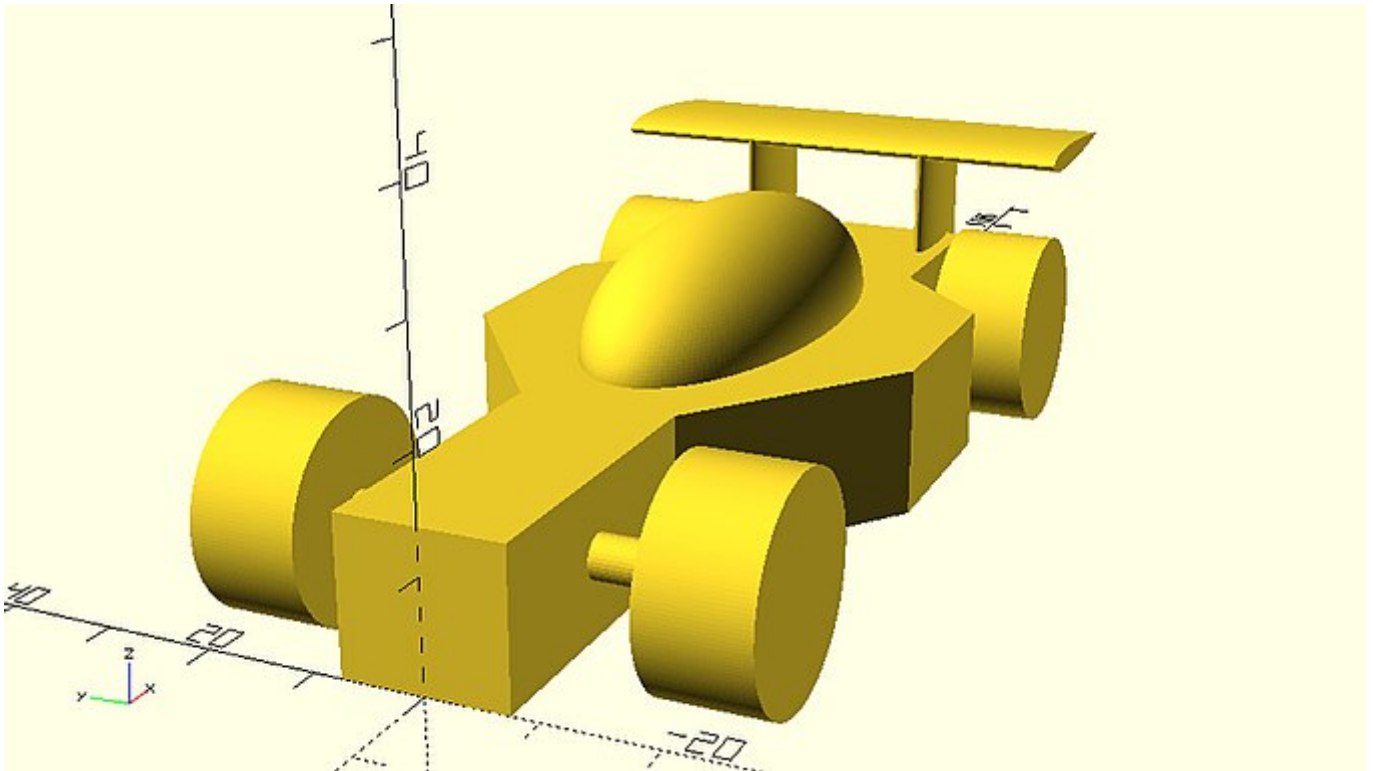# OpenSCAD Tutorial

About OpenSCAD

OpenSCAD is a solid 3D CAD modelling software that enables the creation of CAD models through a scripting file. The domain specific language designed for this purpose allows the creation of fully parametric models by combining and transforming available primitives as well as custom objects.

About this tutorial

This tutorial assumes zero programming or CAD knowledge and is designed to guide you step by step through examples and exercises that will quickly build your understanding and provide you with the right tools to create your own models. Emphasis is placed on parametric design principles that will allow you to rapidly modify your creations and build your own library of reusable and combinable models.

The majority of presented examples and solutions to exercises are available as separate OpenSCAD scripts here (https://github.com/openscad/documentation/tree/master/OpenSCAD_Tutorial/Tutorial_Files).

As of 29-11-2019 this tutorial as well as all accompanying material were completely developed as a Google Season of Docs project.

Useful links

OpenSCAD's website (https://www.openscad.org/index.html)

Download OpenSCAD (https://www.openscad.org/downloads.html)

Syntax cheat sheet (https://www.openscad.org/cheatsheet/index.html)

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