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This is a typical PartDesign workflow: Create a new sketch Draw a closed shape (make sure all points are joined) Close the sketch Expand the sketch into a 3D solid by using the pad tool Select one face of the solid Create a second sketch (this time it will be drawn on the selected face) Draw a ...

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00 Getting Started with FreeCAD - Some basics. - YouTube

This article could be a FreeCAD ebook. Let's get started! In this tutorial we will prepare simple 3D model and after that you could be able to create simple parts by yourself. Let's start! To start modeling in FreeCAD, select File | New and choose the Part Design module from the drop-down menu.

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Getting Started with FreeCAD. FreeCAD is an open source and free parametric 3D design software aimed at mechanical designers though there are some modules that would extend the capability of the software like the Arch module which allows the user to use FreeCAD as an architecture design software. FreeCAD is available on all platforms Windows, MacOS, and Linux.

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Start with the Part Workbench. If you haven't opened a new FreeCAD document (most of the FreeCAD window looks greyed-out), from the pull-down menu click File → New or click the Create a new empty document icon. Click on the Box button to create a box

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FreeCAD: Your own 3D parametric modeler

The FreeCAD Basics Tutorial book is the essential guide for engineers and designers without any experience in computer aided designing. This book will teach you the basics you need to know to start using FreeCAD with easy to understand, step-by-step tutorials. 173 People Used

Freecad Training Book - 08/2020

The workshop facilitator will give a quick walk through to help you get started with working in FreeCAD. Participants must bring their own laptop and have FreeCAD already installed. An external mouse with a scroll wheel is highly advisable (but not required) to help make navigation with the application much easier.

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