How Does OpenSCAD Work?

THE BUILDING BLOCKS

1

Use primitive shapes as building blocks to build more complex objects.

TRANSFORMATIONS

2
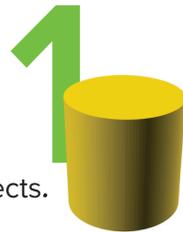
Change the orientation and position of shapes relative to each other.

COMBINATIONS

3

Combine shapes to create new and more complex shapes.

CREATIVE CODING

4
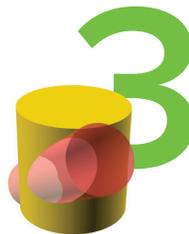
Anything is possible. Even simple code can create surprisingly complex objects.

OpenSCAD was created to help explore how to use 3D printing to create useful parts and objects.

We found that existing solutions tended to require a lot of repetitive labour using graphical user interfaces.

We created an engine and language to describe shapes in a more concise and flexible way.

COMMUNITY

OpenSCAD is developed and maintained by a small group of enthusiasts. OpenSCAD relies on the larger community to help develop, test, document, suggest features, or discuss future directions, or donate.

GET INVOLVED

www.openscad.org/community.html

🌐 **www.openscad.org**
✉ **info@openscad.org**
🐦 **@openscad**
🐙 **github.com/openscad**
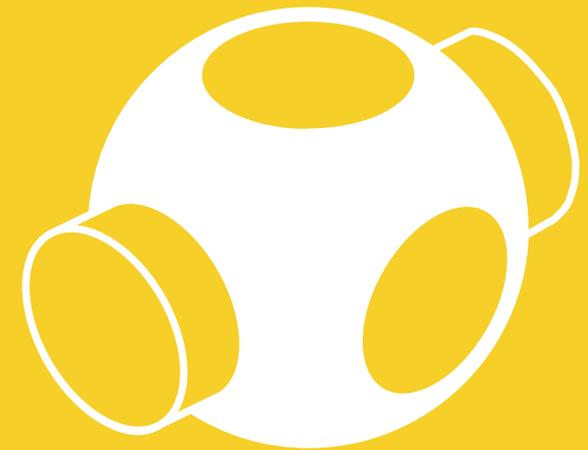
OpenSCAD

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design. code. create.

Why OpenSCAD?

OpenSCAD is a free and easy programming language for expressing ideas in 3D for 3D printing.

- Designing with code is easy to learn and it's easy to share, collaborate, and reuse designs.

- Designing with code makes it convenient to create customizable or parametric parts.

- Designing with code makes collaboration easy. It's text-based and human readable.

- Designing with code is digital literacy.

OpenSCAD is Free, Open Source and multi-platform.
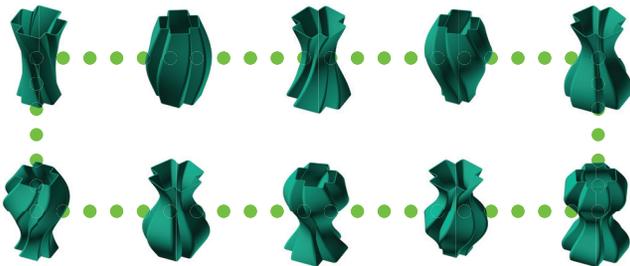
People use OpenSCAD for many purposes

Design personal objects

Teach kids programming

Scientific research

Develop medical instruments and prosthetics

Create robots

Create new 3D printers

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