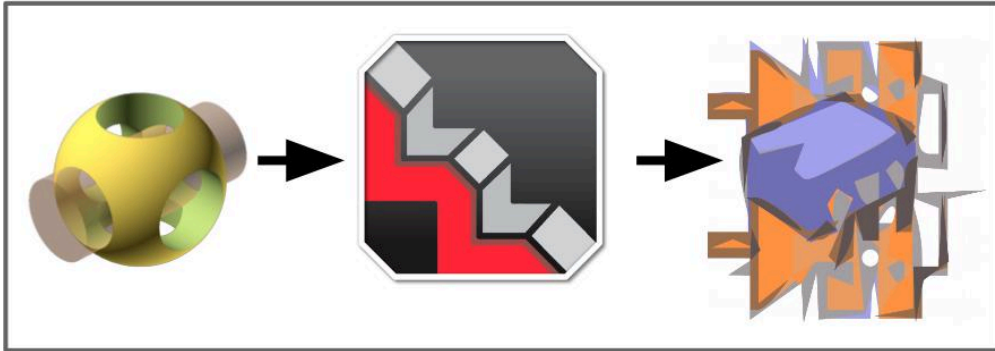


Tutorial: 3D KiCAD Parts using OpenSCAD and Wings3D

happyrobotlabs.com/posts/tutorials/tutorial-3d-kicad-parts-using-openscad-and-wings3d

Chris W.

2012/11/17



Overview

[KiCAD](#) is an Open Source EDA (Electronic Design Automation) suite, which I use for schematic capture and PCB layout. Like many other EDA tools that are floating around KiCAD can make 3D renderings of your circuit boards. If you use stock PCB symbols this is great; however, when you make your own PCB footprints you need to define your own 3d models.

I personally find it convenient to utilize [OpenSCAD](#) to make 3D models of most electrical components, which are simple and easily defined parametrically in OpenSCAD's language. OpenSCAD allows you to write code that translates directly to a 3D model and export this to an STL. However, KiCAD expects VRML files which can be easily generated in Wings3D (which can import STL files). On the plus side Wings3D will allow us to add color elements and surfaces for nice rendering by KiCAD.

However, there is one pretty terrible problem: Wings3D can't for the life of it read in STL files read by OpenSCAD. Actually, Wings3D will just crash. To fix this you can use [meshconv](#); however, use of meshconv does require use of the command line (in either Windows, OS X or Linux). I can only vouch for its usability under Linux. This will modify our workflow a little bit, but not by much.

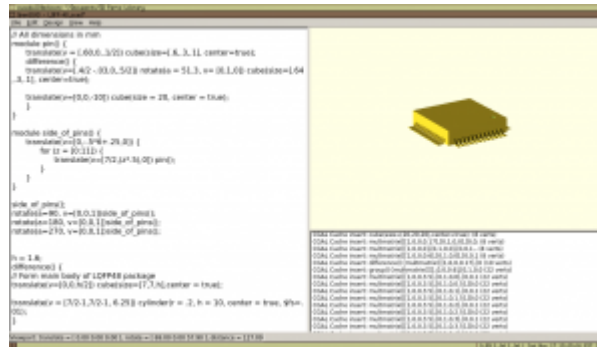
The question now is how does this workflow go down?

Workflow

My workflow is as follows:

1. Get Datasheet of Footprint (not covered here)
2. Make Footprint in KiCAD (not covered here)

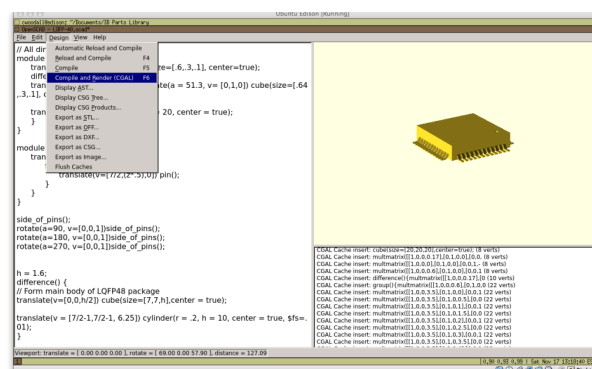
Using OpenSCAD Model the component



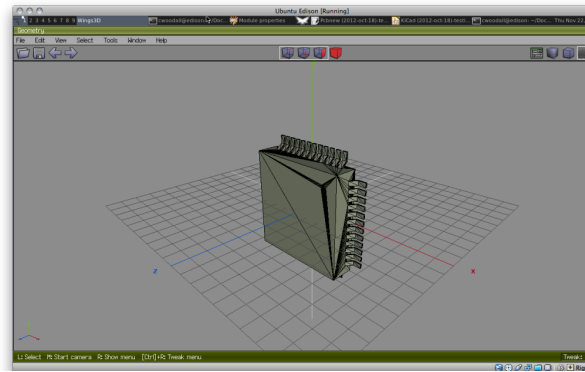
Modeling simple shapes inside of OpenSCAD is pretty easy and there are a few tricks to it. I may eventually write a tutorial for using OpenSCAD, but this will not be it. Learning OpenSCAD, unlike most CAD software, is essentially learning a programming language. I found the [User Manual](#) to be very useful. However, here are some tips.

1. Use modules for everything, especially pins. They are similar to functions and can be reused, positioned and if designed right scaled.
2. Math can be tricky, make sure you have a good concept of how primitives get moved around.
3. When adapting a component follow the datasheet as closely as possible. Take into account your tolerances appropriately.
4. OpenSCAD doesn't really have units. However, use consistent units yourself, because when we go to KiCAD we will scale.

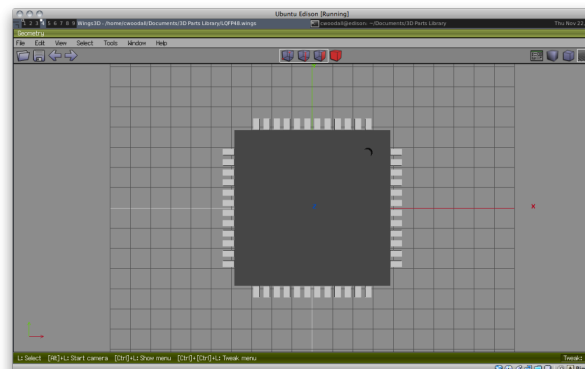
Export component as STL file



Now that we have an OpenSCAD model we need to export it. First, you need to do "Design >> Compile and Render GCAL", which will make our nice mesh model. Then you can export it as a STL file by using "Design >> Export as STL". You can take the STL file and 3d print it. This can be useful for testing placement and pin spacing of expensive parts on PCBs.



Tweak and Export VRML File

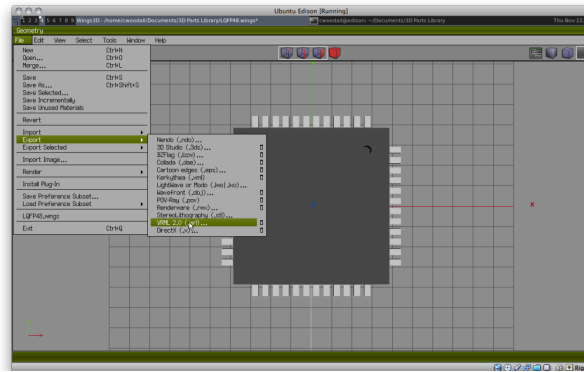


Once you have imported your STL file into Wings3D its all about making it look nice. I am not good at Wings3D, but I know how to make my components look passable.

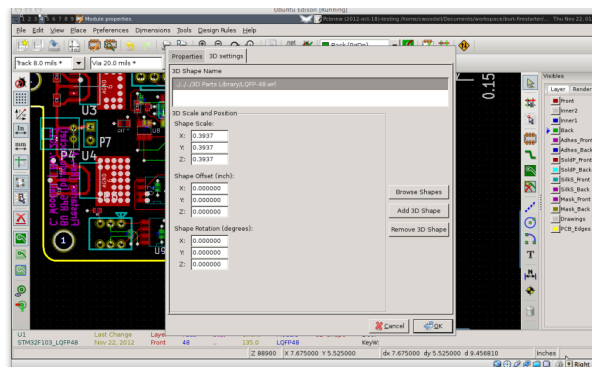
To apply a texture (or as Wings3D calls it a *Material*) to something you need to set it up first. Make sure everything is unselected by hitting "space" and right click somewhere where there are no polygons and click *Materials*. I can't advise you on what material values you want. Most of the time you can just move the bars around until your model looks good. Once all of the materials are created they need to be applied.

To apply a *Material* you need to select all the polygons you want the material to be applied to. Then right click and apply the material by clicking material again and selecting the material you want. Getting a good selection can be tough due to the interface. I suggest taking a glance at the [WikiBooks Wings3D Tutorial](#) for further tips on getting a good selection and tweaking your models.

After your little black box looks good we can export it by selecting "File >> Export >> VRML 2.0 (.wrl)".

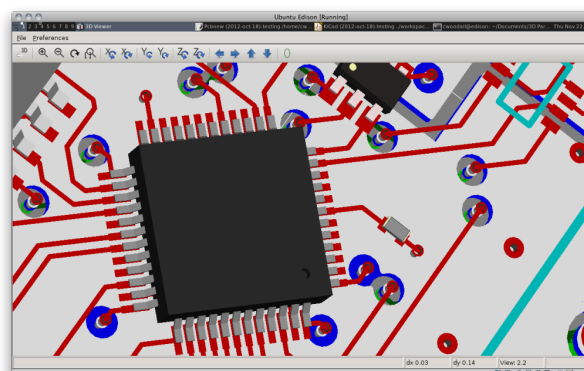


Import to KiCAD with proper scaling



We are almost there. Open up your PCB layout right click on the module you want to add the 3D model to and edit the properties. Click "3D Settings" now "Browse Shapes" and find your 3D .wrl file and select it. I suggest putting your 3D files in a folder in your project directory (or better yet library directory). Once you have the shape imported you need to scale it and rotate it. Rotation is pretty easy to figure out, as is the scaling. If you did your work in mm you will need to scale by 0.393700 (1WU/2.54mm, where WU stands for Wings Units). If you worked in inches then you may need to scale a bit because 1 WU is equal to .1" by default.

Enjoy your beautiful 3D models!



Now you have beautiful 3D models of your circuit boards that allow you to feel semi-competent around MechE's showing off their 3D models. To make yourself feel even better your 3D models have little traces of copper on them, thats pretty cool. To make things

better you can export as a VRML file and bring your design anywhere a VRML file can be read! I am working on a method for going from KiCAD 3D models to AutoDesk Inventor and Solidworks, for including PCBs with all the components on in mechanical drawings.

~~You can get all of my OpenSCAD and Wings3d files [here](#)~~

EDIT: I finally found the 3D parts and have moved them to github to be distributed and managed there. Sorry to all for the inconvenience. Find them on [GitHub](#)