

Freecad A Manual

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Freecad A Manual

FreeCAD

FreeCAD

Manual start FreeCAD is a free, open source parametric 3D modeling application. It is made primarily to model real-world objects, ranging from small electronic components up to large objects such as buildings and civil engineering projects, with a strong focus on 3D-printable objects.

Manual:Introduction - FreeCAD Documentation

A FreeCAD manual Note: The manual has been moved to the official FreeCAD wiki which is now its new home. If you wish to propose edits, please do them there, as this repository will be kept only for generating the ebook versions and will not be directly edited anymore.

Introduction · A FreeCAD manual

FreeCAD features tools similar to Catia, SolidWorks or Solid Edge, and therefore also falls into the category of MCAD, PLM, CAx and CAE. It is a feature based parametric modeler with a

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modular software architecture which makes it possible to provide additional functionality without modifying the core system.. As with many CAD modelers it has many 2D components in order to sketch planar shapes ...

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Manual start FreeCAD is a general purpose parametric 3D computer-aided design (CAD) modeling application, made primarily to design real-life objects. Parametric modeling is a type of modeling where the shapes of the 3D objects you design are controlled by parameters.

Manual:What is FreeCAD - FreeCAD Documentation

The FreeCAD manual is another, more linear way to present the information contained in this wiki. It is made to be read like a book, and will gently introduce you to many other pages from the hubs above. ebook versions are also available, as well as a couple of translations in pdf format.

FreeCAD Documentation

FreeCAD ist auch grundsätzlich ein soziales Projekt, da es von einer Gemeinschaft von Entwicklern und Benutzern entwickelt und gepflegt wird, die durch ihre Leidenschaft für FreeCAD vereint sind. Dieses Manual ist für Neulinge unter den FreeCAD-Benutzern gedacht und soll einen kurz gefassten Gesamtüberblick geben.

Handbuch: Einführung - FreeCAD Documentation

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The FreeCAD interface (FreeCAD uses the Qt framework) to draw and manage its interface. This framework is used in a wide range of applications, so the FreeCAD interface is very classical and presents no particular difficulty to understand. Most buttons are

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standard and will be found where you expect them (File -> Open, Edit -> Paste, etc).

The FreeCAD interface · A FreeCAD manual

Traditional 2D drafting. You might be interested in FreeCAD because you already have some technical drawing experience, for example with software like AutoCAD, or you already know something about design, or you prefer to draw things before building them. In any case, FreeCAD features a more traditional workbench, with tools found in most 2D CAD applications: The Draft Workbench.

Traditional 2D drafting · A FreeCAD manual

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FreeCAD: Select your platform

Modeling for product design. Product design is originally a commercial term, but in the 3D world, it often means modeling something with the idea to have it 3D-printed or, more generally, manufactured by a machine, being a 3D printer or a CNC machine.. When you print objects in 3D, it is of ultimate importance that your objects are solid. As they will become real, solid objects, this is obvious.

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FreeCAD allows you to sketch geometry constrained 2D shapes and use them as a base to build other objects. It contains many components to adjust dimensions or extract design details from 3D models to create high quality production ready drawings.

FreeCAD: Your own 3D parametric modeler

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