



FreeCAD [How-to]

By *Daniel Falck, Brad Collette*

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Solid Modeling with the power of Python

Overview

- Packed with simple and interesting examples of python coding for the CAD world.
- Understand FreeCAD's approach to modeling and see how Python puts unprecedented power in the hands of users.
- Dive into FreeCAD and its underlying scripting language.

In Detail

FreeCAD is a general purpose platform for CAD development. It can be extended to use different workbenches to solve different problems. Targeted squarely at the engineering community, FreeCAD is an open source design application built to be extended. Python, the powerful scripting language, is woven deeply into FreeCAD's DNA and gives users unprecedented power to automate and invent.

"FreeCAD [How-to]" is a lean, fast tour of FreeCAD's major workbenches and design philosophy. The reader will get a hands-on introduction to several modeling techniques and learn how Python makes macro automation and design possible.

FreeCAD allows users to design in a variety of ways; from drawing 2D designs as lines and circles to combining primitive solids into complex 3D shapes. Objects can even be created or modified by code you write in Python. FreeCAD even allows power users to extend the application itself with new dialogs, panels, and scripts. This book will not only show you how to get started using FreeCAD in a traditional GUI mode, but will teach you how to harness its powerful scripting language for more power.

What you will learn from this book

- Creating 3D solid objects with python
- 2D draft design and constraint modeling

- Turning 2D drawings into 3D objects
- Combining simple primitives into complex solids
- Recording Macros
- Creating objects with code
- Importing and exporting data
- Creating a Custom python dialog to automate a task
- Harnessing the constraints engine with python

Approach

Written in cookbook style, this book offers many recipes to create objects, import and export data, create 3D solid objects. Each recipe contains step-by-step instructions followed by analysis of what was done in each task and other useful information.

Who this book is written for

If you've been toying around with FreeCAD and want to have more control over your work flow then this book is for you.

The reader needs to have basic knowledge of modeling.



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Editorial Review

About the Author

Daniel Falck

Daniel Falck has over 25 years experience in manufacturing, machining, CAD, CAM, and computer programming. He has worked for Gibson Guitar doing design and prototyping for over 14 years. Currently, he runs the Prototype machine shop at King Cycle Group, in Portland, Oregon (USA), where bicycle components are manufactured. His home shop is full of CNC machining equipment that he uses to create guitar parts for customers, using open source software running on Linux. Over the past 10 years he has worked with open source manufacturing software such as Linuxcnc, APT360, HeeksCAD, FreeCAD, and a myriad of specialized python scripts.

Brad Collette

Brad Collette once designed software for a big company but doesn't like to remember that. These days, he is an entrepreneur, hobbyist, jack-of-all-trades, and a gentleman farmer. He is engaged in a multi-year project to raise two hacker sons. He has contributed to numerous open source projects and is an organizing member of Columbia Gadget Works, central Missouri's finest hackerspace.

Users Review

From reader reviews:

Christopher Arredondo:

The publication with title FreeCAD [How-to] has lot of information that you can learn it. You can get a lot of gain after read this book. This book exist new understanding the information that exist in this book represented the condition of the world at this point. That is important to yo7u to find out how the improvement of the world. That book will bring you throughout new era of the the positive effect. You can read the e-book on your smart phone, so you can read that anywhere you want.

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