe in ./release and you can build an installer with it as described in the instructions for building with Microsoft Visual C++, described here (http://en.wikibooks.org/wiki/OpenSCAD_User_Manual/Building_on_Windows#Building_an_installer).

The difference is that instead of right-clicking on the *.nsi file you will run:

```
makensis installer.nsis
```

Note that as of early 2013, OpenSCAD's 'scripts/release-common.sh' automatically uses the version of nsis that comes with the MXE cross build system, so you may wish to investigate the release-common.sh source code to see how it works, if you have troubles. This is a set of instructions for building OpenSCAD with the Microsoft Visual C++ compilers. It has not been used since circa 2012 and is unlikely to work properly. It is maintained here for historical reference purposes.

A newer build is being attempted with the Msys2 system, please see
http://en.wikibooks.org/wiki/OpenSCAD_User_Manual/Building_on_Microsoft_Windows

---

This MSVC build is as static as reasonable, with no external DLL dependencies that are not shipped with Windows

Note: It was last tested on the Dec 2011 build. Newer checkouts of OpenSCAD may not build correctly or require extensive modification to compile under MSVC. OpenSCAD releases of 2012 were typically cross-compiled from linux using the Mingw & MXE system. See Cross-compiling for Windows on Linux or Mac OS X.

Downloads

start by downloading:

- Visual Studio Express http://download.microsoft.com/download/E/8/E/E8EEB394-7F42-4963-A2D8-29559B738298/VS2008ExpressWithSP1ENUX1504728.iso
- QT (for vs2008) http://get.qt.nokia.com/qt/source/qt-win-opensource-4.7.2-vs2008.exe
- git http://msysgit.googlecode.com/files/Git-1.7.4-preview20110204.exe
- glew https://sourceforge.net/projects/glew/files/glew/1.5.8/glew-1.5.8-win32.zip/download
- cmake http://www.cmake.org/files/v2.8/cmake-2.8.4-win32-x86.exe
- boost http://www.boostpro.com/download/boost_1_46_1_setup.exe
- cgal https://gforge.inria.fr/frs/download.php/27647/CGAL-3.7-Setup.exe

- OpenCSG http://www.opencsg.org/OpenCSG-1.3.2.tar.gz
- eigen2 http://bitbucket.org/eigen/eigen/get/2.0.15.zip
- gmp/mpfr http://holoborodko.com/pavel/downloads/win32_gmp_mpfr.zip
- MinGW http://netcologne.dl.sourceforge.net/project/mingw/Automated%20MinGW%20Installer/mingw-get-inst/mingw-get-inst-20110316/mingw-get-inst-20110316.exe

Installing

- Install Visual Studio
  - No need for siverlight or mssql express
  - You can use a virtual-CD program like MagicDisc to mount the ISO file and install without using a CD
- Install QT
  - Install to default location `C:\Qt\4.7.2\`
- Install Git
  - Click `Run Git and included Unix tools from the Windows Command Prompt` despite the big red letters warning you not to.
- Install Cmake
  - Check the 'Add cmake to the system path for the current user' checkbox
  - Install to default location `C:\Program Files\CMake 2.8`
- Install Boost
  - Select the VC++ 9.0 vs2008 radio
  - Check the 'multithreaded static runtime' checkbox only
  - Install into `C:\boost_1_46_1\`
- Install CGAL
  - Note - CGAL 3.9 fixes several bugs in earlier versions of CGAL, but CGAL 3.9 will not compile under MSVC without extensive patching. Please keep that in mind when compiling OpenSCAD with MSVC - there may be bugs due to the outdated version of CGAL required to use MSVC.
  - Note its not a binary distribution, just an installer that installs the source.
  - No need for CGAL Examples and Demos
  - Make sure mpfr and gmp precompiled libs is checked
  - The installer wants you to put this in `C:\Program Files\CGAL-3.7\` I used `C:\CGAL-3.7\`
  - Make sure CGAL_DIR environment checked.
- Install MinGW
  - Make sure you select the MSYS Basic System under components
- Extract downloaded win32_gmp_mpfr.zip file to `C:\win32_gmp_mpfr\`
- Replace the mpfr and gmp .h files in CGAL with the ones from win32_gmp_mpfr
  - Delete, or move to a temp folder, all files in `CGAL-3.7\auxiliary\gmp\include` folder
  - Copy all the .h files in `C:\win32_gmp_mpfr\gmp\Win32\Release` to `CGAL-3.7\auxiliary\gmp\include`
  - Copy all the .h files in `C:\win32_gmp_mpfr\mpfr\Win32\Release` to `CGAL-3.7\auxiliary\gmp\include`
- Replace the mpfr and gmp libs in CGAL with the ones from win32_gmp_mpfr
  - Delete, or move to a temp folder, all (06/20/2011 libmpfr-4.lib is needed 7/19/11 - i didnt need it) files in `CGAL-3.7\auxiliary\gmp\lib` folder.
  - Copy `C:\win32_gmp_mpfr\gmp\Win32\Release\gmp.lib` to `CGAL-3.7\auxiliary\gmp\lib`
  - Copy `C:\win32_gmp_mpfr\mpfr\Win32\Release\mpfr.lib` to `CGAL-3.7\auxiliary\gmp\lib`
  - Go into `CGAL-3.7\auxiliary\gmp\lib` and copy `gmp.lib` to `gmp-vc90-mt-s.lib`, and `mpfr.lib` to `mpfr-vc90-mt-s.lib` (so the linker can find them in the final link of openscad.exe)

To get OpenSCAD source code:

- Open "Git Bash" (or MingW Shell) (the installer may have put a shortcut on your desktop). This launches a command line window.
- Type **cd c:** to change the current directory.
- Type **git clone git://github.com/openscad/openscad.git** This will put OpenSCAD source into C:\openscad\

Where to put other files:

I put all the dependencies in C:\ so for example,

- C:\eigen2\
- C:\glew-1.5.8\
- C:\OpenCSG-1.3.2\

.tgz can be extracted with `tar -zxvf` from the MingW shell, or Windows tools like 7-zip. Rename and move sub-directories if needed. I.e eigen-eigen-0938af7840b0 should become c:\eigen2, with the files like COPYING and CMakeLists.txt directly under it. c:\glew-1.5.8 should have 'include' and 'lib' directly under it.

Compiling Dependencies

For compilation I use the QT Development Command Prompt

Start->Program Files->Qt by Nokia v4.7.2 (VS2008 OpenSource)->QT 4.7.2 Command Prompt

Qt

Qt needs to be recompiled to get a static C runtime build. To do so, open the command prompt and do:

```
configure -static -platform win32-msvc2008 -no-webkit
```

Configure will take several minutes to finish processing. After it is done, open up the file Qt\4.7.2\mkspecs\win32-msvc2008\qmake.conf and replace every instance of -MD with -MT. Then:

```
nmake
```

This takes a very, very long time. Have a nap. Get something to eat. On a Pentium 4, 2.8GHZ CPU with 1 Gigabyte RAM, Windows XP, it took more than 7 hours, (that was with -O2 turned off)

CGAL

```
cd C:\CGAL-3.7\
set BOOST_ROOT=C:\boost_1_46_1\
cmake .
```

Now edit the `CMakeCache.txt` file. Replace every instance of `/MD` with `/MT` . Now, look for a line like this:

`CMAKE_BUILD_TYPE:STRING=Debug`

Change `Debug` to `Release`. Now *re-run* cmake

```
cmake .
```

It should scroll by, watch for lines saying `"--Building static libraries"` and `"--Build type: Release"` to confirm the proper settings. Also look for `/MT` in the `CXXFLAGS` line. When it's done, you can do the build:

```
nmake
```

You should now have a `CGAL-vc90-mt-s.lib` file under `C:\CGAL-3.7\lib` . If not, see Troubleshooting, below.

OpenCSG

Launch Visual Express.

```
cd C:\OpenCSG-1.3.2
vcexpress OpenCSG.sln
Substitute devenv for vcexpress if you are not using the express version
```

- Manually step through project upgrade wizard
- Make sure the runtime library settings for all projects is for Release (not Debug)
  - Click Build/Configuration Manager
  - Select "Release" from "Configuration:" drop down menu
  - Hit Close
- Make sure the runtime library setting for OpenCSG project is set to multi-threaded static
  - Open the OpenCSG project properties by clicking menu item "Project->OpenCSG Properties" (might be just "Properties")
  - Make sure it says "Active(Release)" in the "Configuration:" drop down menu
  - Click 'Configuration Properties -> C/C++ -> Code Generation'
  - Make sure "Runtime Library" is set to "Multi-threaded (/MT)"
  - Click hit OK
- Make sure the runtime library setting for glew_static project is set to multi-threaded static
  - In "Solution Explorer - OpenCSG" pane click "glew_static" project
  - Open the OpenCSG project properties by clicking menu item "Project->OpenCSG Properties" (might be just "Properties")
  - Make sure it says "Active(Release)" in the "Configuration:" drop down menu
  - Click C/C++ -> Code Generation
  - Make sure "Runtime Library" is set to "Multi-threaded (/MT)"

    - Click hit OK
- Close Visual Express saving changes

Build OpenCSG library. You can use the GUI Build/Build menu (the Examples project might fail, but glew and OpenCSG should succeed). Alternatively you can use the command line:

```
cmd /c vcexpress OpenCSG.sln /build
Again, substitute devenv if you have the full visual studio
```

The `cmd /c` bit is needed otherwise you will be returned to the shell immediately and have to Wait for build process to complete (there will be no indication that this is happening appart from in task manager)

OpenSCAD

- Bison/Flex: Open the mingw shell and type `mingw-get install msys-bison`. Then do the same for flex: `mingw-get install msys-flex`
- Open the QT Shell, and copy/paste the following commands

```
cd C:\openscad
set INCLUDE=%INCLUDE%C:\CGAL-3.7\include;C:\CGAL-3.7\auxiliary\gmp\include;
set INCLUDE=%INCLUDE%C:\boost_1_46_1;C:\glew-1.5.8\include;C:\OpenCSG-1.3.2\include;C:\eigen2
set LIB=%LIB%C:\CGAL-3.7\lib;C:\CGAL-3.7\auxiliary\gmp\lib;
set LIB=%LIB%C:\boost_1_46_1\lib;C:\glew-1.5.8\lib;C:\OpenCSG-1.3.2\lib
qmake
nmake -f Makefile.Release
```

Wait for the nmake to end. There are usually a lot of non-fatal warnings about the linker. On success, there will be an openscad.exe file in the release folder. Enjoy.

Building an installer

- Download and install NSIS from http://nsis.sourceforge.net/Download
- Put the FileAssociation.nsh macro from http://nsis.sourceforge.net/File_Association in the NSIS Include directory, C:\Program Files\NSIS\Include
- Run 'git submodule init' and 'git submodule update' to download the MCAD system (https://github.com/elmom/MCAD) into the openscad/libraries folder.
- Copy the OpenSCAD "libraries" and "examples" directory into the "release" directory
- Copy OpenSCAD's "scripts/installer.nsi" to the "release" directory.
- Right-click on the file and compile it with NSIS. It will spit out a nice, easy installer. Enjoy.

Compiling the regression tests

- Follow all the above steps, build openscad, run it, and test that it basically works.
- Install Python 2.x (not 3.x) from http://www.python.org
- Install Imagemagick from http://www.imagemagick.org
- read openscad\docs\testing.txt
- Go into your QT shell

```
set PATH=%PATH%;C:\Python27 (or your version of python)
cd c:\openscad\tests\
cmake . -DCMAKE_BUILD_TYPE=Release
Edit the CMakeCache.txt file, search/replace /MD to /MT
cmake .
nmake -f Makefile
```

- This should produce a number of test .exe files in your directory. Now run

```
ctest
```

If you have link problems, see Troubleshooting, below.

Troubleshooting

Linker errors

If you have errors during linking, the first step is to improve debug logging, and redirect to a file. Open Openscad.pro and uncomment this line:

```
QMAKE_LFLAGS   += -VERBOSE
```

Now rerun

```
nmake -f Makefile.Release > log.txt
```

You can use a program like 'less' (search with '/') or wordpad to review the log.

To debug these errors, you must understand basics about Windows linking. Windows links to its standard C library with basic C functions like malloc(). But there are four different ways to do this, as follows:

```
compiler switch - type - linked runtime C library
/MT - Multithreaded static Release - link to LIBCMT.lib
/MTd - Multithreaded static Debug - link to LIBCMTD.lib
/MD - Multithreaded DLL Release - link to MSVCRT.lib (which itself helps link to the DLL)
/MDd - Multithreaded DLL Debug - link to MSVCRTD.lib (which itself helps link to the DLL)
```

All of the libraries that are link together in a final executable must be compiled with the same type of linking to the standard C library. Otherwise, you get link errors like, "LNK2005 - XXX is already defined in YYY". But how can you track down which library wasn't linked properly? 1. Look at the log, and 2. dumpbin.exe

dumpbin.exe

dumpbin.exe can help you determine what type of linking your .lib or .obj files were created with. For example, `dumpbin.exe /all CGAL.lib | find /i "DEFAULTLIB"` will give you a list of DEFAULTLIB symbols inside of CGAL.lib. Look for LIBCMT, LIBCMTD, MSVCRT, or MSVCRTD. That will tell you, according to the above table, whether it was built Static Release, Static Debug, DLL Release, or DLL Debug. (DLL, of course means Dynamic Link Library in this conversation.) This can help you track down, for example, linker errors about conflicting symbols in LIBCMT and LIBCMTD.

dumpbin.exe can also help you understand errors involving unresolved external symbols. For example, if you get an error about unresolved external symbol `___GLEW_NV_occlusion_query`, but your VERBOSE error log says the program linked in glew32.lib, then you can `dumpbin.exe /all glew32.lib | find /i "occlusion"` to see if the symbol is actually there. You may see a mangled symbol, with __impl, which gives you another clue with which you can google. In this particular example, glew32s.lib (s=static) should have been linked instead of glew32.lib.

CGAL

CGAL-vc90-mt-s.lib

After compilation, it is possible that you might get a file named `CGAL-vc90-mt.lib` or `CGAL-vc90-mt-gd.lib` instead of `CGAL-vc90-mt-s.lib`. There are many possibilities: you accidentally built the wrong version, or you may have built the right version and VCExpress named it wrong. To double check, and fix the problem, you can do the following:

```
cd C:\CGAL-3.7\lib
dumpbin /all CGAL-vc90-mt.lib | find /i "DEFAULTLIB"
(if you have mt-gd, use that name instead)
```

If this shows lines referencing `LIBCMTD, MSVCRT, or MSVCRTD` then you accidentally built the debug and/or dynamic version, and you need to clean the build, and try to build again with proper settings to get the *multi-threaded static release* version. However, if it just says `LIBCMT`, then you are probably OK. Look for another line saying `DEFAULTLIB:CGAL-vc90-mt-s`. If it is there, then you can probably just rename the file and have it work.

```
move CGAL-vc90-mt.lib CGAL-vc90-mt-s.lib
```

Visual Studio build

You can build CGAL using the GUI of visual studio, as an alternative to nmake. You have to use an alternate cmake syntax. Type 'cmake' by itself and it will give you a list of 'generators' that are valid for your machine; for example Visual Studio Express is `cmake -G"Visual Studio 9 2008" ..`. That should get you a working `.sln` (solution) file.

Then run this:

```
vcexpress CGAL.sln
```

Modify the build configure target to Release (not Debug) and change the properties of the projects to be '/MT' multithreaded static builds. This is the similar procedure used to build OpenCSG, so refer to those instructions above for more detail.

Note for Unix users

The 'MingW Shell' (Start/Programs) provide tools like bash, sed, grep, vi, tar, &c. The C:\ drive is under '/c/'. MingW has packages, for example: `mingw-get install msys-unzip` downloads and installs the 'unzip' program. Git contains some programs by default, like perl. The windows command shell has cut/paste - hit `alt-space`. You can also change the scrollback buffer settings.

References

- Windows Building, OpenSCAD mailing list, 2011 May (http://rocklinux.net/pipermail/openscad/2011-May/thread.html).
- C Run-Time Libraries linking (http://msdn.microsoft.com/en-us/library/abx4dbyh(v=vs.80).aspx), Microsoft.com for Visual Studio 8 (The older manual is good too, here (http://msdn.microsoft.com/en-us/library/aa278396(VS.60).aspx))
- old nabble (http://old.nabble.com/flex-2.5.35-1:-isatty()-problem-(and-solution)-td17659695.html) on _isatty, flex
- Windows vs. Unix: Linking dynamic load modules (http://xenophilia.org/winvunix.html) by Chris Phoenix
- Static linking in CMAKE under MS Visual C (http://www.cmake.org/Wiki/CMake_FAQ#How_can_I_build_my_MSVC_application_with_a_static_runtime.3F) (cmake.org)
- __imp , declspec(dllimport), and unresolved references (http://stackoverflow.com/questions/3704374/linking-error-lnk2019-in-msvc-unresolved-symbols-with-imp-prefix-but-should) (stackoverflow.com)

For building OpenSCAD, see https://github.com/openscad/openscad/blob/master/README.md

For making release binaries, see http://svn.clifford.at/openscad/trunk/doc/checklist-macosx.txt

Chapter 8 -- Frequently Asked Questions

OpenSCAD User Manual

General

How is OpenSCAD pronounced?

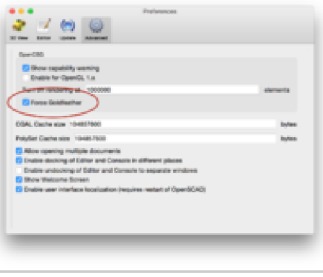
The intended pronounciation is: **Open - ESS - CAD**

Display

Preview doesn't appear to work at all

Some systems, in particular Intel GPUs on Windows, tend to have old or broken OpenGL drivers. This affects preview rendering when using difference or intersection operators.

The following tends to improve the situation: Edit->Preferences->Advanced->Force Goldfeather (see screenshot).



Force Goldfeather

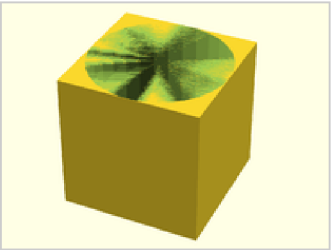
What are those strange flickering artifacts in the preview?

This is typically caused by differencing objects that share one or more faces, e.g.:

```
difference() {
  cube(20, center = true);
  cylinder(r = 10, h = 20, center = true);
}
```

In most cases the final render will be fine, but it's recommended to make the cuts a little bit larger to prevent this type of issues.

See also: https://en.wikipedia.org/wiki/Z-fighting

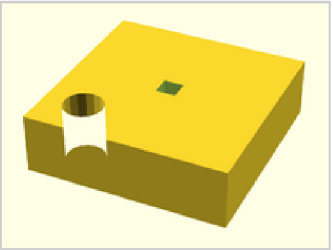


OpenSCAD display issue with coincident faces

Why are some parts (e.g. holes) of the model not rendered correctly?

This can happen when using features like `linear_extrude()` or when importing objects. The convexity of the objects is not known. For more complex objects, the `convexity` parameter can be used to specify the value. Note that higher values will cause a slowdown in preview.

```
difference() {
    linear_extrude(height = 15) {
        difference() {
            square([50, 50]);
            translate([10, 10]) circle(5);
        }
    }
    translate([25, 25]) cube([5, 5, 40], center = true);
}
```



OpenSCAD display issue with convexity setting too low

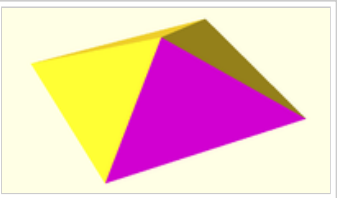
See also https://en.wikipedia.org/wiki/Convexity_%28mathematics%29

Why is my model showing up with F5 but not F6?

This can be caused by polyhedrons with flipped faces.

This can be visualized in "Thrown Together" display mode. See polygon orientation issues for details.

```
points = [[5,5,0],[5,-5,0],[-5,-5,0],[-5,5,0],[0,0,3]];
faces = [[0,1,4],[1,2,4],[2,3,4],[3,4,0],[1,0,3],[2,1,3]];
polyhedron(points, faces);
```



OpenSCAD polyhedron with flipped face

If the model imports external STL files, see also import related question.

Why is the preview so slow?

http://forum.openscad.org/Why-is-for-so-slow-tp11511p11531.html

This is hard to explain, but in essence, having intersections as the negative objects in difference is expensive. The preview rendering algorithm only allows having primitive objects as negatives, and everything else has to be unpacked.

For example (using `A+B` = `union()` / `A-B` = `difference()` / `A*B` = `intersection()`):

```
A - B*C - D*E
```

becomes: `A-B-D + A-B-E + A-C-D + A-C-E`

..and if A is more complex:

```
A+B - C*D - E*F
```

becomes: `A-C-E + A-C-F + A-D-E + A-D-F + B-C-E + B-C-F + B-D-E + B-D-F`

All combinations have to be rendered, which can take some time, especially on older GPUs, and especially on low-end Intel GPUs.

Import

Why is my imported STL file only showing up with F5 but not F6?

This is mostly caused by bad STL files, the best bet is to verify the STL file in a tool like Blender MeshLab or NetFabb and fix the issues. In essence the model needs to be manifold to be processed in OpenSCAD.

The reason for the model still showing up in preview mode is that there is no real geometry calculation going on yet. The preview simply draws the triangles from the STL.

There is one specific issue that causes problems called "Zero faces" (meaning the STL contains triangles with zero area because all 3 points are on one line) which are currently not handled well in OpenSCAD.

Using MeshLab

MeshLab has a filter to remove zero faces by flipping edges of polygons

```
Filters -> Cleaning and Repairing -> Remove T-Vertices by Edge-Flip.
```

Set the Ratio to a very high value (e.g. 1000000), otherwise it's possible the model gets distorted.

Using Blender

Blender has a 3D-Printing-Toolbox Plug-in (needs to be enabled in the UserSettings) which can show issues with the STL file. See http://wiki.blender.org/index.php/Extensions:2.6/Py/Scripts/Modeling/PrintToolbox

Using NetFabb/Microsoft cloud service

The Microsoft 3D Model Repair service can help with fixing STL files. Make sure to read the service conditions before posting files. See https://modelrepair.azurewebsites.net/

I'm getting "Unsupported DXF Entity" warnings when importing DXF files, what

does that mean?

DXF import sometimes produces warning messages like `Unsupported DXF Entity 'SPLINE' (1c1) in "file.dxf"`. This means the DXF file is using features that the OpenSCAD importer does not know how to handle. The importer will simply ignore those unknown entities which could result in an incomplete model.

When using Inkscape, the easiest way to produces DXF files without unsupported entities is to convert all Bezier curves to short line segments using

`Extensions -> Modify Path -> Flatten Beziers`

The value given in the dialog will determine the length of the line segments. Lower values will produce smoother results, but also much more line segments. As export file format, use "Desktop Cutting Plotter (AutoCAD DXF R14)".

A more detailed tutorial is available at http://repraprip.blogspot.de/2011/05/inkscape-to-openscad-dxf-tutorial.html

Export

How can I export multiple parts from one script?

Answer based on comments in related issue on github https://github.com/openscad/openscad/pull/1534#issuecomment-227024209

There is a way to generate a bunch of geometric primitives and export them as STL files from a single script, without commenting/uncommenting code.

There is a variable, PARTNO, that indicates which part is being exported in the current run. If PARTNO is 'undef', then nothing is exported.

```
PARTNO = undef; // default part number

module tree() {
  color("green") cylinder(r1 = 12, r2 = 1, h = 30);
  // ...
}

module trunk() {
  color("brown") cylinder(r = 3, h = 10);
  // ...
}

module base() {
  color("white") translate([-10, -10, 0]) cube([20, 20, 5]);
  // ...
}

if (PARTNO == 1) tree();
if (PARTNO == 2) trunk();
if (PARTNO == 3) base();

// optionally use 0 for whole object
if (PARTNO == 0) {
  base();
  translate([0, 0, 5]) trunk();
  translate([0, 0, 15]) tree();
}
```

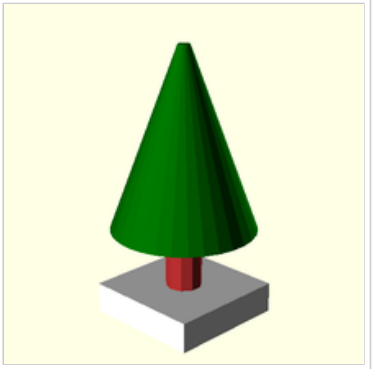


Image exported with PARTNO=0

When working interactively, the PARTNO variable at the top of the file can be set to the number of the part that will be shown/exported from the GUI.

It's possible to automate the process of exporting all of the parts by writing a shell script on MacOS or Linux, or a batch file on Windows. The shell script would look something like this:

```
# export parts as STL
openscad -DPARTNO=1 -o tree.stl model.scad
openscad -DPARTNO=2 -o trunk.stl model.scad
openscad -DPARTNO=3 -o base.stl model.scad

# export image of all the parts combined
openscad -DPARTNO=0 -o model.png model.scad
```

Running this script once from the command line will export all of the parts to separate files.

Language

Why am I getting an error when writing a = a + 1?

http://forum.openscad.org/A-A-1-tp11385p11411.html

First of all, the question **why** we have these "limitations" will become more clear once we start better exploiting the opportunities.

- We need a "reduce" function to help collecting information depending on a list of input. Recursion is fine, but people tend to struggle with it and we could offer some help.
- We should probable disallow any attempt of reassignment, to make it more clear what's going on. The only real reason we partially allow it is to allow cmd-line variable overrides.

To help think about things:

- Imagine every expression in OpenCAD being executed in parallel. Any dependency of existing expressions must be made explicit by hierarchical grouping. This will kill the idea of iterating in order to accumulate information.
- In terms of functions: Imagine a function expression being something you'd type into a spreadsheet cell. Not totally mappable, but it might help framing it.

Now, we *could* add all kinds of sugar to help people apply their existing programming problem solving skills. Question is more if it really helps us, secondary who will spearhead the design of such language extensions, as we currently don't really have attachment for these ideas on the dev-team.

If you think about the OpenSCAD language as something similar to HTML, but for 3D modeling, you'd still have a need for various programs generating code in this language (similar to the plethora of HTML generators out there). There exist a number of tools for helping with OpenSCAD code generation from existing programming languages (python, ruby, C++, haskell, clojure off the top of my head) and there are tools offering Javascript interfaces for similar purposes (OpenJSCAD, CoffeeSCAD). Until we have a really good reason to do so in OpenSCAD proper, and a really good candidate for which language to support, I think it's better to keep these things separate.

see also for help: List Comprehension

Recursive Functions (https://en.wikibooks.org/wiki/OpenSCAD_User_Manual/User-Defined_Functions_and_Modules#Recursive_functions)

User Interface

I'm not getting any menubar when running OpenSCAD in Ubuntu, how can I get it back?

This seems to be caused by Ubuntu messing with Qt to move the menubar somewhere else (e.g. top of the screen).

That problem hits other applications too, see https://bugs.launchpad.net/ubuntu/+source/appmenu-qt5/+bug/1307619

There are two things that could help:

- Set the `QT_QPA_PLATFORMTHEME` environment variable to an empty string (the default value is probably "appmenu-qt5") or simply run OpenSCAD with `QT_QPA_PLATFORMTHEME= openscad`
- Remove the `appmenu-qt5` package to globally disable menubar changes for all applications

Why are the error line numbers wrong?

That is a limitation/bug in the current parser that handles `include<>` basically as copy&paste of content. In some cases it's possible to work around the issue by placing the `include<>` statements at the end of the file.

When depending on libraries, it's recommended to use `use<>` instead which does not have that problem and also automatically inhibits any top-level geometry of that file (which might be there as demo for the library).

Errors

Why am I getting "no top level geometry to render"?

This can have different reasons, some common ones include

Missing / Commented out module call

```
module model() {
  cube(20);
}
%model();
```

Using the % modifier does not only make the part transparent, it's not included in the final render!

Difference / Intersection with wrong translated objects

The easiest way to solve this type of issues is to highlight the objects using the # modifier and see if the objects are placed at the position where they should be.

Importing broken STL files

See Why is my imported STL file only showing up with F5 but not F6?

Reporting bugs, Requesting features

How do I report bugs?

Bugs in OpenSCAD are best reported in the github (http://github.com/) issue tracking system at https://github.com/openscad/openscad/issues. If you are not sure it's a bug, asking on the mailing list/forum (http://www.openscad.org/community.html) can help clarifying things.

Please try searching through the existing issues if the bug was already reported. If you find something similar or if you are unsure, create a new issue, but mention the (possibly) related one.

The bug report should give as much information as possible to help with reproducing it, including but not limited to

- The OpenSCAD version
- The Operating System name and version
- A description of the scenario that produces the issue
- In case of graphics issues, the OpenGL driver information
- If possible, a trimmed down script reproducing the issue

Most of the technical version information can be found in menu `Help -> Library Info`.

How do I request new features?

New features or changes/extensions to existing features can be requested in the github (http://github.com/) issue tracking system at https://github.com/openscad/openscad/issues too.

Please make an effort to clearly explain the new feature / change as detailed as possible. Including some background about why you think this feature would be useful to you and other people helps a lot and increases the chances of it being implemented.

How do I report bugs that are related to the Operating System I use?

Windows

The Windows version is currently maintained by the OpenSCAD team, so please use the github issue tracker (https://github.com/openscad/openscad/issues) for reporting bugs.

Mac OSX

The Mac OSX version is currently maintained by the OpenSCAD team, so please use the github issue tracker (https://github.com/openscad/openscad/issues) for reporting bugs.

Linux

The OpenSCAD versions included in / distributed by the various Linux distributions are usually maintained by people/teams working with the distributions.

Specific bugs can be reported in the respective bug tracking systems, e.g.

- **Debian** - See "please report it" directions at https://bugs.debian.org/cgi-bin/pkgreport.cgi?package=openscad
- **Ubuntu** - See "Report a bug" directions at https://launchpad.net/ubuntu/+source/openscad
- **Fedora / Red Hat** - See https://apps.fedoraproject.org/packages/openscad/bugs and https://bugzilla.redhat.com/buglist.cgi?component=openscad
- **Arch Linux** - See "reporting bug guidelines" directions at https://bugs.archlinux.org/index.php?string=openscad&status%5B%5D=

The nightly builds (https://build.opensuse.org/package/show/home:t-paul/OpenSCAD) hosted on the openSUSE build service (https://build.opensuse.org/) are maintained by the OpenSCAD team, so please use the github issue tracker (https://github.com/openscad/openscad/issues) for reporting issues with those packages.

Chapter 9 -- Libraries

OpenSCAD User Manual

Library Locations

OpenSCAD uses three library *locations*, the installation library, built-in library, and user defined libraries.

1. The *Installation* library location is the `libraries` directory under the directory where OpenSCAD is installed.
2. The *Built-In* library location is O/S dependent. Since version 2014.03, it can be opened in the system specific file manager using the "File->Show Library Folder..." menu entry.
   - Windows: `My Documents\OpenSCAD\libraries`
   - Linux: `$HOME/.local/share/OpenSCAD/libraries`
   - Mac OS X: `$HOME/Documents/OpenSCAD/libraries`
3. The *User-Defined* library path can be created using the `OPENSCADPATH` Environment Variable to point to the library(s). `OPENSCADPATH` can contain multiple directories in case you have library collections in more than one place, separate directories with a semi-colon for Windows, and a colon for Linux/Mac OS. For example:

   Windows: `C:\Users\A_user\Documents\OpenSCAD\MyLib;C:\Thingiverse Stuff\OpenSCAD Things;D:\test_stuff`
   (Note: For Windows, in versions prior to 2014.02.22 there is a bug preventing multiple directories in `OPENSCADPATH` as described above, it uses a colon (:) to separate directories. A workaround, if your libraries are on C: is to leave off the drive letter & colon, e.g. `\Thingiverse Stuff\OpenSCAD Things:\stuff`
   Linux/Mac OS: `/usr/lib:/home/mylib:.`

   OpenSCAD will need to be restarted to recognise any change to the `OPENSCADPATH` Environment Variable.

   Where you specify a *non-fully qualified* path & filename in the **`use <...>`** or **`include <...>`** statement that path/file is checked against the directory of the calling .scad file, the *User-Defined* library paths (`OPENSCADPATH`), the *Built-In* library (i.e. the O/S dependent locations above), and the *Installation* library, ***in that order***. NOTE: In the case of a library file itself having **`use <...>`** or **`include <...>`** the directory of the library .scad file is the 'calling' file, i.e. when looking for libraries within a library, it does not check the directory of the top level .scad file.

For example, with the following locations & files defined: (with `OPENSCADPATH=/usr/lib:/home/lib_os:.`)

```
1. <installation library>/lib1.scad
2. <built-in library>/lib2.scad
3. <built-in library>/sublib/lib2.scad
4. <built-in library>/sublib/lib3.scad
5. /usr/lib/lib2.scad
6. /home/lib_os/sublib/lib3.scad
```

The following **`include <...>`** statements will match to the nominated library files

```
include <lib1.scad>  // #1.
include <lib2.scad>  // #5.
include <sublib/lib2.scad>  // #3.
include <sublib/lib3.scad>  // #6.
```

Since 2014.03, the currently active list of locations can be verified in the "Help->Library Info" dialog.

The details info shows both the content of the `OPENSCADPATH` variable and the list of all library locations. The locations will be searched in the order they appear in this list. For example;

```
OPENSCADPATH: /data/lib1:/data/lib2
OpenSCAD library path:
  /data/lib1
  /data/lib2
  /home/user/.local/share/OpenSCAD/libraries
  /opt/OpenSCAD/libraries
```

Setting `OPENSCADPATH`

In Windows, Environment Variables are set via the `Control panel`, select `System`, then `Advanced System Settings`, click `Environment Variables`. Create a new `User Variable`, or edit `OPENSCADPATH` if it exists.

On Linux (probably also on Mac), to simply add the environment variable to all users, you can type in terminal: `sudo sh -c 'echo "OPENSCADPATH=$HOME/openscad/libraries" >>/etc/profile'` to set the `OPENSCADPATH` to `openscad/libraries` under each user's home directory. For more control on environment variables, you'll need to edit the configuration files; see for example this page (http://unix.stackexchange.com/questions/117467/how-to-permanently-set-environmental-variables).

MCAD

OpenSCAD bundles the MCAD library (https://github.com/openscad/MCAD).

There are many different forks floating around (e.g.[2] (https://github.com/SolidCode/MCAD), [3] (https://github.com/elmom/MCAD), [4] (https://github.com/benhowes/MCAD)) many of them unmaintained.

MCAD bundles a lot of stuff, of varying quality, including:

- Many common shapes like rounded boxes, regular polygons and polyeders in 2D and 3D
- Gear generator for involute gears and bevel gears.
- Stepper motor mount helpers, stepper and servo outlines
- Nuts, bolts and bearings
- Screws and augers
- Material definitions for common materials
- Mathematical constants, curves
- Teardrop holes and polyholes

The git repo also contains python code to scrape OpenSCAD code, a testing framework and SolidPython, an external python library for solid cad.

Other Libraries

- BOLTS tries to build a standard part and vitamin library that can be used with OpenSCAD and other CAD tools: [5] (http://www.bolts-library.org/)
- Obiscad contains various useful tools, notably a framework for attaching modules on other modules in a simple and modular way: [6] (https://github.com/Obijuan/obiscad)
- This library provides tools to create proper 2D technical drawings of your 3D objects: [7] (http://www.cannymachines.com/entries/9/openscad_dimensioned_drawings)
- Stephanie Shaltes (https://plus.google.com/u/0/101448691399929440302) wrote a fairly comprehensive fillet library (https://github.com/StephS/i2_xends/blob/master/inc/fillets.scad)
- The shapes library (http://svn.clifford.at/openscad/trunk/libraries/shapes.scad) contains many shapes like rounded boxes, regular polygons. It is also included in MCAD.
- Also Giles Bathgates shapes library (https://github.com/elmom/MCAD/blob/master/regular_shapes.scad) provides regular polygons and polyeders and is included in MCAD.
- The OpenSCAD threads (http://dkprojects.net/openscad-threads/) library provides ISO conform metric and imperial threads and support internal and external threads and multiple starts.
- Sprockets for ANSI chains and motorcycle chains can be created with the Roller Chain Sprockets OpenSCAD Module (http://www.thingiverse.com/thing:197896). Contains hard coded fudge factors, may require tweaking.
- The Pinball Library (http://code.google.com/p/how-to-build-a-pinball/source/browse/trunk/scad/pinball) provides many components for pinball design work, including models for 3d printing of the parts, 3d descriptions of mount holes for CNC drilling and 2d descriptions of parts footprint
- For the generation of celtic knots there is the Celtic knot library (https://github.com/beanz/celtic-knot-scad)
- The 2D connection library (https://www.youmagine.com/designs/openscad-2d-connection-library) helps with connections between 2D sheets, which is useful for laser cut designs.
- local.scad (https://github.com/jreinhardt/local-scad) provides a flexible method for positioning parts of a design. Is also used in BOLTS.
- SCADBoard (http://scadboard.wordpress.com/) is a library for designing 3D printed PCBs in OpenSCAD.
- A Ruler (http://www.thingiverse.com/thing:30769) for determining the size of things in OpenSCAD.
- A colorspace converter for working with colors in HSV and RGB: http://www.thingiverse.com/thing:279951/
- A number of utility functions is collected in https://github.com/oampo/missile
- Unit test framework https://github.com/oampo/testcard
- Knurled surface library by aubenc http://www.thingiverse.com/thing:9095
- Text module based on technical lettering style https://github.com/thestumbler/alpha

There is also a list with more libraries here: [8] (https://github.com/openscad/openscad/wiki/Libraries)

Chapter 10 -- Command Glossary

OpenSCAD User Manual

This is a Quick Reference; a short summary of all the commands without examples, just the basic syntax. The headings are links to the full chapters.

Please be warned: The Command Glossary is presently outdated (03 2015).

Please have a look at the Cheatsheet, instead:

http://www.openscad.org/cheatsheet/

Mathematical Operators

```
+
-   // also as unary negative
*
/
%
```

```
<
<=
==
!=
>=
>
```

```
&&   // logical and
||   // logical or
!    // logical not
```

```
<boolean> ? <valIfTrue> : <valIfFalse>
```

Mathematical Functions

```
abs ( <value> )
```

```
cos ( <degrees> )
sin ( <degrees> )
tan ( <degrees> )
asin ( <value> )
acos ( <value> )
atan ( <value> )
atan2 ( <y_value>, <x_value> )
```

```
pow( <base>, <exponent> )
```

```
len ( <string> )   len ( <vector> )   len ( <vector_of_vectors> )
min ( <value1>, <value2> )
max ( <value1>, <value2> )
sqrt ( <value> )
round ( <value> )
ceil ( <value> )
floor ( <value> )
lookup( <in_value>, <vector_of_vectors> )
```

String Functions

```
str(string, value, ...)
```

Primitive Solids

```
cube(size = <value or vector>, center = <boolean>);
```

```
sphere(r = <radius>);
```

```
cylinder(h = <height>, r1 = <bottomRadius>, r2 = <topRadius>, center = <boolean>);
cylinder(h = <height>, r = <radius>);
```

```
polyhedron(points = [[x, y, z], ... ], triangles = [[p1, p2, p3..], ... ], convexity = N);
```

Transformations

```
scale(v = [x, y, z]) { ... }
```

```
(In versions > 2013.03)
resize(newsize=[x,y,z], auto=(true|false) { ... }
resize(newsize=[x,y,z], auto=[xaxis,yaxis,zaxis]) { ... }  // #axis is true|false
resize([x,y,z],[xaxis,yaxis,zaxis]) { ... }
resize([x,y,z]) { ... }
```

```
rotate(a = deg, v = [x, y, z]) { ... }
rotate(a=[x_deg,y_deg,z_deg]) { ... }
```

```
translate(v = [x, y, z]) { ... }
```

```
mirror([ 0, 1, 0 ]) { ... }
```

```
multmatrix(m = [tranformationMatrix]) { ... }
```

```
color([r, g, b, a]) { ... }
color([ R/255, G/255, B/255, a]) { ... }
color("blue",a) { ... }
```

Conditional and Iterator Functions

```
for (<loop_variable_name> = <vector> ) {...}
```

```
intersection_for (<loop_variable_name> = <vector_of_vectors>) {...}
```

```
if (<boolean condition>) {...} else {...}
```

```
assign (<var1>= <val1>, <var2>= <val2>, ...) {...}
```

CSG Modelling

```
union() {...}
```

```
difference() {...}
```

```
intersection() {...}
```

```
render(convexity = <value>) { ... }
```

Modifier Characters

```
! { ... } // Ignore the rest of the design and use this subtree as design root
* { ... } // Ignore this subtree
```

```
% { ... } // Ignore CSG of this subtree and draw it in transparent gray
# { ... } // Use this subtree as usual but draw it in transparent pink
```

Modules

```
module name(<var1>, <var2>, ...) { ...<module code>...}
```

Variables can be default initialized `<var1>=<defaultvalue>`

In module you can use `children()` to refer to all child nodes, or `children(i)` where `i` is between `0` and `$children`.

Include Statement

After 2010.02

```
include <filename.scad> (appends whole file)
```

```
use <filename.scad>  (appends ONLY modules and functions)
```

filename could use directory (with / char separator).

Prior to 2010.02

```
<filename.scad>
```

Other Language Features

```
$fa is the minimum angle for a fragment. The default value is 12 (degrees)
```

```
$fs is the minimum size of a fragment. The default value is 1.
```

```
$fn is the number of fragments. The default value is 0.
```

When $fa and $fs are used to determine the number of fragments for a circle, then OpenSCAD will never use less than 5 fragments.

```
$t
```

The $t variable is used for animation. If you enable the animation frame with view->animate and give a value for "FPS" and "Steps", the "Time" field shows the current value of $t.

```
function name(<var>) = f(<var>);
```

```
echo(<string>, <var>, ...);
```

```
render(convexity = <val>) {...}
```

```
surface(file = "filename.dat", center = <boolean>, convexity = <val>);
```

2D Primitives

```
square(size = <val>, center=<boolean>);
square(size = [x,y], center=<boolean>);
```

```
circle(r = <val>);
```

```
polygon(points = [[x, y], ... ], paths = [[p1, p2, p3..], ... ], convexity = N);
```

3D to 2D Projection

```
projection(cut = <boolean>)
```

2D to 3D Extrusion

```
linear_extrude(height = <val>, center = <boolean>, convexity = <val>, twist = <degrees>[, slices = <val>, $fn=...,$fs=...,$fa=...]){...}
```

```
rotate_extrude(convexity = <val>[, $fn = ...]){...}
```

DXF Extrusion

```
linear_extrude(height = <val>, center = <boolean>, convexity = <val>, twist = <degrees>[...])
import (file = "filename.dxf", layer = "layername")
```

```
rotate_extrude(origin = [x,y], convexity = <val>[, $fn = ...])
import (file = "filename.dxf", layer = "layername")
```

STL Import

```
import("filename.stl", convexity = <val>);
```