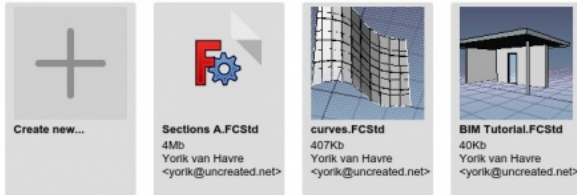




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Recent files



Tip: Adjust the number of recent files to be shown here in menu Edit -> Preferences -> General -> Size of recent file list

Examples



You can configure a custom folder to display here in menu Edit -> Preferences -> Start -> Show additional folder

General documentation

User hub

This section contains documentation useful for FreeCAD users in general: a list of all the workbenches, detailed instructions on how to install and use the FreeCAD application, tutorials, and all you need to get started.

Power users hub

This section gathers documentation for advanced users and people interested in writing python scripts. You will also find there a repository of macros, instructions on how to install and use them, and more information about customizing FreeCAD to your specific needs.

Developers hub

This section contains material for developers: How to compile FreeCAD yourself, how the FreeCAD source code is structured + how to navigate in it, how to develop new workbenches and/or embed FreeCAD in your own application.

Manual

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. [e-book versions](#) are also available.

Workbenches documentation

These are the help pages of all the workbenches currently installed on this computer.



Getting help from the community

The [FreeCAD forum](#) is a great place to get help from other FreeCAD users and developers. The forum has many sections for different types of issues and discussion subjects. If in doubt, post in the more general [Help on using FreeCAD](#) section.

If it is the first time you are posting on the forum, be sure to [read the guidelines](#) first!

FreeCAD also maintains a public [bug tracker](#) where anybody can submit bugs and propose new features. To avoid causing extra work and give the best chances to see your bug solved, make sure you read the [bug submission guide](#) before posting.

Available addons

Below is a list of available extra workbenches that can be added to your FreeCAD installation. Browse and install them from menu Tools -> Addons manager. You can learn more about any of them by clicking the links below.

Cannot fetch information from GitHub. [Authorize FreeCAD to access the internet](#) and reload the Start page.

Tip: If not bundled with your FreeCAD version, install the FreeCAD documentation package to get documentation hubs, workbench help and individual command documentation without an internet connection.

Tip: To open any of the links above in your desktop browser, Right-click -> Open in external browser

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