

Frequently Asked Questions

Did you check the documentation ?

Please have a look at the [LibreCAD User Manual](#). Since release 2.2, extensive work has been done to **improve the documentation to make it more readable, better organized and more thorough**.

FAQ sections

Below you will find some of most frequent questions asked on forum organized by topics. If you do not find the answer to you issue, you can ask for support on the [LibreCAD forum](#).

Drawing issues

- [Is the line width feature broken?](#)
- [Is the unit precision broken? All the lines seem to have the same width.](#)
- [Why do I not see the text associated with dimensions?](#)
- [How can I copy a part of a drawing to another drawing?](#)
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- [My drawing had some hatches. How can I see them?](#)

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Drawing issues

Is the line width feature broken?

You might need to zoom in to see the width or increase the line width. The line width is not shown when Draft Mode is enabled.

Is the unit precision broken? All the lines seem to have the same width.

If the issue is on the *Status bar* or the information given in the *Command line window* then check *Current Drawing Preferences* → *Units*. If the issue affects the dimensions, check *Current Drawing Preferences* → *Dimensions*.

Why do I not see the text associated with dimensions?

You might need to raise the value of *Current Drawing Preferences* → *Dimensions* → *Text Height*. You might also need to adjust the value of *Current Drawing Preferences* → *Dimensions* → *General Scale*. If you see an **empty rectangle** where the dimension text should be, then check that *View* → *Draft* is not active.

How can I copy a part of a drawing to another drawing?

See the [How to](#) section.

Is there a library of drawing elements?

Yes. There is a [Library Browser](#) which allows [insertion](#) in a similar way as for blocks. You can also [add elements](#) to create your own library.

My drawing had some hatches. How can I see them?

If *Draft* mode is selected then hatches cannot be seen. Check that *View* → *Draft* is not active.

Graphical User Interface issues

Some "tools" are missing. How can I get them back?

First, determine the type of "tool" which is missing by check this [image of LibreCAD GUI](#). The *toolbars* and *widgets* are respectively activated through the menu *Widgets* → *Toolbars* and *Widgets* → *Dock*

widgets. The *Status bar* can be activated by clicking in the menu *View → Statusbar*. You may refer to the [layout description](#) for more information.

DWG/DXF import issues

Why does the DWG import fail?

The support of DWG format by LibreCAD is basic. If the DWG import fails, you can convert from DWG to DXF format. Free converters are available on the web.

The DXF import process failed, why?

LibreCAD does not support all features of Autodesk DXF format. Currently, only the following DXF formats are supported: R12, R14, 2000, 2004 and 2007. If you have issues to import some DXF files, please report them on the [LibreCAD forum](#).

Open Design Alliance File Converter

Try the [Online Converter](#) or download [ODA File Converter](#), a freeware application for converting between different versions of .dwg and .dxf. Other [Open Design Alliance downloads](#) may help identify file issues.

Printing/exporting issues

Why does exporting a PDF produce a blank page?

You need to select a paper format (e.g. A4) in *Current Drawing Preferences*.

How do I print a drawing with dimensions to a specific scale?

Make sure the *Widgets → Toolbars → Tool Options* bar is active and see the [Printing guide](#).

LibreCAD release/build

Where can I find the upcoming release of LibreCAD?

For most OS, it is possible to build the release under development with this [procedure](#).

Host User Manual Locally

How can I use User Manual Offline?

Purpose:

- Allow Read the Docs files to be viewed offline
- Make edits to the docs themselves and render them locally so you can inspect changes before pushing back to the fork

Install:

- [Python 3.7.6](#) - Note: current bug in v3.8
- `pip install Sphinx`
- `pip install sphinx-autobuild`
- `pip install sphinx-rtd-theme`
- Download <https://github.com/LibreCAD/docs/archive/master.zip> from a fork of your choice and Unzip to a directory

Run:

- Navigate to the directory in Explorer and on the address bar enter `make livehtml` to start the local server
- In your browser enter <http://127.0.0.1:8000>

If you make any changes to the local User Manual files they will be updated in the browser.

Contribute on GitHub

Can I easily contribute on GitHub?

Yes, with a simple *pull request*. You will need to register a GitHub account, login then go to the page. For example, click the *Edit on GitHub* link in the top right-hand corner of any User Manual page to go to the page.

- Click the **pencil** button, make any edits you want and preview them to confirm. Give the proposed change a description up to 50 characters and optionally an extended description then click the **Propose file Change** button.
- If you're reading this you won't have write access to the repository so GitHub will create a copy of the repository as a branch of a fork in your account. Click the **Create pull request** button.
- Give the *Pull Request* a description and add any comments then click **Create pull request**.

Many GitHub files are in a special format, for example, the User Manual uses reStructuredText. If the change is simple you might be able to stay within existing formatting or perhaps copy a similar formatted section from elsewhere and change it. If you mess it up get help or submit it anyway and let the repository owner fix it.

Each contribution should have its own branch so just start again. If you missed something in the *pull*

request just open up your repository fork branch, change it and commit a new change.

This process will only work for one file and you can repeat it for two. Complex Pull Requests involving multiple files should be submitted using Git.

[Managing files on GitHub](#)

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Last update: **2020/04/01 12:44**

