

nature of FreeCAD will allow the user to adapt its workflow for such environments.

FreeCAD uses open-source libraries from the field of computing science; among them are Open CASCADE (a CAD kernel), Coin3D (an incarnation of Open Inventor), the Qt GUI framework, and Python, a popular scripting language. FreeCAD itself can also be used as a library by other programs.^[2]

Interior of 3D house

There are moves to expand FreeCAD in the architecture, electrical, and construction (AEC) engineering sectors and to add building information modeling (BIM) functionality with the Arch Module.^[3]

Supported file formats

FreeCAD's own main file format is **FreeCAD Standard file format** (.FCStd).^[4] It is a standard zip file that holds files in a certain structure.^[4] Document.xml file has all geometric and parametric objects definitions.^[4] GuiDocument.xml then has visual representation details of objects.^[4] Other files include brep-files for objects and thumbnail of drawing.^[4]

Arduino board imported from Eagle
PCB software

Besides FreeCAD's own file format, files can be exported and imported in the following file formats: DXF, SVG (Scalable Vector Graphics), STEP, IGES, STL (STereoLithography), OBJ (Wavefront), DAE (Collada), SCAD (OpenSCAD), IV (Inventor) and IFC.^[5]

DWG support

FreeCAD's support for the important DWG file format has been problematic due to software license compatibility problems with the GNU LibreDWG library. The GNU LibreDWG library started as a real free alternative to the source-available OpenDWG library (later Teigha Converter and now ODA File Converter) and is licensed under the GPLv3. As FreeCAD (and also LibreCAD) has dependencies on Open Cascade, which prior to version 6.7.0 was only compatible with GPLv2,^[6] it couldn't use the GNU LibreDWG library as GPLv2 and GPLv3 are essentially incompatible.^{[7][8]} Open CASCADE technologies were contacted by Debian team in 2009, and 2012 got a reply that Open CASCADE technologies was considering dual-licensing OCCT (the library), however they postponed that move. A request also went to the FSF to relicense GNU LibreDWG as GPLv2 or LGPLv3, which was rejected.^[9]

As of 2014 the 0.14 release of FreeCAD, including the new LGPL release of Open Cascade, the BSD-licensed version of Coin3D, and the removal of PyQT, FreeCAD is now completely GPL-free.^[10] However, LibreDWG has not been adopted. FreeCAD is able to import and export a limited subset^[11] of the DWG format via the Teigha Converter (the former OpenDWG library).

Release history

- GitHub Files section (github.com/FreeCAD/FreeCAD (<https://github.com/FreeCAD/FreeCAD>))
- SourceForge Files section (sourceforge.net/projects/free-cad/files/ (<http://sourceforge.net/projects/free-cad/files/>)) (on this SourceForge page, it is posted that this project has moved to GitHub.)
- SourceForge Old Files section (sourceforge.net/projects/free-cad/files/OldFiles/ (<http://sourceforge.net/projects/free-cad/files/OldFiles/>))

Version	Release date	Information
0.0.1	October 29, 2002	Initial release
0.1	January 27, 2003	
0.2	August 9, 2005	
0.3	October 31, 2005	
0.4	January 15, 2006	
0.5	October 5, 2006	
0.6	February 27, 2007	
0.7	April 24, 2009	
0.8	July 10, 2009	
0.9	January 16, 2010	
0.10	July 24, 2010	
0.11	May 3, 2011	[12]
0.12	November 20, 2011	[13]
0.13	January 29, 2013	[14]
0.14	July 1, 2014	License changed to LGPLv2+[15]
0.15	April 8, 2015	<u>Oculus Rift</u> support[16]
0.16	April 18, 2016	[17]
0.17	April 6, 2018	[18]
0.18	March 12, 2019	[19]
0.19		Current Developer Release
Legend: ■ Old version ■ Older version, still maintained ■ Latest version ■ Latest preview version ■ Future release		

See also

- [Open CASCADE Technology](#)
- [Open-source software](#)
- [Comparison of CAD editors](#)
- [Parametric modeling](#)
- [Constructive solid geometry](#)
- [Boundary representation](#)
- [List of free and open source software packages](#)

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19. "Release notes 0.18" (https://www.freecadweb.org/wiki/Release_notes_0.18). Retrieved 15 March 2019.

Further reading

- Falck, Daniel; Collette, Brad (2012): *FreeCAD [How-to]. Solid Modeling with the Power of Python*, Packt Publishing, Birmingham, ISBN 978-1-84951-886-4.

External links

- [Official website \(https://www.freecadweb.org/\)](https://www.freecadweb.org/)
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