

DARA 3D Modelling Workshop OpenSCAD Overview & Topic Outline

Including Lesson Plans 1 & 2 and Resource Reference List

OpenSCAD Lesson Plan One: Page of 1 of 3.

OpenSCAD Lesson Plan One - Introductorily Remarks & Preparatory Tasks

Basically a hands-on discovery method of instruction with demonstrations and over-the-shoulder tutored practice sessions of OpenSCAD 3D modeling software.

Three Step Process:

Design = Drawing (.scad) & Modelling (.stl)

Processing (Slicing) = Toolpath Generation (.gcode)

Production = 3D Printing

SCAD = S-CAD = Scripted Computer Aided Design

Uses Computational Geometry Algorithms Library (CGAL) as its basic
Constructive Solid Geometry (CSG) engine

Authors Note:

Great for those who like it and view the world from a similar perspective;
but not for everyone - some folks just dismiss it entirely

https://en.wikipedia.org/wiki/Constructive_solid_geometry

Preparatory Tasks - Prior to First Class Get Together

Down Load and Install OpenSCAD

<https://openscad.org/>

Check out "Cheat Sheet"

OpenSCAD Help Menu or

<https://openscad.org/cheatsheet/index.html?version=2021.01>

Note: Clicking on a Cheat Sheet topic redirects to OpenSCAD Documentation.

Added Note: In case there is limited DARA Wi-Fi access during training session , highly recommend saving the **OpenSCAD Cheat Sheet** as a separate .html file to your computer.

Same can be said for downloading a .pdf copy of the **OpenSCAD User Manual/General** from

https://en.wikibooks.org/wiki/OpenSCAD_User_Manual

Warning - Program Anomaly

OpenSCAD does not consistently shut down and exit properly.

May still take up computer resources even after existing/quitting the program.

Use Task Manager (CTRL-ALT-Delete) to end the application.

Video Tutorial and Other Helpful Resources

See **Resources** page for a list of the informative and mostly to-the-point OpenSCAD video presentations.

OpenSCAD Lesson Plan One - Getting Started

Menu Commands: File, Edit, Design, View, Windows, Help

File: Open, Save, Examples, Export, New Window
 Edit: Cut Copy/Paste, Redo, Indent, Comment, Find/Replace, Preferences
 View: Perspective and "Camera Positioning"
 Window: Editor & Console
 Help: Cheat Sheet & Documentation

Bottom of Screen Menu Commands (View Perspectives)

Punctuation Syntax

lower case

delimiters () { } [] ; ,

Commenting/Remarks

// Single Line Comment

End of line comment `circle(r=10); // my comment`

Multiple Lines Comments

```
/*
  blah
  blah
  blah
*/
```

Author's Notes:

Take advantage of: "*IntelliSense*" (entry completion), Cheat Sheet, Documentation, Examples

Pay attention to Error Messages (Console Window)

Be diligent in searching for typos:

especially misspellings or upper case commands

missing punctuation

including semicolons, braces inclosing command lines, parentheses, brackets enclosing vector variable,

Be aware that variable names are case sensitive.

Quotation marks are generally only used for color names and strings.

Use the *echo* command to help troubleshoot

`echo("The value of n = ",n);` may help with debugging.

Be aware that error messages may be confusing, especially if there are missing punctuation marks.

OpenSCAD is a *Functional* programming language, and as such, variables are bound to expressions and keep a single value during their entire lifetime (*immutability*). That is to say, variables are set at compile time and cannot be modified during run time. Attempting to reassign variable values throws a **Warning Message**.

Features:

Special Variables

\$fa; minimum angle
 \$fs; minimum size
 \$fn; number of fragments
 \$t; animation step

Character Modifiers

* disable
 ! show only (do not include in model)
 # highlight / debugging
 % transparent / background

Built-In Functions

text
 string
 mathematical

Negation = NOT = !

Logical Operators == && ||

Major Sources of Confusion & Consternations

Cryptic Error Messages
 Overlooked misspellings and missing punctuation
 Forgetting to call module

Transformations

translate([x,y,z])
 rotate([x,y,z])
 offset(r,delta,chamfer)
 hull()
 minkowski()

color("colorname",alpha) color("#hexvalue") color([r,g,b,a])

scale([x,y,z])
 resize([x,y,z])
 mirror([x,y,z])

Program Flow Control

for & if statements

Additional Elements

include & use statements
 Functions
 Modules
 Libraries

Note: Lengths are dimensionless, angles are measured in degrees.

OpenSCAD Lesson Plan Two Demonstrations & Hands-On Tutoring

Vectors and Variables

```

Vector (n dimensional)    myStuff = [10,30,40,50]; contains 4 elements number 0,1,2,3
A = myStuff(3)           // A = 3

Vector of Vectors        // myThings = [[10,5,20],[3,4,9],[2.5,8],[6,9,2]];
B = myThings(0);         // B = [10,5,20]
C = B(2)                 // C = 20

```

Variables are bound to expressions and keep a single value during their entire lifetime (immutability). That is to say, variables are set at compile time and cannot be modified during run time.

Variable names are case sensitive.

Creating Shapes with Primitives & Boolean Operations

Demonstrate 2D & 3D Primitives

Demonstrate 2D & 3D Boolean Operations

Demonstrate 2D to 3D Transformations (*linear_extrude* & *rotate_extrude*, *scale*, and *resize*)

2D Primitives

- Circle
- Square
- Regular Polygons
- Polygon (points & paths)

3D Primitives

- Cube
- Sphere
- Cylinder (cones, frustums, regular polyhedrons)

Boolean Operatives

- Union
- Difference
- Intersection

Transformations

```

translate([x,y,z])
rotate([x,y,z])
scale([x,y,z])
resize([x,y,z])
color("colorname",alpha)  color("#hexvalue")  color([r,g,b,a])

```

Pretty Neat Stuff

- Rounded Corners
- Chamfers & Fillets
- linear_extrude 2D to 3D (square to circle)
- rotate_extrude (fancy spanzy)
- offset(r|delta,chamfer)
- hull()
- minkowski()
- projection(cut)
- surface(file = "path/filename.ext")

Note: Use forward slash /, not backward slash \.

Here We Go - Note to Instructor: See Instructors Notes for step-by-step examples and comments.

DARA 3D Modelling Workshop Resources & References

Fused Deposition Modelling (FDM)

An FDM 3D printer works by depositing melted filament material over a build platform layer by layer until you have a completed part. (Prints from the bottom up.)

<https://3dinsider.com/what-is-fdm/>

Resin 3D Printing

This 3D printing type is also known as vat photo polymerization: the process uses short wavelength light to solidify liquid resin into 3D pieces. (Prints from the top down.)

<https://www.youtube.com/watch?v=TuubPMWeQQE>

Getting Into Resin 3D Printing - The Ultimate SLA Beginners Guide
Nerdtronic

<https://www.youtube.com/watch?v=gSvjzGnAosl>

FDM vs Resin 3D Printing - Which is Better?

MWG Forge

Constructive Solid Geometry

https://en.wikipedia.org/wiki/Constructive_solid_geometry

File Formats & CAD Types

<https://blog.zwsoft.com/cad-file-format/>

OpenSCAD The Programmers Solid 3D CAD Modeler

<https://openscad.org/>

OpenSCAD User Manual/General

https://en.wikibooks.org/wiki/OpenSCAD_User_Manual/General

OpenSCAD Help Menu "Cheat Sheet"

<https://openscad.org/cheatsheet/index.html?version=2021.01>

3D Scanning Photogrammetry

3D Scanning for Free Holocreators GmbH · Jun 24, 2019

<https://www.youtube.com/watch?v=1s8dZAk6dy8>

OpenSCAD Videos

Author's Note: - I am reluctant to recommend any beginning OpenSCAD video/tutorials since most of them seem to be long-winded, sometimes far-a-field and in my opinion, possibly discouraging to new users.

However, at least one is pretty much on target.

3D Modeling with Code! The best demo (OpenSCAD)

<https://www.youtube.com/watch?v=KrFtd5D1cw>

MostlyBuilds

There is another series that has some neat stuff, but it takes the commentator forever to get to the point.

Open-SCAD DIY3DTech

<https://www.youtube.com/@OpenSCADDIY3DTech>

and

OpenSCAD: Introduction

https://www.youtube.com/watch?v=dzzFOU_Ggrs

cilynx

OpenSCAD Added Notes

1. Version 2024.08.30
Measurement Tools (Lengths & Angles)
2. OpenSCAD & FreeCAD
Comparisons
<https://www.findtop.com/openscad-vs-freecad-3d-modeling-software-comparison/>
Import OpenSCAD Code into FreeCAD
https://wiki.freecad.org/Import_OpenSCAD_code
3. PythonSCAD
<https://pythonscad.org/>

OpenSCAD Libraries

https://en.wikibooks.org/wiki/OpenSCAD_User_Manual/Libraries

https://en.wikibooks.org/wiki/OpenSCAD_User_Manual/MCAD

<https://openscad.org/libraries.html>

Belfry OpenSCAD Library shapes, masks, manipulators, and support for threading, gears, beziers.

BOLTS standard parts

Dimensioned Drawings provides tools to create proper 2D technical drawings of your 3D objects.

DotSCAD comprehensive library of 2D and 3D operations and transforms including extrusion along arbitrary paths, shape bending, etc.

Fillets a comprehensive fillets library

Michigan Tech's Open Sustainability Technology Lab (MOST) libraries

Obiscad contains various useful tools, notably a framework for attaching modules on other modules in a simple and modular way.

OpenSCAD Threads Library: provides ISO conform metric and imperial threads and support internal and external threads and multiple starts.

SCADBoard is a library for designing 3D printed PCBs in OpenSCAD.

Shapes library contains many shapes like rounded boxes, regular polygons. It is also included in

include <./lib/bosl/beziers.scad>