FreeCAD

Overview

FreeCAD is a general purpose parametric 3D CAD modeler, whose development is completely open source (LGPL License). FreeCAD is aimed directly at mechanical engineering and product design but also fits in a wider range of uses around engineering, such as architecture, finite element analysis, 3D printing, and other tasks.

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 modular software architecture which makes it possible to provide additional functionality without modifying the core system.

Supported File Formats for Import/Export include

- STEP
- IGES
- OBJ
- STL
- DXF
- SVG
- STL
- DAE
- IFC or OFF
- NASTRAN
- VRML
- FreeCAD's native FCStd file format

The level of compatibility between FreeCAD and a given file format can vary, since it depends on the module that implements it.

Documentation

- Feature List
- Getting Started

Known Issues

Upon opening a file, you may receive a FreeCAD is not responding dialogue. Click "Wait" and then your file should open.

Invoking

FreeCAD is run from the CNF Conversion Computers or from the CNF 3D conversion server. You will get better graphical performance for the CNF Linux conversion computers if you connect and run FreeCAD with VirtualGL from CNF Thin:

```
$ vglconnect -s <netid_or_guestid>@<korat_or_minx> -S ${TLSESSIONDATA}/korat_or_minx

(korat_or_minx) $ setenv VGL_PROBEGLX 0
(korat_or_minx) $ vglrun freecad
```