# OpenSCAD Basic Terminology

Shape - Any graphical 2D or 3D object created by OpenSCAD.

Design - An OpenSCAD creation (that is, an OpenSCAD program), which usually consists of a combination of multiple shapes.

Operation - An OpenSCAD command that changes the appearance/properties of one or more shapes.

Parameter - Any value that specifies characteristics of shapes, operations, modules, or functions.

Preview - The process of quickly displaying a design on-screen.

Render - The process of fully evaluating the geometry of a design (and showing it on-screen). Once it’s rendered, you can export a design.

Units - All dimensions in OpenSCAD are specified in units. A unit is usually a millimeter (by 3D-printing industry convention), but OpenSCAD is technically unit-less. All models should be explicitly sized in 3D-printing preparation software just prior to printing.

Width - The dimension associated with the x-axis, which is the “left right” axis when 3D printing.

Length - The dimension associated with the y-axis, which is the “forward-backward” axis when 3D printing.

Height - The dimension associated with the z-axis, which is the “up down” axis when 3D printing.

2D shapes - Shapes with a width and length, but no height.

3D shapes - Shapes with a width, length, and height

Comment - A piece of code that does not execute it is preceded with // or for a multiline comment any text between /\* and \*/