

OpenSCAD

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OpenSCAD is free coding software that was developed specifically for designing 3D models. With some basic programming, users can construct combinations of extrusions and Boolean operations to build up complex parametric model files that can be exported for 3D printing. The advantage of parametric modeling is that design variables can be easily modified to change clearances, sizes, and styles, leading to highly customizable models. In addition, OpenSCAD code is particularly good at handling mathematical equations, data, and algorithms so that users can produce mathematically meaningful designs. In this workshop we'll start at the beginning and get everyone started with OpenSCAD, and then move into more complex examples that will include knot models and algorithmic designs.

Attendees are encouraged to come to the workshop with OpenSCAD already installed:

<http://www.openscad.org/downloads.html>