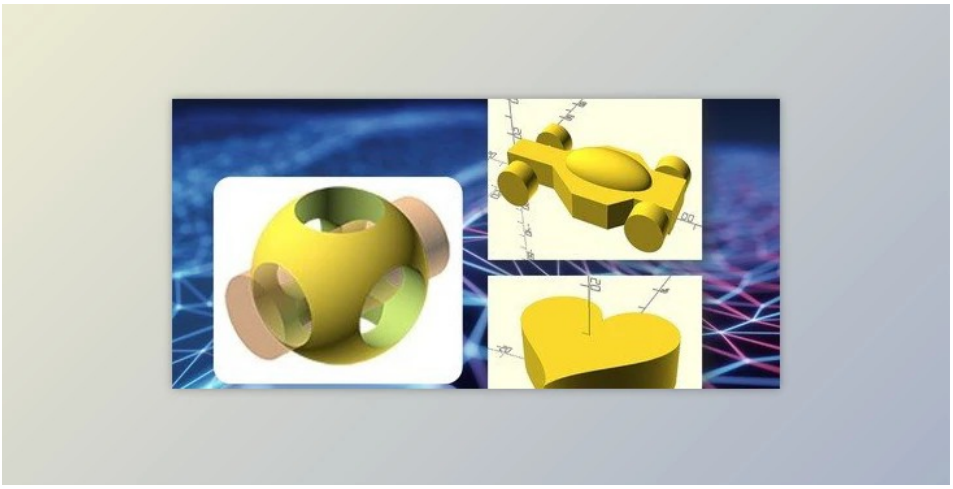
Learning and Praticing OpenSCAD on 3D Modeling

2025-02-10 16:37:17 label 我要反馈 下载页面



Learning and Praticing OpenSCAD on 3D Modeling: OpenSCAD is software designed for creating solid 3D CAD models, available for Linux/UNIX, Windows, and Mac OS X. Unlike the freest software for creating 3D models (such as Blender), it does not focus on the artistic aspects of 3DModelingng but instead on the CAD aspects. OpenSCAD is the go-to application for creating 3D models of machine parts, but it's not intended for creating computer-animated movies.

OpenSCAD is not an interactive modeller; instead, it functions like a 3D compiler that reads a script file describing the object and renders the 3D model from it. This approach gives the designer complete control over the modelling process, enabling easy changes at any step or creating designs defined by configurable parameters.

OpenSCAD offers two main modelling techniques: Constructive Solid Geometry (CSG) and extrusion of 2D outlines. Autocad DXF files can be used as the data exchange format for 2D outlines. In addition to 2D paths for extrusion, it is also possible to read design parameters from DXF files. Besides DXF files, OpenSCAD can read and create 3D models in the STL and OFF file formats.

This course aims to provide detailed guides and demos based on a formal OpenSCAD tutorial, covering every single coding detail with a thorough explanation. After completing the course, you'll be able to use OpenSCAD to design and build creative 3D CAD objects, and you'll be ready to enrol in the next level course, "Mastering OpenSCAD in 10 Projects". Enjoy!

Who this course is for:

- Anyone interested in 3D printing
- Anyone interested in building hands-on 2D / 3D shapes
- Anyone interested in learning a new tool for building 3D CAD models via coding

去下载

标签

平面设计 Tutorial