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~~FreeCAD Tutorial
for Beginners~~

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#1 Introduction
- Tutorial

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Tutorial 1: The

Basics ~~FreeCAD~~

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~~Intro - 01~~

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complicated plot
in the Plot
Workbench with
two different
input lists (X,
T), and three
output lists
(Y1, Y2, Y3),
then create two
sets of axes in
the same plot,
and plot the
pairs of lists
(Y1 against X,

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and Y2 and Y3
against T); set
the axes labels
and grid, and
move the legends
and title.

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Yep, that's
going to happen
a lot: in
FreeCAD you

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often jump back and forth between sketches and 3D models. You create a 3D model based on a sketch. Then click on one face (surface) of the 3D model and continue with the next sketch. After that, you turn

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this sketch in a new 3D shape, etc.

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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

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architecture
design software.
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Getting Started
with FreeCAD -
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Within FreeCAD,
locate the macro
path by choosing
Edit ?

Preferences ?

General ? Macro
and look for the
"Macro path". By
default, the
macro directory
is the (hidden)
./FreeCAD/
directory in
your home-

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directory.

Within the macro-directory create (if not already present) a folder called “Mod ”.

How to install additional workbenches - FreeCAD Documentation

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computer-aided design (CAD) modeler and a building information modeling (BIM) software with finite element method (FEM) support. FreeCAD is intended for mechanical engineering product design

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but also expands to a wider range of uses around engineering, such as architecture or electrical engineering.

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[Wikipedia](#)

FreeCAD 0.17.

This article explains how to

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set display settings of parts. Parts's color, material and transparency can be changed in display settings.

Default display settings. Part's material is set to Chrome.

Part's transparency is

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set to 70%.

FreeCAD: Display settings of parts (color, material ...

First, create a rectangle on the existing rim.

Cut it by 2.5mm and then select the outward rim of your box to create a fillet.

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Switch to top view and create a new sketch on the rim you created in Lesson 2. Now draw a rectangle on the right side starting at 35.0, 37.5 and make it 2.5 x -75.0.

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shapes can
depend on those
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even be based on
specific
properties.

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images is a breeze, and users can even fully program them in Python. FreeCAD comes with a complete Graphical User Interface (GUI) that is based on the Qt framework.

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Extract the file using 7zip and start the application by double clicking on FreeCAD.exe in the bin sub-directory of the extracted package. For

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recipes described in this book will introduce most of the concepts used throughout FreeCAD so that knowledge gained in one area will help you explore the other areas more efficiently.

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[How-to] [Book]

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FreeCAD has been a Godsend for developing and evaluating my ideas and I would definitely recommend it if you are a small business with minimal or no budget for expensive CAD

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software. Pros :

- Price (free),
- functionality:
 - solid modeling (not just a collection of lines, arc and circles) ,
 - features,
 - (eventual) ease of use, ability to create .stl

...

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FreeCAD Reviews and Pricing - 2020

jayanam writes:
In this tutorial
I show how to
export a FreeCAD
model as STL
file and then
import it into
Blender 2.9.
Although it is a
straight forward
process there

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are a few small issues that have to be solved to be able to work well with the mesh in Blender.

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