LibreCAD User Manual

Release 2.2.0.2

LibreCAD Documentation Team

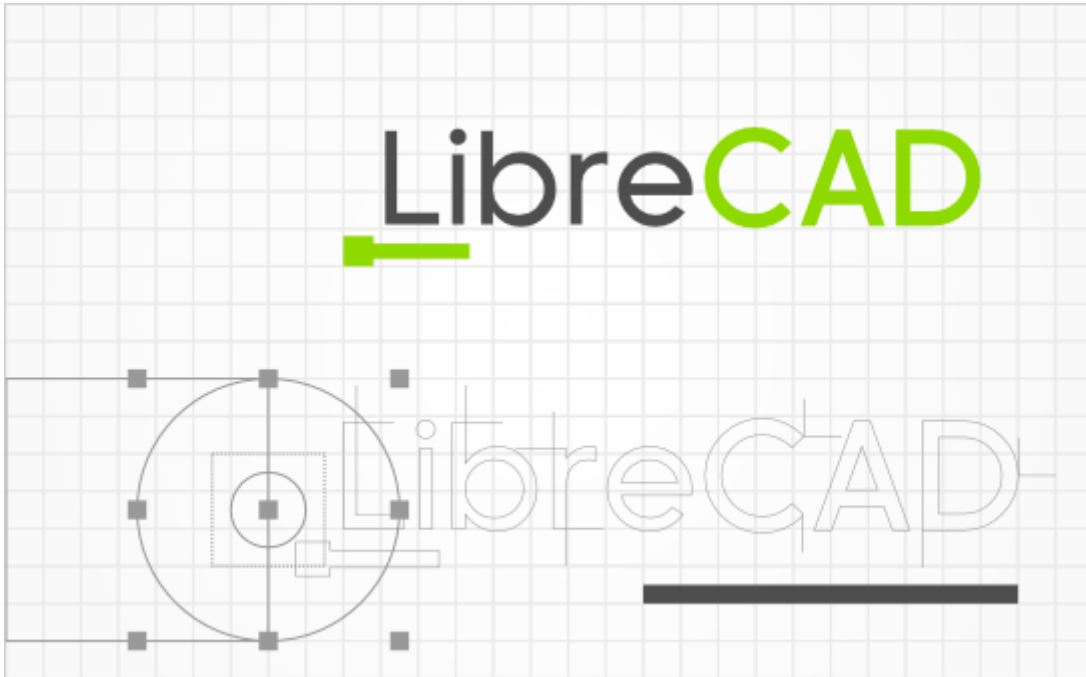**May 10, 2024**

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LibreCAD v2.2.0 - User Manual

This is an **interim release** *of the of the* **LibreCAD User Manual** *and is subject to change.*

The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

LibreCAD is a free Open Source Computer Aided Drafting (CAD) application licensed under GPLv2.

The latest information about LibreCAD can be found at librecad.org.

** LibreCAD User Manual - PDF **

The PDF conversion process on Read the Docs has limitations that affect the PDF version of this document. Our best efforts has been made to address those limitations where possible, but some issues with tables and images/figures remain.

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ABOUT

LibreCAD, the web site, wiki and the User Manual are all user supported and represents the efforts numerous *volunteers* committing countless hours of time to create, improve and support the application and documentation. Support is free directly from the large dedicated community of users, contributors and developers.

1.1 The Application

1.1.1 History

LibreCAD is a **free Open Source** 2D CAD application using the cross-platform framework **Qt**. That means it works with multiple operating systems; Windows, Apple and Linux.

The project started around 2010 as a fork of QCAD 2.0.5.0. It began as a project to build CAM capabilities into the community version of QCAD for use with a Mechmate CNC router. This gave rise to CADuntu. The project was known as CADuntu only for a couple of months before the community decided that the name was inappropriate. After some discussion within the community and research on existing names, CADuntu was renamed to **LibreCAD**.

Since QCAD CE was built around the outdated Qt3 library, it had to be ported to Qt4 before additional enhancements. Porting the rendering engine to Qt4 proved to be a large task, so initially LibreCAD, the 1.0.0 series, still depended on the Qt3 support library. Thanks to our master developer Rallaz, the Qt4 porting was completed during the development of 2.0.0 series and LibreCAD has become Qt3 free. The latest version of LibreCAD, the 2.2.0 series, requires the Qt5 framework.

1.1.2 Features

LibreCAD has the following features:

- reads DWG and DXF files
- writes DXF, SVG, PDF, and more…
- drawing entities include line, polyline, spline, circle, ellipse, text, dimension, blocks and hatches
- advanced snapping system
- custom toolbars and menus
- highly customizable user interface
- plugin system

As free software you can redistribute it and/or modify it under the terms of the see GNU General Public License version 2 (GPLv2) as published by the Free Software Foundation.

1.2 The User Manual

The user manual is a compilation from many sources. The manual includes detailed instructions on obtaining, installing and configuring LibreCAD in the *Getting Started* section. It contains the technical descriptions of the tools, functions, widgets, etc. in the *Reference* section and generic instructions on how to do a few things with LibreCAD in the *User Guide* section. There is also further information and links to additional resources in the *appendices*.

The manual is best viewed with a minimum screen width of about 1100 pixels to display the menu and content in a browser. Clicking the "LibreCAD" text or icon at the top of the menu will return to the User Manual's home page. On smaller devices, such as a mobile device, a minimum screen width of 800 pixels is recommended to display to content. At screen resolutions of 800 pixels or less, the navigation menu is hidden. It can be made visible by clicking the "hamburger" icon. Clicking the "LibreCAD" text top of the window will return to the home page.

This manual uses screen captures of LibreCAD installed on Linux. While the images may appear slightly different on Windows or MacOS, the application layout and menu commands will be the same.

1.2.1 Conventions

- Internal and external links appear in blue.
- Clicking on the embedded images will display them full size. Click the browser's "back button " to return to the manual.
- Application menu paths are shown in bold and levels are separated with "->", e.g. **File -> New.**
- Dialogue box titles are shown in **bold** with matching case.
- Dialogue box labels are enclosed in quotes, " ", with matching case.
- Tab titles are enclosed in quotes with matching case.
- Button labels are enclosed in quotes with matching case.
- Key combinations are shown with the keyboard labels enclosed in square brackets with a "+" between keys, e.g. [Ctrl]+[C].
- Text typed at the command line (OS commands, CAD commands or coordinates) is shown in a text box:

```
li
0,0
0,50
k
```

1.2.2 Contributors

There are many people who have contributed to the **LibreCAD User Manual**. Those contributions have come via LibreCAD's forum, wiki and source code.

Some of those that have contributed, directly or indirectly, include:

Armin Stebich	Bob Woltz
Chris G	Clive Tubb
David Huff	dellus
Dli,	Fabrice
Ferdi	Gary S (Maintainer)
R. van Twisk	Ravas
Richard M Brown	Stano Sitar

1.2.3 Copyright

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GETTING STARTED

LibreCAD is available for multiple operating systems and from a variety of sources. It can be used "straight out of the box" or can be customized to suit the user's preferences and needs.

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2.1 Download and Install

LibreCAD is available as two variants; as a stable version (i.e. general release), or as a unstable (also referred to as "daily" or "nightly" depending on the repository) release. The stable version is recommended for most users. The unstable build represents the cutting edge of LibreCAD development and *might* contain bugs or new, incomplete features under development. Those new features may or may not be included in next general release version. LibreCAD can also be built from the source code so users can compile the most current code base for their OS / platform.

2.1.1 Windows

Links to the MS Windows installers can be found in the *Resources* section of the appendix. Download the desired version, stable or a "NightlyBuild" of the installer from the build directory and run the installer (*exe* file).

2.1.2 Mac OS/X

Links to the installers for OS/X 10.9 can be found in the *Resources* section. Download the desired version of the installer from the build directory and run the installer (*dmg* file).

For other versions of OS/X, please follow build instructions in the *appendix*.

Note: If you are an OS/X developer, please help us improve the DMG installers and MacPorts LibreCAD package.

2.1.3 Linux

LibreCAD is available in the software repository of many Linux distributions, however the versions in the repositories may not be the most recent release of LibreCAD. Some distributions may have community supported builds that may be more recent than what is available in the official software repository.

Packages are available for the following Linux distributions through their respective repositories:

- Debian (Stable and Unstable)
- Ubuntu (Stable and Daily PPAs)
- Arch Linux
- Fedora
- Gentoo
- OpenSUSE

Links to the repository can be found in the *Resources*.

Ubuntu

Official Ubuntu Repository

LibreCAD can be found in Ubuntu's "Software Center" for Ubuntu 11.04 (Natty) and later. Search for "librecad" in the software manager and then download and install it for your system, or to install it from the command line type:

```
$ sudo apt-get update
$ sudo apt-get install librecad
```

Debian

LibreCAD is available in the main repository of Debian 7.0 (Wheezy) and later. Use your favorite package manager (e.g. aptitude, synaptic, etc.) and search for "librecad" to install and configure it, or simply from the command line type:

```
$ sudo apt install librecad
```

Debian Unstable

If you are not running unstable (i.e. Sid), and still want to upgrade LibreCAD to a newer version unstable package, download the LibreCAD debs from "Debian unstable" (*Resources*), and manually install them in your system by "dpkg -i":

```
$ sudo dpkg -i /path/to/librecad_1.0.0~rc3+nolibs-1_i386.deb
$ sudo dpkg -i /path/to/librecad-data_1.0.0~rc3+nolibs-1_i386.deb
```

2.1.4 Other

FreeBSD

LibreCAD is available from [ports], and can be installed as a binary package:

```
# pkg install librecad
```

2.1.5 Build from Source Code

For the most current up-to-date version of LibreCAD with the latest enhancements and fixes, it can be built for source. the instructions are in the *Build from Source* section of the appendix.

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2.2 Configuration

2.2.1 Initial Setup

LibreCAD's "Welcome" dialog is shown the first time LibreCAD is launched after installation. The dialog prompts the user to select the *Default Unit* and the languages to be used for the GUI and for the commands:

LibreCAD then starts with its default configuration and is ready use. Additional configuration as required can be completed by changing the **Application Preferences** as shown *below*.

2.2.2 User Interface

The user interface consists of several major elements:
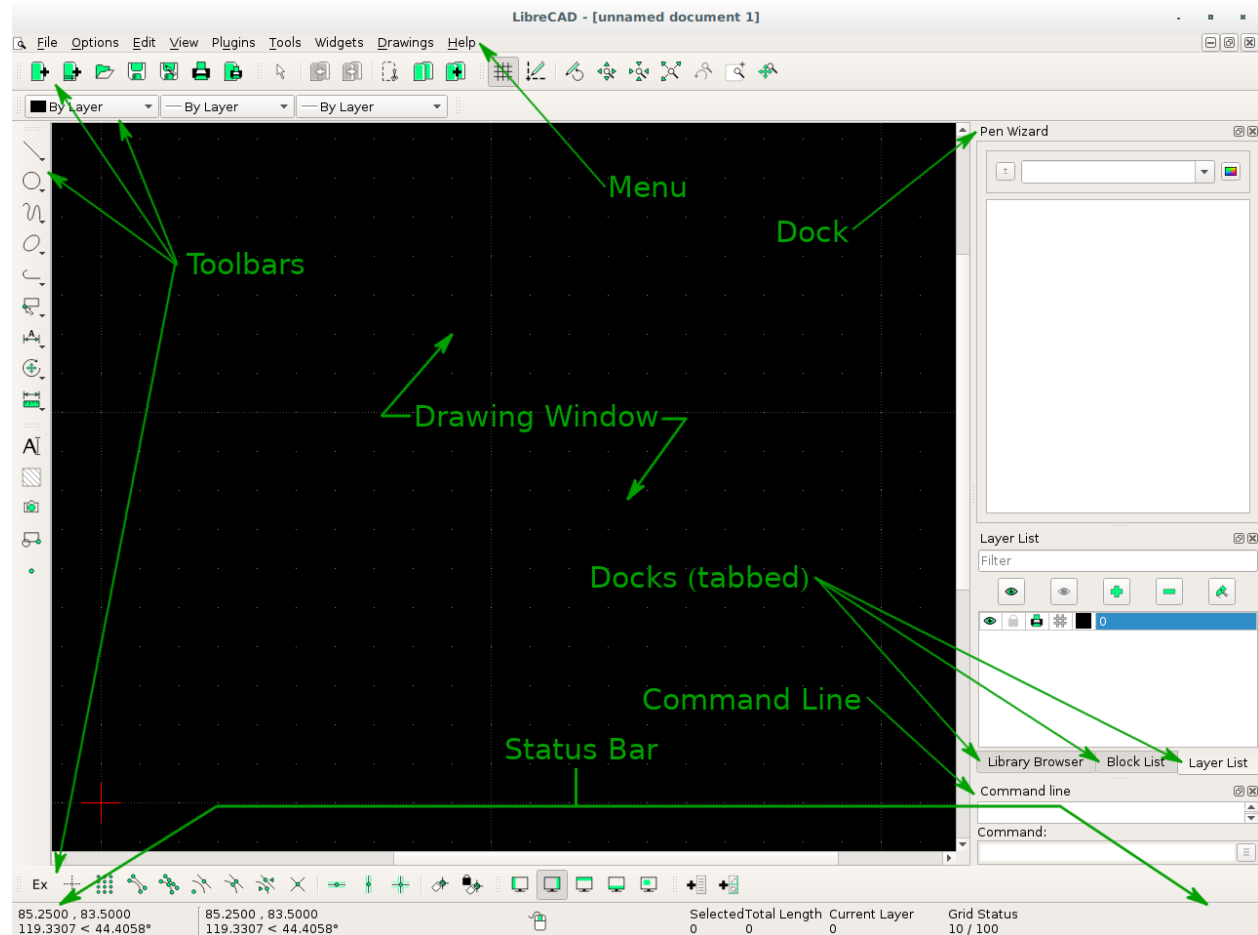
LibreCAD's default applications window

- *Menu*: provides access to application functions (open, close, etc.) and drawing tools. Menus can be customized to suit user preferences. Refer to *Application Menu* in the **Reference** section.

- *Toolbars*: provides access to drawing tools and functions. Refer to *Toolbars*.

- *Docks*: provides access to drawing tools and functions. Refer to *Docks*.

- *Status bar*: displays information about current drawing operation. (See below.)

- *Drawing window*: displays the active drawing.

Status Bar

There are five sections in the status bar. Those sections displays information about current drawing operation and include coordinates, prompts for the next action, information about the selected entity or entities, and the grid status.

Section	Description
Coordinates	Displays the absolute coordinates (left) and the relative coordinates (right) of the cross-hair / cursor. Cartesian coordinates are shown on top and and polar coordinates are bottom. Refer to *Coordinates* in **Fundamentals** of the **Reference** section for more details on the type of coordinates. 100.0000 , 10.0000 0.0000 , -90.0000 100.4988 < 5.7106° 90.0000 < 270.0000°
Next Action	Displays a prompt for the next action required for the current command; e.g. "Specify next point", "Specify center", etc. Prompts to the left of the mouse icon represent left mouse button actions and the right represent right mouse button actions. Specify next point Back
Selected	Display the total number and the combined lengths of one or more selected entities. Selected Total Length 1 127.279
Current Layer	Displays the active layer. Refer to *Layers* in **Fundamentals** of the **Reference** section for more details. Current Layer 0
Grid	Displays the minor and major grid spacing for the X and Y axes. Minor grid spacing is represented by dots, and the major by dotted lines. Grid Status 10 / 100

Layout

LibreCAD's layout and appearance is highly configurable:

- *Menus*: drop down from the menu bar or can be "torn off" and float anywhere on the display. Click the dashed line at the top of a menu to detach it.

- *Toolbars*: can be dragged and dropped to the top, bottom, left, right, or float anywhere on the display.

- *Docks Widgets*: (e.g. command line or layer list) can also be dragged and dropped to the top, bottom, left, right, or float within the drawing window. In addition they can be stacked in the same region of the application window where they will be "tabbed". Optionally docks can be placed outside of the application window, such as when using multiple monitors. Drawing tools are also available as dock widgets, but are suited as floating "toolboxes". Widgets can also be resized by dragging their edges.

- *Style sheets*: allow users to change the visual elements of the application's window decorations; title bars, fonts, colors, etc. Refer to the *Customizing* in the Reference section for more details.
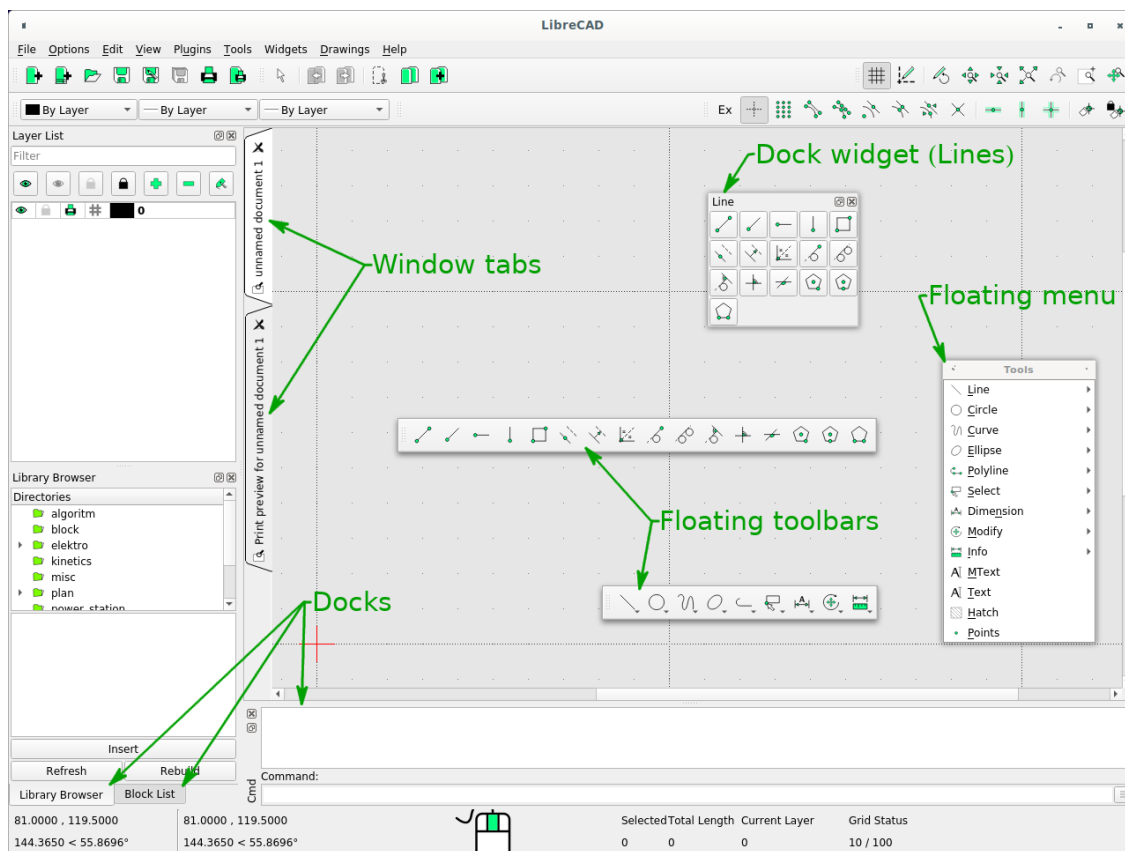

Fig. 1: LibreCAD applications window - custom layout

2.2.3 Application Preferences

In addition to the layout, LibreCAD has many preferences that will change other aspects of the appearance or behavior of the application. The preferences can be configured by selecting *Options -> Application Preferences*. Different elements of the preferences can be set; Appearance, Paths and Defaults.

Appearance

There are three categories on the "Appearance" tab that allows the user to change the look and behavior of LibreCAD.

The *Graphic View* category has options for the snap indicator style and shape, scroll-bars and grid. Use the *Snap Indicator Lines* to select the style for orthogonal (Crosshair, Crosshair 2 or Spiderweb) or isometric (Isometric) projections. The *Anti-alias* setting, if supported by the hardware, when checked will reduce jagged edges of diagonal lines, circles, etc.

The *Language* categories allows the the user to select the language used in the GUI and command line. Supported languages can be found in the *appendix*.

Thirdly, the *Graphic Colors* section allow custom colors to be selected for the snap indicator, drawing background, grid, and other indicators (selections, highlighted items and Handles). Users can select predefined colors from the drop down menu or select their own from the color selector.
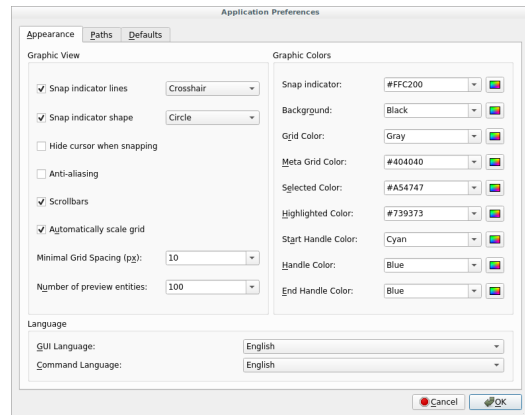
Path

The *Path* tab allows users to specify the directory paths to additional resources; language ("Translations") and user created or installed Hatch Patterns, Fonts, Parts libraries and Templates and a "Variable file". These paths do not override the defaults paths, but are appended so the default resources are still available.

It is recommended that user defined resource be placed in a user directory (e.g. home directory on Linux: ~/LibreCAD/Translations, etc.)

- *Translations*: Language files for the GUI and / or command languages.

- *Hatch Patterns, Fonts, Parts Libraries*: user created or obtained from other sources such as the Parts Library wiki

- *Template*: specify the full path and filename of a user-defined drawing template to load when launching the application or starting a *new* drawing

- *Variable File*: load a user-defined variable file when starting the application (see the *Command Line* guide for details on using commands / variables files.)
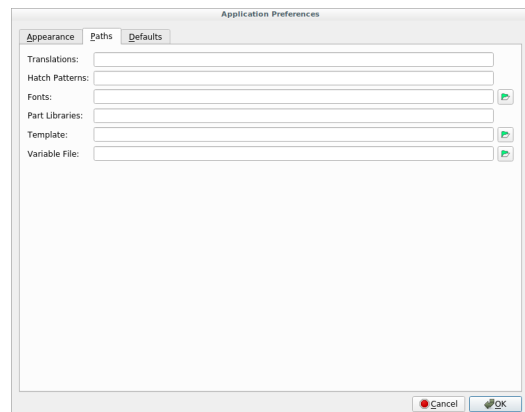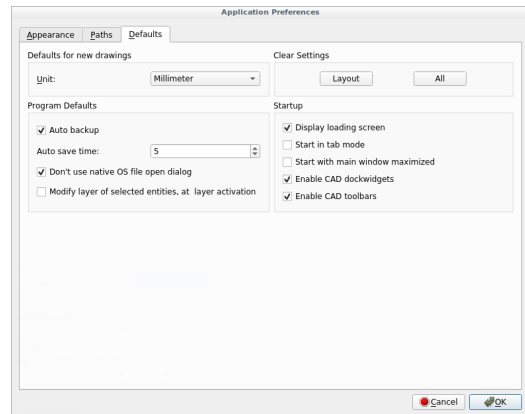
Defaults

The *Defaults* tab allows users to specify application-wide defaults.

Drawing Defaults

Unit: Defines the *default unit of measure* for all new drawings. The default can be overridden by setting the unit of measure in the Drawing preferences or template.

Program Defaults

- *Auto backup*: When checked, a backup will be created when closing the file. Backup files are saved to the same directory as the drawing file with a tilde (~) appended to the file name.

- *Auto save time*: The time in minutes to perform an automatic save of the open files. Auto files are saved to the same directory as the drawing file with a hash symbol (#) prefixed to the file name.

- *Don't use native OS file open dialog*: When checked, LibreCAD's file open dialog is displayed when opening files.

- *Modify layer of selected entities, at layer activation*: If checked, entities can be assigned to a layer by selecting the entities and then selecting the destination layer.

Clear Settings

LibreCAD's configuration can be partially or entirely reset back to a defaults:

- *Layout*: Resets the application window *layout* to the default configuration.

- *All*: Resets the application to the default configuration. Window layout, color settings, custom menus and toolbars, etc. are all reset. The "Welcome" dialog will be displayed next time the application is launched.

Startup

When checked the following items will:

- *Display loading screen*: LibreCAD's load screen (e.g. splash screen) is displayed when launching the application.

- *Start in tab mode*: the drawing window is tabbed (same as selecting Drawings -> Tab mode from the main menu).

- *Start with main window maximized*: LibreCAD will start with the application window full screen.

- *Enable CAD dockwidgets*: show drawing tools (Circle, Curve, etc.) in the widget menu (Widgets -> Dockwidgets)

- *Enable CAD toolbars*: show drawing tools (Circle, Curve, etc.) in the toolbar menu (Widgets -> Toolbars)

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REFERENCE

The Reference section provides an explanation of LibreCAD's user elements, commands, drawing tools and other features that are needed to create and print drawings. The reference manual provides a brief description of the command and what parameters are necessary.

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3.1 Fundamentals

To be able to use LibreCAD effectively, there are a few concepts that need to be understood. While a basic drawing or sketch can be created after the initial setup, as drawings become more complex further consideration to the elements of a complete drawing is important. This section offers an introduction to some concepts that are required to create a drawing, but by no means is it exhaustive. The rest of the Reference section provides a description of the tools used to configure, create and modify drawings. Further general examples and guidance is offered in the *User Guide* section.

LibreCAD supports two drawing perspectives; orthogonal and isometric projections. Orthogonal projection is the default perspective for creating two dimensional (2D) drawings. An *isometric* projection allows LibreCAD to represent a three-dimensional object in two dimensions, sometimes referred to as "2.5D". Both projections use coordinates to locate drawing elements.

3.1.1 The Coordinate System

Understanding the coordinate systems and how coordinates work in LibreCAD is necessary to produce precise drawings. Points are used to describe an aspect of an entity (e.g. the end of a line, the center of a circle, etc.), and can be placed accurately using coordinates.

There are two coordinate systems used in LibreCAD to place a point in a drawing. A point can be placed by specifying:

- a horizontal distance and vertical distance from a reference point (*Cartesian*), or

- an angle and distance from a reference point (*Polar*).

All coordinates in a drawing are relative to the *origin* or *absolute zero*. It is where the X and Y axes cross each other and is represented by crossed red lines. The coordinates of this point are **0,0**. Coordinates to the right and above the origin are positive. Coordinates to the left or below are negative. Every entity drawn can be located in relation to the origin.

LibreCAD also uses a *Relative Zero* point. It is the last point set having created an entity. It is represented within the drawing by a small red circle with a cross in it. The Relative Zero point is set temporarily to a new position in a drawing so that a subsequent coordinates of the next entity can be placed using *relative coordinates*.
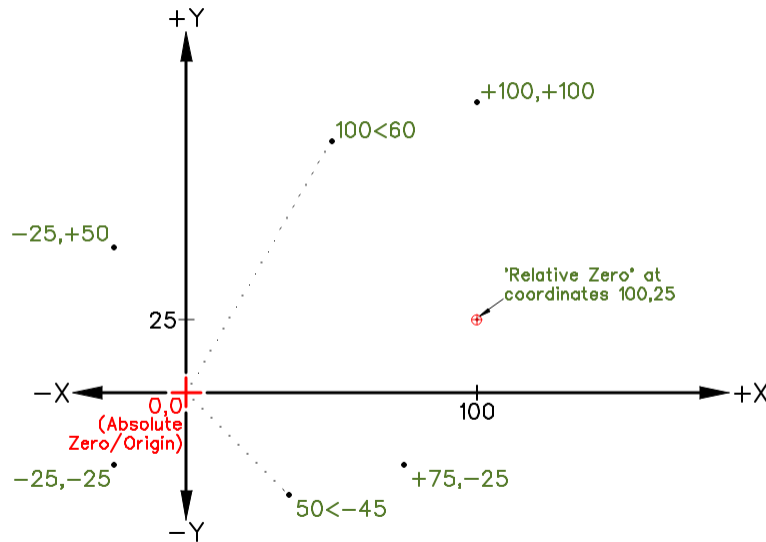
Angles
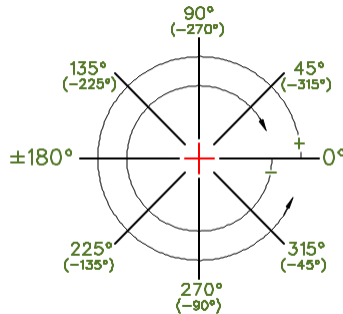
Angles are also used in LibreCAD. While horizontal or vertical distances are measured in the specified *unit*, angles in LibreCAD are always measured in degrees, beginning from 0 degrees horizontally to the right of the origin, or at the 3 o'clock position. Angles entered as a positive value are measured in an anti-clockwise direction. Angles entered as a negative value are measure in a clockwise direction.

Types of Coordinates

Cartesian

The *Cartesian* coordinate system is commonly used in most CAD programs. Cartesian coordinates take the form *X,Y* where X is measured along the horizontal axis and Y on the vertical axis. Coordinates can also be shown as "positive" (+) or "negative" (-) values. A specific point in a drawing is located by exact distances from the X and Y axis - for example a point in a drawing could be "100,75", as shown here.
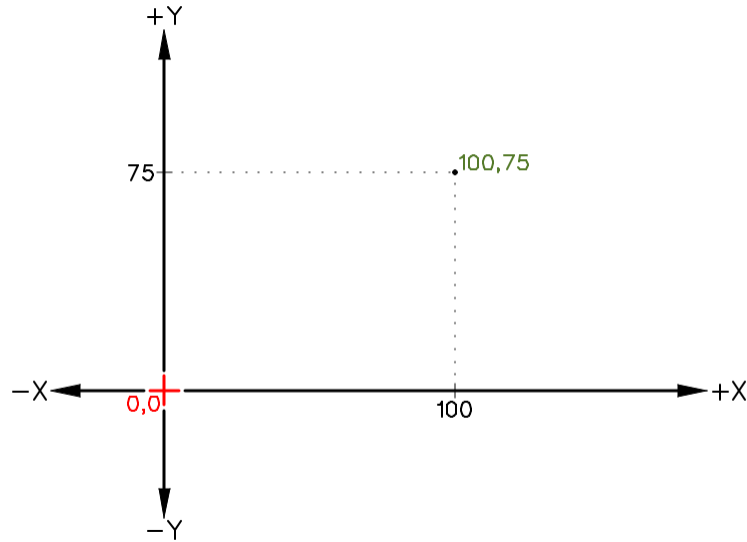
Polar

The *Polar* coordinate system uses a distance and an angle to locate a point in a drawing. In LibreCAD the < symbol is used to designate an angle when using polar coordinates. Polar coordinates take the form *100<45*, indicating a line 100 units long and with an angle of 45 degrees as shown.
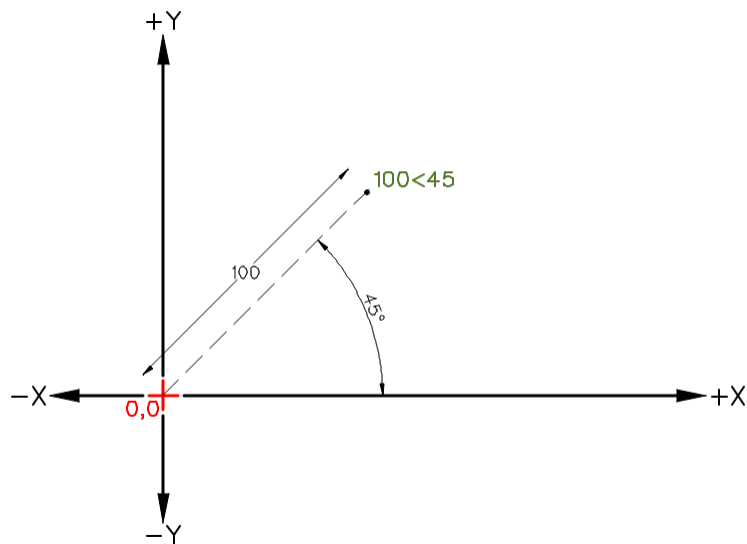
Defining Coordinate Locations

In LibreCAD lines, points, arcs, polylines, circles, and many more entities can be placed in a drawing using either *absolute* or *relative* coordinate input.

Absolute

When using absolute coordinates, whether Cartesian or polar, points are entered in direct relation to the origin (0,0). To do this in LibreCAD enter in the desired point, e.g. "100,75" or "100<45" as shown in the two images above.

Relative

The next point can also be placed *relative to the previously placed* point. The last point placed when creating an entity becomes a temporary reference for the next point. The newly set temporary reference is the "*Relative Zero* point" and the next coordinates can be entered relative to that point. To define the next point relative to the Relative Zero point, either cartesian or polar, prefix the coordinates with the "@". Points without the @ prefix are always interpreted as absolute coordinates.

For example, when using cartesian coordinates to set a point 75 units to the right and 65 units above of the previous point, use "@75,65". In the example shown, the previous point was set at 25 units horizontally and 35 vertically (25,35) from the origin (0,0). The next point can be set @75,65 from the Relative Zero point at 25,35, resulting in a point at 100,100 absolute (100 units horizontally and 100 vertically from the origin).

Hint: Relative coordinates can also be entered using the `x..y` format. This format is equivalent to prefixing the coordinates with the @ symbol, but allows for *numeric keypad* input when using the *command line*. As an example, entering coordinates as `75..65` is equivalent to `@75,65`.
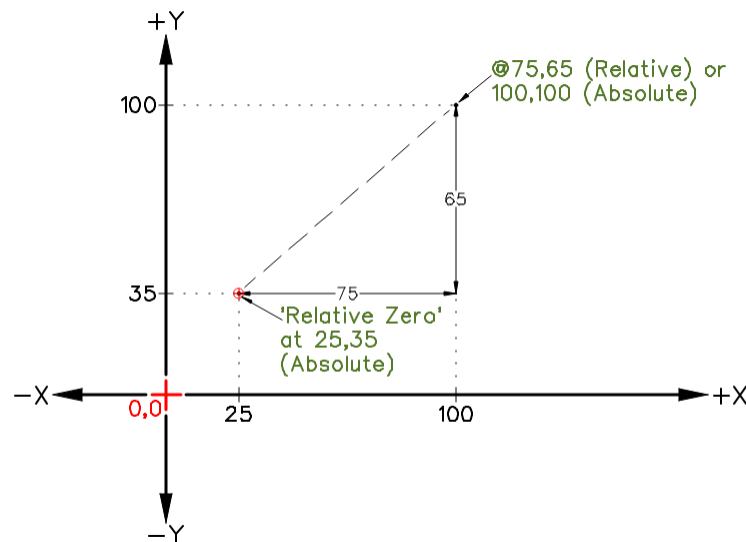

Fig. 1: Absolute & Relative Cartesian Coordinates

As an example when using a polar coordinates, to place a point 100mm and 45 degrees from the last point drawn at 25,35 (absolute cartesian coordinate) use "@100<45" (relative polar coordinate).
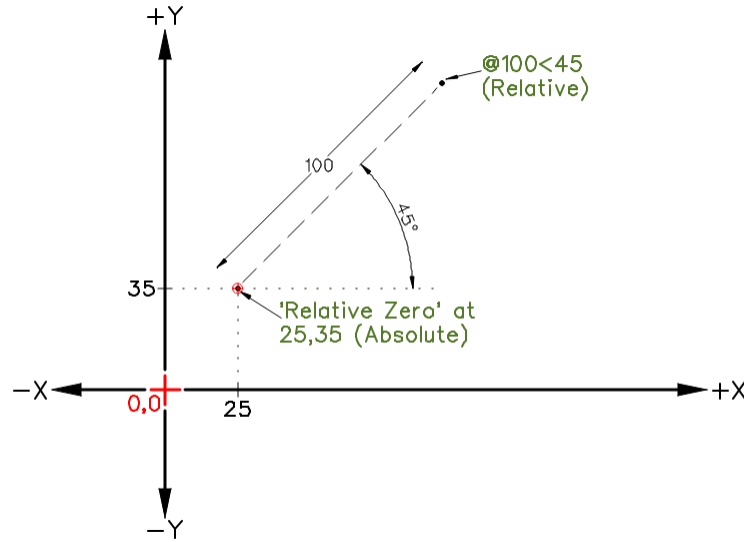

Fig. 2: Absolute Cartesian & Relative Polar Coordinates

3.1.2 Entities

An entity is a geometric shape; a line, circle, arc, etc. A collection of entities is what forms a drawing. In addition to the basic information that describes the geometry of an entity, there are two more *attributes* that further define an entity:

- *Pens* describes the appearance of an entity, either on screen or in printed output with three additional properties:
    - Color
    - Width
    - Line Type
- *Layers* provide a means to organize drawing and manage the properties of multiple entities.

Note: *Pen* or *Layers* properties *can* have a specific meaning, but vary by industry or an organization's standards and a complete description is beyond the scope of this manual.

Pen

As with many other aspects of drafting line color, thickness and type assigned to an entity, such as a line or circle are determined by drafting conventions or common practices. Within LibreCAD, the three attributes are grouped together as a "Pen":

- **Color** - LibreCAD has 16 default colors, but supports the RGB color space (#000000 to #FFFFFF or 16,777,215 colors). The initial color for entities is black.
- **Width** - The default line width is 0.00mm. Line widths of up to 2.11mm are supported.
- **Line Type** - The default line type is "Continuous" (e.g. solid). Other line types included with LibreCAD are Dot, Dash, Divide, Center, and Border.

The pen attributes can be defined for a single entity (via the *Properties* tool) , by a group of selected entities (via the *Attribute* tool), or by layer.

Note: Just as with entities, "pens" can also be applied to layers. See *Layer List Dock* for details on setting a layer's attributes.

Color

The color for an entity can be selected from the "Color" selection drop-down menu. The drop-down menu allows the color to be selected "By Layer", "By Block", from the "Custom" color selector, or chosen quickly from one of the 16 pre-defined colors:

Selecting "By Layer" will assign the color that was defined for the layer (see above) to the entity. If the layer's selected color is subsequently changed all entities on the layer will be assigned the layer's color.

When editing a *block*, selecting "By Block" will assign the color that was defined for the block to the added entity. If the block's color is subsequently changed all entities in the block will be assigned the block's color.

Selecting "Custom" will allow a selection from a palette of 36 colors and shades of grey or from a user defined colors. User defined colors are created by clicking the Add button `+` and then selecting the *hue* and *value* from the color selection tool. User defined colors can be modified by right-clicking on a user defined color and selecting a new *hue* and *value*. A maximum of eight user defined colors can be added.
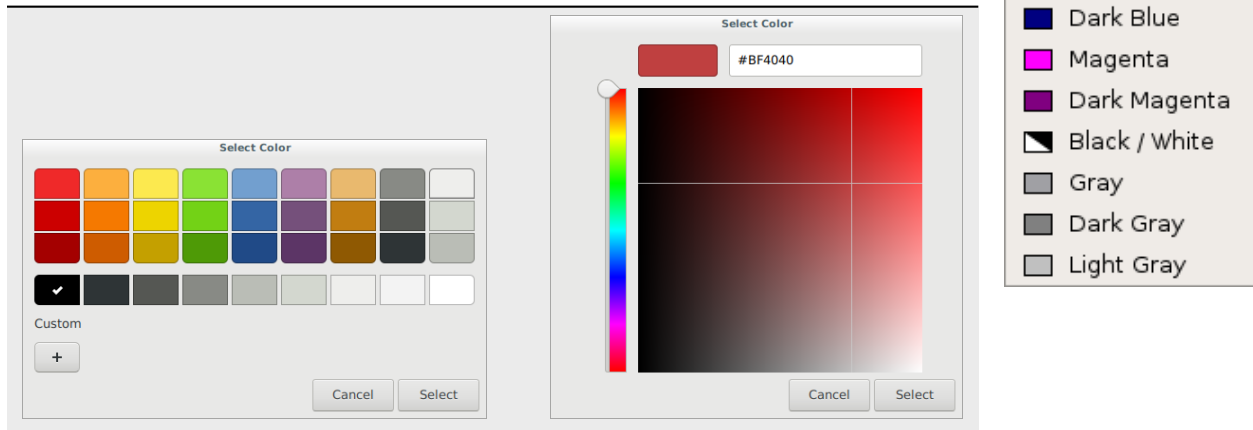
Width
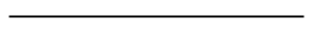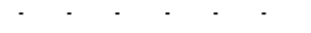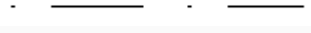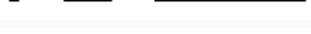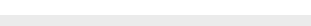
The LibreCAD default line thickness is 0.00 mm (hairline) and range up to 2.11 mm. The range include ISO standard width lines. Line widths vary by purpose; outline, hidden, section, etc., and by drawing size; larger drawings utilize thicker lines. In addition to the line widths, four additional options will show on the drop-down list:

- - Unchanged -: Leave the line width as previously selected.

- By Layer: Adopts the line width set in the *layer's attributes*.

- By Block: Adopts the line width used when the *block* was created.

- Default: Returns the line width to the default (0.00 mm).

Refer to *Common Line Widths* in the appendix for examples of line thickness by purpose.

Line Type

LibreCAD includes several commonly used line types:

Line Type	Example	Purpose
Continuous		Object or visible, dimension, extension and construction lines.
Dot		
Dash		Hidden lines and phantom lines (long dash).
Dash Dot		
Divide		Marks location (cross) section of object.
Center		Marks center of circle, arc or any symmetrical object.
Border		Used for drawing border around perimeter of sheet.

Other than "Continuous", the other non-continuous lines are available in default, "tiny" (1/6x default), "small" (1/2x) and "large (2x)". As with line thickness, different types of lines are used for different purposes. A complete list of *line types* are shown in the appendix.

Note: Intervals for non-continuous line types with white spaces remain constant when scaled. "Tiny" should be used in most cases.

Fig. 3: Available Line Widths (Attributes)

Layers

One of LibreCAD's key features is the ability to use layers. Layers help organize drawings by allowing users to place and manage related entities. Traditional manual drafting used a similar approach. The layers were often drawn on separate transparent sheets of paper. These sheets were then overlaid one on top of another to produce final drawings.

Whether for engineering, architectural, construction, manufacturing or other types, layers were used to show different aspects on the drawing. Layers could be added to show centre lines or dimensions on an engineering or manufacturing drawings, or to show different building systems on architectural drawings such as exterior walls, partitions, electrical, HVAC, grid lines, etc. While one layer can contain multiple entities, every entity in a drawing can only be associated with single layer. Typically entities with common functions or attributes are put on the same layer. For example, all the walls in a floor plan drawing would be put on a layer named "Walls".

Layers have an added advantage that all the pen attributes can be assigned to a layer. Every entity on that layer will adopt the attributes that have been assigned to that layer. However, the attributes assigned by the layer can be overridden for entities if necessary. In the above example a line thickness can applied to all entities on the "Walls" layer by changing the "Layer Settings" for that layer.

Creating a Layer

Layers are usually created to hold entities with common attributes. Creating a layer is simple:

- Click the **Add a layer** icon .

- Specify a *Layer Name*.

- Optionally specify the Color, Width and Line Type for the layer.

- Click **Ok**.

Construction Layers

A construction layer is designed to hold geometry construction lines:

- A construction layer won't appear on printout.

- All lines of a construction layer are infinite in length.

  You can toggle between construction and normal mode three ways:

- When creating or modifying a layer, click the *Construction Layer* checkbox in the *Layer Settings* dialog.

- Right-click on a named layer in the *Layer List* and choose "Toggle Construction Layer".

- Click the "Toggle construction lines" icon / in the *Layer List*.

  For more details on hiding, locking and deleting layers, refer to *Layer List Dock* in the Dock Widgets Reference section.

3.1.3 Drawing and Editing Entities

Creating Entities

There are two methods for defining coordinates when drawing entities in LibreCAD. Users can either use a mouse or other pointing devices, or the keyboard and enter coordinates.

Using a Mouse

Entities' coordinates can also be located graphically using a mouse or other pointing device. Using a mouse is less precise, but may be acceptable for rough sketches or other freehand work. However, the accuracy of using a mouse can be enhanced through the use of *snaps*.

Using the Keyboard

The keyboard can be used to select tools and enter coordinates using the *command line*.
Tools can be selected by typing a command and then entering the coordinate values.
The commands are shown in the *Drawing Tools*. Coordinates can be entered at the
command line after selecting a drawing tool using the formats as shown previously;
`100,75` (*Cartesian*) or `100<45` (*polar*), *absolute* or *relative* (prefixed with `@`).

LibreCAD offers an additional method for entering *relative coordinates* when using the
numeric keypad; type two decimal points `..` in lieu of the comma between the X and Y
values. For example, `10..20` is equivalent to typing `@10,20`. Refer to the **User Guides**
for more details on using the *command line*.

Hint: The command line also accepts arithmetic expressions in coordinate values; e.g. `100<45+15` is a valid input
for a 60° polar coordinate. For more details see *The Command Line* section or *Calculator* appendix.

Text input may also be required for **Tool Options** where distance, angle, etc. are needed
with some of the **Drawing Tools**.

Selecting Entities

Selecting entities allow them to be modified or deleted. Some operations can be applied
to groups of selected entities and other can only be applied to one entity at a time. There
are a variety of ways that entities can be selected:

- Single click on an entity. Holding the [Shift] key while clicking will allow additional
  entities to be selected.

- Click and drag a selection box:

– Left to right while moving down or up to select entities enclosed within the selection
  window's boundary (blue selection box).

– Right to left while moving down or up to select entities enclosed within the window's
  boundary and crossed by the selection boundary (green selection box)

- type "sa" at the command line to select all entities.

  Deselect selected entities by typing "tn" at the command line or pressing [Esc]. Note
  that it might be might be necessary to press [Esc] twice if a command it active.

  Also see the *Select* tools for additional methods to select and deselect entities.

Entity Handles

Selected entities display "handles". Left-clicking on and dragging a handles allow the
entities to be manipulated; lengthened, moved or enlarged depending on the type of
entity:

- Entities that consist of a single segment, such as lines, arcs and polyline segments, have
  a start handle and an end handle. Either handle can be clicked on and dragged into a
  new position.

- Handles on circles or other entities that consist of multiple segments allow it to be ma-
  nipulated in a variety of ways depending on the type of entity. For example:
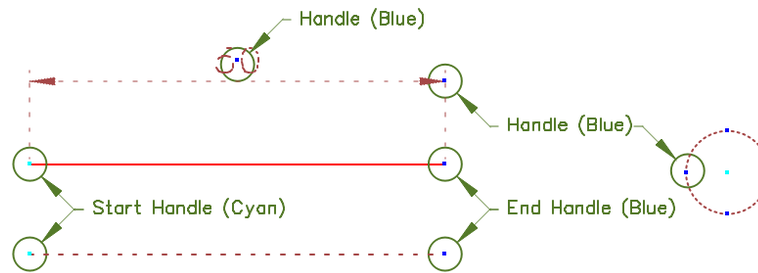
- A rectangle's corner handle can be dragged to a new position creating other quadrilaterals.

- A circle's perimeter handle can be dragged to increase or decrease the diameter.

- The end points on the edges of a polygon can be repositioned.

- Dimension text can be moved, or dimension and extension lines can be repositioned by dragging a handle.

Ordering Entities

Entities are drawn on a 2D plane, but can overlay each other and have levels that are independent of layers. Entity can be raised, *moved up*, or lowered, *moved down*, with respect to other entities. Each layer can contain entities that are at different levels. Use the commands in **Tools -> Modify -> Order** to move entities up or down.

Action	Key	Result
move to top	[Home]	Moves the selected entity to the *top* most position.
move to bottom	[End]	Moves the selected entity to the *bottom* most position.
raise over entity	[Page Up]	Moves the selected entity *up* one relative position.
lower after entity	[Page Down]	Moves the selected entity *down* one relative position.

3.1.4 Isometric Drawings

LibreCAD can also be used to create drawings with an *isometric perspective*. Creating isometric drawings is similar to creating orthogonal drawings, but with an additional consideration towards combining multiple views in a single perspective. Instead of the two axes (X,Y) used with orthogonal drawings, isometric drawing are represented in a two dimensional drawing on three axes (X,Y,Z) with 120 ° between each axis.

See *Drawing an Isometric View* in **Drawing and Editing** section of the **User Guide** for details on configuring LibreCAD for isometric drafting.
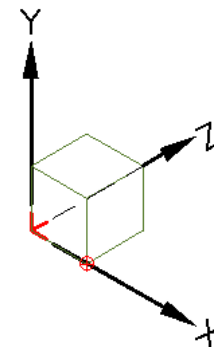
Fig. 4: Isometric X, Y ,Z Axes

LibreCAD v2.2.0 - User Manual

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The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

3.2 Main Menu

Most of LibreCAD's commands are accessible through the menu bar. Drop-down menus provides access to file and print operations, application and drawing configurations, drawing and editing tools, and other functions.

The drop-down menus can be changed into a floating menu. Each drop-down menu has a dashed line at the top. Clicking the dashed line allows the menu to be "torn off" and moved to another location anywhere on-screen. Clicking the x on the upper right corner of the floating menu closes it.

Hint:

The windows top bar is created and handled by your operating system.

Thus appearance and handling may vary.

If just click doesn't work, try to drag the dashed line to create the floating menu.

The x may be any other symbol and it may also be on the left side, just follow your intuition and close the menu like any other window on your system.

3.2.1 File

Menu Item	Icon	Short-cut	Description
New		[Ctrl]+n	Creates a new drawing file.
New from Template			Creates a new drawing file from a template. See *Templates* in **User Guides** for details.
Open		[Ctrl]+o	Open existing drawing file.
Save		[Ctrl]+s	Save current drawing file.
Save as		[Ctrl] [Shift]+s	Save current drawing file to a different location or with a new file name.
Import			Import a *block*, or bit mapped or vector images into the current drawing. Supported bit mapped formats include: bmp, cur, gif, ico, jpeg, pbm, pgm, png, ppm, xbm, and xpm. Vector images supported include svg, and svgz.
Export			Export the current drawing as a CAM, pdf or image file. Supported bitmapped formats include: cur, jpeg, pbm, pgm, png, ppm, bmp, ico, xbm, and xpm. Vector images supported include svg, and svgz. Use the ''CAM'' export to save SVG (Scalable Vector Graphics) suitable for MakerCAM, EleskCAM, LaserWeb, …
Print		[Ctrl]+p	Produce output of the current drawing. See *Printing* in the **User Guides** section.
Print Preview			View output on screen of the current drawing.
Close		[Ctrl]+w	Close the active drawing.
Quit		[Ctrl]+q	Close the application.
Recent Files			Open existing drawing file from a list of previously opened drawing files.

3.2.2 Options

Menu Item	Icon	Short-cut	Description
Application Preferences			See *Application Preferences* in **Getting Started** for details.
Current Drawing Preferences			See *Drawing Preferences* for details.
Widget Options			See *Widget Options* for in **Customizing** details.
Device Options			Select input device: mouse, tablet, trackpad or touchscreen.
Reload Style Sheet		[Ctrl]+t	See *Style Sheets* in **Customizing** for details.

3.2.3 Edit

Menu Item	Icon	Short-cut / *Command*	Description
Selection Pointer		[Esc] / *k, kill*	Reverts from current operation to the selection pointer (e.g. cancels the current operation)
Undo		[Ctrl]+z / *u, undo, oo*	Sequentially reverses the previous operations.
Redo		[Ctrl]+[Shift]+z / *r, redo, uu*	Sequentially reverses the previously reversed operations.
Cut		[Ctrl]+x	Removes the selected entity (or entities) and places it in temporary memory, e.g. ''clipboard'' for later recall. A reference point needs to be placed for subsequent paste operations.
Copy		[Ctrl]+c	Creates a copy of the selected entity (or entities) in temporary memory to be recalled. A reference point needs to be placed for subsequent paste operations.
Paste		[Ctrl]+v	Recalls the entity (or entities) from temporary memory and place it at a location defined by a reference point.
Delete Selected		[Del]	Removes the selected entity (or entities) from the current drawing.

3.2.4 View

Menu Item	Icon	Short-cut / *Command*	Description
Fullscreen		[F11]	Hides the application title bar and toggles LibreCAD to use the entire display.
Statusbar		[Ctrl]+[I]	Toggles the visibility of the status bar at the bottom of the application window.
Grid		[Ctrl]+[G]	Toggles the visibility of the grid.
Draft		[Ctrl]+[D]	Toggles to or from ''Draft Mode''. Draft mode displays hatches as invisible, and only displays bounding boxes for images and text. Drawings will display faster, particularly on slow computers.
Redraw		[Ctrl]+[R] / *zr, rg, regen, redraw*	Refreshes the view of the current drawing.
Zoom In		[Ctrl]+[+]	Increase view of drawing by 25% increments.
Zoom Out		[Ctrl]+[-]	Decrease view of drawing by 20% increments.
Auto Zoom		*za*	Resize the view of the drawing to fill the drawing window.
Previous View		*zv*	Revert to the previous zoom level of the drawing.
Window Zoom		*zw*	Increase the view of the selected area to fill the drawing window.
Zoom Panning		*zp*	Move the view of the drawing in the window.

3.2.5 Plugins

Menu Item	Icon	Short-cut	Description
Align			Align selected entities to a reference by defining the final positions of 2 initial points.
Read ascii points			Read points from a text file. Each line of the file is a point defined by an ID, X coordinate, Y coordinate, Z coordinate and an optional code. Each field can be separated by a comma, a tab or a space. The decimal separator is the point (.). The points can be connected with a line, ID, or coordinate and code fields can be plotted as text.
Divide			Divide a line or a circle with *n* sections. A tick can be located at the limit of each section to show each limit. The size of this tick can be defined as a percentage of the segment length. The line or the circle can be broken at the limit of each section using *Divide* tool.
Gear plugin			Draw a gear by selecting the center of gear and defining parameters such as number of teeth, modulus, etc.
ESRI Shapefile			Import GIS geospatial vector data shapefile (i.e. maps). **Warning:** The import process will lock LibreCAD until it is complete and large files can be very time consuming.
List entities			List the selected entities along with their properties such as ID, layer, color, line type, line thickness, coordinates.
Read PIC file			Import Pic graphics language diagrams.
Plot plugin			Plot a mathematical function or a parametric function using the drawing coordinate system. The formula, start value, end value and step value are required. The plot can be lines, a polyline or a spline.
Same properties			Apply the properties of a reference entity to selected entities. The modified properties are layer, color, line type and line thickness.
Sample plugin			Draw a line by specifying the X and Y coordinates of end points.

3.2.6 Tools

See *Drawing Tools* for a description of the drawing tools.

3.2.7 Widgets

Menu Item	Icon	Short-cut	Description
Dock Areas			Toggles the visibility of the left, right, top, bottom and /or floating *Dock Widgets*.
Dock Widgets			See *Dock Widgets* for descriptions.
Toolbars			Toggles the visibility of the *toolbars*.
Menu Creator			Create custom menus. See *Menu Creator* in **Getting Started** for details.
Toolbar Creator			Create custom toolbars. See *Toolbar Creator* in **Getting Started** for details.

3.2.8 Drawings

Menu Item	Icon	Short-cut	Description
Tab mode			When selected, each open drawing is in its own tabbed drawing or preview window. Tabbed windows cannot be moved, resized or arranged with in the LibreCAD application window. The selected tab is the active window.
Window mode			When selected, each open drawing is in its own independent drawing or preview window that can be moved, resized or otherwise arranged (see below). "Window mode" is the default mode.
Layout			*Tab mode only*. Allows the selection of the tab's appearance ("Rounded" or "Triangular") and the location of the drawing windows tab ("North", "South", "East" or "West" (top, bottom, right, or left respectively)).
Arrange			*Window mode only*. Drawing windows can be "Maximize", "Cascade", "Tiled", "Tiled Vertically", "Tiled Horizontally"
Currently opened drawings			List the current open drawing(s). The item with the checked box is the active drawing.

3.2.9 Help

Menu Item	Icon	Short-cut	Description
Online			Displays links to online resources; Wiki, User's Manual, Command, Style Sheets, Widgets, Forum and Release Information.
About			Displays with information about the current version of LibreCAD and web links: to the "Contributors", License and "The Code" repository.
License			Displays the license text (GNU General Public License version 2).

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3.3 Drawing Tools

The drawing tools are used to create and modify entities such as lines, circles, etc. in a drawing. Commands can be selected from the menu, toolbars or entered via the *command line*. Right-click anywhere in the drawing window to display a popup menu showing the history of commands launched from the menus or the toolbars with the latest command being on top of the list. Note that the command line input is not shown.

Several drawing tools require additional parameters and will provide prompts on the **Tool Options** toolbar. *This toolbar should always be enabled.* If the tool options do not appear, from the menu select **Widgets -> Toolbar ->** and enable

Tool Options. If using the *command line*, the same tool options are available via the toolbar or the command line. The tools that have options are shown in the table below.

3.3.1 Line

Tool	Icon	Command	Description
2 points		l, li, line	Draw a line between two assigned points. **Tool Options:** Close Undo Redo
Angle			Draw a line from an assigned point defining the start, middle or end of the line and with an assigned length and angle. **Tool Options:** Angle: 30 Length: 10 Snap Point: Start
Horizontal		hor, horizontal	Draw a horizontal line from an assigned point defining the start, middle or end of the line and with an assigned length. **Tool Options:** Length: 10 Snap Point: Start
Vertical		ver, vertical	Draw a vertical line from an assigned point defining the start, middle or end of the line and with an assigned length. **Tool Options:** Length: 10 Snap Point: Start
Rectangle		rec, rect, rect-angle	Draw a rectangle by assigning the points of two diagonally opposite corners.
Parallel through point		pp, ptp	Draw a given number of lines parallel to a selected existing line through an assigned point. **Tool Options:** Number: 1
Parallel		o, pa, offset, parallel	Draw a given number of lines parallel to a selected existing line with a given distance between lines. **Tool Options:** Distance 10 Number: 1

Bisector		bi, bisect	Draw a given number of lines bisecting two existing non-parallel lines (e.g. at an angle to each other with or

3.3.2 Circle

Tool	Icon	Command	Description
Centre, Point		ci, circle	Draw a circle with a given radius by assigning a centre point and a point on the circumference.
2 Points		c2, circle2	Draw a circle with a given diameter by assigning two opposite points on the circumference.
2 Points, Radius			Draw a circle with two points on the circumference and with an assigned radius. **Tool Options:** Radius: 10
3 Points		c3, circle3	Draw a circle assigning three points on the circumference.
Centre, Radius		circlecr	Draw a circle with a given radius centred at an assigned point on the circumference. **Tool Options:** Radius: 10
Tangential, 2 Circles, 1 Point			Draw a circle tangential to two existing circles and assigning a centre point to establish the radius.
Tangential, 2 Points			Draw a circle tangential to an existing circle and define the diameter and placement by assigning two points on the circumference.
Tangential, 2 Circles, Radius			Draw a circle tangential to two existing circles with a given radius. **Tool Options:** Radius: 10
Tangential, 3 Circles		ct3, tan3	Draw a circle tangential to three existing circles and/or lines.

3.3.3 Curve

Tool	Icon	Command	Description
Center, Point, Angles			Draw a curve (arc) with a given radius defined by a center point and a point on the circumference, the direction of rotation (clockwise or counter-clockwise), a point defining the start position of the arc and a point defining the end position of the arc. **Tool Options:**
3 Points		a, ar, arc	Draw a curve (arc) by assigning three points on the circumference of the arc defining the start position, a point on the circumference and end position of the arc.
Arc Tangential			Draw a curve (arc) tangential to the end of an existing line segment with a defined radius or angle (deg). **Tool Options:** or
Spline		spl, spline	Draw an open or closed spline (curve) by assigning control points and a given degree of freedom (1 - 3). **Tool Options:**
Spline through points		stp, spline2	Draw an open or closed spline (curve) by defining points on the spline. **Tool Options:**
Ellipse Arc (Axis)			N/A
Freehand Line		fhl, free	Draw a non-geometric line.

3.3.4 Ellipse

Tool	Icon	Command	Description
Ellipse (Axis)			Draw an ellipse by assigning a centre point, a point on the circumference of major access and a point on the circumference the minor access.
Ellipse Foci Point			Draw an ellipse by assigning two foci points and a point on the circumference.
Ellipse 4 Point			Draw an ellipse assigning four points on the circumference.
Ellipse Center and 3 Points			Draw an ellipse by assigning a centre point three points on the circumference.
Ellipse Inscribed		ei, ie	Draw a Ellipse constrained by four existing non-parallel line segments.

3.3.5 Polyline

Tool	Icon	Command	Description
Polyline		pl, polyline	Draw an open or closed continuous line consisting of one or more straight line or arc segments defined by endpoints and / or radius or angle for arcs. **Tool Options:** Options include Line and for curved segments; Tangential, Tan Radius and Angle.
Add node			Add node to existing polyline. (Use "Snap on Entity" to place new node on segment.)
Append node			Add one or more segments to an existing polyline by selecting polyline and adding new node endpoint.
Delete node			Delete selected node of an existing polyline.
Delete between two nodes			Delete one or more nodes between selected nodes of an existing polyline.
Trim segments			Extend two separate non-parallel segments of an existing polyline to intersect at a new node.
Create Equidistant Polylines			Draw a given number of polylines parallel to a selected existing polyline with a given distance between lines. **Tool Options:**
Create Polyline from Existing Segments			Create polyline from two or more existing separate line or arc segments forming a continuous line.

3.3.6 Select

Hint: When the pointer is in the shape of an arrow, i.e. no tool selected, it is, by default, in a select mode. One can select single or multiple entities by clicking on them or select multiple entities by mean of a window:

- Drawn from left to right (blue), it will select all entities fully included in it.
- Drawn from right to left (green), it will also select intersecting entities.

Deselecting can be done by pressing the esc key, or *Selection Pointer* from *Edit menu*

Tool	Icon	Command	Description
Deselect all		tn	Deselect all entities on visible layers ([Ctrl]-[K] or default [Esc] action).
Select All		sa	Select all entities on visible layers ([Ctrl]-[A]).
Select Entity			Select, or deselect, one or more entities (default cursor action).
(De-)Select Contour			Select or deselected entities connected by shared points.
Select Window			Select one or more entities enclosed by selection window (L to R), or crossed by selection window (R to L) (default cursor "drag" action).
Deselect Window			Deselect one or more entities enclosed by selection window (L to R), or crossed by selection window (R to L).
Select Intersected Entities			Select one or more entities crossed by selection line.
Deselect Intersected Entities			Deselect one or more entities crossed by selection line.
(De-)Select Layer			Select or deselected all entities on the layer of the selected entity.
Invert Selection			Select all un-selected entities will deselecting all selected entities.

3.3.7 Dimension

Tool	Icon	Command	Description
Aligned		da	Apply dimension lines and text aligned to an existing entity by selecting start and end points on a line segment and placement point for the text. **Tool Options:**
Linear		dr	Apply dimension lines and text at an defined angle to an entity by selecting start and end points on a line segment and placement point for the text. **Tool Options:**
Horizontal		dh	Apply dimension lines and text aligned to an entity by selecting start and end points on a line segment and placement point for the text. **Tool Options:**
Vertical		dv	Apply dimension lines and text aligned to an entity by selecting start and end points on a line segment and placement point for the text. **Tool Options:**
Radial		dimradial	Apply dimension lines and text a circle's or arc's radius by selecting entity and placement point for the text. **Tool Options:**
Diametric		dimdiameter	Apply dimension lines and text a circle's or arc's diameter by selecting entity and placement point for the text. **Tool Options:**
Angular		dimangular	Apply angular dimension by selecting two existing non-parallel line segments and placement point for the text. **Tool Options:**

3.3.8 Modify

Tool	Icon	Command	Description
Order			Order entities within a layer. Selected entities can be moved to top, bottom, *raised* (moved forward) over another entity or *lowered* (moved backwards) behind an entity.
Move / Copy		mv	Move a selected entity by defining a reference point and a relative target point. Optionally keep the original entity (Copy), create multiple copies and / or alter attributes and layer.
Rotate		ro	Rotate a selected entity around a rotation point, moving the entity from a reference point to a target point. Optionally keep the original entity, create multiple copies and / or alter attributes and layer.
Scale		sz	Increase or decrease the size of a selected entity from a reference point by a defined factor for both axis. Optionally keep the original entity, create multiple copies and / or alter attributes and layer.
Mirror		mi	create a mirror image of a selected entity around an axis defined by two points. Optionally keep the original entity and / or alter attributes and layer.
Move and Rotate			Move a selected entity by defining a reference point and a relative target point and rotating the entity at a given angle. Optionally keep the original entity, create multiple copies and / or alter attributes and layer.
Rotate Two			Rotate a selected entity around an absolute rotation point, while rotating the entity around a relative reference point to a target point. Optionally keep the original entity, create multiple copies and / or alter attributes and layer.
Revert direction		revert	Swap start and end points of one or more selected entities.
Trim		tm, trim	Cut the length of a line entity to an intersecting line entity.
Trim Two		t2, tm2	Cut the lengths of two intersecting lines to the point of intersection.
Lengthen		le	Extend the length of a line entity to an intersecting line entity.

Tool Options:

Length: 10

Offset   o, pa, offset,

3.3.9 Info

Tool	Icon	Command	Description
Distance Point to Point		dpp, dist	Provides distance, cartesian and polar coordinates between two specified points.
Distance Entity to Point			Provides shortest distance selected entity and specified point.
Angle between two lines		ang, angle	Provides angle between two selected line segments, measured counter-clockwise.
Total length of selected entities			Provides total length of one or more selected entities (length of line segment, circle circumference, etc.).
Polygonal Area		ar, area	Provides area and circumference of polygon defined by three or more specified points.

3.3.10 Others

Tool	Icon	Command	Description
MText		mtxt, mtext	Insert multi-line text into drawing at a specified base point. Optionally define font, text height, angle, width factor, alignment, angle, special symbols and character set. **Tool Options:** Text: A line of text Angle: 0
Text		txt, text	Insert single-line text into drawing at a specified base point. Optionally define font, text height, alignment, angle, special symbols and character set. **Tool Options:** Text: A line of text Angle: 0
Hatch		ha, hatch	Fill a closed entity (polygon, circle, polyline, etc) with a defined pattern or a solid fill. Optionally define scale and angle.
Points		po, point	Draw a point at the assigned coordinates.

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3.4 Snapping

Snaps provide the ability to pick precise locations when using a mouse. Various snap tools are available to allow the user to select different locations on entities or elsewhere in the drawing space when using the grid.

Menu Item	Icon	Command	Description
Exclusive Snap Mode			**On**: only one snap mode is allowed. **Off**: multiple snap modes are allowed The snap modes are remembered in each state.
Free Snap		os, sf	Allows for the crosshair to move freely while other snap modes are enabled.
Snap on Grid		sg	Snap to a grid intersection.
Snap on Endpoints		se	Snap to the endpoints of a line segment, the quadrants of a circle, a point, or the alignment point of a text or mtext object.
Snap on Entity		np, sn	Snap to the path of an entity.
Snap Center		sc	Snap to the center of a circle or ellipse. It will also snap to the foci of an ellipse.
Snap Middle		sm	Snap to the middle of a path. Enabling this mode displays a ''Middle points" input. If you change the value to 2 then you can snap to the trisection points of a line segment.
Snap Distance		sd	If you snap to the endpoint of a line segment then activate ''snap distance" and input 50, then it will snap to a point 50 units from the endpoint on the line segment. However, it will also snap to a point that is 50 units from the other endpoint.
Snap Intersection		si	Snap to the intersection of two entities. Note this does not currently work for polylines.
Restrict Horizontal		rh	Restricts the crosshairs to the x-axis (horizontal movement).
Restrict Vertical		rv	Restricts the crosshairs to the y-axis (vertical movement).
Restrict Orthogonal		rr	Restricts the crosshairs to the x **or** y-axis. (either horizontal **or** vertical movement).
Restrict Nothing		rn	Turns off restricted cursor movements.
Set relative zero position			Manually sets the Relative Zero Point at the selected coordinate.
Lock relative zero position			Locks the Relative Zero Point to the current coordinate.

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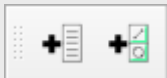
The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

3.5 Toolbars

Toolbars provide an alternative to the menus for accessing application functions and drawing tools. See *Application Menu* or *Drawing Tools* for complete descriptions.

Toolbars can be moved any where on the display and left floating, or docked to any of the four sides of the drawing window, similar to *Dock Widgets*. Unlike Dock Widgets, icons on a Toolbar are a single row when floating or docked to the top or bottom of the drawing window and verticle when docked to either side. Also, Toolbars cannot be resized.

3.5.1 Drawing Tools Toolbars

Menu Item	Description and/or *Menu Equivalent*
Categories	Groups commonly used drawing tools; *Tools -> Line / Circle / Curve / Ellipse / Polyline / Select / Dimension / Modify* and *Info*. Clicking an icon will display the drawing tool in the category. See *Drawing Tools* for details on each tool. **Toolbar:**
Circles	Drawing tools for circles. See *Tools -> Circle*. **Toolbar:**
Creators	Access menu and toolbar creators to modify LibreCAD's interface. *Widgets -> Menu Creator / Toolbar Creator*. Also see *Customizing LibreCAD's Interface*. **Toolbar:**
Curves	Drawing tools for curves. See *Tools -> Curve*. **Toolbar:**
DefaultCustom	Custom toolbar example. Includes MText, Hatch, Insert Image, Create Block, and Points. **Toolbar:**
Dimension	Drawing tools for dimensioning. See *Tools -> Dimension*. **Toolbar:**
Dock Areas	Show or hide widgets. *Widgets -> Dock Areas*. See *Dock Widget Areas*. **Toolbar:**

3.5.2 Tool Options

A variety of LibreCAD's tools require additional parameters or have options that are shown on the **Tool Options** toolbar. Several drawing tools use tool options. Those toolbars are shown in the *Drawing Tools* reference. Two additional operations that require the use of the **Tool Options** toolbar are **Print Preview** and block operations.

Important: Print Preview and Block operations require the use of the **Tool Options** toolbar as there are no menu or command line equivalencies. Ensure the Tool Option toolbar is enabled.

The **Tool Options** toolbar should always be enabled. If the tool options do not appear, from the menu select **Widgets -> Toolbar** and enable **Tool Options** (place checkmark in checkbox). If using the *command line*, the same tool options are available via a command line prompt or the **Tool Options** toolbar.

Print Preview Tool Options

The **Print Preview Tool Options** toolbar is used to set up the print output regardless of output format (pdf or paper). The toolbar is displayed after clicking the **Print Preview** icon or selecting **File -> Print Preview** from the menu. A combination of scale value, color status and drawing position relative to paper allows for customized print output. The steps are detailed in *printing guide*.
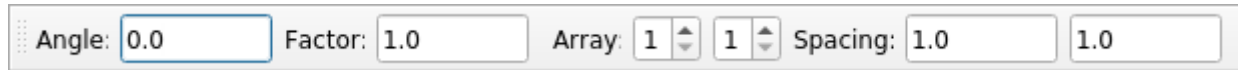
Option Item	Icon	Description
Scale		Displays the current scale of the drawing. The scale can also be changed to a predefined value selected from the drop-down list, or entered as a custom ratio.
"fixed"		Checking the "fixed" checkbox locks the scale to value shown in the text-box.
Apply Print Scale to line width		When enabled, line width is scaled on *reduced* drawings (drawings reduced in size to fit a smaller page).
Toggle Black/White		Toggles the colors of all entities from color to black/white.
Center to page		Centers the drawing and paper to the Print Preview window.
Fit to page		Fits the drawing to the margins of the paper and centers page to the Print Preview window. (Not available when "fixed" is selected.)
Multipages		Determines the number of pages needed to print the drawing based on the defined scale and the paper size selected in *Drawing Preferences*.

Block Operations Tool Options

There are two block insert operations with corresponding Tool Option toolbars. For further details on using blocks refer to *Blocks* in the User Guides.

From Block List

The **Block Insert** capability can be expanded through the *Tool Option* bar features before the block is inserted.

| Angle: 0.0 | Factor: 1.0 | Array: 1 ⇕ 1 ⇕ | Spacing: 1.0 | 1.0 |

Option Item	Description
Angle	Defines the angle of rotation, if any. See *Angles in LibreCAD*.
Factor	Defines the scale factor, if any. It is the same scale factor as in *Modify*.
Array	Defines the numbers of columns and rows to create a pattern of selected block. Otherwise keep 1 for columns and rows.
Spacing	Defines the distance between each column of the array and the distance between each row. The distance is measured between 2 insertion points of 2 adjacent blocks.

From Block Library

Inserting a block from a library can be enhanced through the *Tool Option* bar features before the block is inserted.

| Angle: 0.0 | Factor: 1 |

Option Item	Description
Angle	Defines the angle of rotation, if any. See *Angles in LibreCAD*.
Factor	Defines the scale factor, if any. It is the same scale factor as in *Modify*.

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3.6 Dock Widgets

Dock widgets are small movable windows that serve two purposes: (1) quick access to the drawing tools, and (2) access to additional features not available from a menu including:

- Block List,
- Command Line,
- Layer List,
- Library Browser, and
- Pen Wizard.

As an alternative to the **Tools** menu, the Dock widgets provide more convenient way to access the *Drawing Tools*:

- Circle,

- Curve,

- Dimension,

- Ellipse,

- Info,

- Line,

- Modify, and

- Polyline.
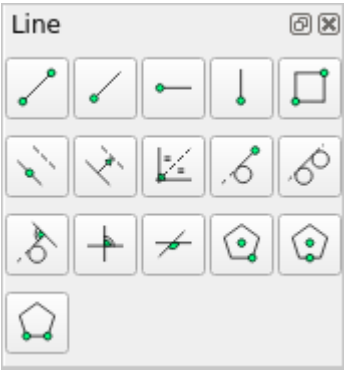
Fig. 5: Drawing tool dock example: Line

3.6.1 Dock Widget Areas

Dock widgets can be moved to different areas by "grabbing" (left clicking and dragging) the title bar of the widget and releasing it in a new location. A widget can be left *floating* inside or outside of the drawing window, or placed in one of the four dock areas (left, right, top or bottom). The Dock Areas *toolbar* allows widgets to be hidden or made visible depending on their location:

Tool	Icon	Description
Left		Shows / hide the dock widgets located on the left side of the drawing window.
Right		Shows / hide the dock widgets located on the right side of the drawing window.
Top		Shows / hide the dock widgets located on the top of the drawing window.
Bottom		Shows / hide the dock widgets located on the bottom of the drawing window.
Floating		Shows / hide the dock widgets floating within the drawing window or outside of the drawing window.

Widgets can be placed either on top of an existing widget in any of the dock areas creating a tab for each of the widgets. Widgets can also be place above or below an existing widget dividing the area into multiple sections.

In addition, dock widgets can be resized by clicking and dragging the edge of the widget's box. A widget has a minimum width of five icons and can be no shorter than the default height.

3.6.2 Block List Dock

The Block List Dock provides the functions to manage blocks and a list of blocks that are active in the drawing. Block functions include:
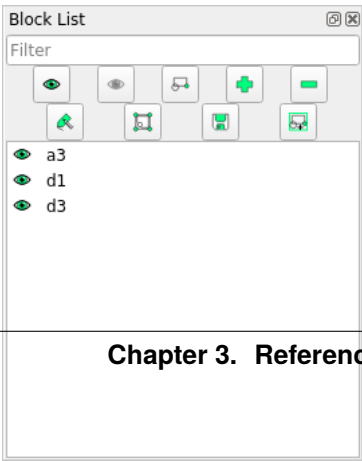
Icon	Description
	"Show all blocks" - *Defreeze*, makes all the blocks in the current drawing visible.
	"Hide all blocks" - *Freeze*, hides all blocks in the active drawing.
	"Create a block" - Creates a block from the selected items.
	"Add an empty block" - Creates an empty block that can then be edited is a separate window (see below).
	"Remove the active block" - Deletes the highlighted block.
	"Rename the active block" - Rename the highlighted block.
	"Edit the active block in a separate window" - Open a new drawing window to edit a new or existing block.
	"Save the active block to a file" - Saves the highlighted block to a file.
	"Insert the active block". - Inserts the highlighted block in the current drawing at the specified reference point

The lower portion of the dock shows a list of blocks in the current drawing. The blocks in the above example are named "a3", "d1", and "d3". More details on creating and using *blocks* can be found in the **User Guides**.

3.6.3 Command Line Dock

The *Command Line* is for users that want to draw by using keyboard commands. Commands, such as "li" for a line, "cir" for a circle, etc, are entered at the command line along with the required parameters (e.g. start and end coordinates for a line). Using the command line can be faster and/or more precise than drawing using exclusively a mouse and toolbars. The available commands are listed with the *Drawing Tools* and *Snapping* tools. There are also commands available for *Edit* and *View* operations.
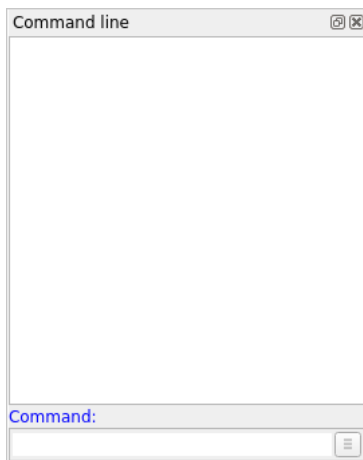
Fig. 7: Command Line dock

The dock includes a drop-down menu that offers additional functions:

- Toggle Keycode Mode off or on.

- Load a Command file.

- Paste commands.

Fig. 8: Command Line dock drop-down menu

Further details on using the *command line* are in the **User Guides**. In addition to command input, the command line provides access to a built in calculator. The calculator can be invoked with the *cal* command. The available *operators and functions* can be found in the **Appendix**.

Note: LibreCAD is designed with emphasis on mouse input and at the moment some options can be only selected by using the mouse as there is no equivalent command.

3.6.4 Layer List Dock

The Layer List Dock provides a list of layers in the current drawing and the functions required to manage them. There are three sections in the dock:

- the list filter,

- the list operators and functions, and

- the layer operators and functions.

The list filter is the input box at the top of the dock. It provides the ability to filter a long list of layer names to help locate a layer. Enter a text string, the name or partial name of a layer, in the input box to filter the layer or layers. Wildcards ("*" or "?") can be used to filter the list to locate similar layer names (e.g. "*01" would show all names ending by "01").

The icons on the top of the layer list allow operations to the entire list of layers. Those operations include:
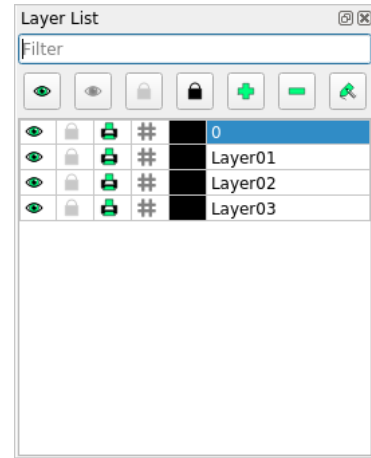
Icon	Function	Description
	Show all layers	*Defreeze*, makes all the layers in the current drawing visible.
	Hide all layers	*Freeze*, hides all layers in the current drawing.
	Unlock all layer	Unlock all layers to allow changes to the entities in the layer. (*)
	Lock all layer	Lock all layers to prevent unintentional changes. (*)
	Add a layer	Add a new layer to the list. (Shortcut [Ctrl]+[L]) (*)
	Remove layer	Remove the highlighted layer from the list. (*)
	Modify layer attributes / rename	Modify the selected layer's attributes and / or rename the layer. (*)

dock example with

The lower portion of the dock shows a list of layers in the current drawing and are listed in alpha-numeric order. In the example above the layers are named "Layer01", "Layer02", and "Layer03". Note that layer **0** is a special layer and should not be used for general drawing purposes.

Icons to the left of each layer act on the layers individually. The layer operations are:

Icon	Description
/	Show / hide the selected layer. (*Defreeze / Freeze*)
/	Lock / unlock the selected layer. (*)
/	Print / don't print the selected layer. (*)
/	Toggle construction lines. (*) A layer designated as a "Construction Layer" is special layer used to create reference geometry to help align other drawing entities. Construction lines are intended as temporary guide lines and drawn to "infinity".
◥	Shows the layer's assigned color (the default is Black).

Layer list or individual layer operations can also be accessed by right-clicking on a layer. Right-clicking in the list window opens a popup menu that provides equivalent operations to the item marked with an asterisk (*).

Clicking the *Attribute* icon allows users to change the *pen attributes* of all entities on the selected layer. Entities that have their own pen attribute set differently will override the layer's attributes.

The attributes include:

Defreeze all Layers
Freeze all Layers
Unlock all Layers
Lock all Layers
Toggle Layer Visibility
Toggle Layer Lock
Toggle Layer Printing
Toggle Construction Layer
...er
...ttributes

Attribute	Description
Layer Name	The default layer name is "O", but any alpha-numeric label can be used. New layers are created with the name of the highlighted layer with a sequence number appended. Layers are sorted in the list alpha- numerically.
Construction Layer	Toggle the construction lines off / on.
Default Pen:	- Color: Select from default or custom colors. - Width: Select from predefined line widths from 0.00 to 2.11 mm. - Type: Select from predefined line types: Continuous, or Dot, Dash, Dash Dot, Divide, Center, or Border (normal, "tiny", "small", or "large").

More details on creating and using *layers* can be found in the "Drawing Setup" section of the **User Guides**.

3.6.5 Library Browser Dock
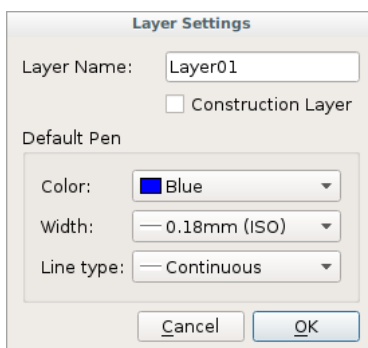
The Library Browser Dock shows blocks available from the defined libraries and allows users to insert blocks into the current drawing. To insert a block, select a block from one of the categories by clicking on it, e.g. "d1" and click the "Insert" button. Specify a reference point in the drawing window with a mouse click or by entering coordinates at the command prompt. Once inserted into the drawing, the block is shown in the *Block List Dock*.

LibreCAD includes several libraries and additional libraries can be specified by defining a path to user libraries in the *Application Preferences*, "Path" tab as shown in **Getting Started**.

Fig. 10: Library Browser dock example

3.6.6 Pen Wizard Dock

The Pen Wizard allows users to create a palette of favorite colors for the drawing tools. Colors can be selected from the existing colors via the drop-down list or created as a custom colors via the [ ] button to the right of the drop-down list. Pressing the "Add to favorites" [+] button to the left will add the color to the list of favorites below. Drag-and-drop the colors in the list to arrange them in the preferred order.

Once colors have been added to the list, set the active pen color by double-clicking a favorite color.

Right-clicking a favorite color allows users to:

- Select all objects of a specific color and clicking on "Select objects"

- Change the color of all selected objects by clicking on "Apply to selected"

- Delete a favorite color by clicking "Remove"

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Fig. 11: Pen Wizard dock example

3.7 Drawing Preferences

Drawing preferences have two purposes, they allow users to :

1. override the application defaults on a drawing by drawing basis.

2. define specifics for the drawing's output, format and other drawing specific configuration.

The preferences can be configured by selecting **Options -> Current Drawing Preferences**. There are five tabs for managing the drawing preferences: *Paper*, *Units*, *Grid*, *Dimensions* and *Splines*.

3.7.1 Paper

The *Paper* tab is used to define the size, orientation and margins of the page used when generating output. The output can be as a physical printed page or an electronic form such as a PDF. Layout of the page with specified settings may be checked in Preview section. The paper format is also used when previewing a drawing (**File -> Print Preview**).

To be able to generate output, users must select a paper size and orientation. Paper sizes include ISO, ANSI and other sizes. Sizes of the select page are shown in the current unit of measurement. Custom sizes can also be select by choosing *Custom* from the drop-down box and specifying the paper width and height.

Orientation can be selected for any page size and is either *Landscape* (long edge horizontal) or *Portrait* (long edge vertical).

Margins determine the printable area of a page. Specified fields at the edges of the page are marked with gray color and always stay empty on the output.

If tiled printing is used to output the drawing, use *Number of pages* section to set a horizontal and a vertical number of pages. For more details about the tiled printing refer to the *Printing Guide*.

3.7.2 Units
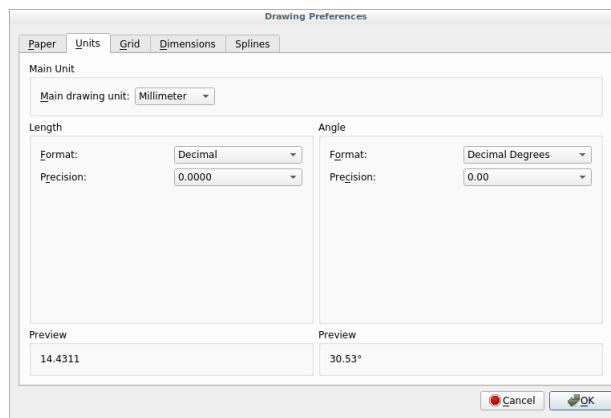
The **Units** tab allows users to set the *Main drawing unit* to the preferred unit of measure and the format of linear and angular dimensions. The "Main Unit" setting overrides the default set during LibreCAD's initial application *configuration*. The same units of measures are as noted in the *appendix* are available for the drawing's preferences.

Note: These preferences format the display of linear and angular units on the *status bar*. They *do not* affect the appearance of dimensions in the drawing. See *Dimensions* below for configuring dimensions' format.

Length Format

Format	Example	Maximum Precision	Description
Scientific	1.44311E+1	0.00000000E+1	Significant x 10 ^{n}
Decimal	14.43112	0.00000000	Integer part separated from the fractional part of a number by a decimal
Engineering	1'-2.43112"	0'-0.00000000"	Feet and decimal inches
Architectural	1'-2 7/16"	0'-0 1/128"	Feet and fractional inches
Fractional	14 7/16"	1/128"	Fractional inches
Architectural (metric)	14.43112 [5]	0.00000000	Decimal metric units (mm, cm, etc...)

Angle Format

Format	Example	Maximum Precision	Description
Decimal Degrees	30.5 °	0.00000000	Integer part separated from the fractional part of a number by a decimal
Deg/Min/Sec	30 ° 32' 0"	0 ° 00' 00.0000"	Degrees [°] / Minutes (', 1/60 of a degree) / Seconds ('', 1/60 of a minute)
Gradians	33.9g	0.00000000g	1/100 of a right angle
Radians	0.5r	0.00000000r	SI unit of measure where the arc of a circle is measured by the length of the radius
Surveyor's units	N30d32'E	N0d00'00.0000'	Cardinal directions measure in deg/min/sec from North and East

3.7.3 Grid
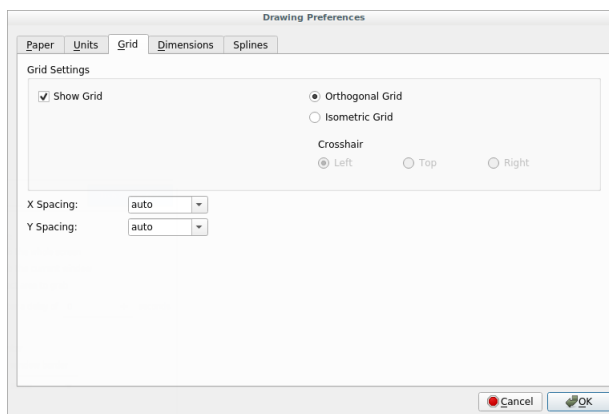
The grid provides an evenly spaced guides to assist with placing entities. When used with *snaps* place can be precise. The **Grid** tab has the following options:

- *Show Grid*: Toggles the grid markers between visible or not visible. The grid can also be toggled with [Ctrl]-g or by using the grid button of the *view* toolbar. This setting does not affect the use of "Snap to Grid".

- Grid X and Y Spacing: Sets the minimum frequency of the grid markers. Values can be selected from the drop-down box. Other values can be typed directly into the text box. *Auto* sets the frequency of markers to a spacing suitable to the current zoom level.

- *Orthogonal* or *Isometric Grid*: Selects the grid to use. *Orthogonal* place the grid at right angles to the X and Y axis. *Isometric* places the grid at 120 ° intervals for guiding *isometric drawings*.

- Cross-hair: Toggles the orientation of the cross-hairs (right, left, or top) when used with *Isometric Snap indicator lines* (see *Application Preferences*).

3.7.4 Dimensions
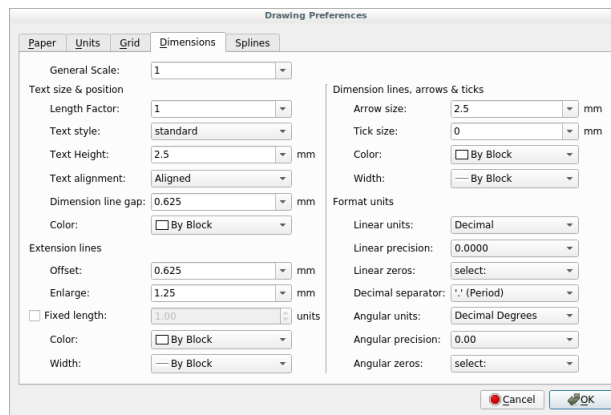
The dimension preferences affect how dimensions appear on the drawings. Settings include:

- General scale:

- Text & position:

- Extension lines:

- Dimension lines, arrows & ticks:

- Format units: setting for linear and angular dimensions. These settings are independent of the preference defined in the *Drawing Prefences*.

Setting	Description
General Scale	Adjusts the *sizes* of the text and arrows by the factor provided.
Text size & position	
Length factor	Adjusts the *dimension value* by the factor provided. The entity remains the length as drawn.
Text Style	Sets the *font* used for dimension text.
Text Height	Sets the text height, measured in the units defined on the *Units* tab.
Text alignment	Aligns the text parallel and offset to the dimension line or horizontal centered on the dimension line.
Dimension line gap	Sets the space between the dimension line and the dimension text.
Color	Set the color of the dimension lines and text.
Extension lines	
Offset	Gap between entity and dimension extension line.
Enlarge	Length of extension line beyond dimension line.
Fixed length	Fixed length of extension line measured from the dimension line towards the dimensioned entity.
Color	Extension line color, independent of layer settings.
Width	Extension line width, independent of layer settings.
Dimension lines, arrows and ticks	
Arrow size	Length of dimension (and leader) arrow.
Tick size	Length of dimension tick to from end of dimension line in each direction, e.g. a length of 1 will result in a total length of 2 units. (Anything greater than ''0" will result in a *tick* instead of a dimension *arrow*).
Color	Tick line color, independent of layer settings.
Width	Tick line width, independent of layer settings.
Format units	
Linear units	(See *Length Format* under **Units** above.)
Linear precision	(See *Length Format* under **Units** above.)
Linear zeros	Remove leading, trailing, 0' and / or 0" from linear dimensions.
Decimal separators	Set the decimal separator to a period [.], or comma [,].
Angular units	(See *Length Format* under **Units** above.)
Angular precision	(See *Length Format* under **Units** above.)
Angular zeros	Remove leading or trailing zeros from angular dimensions.

3.7.5 Splines
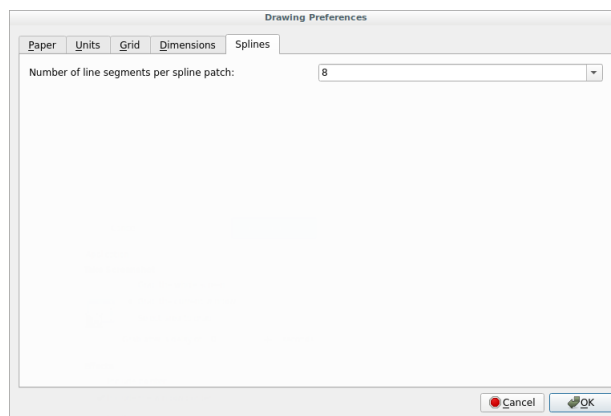
The single parameter *Number of line segments per spline patch* affects the 'smoothness' of a spline. The greater the

value, the 'smoother' the spline will be drawn.

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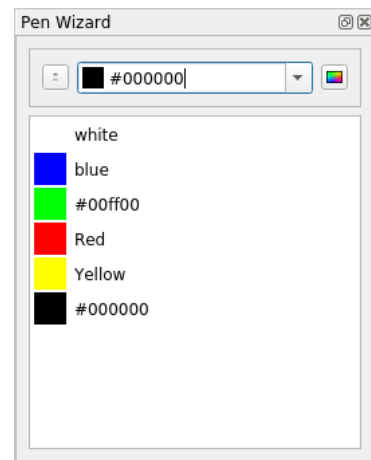

3.8 Customizing LibreCAD's Interface

In addition to be able to *move and reorganize menus and toolbars*, LibreCAD can be customized with user-defined pop-up menus, custom toolbars and alternative "window decorations".

3.8.1 Menu Creator

The Menu Creator is used to create, modify, or delete custom **pop-up menus**. The pop-up menu is actived by using the mouse assigments described below. Open the Menu Creator by selecting **Options -> Widgets -> Menu Creator**
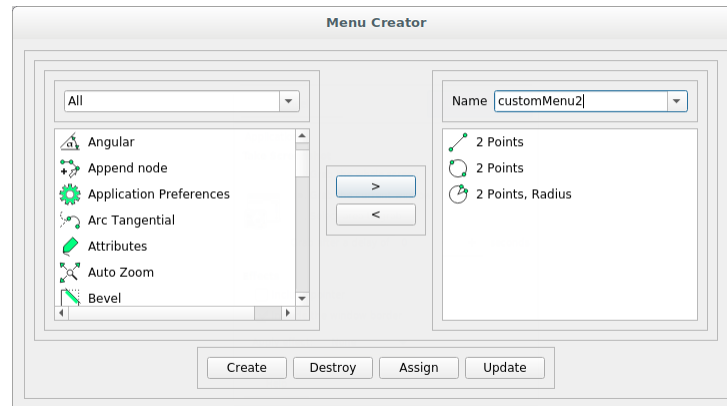
To create a new menu:

1. Input a name for the menu into the **Name** drop-down box.

2. Select an action from the left-hand list and use the right arrow (or double-click on the action) to add it to the new menu (right-hand list).

3. Drag and drop the items on the right-hand list to arrange the actions in the menu.

4. Click the **Create** button.

To edit an existing menu:

1. Select the menu from the **Name** drop-down box.

2. Add additional items as described above.

3. To remove an item from the menu, select an action from the right-hand list and use the left arrow (or double-click on the action) to remove it from the menu.

4. Drag and drop the items on the right-hand list to arrange the actions.

5. Click the **Update** button.

To delete an existing menu:

1. Select the menu from the **Name** drop-down menu.

2. Click the **Destroy** button.

A custom pop-up menu needs to be assigned to a mouse button to make them accessible:

1. Click the **Assign** button

2. Enable one or more of the activators ("Double-Click", "Right-Click", "Ctrl-Right-Click", and / or "Shift-Right-Click") with a check in the check-box.

3. Click the **Save** button
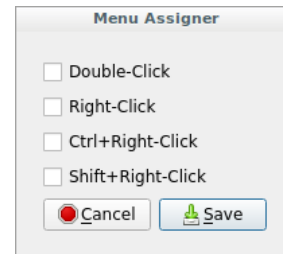
3.8.2 Toolbar Creator

Custom toolbars can be created, modified and deleted using the Toolbar Creator. From the menubar, select **Options ->Widgets -> Toolbar Creator**.
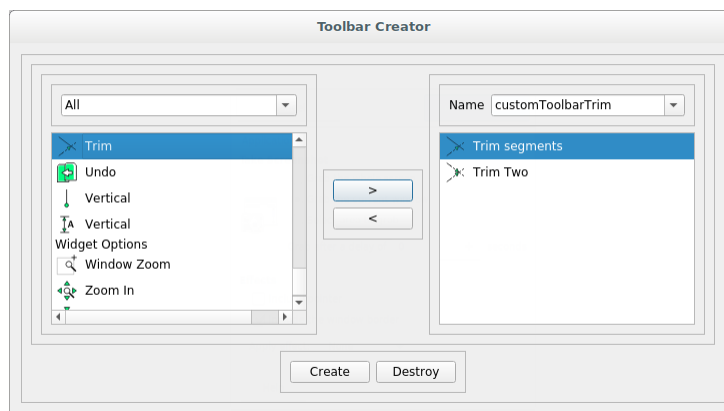
To create a new toolbar:

1. Input a name into the **Name** drop-down box

2. Double-click on an action to add it

3. Drag and drop to arrange actions

4. Click the **Create** button

To modify an existing toolbar:

1. Select the toolbar from the **Name** drop-down box

2. Double-click to remove actions

3. Drag and drop to arrange actions

4. Click the **Create** button

To delete an existing toolbar:

1. Select the toolbar from the **Name** drop-down box

2. Click the **Destroy** button

3.8.3 Widget Options

"Widgets" in this context refer to appearance of LibreCAD's border and title bar, icons and the statusbar. Select **Options - > Widget Options** from the menubar to change these options. The following options are available for:

- General: LibreCAD's application window decorations (borders, buttons, etc.) and permits the use of custom *style sheets* (see below).

- Toolbar: change the *Theme* and size of the toolbars' icons.

- Statusbar: change the height of the statusbar and the size of the font used.

To enable an option, place a check in the checkbox and specify the desired option.
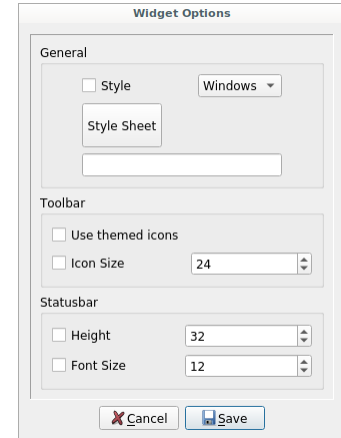
3.8.4 Style Sheets

In "Widget Options" you can also choose a style; the style list is dependent on your operating system and the version of Qt used to build LibreCAD. For example on Windows 7 and using Qt 5 the list is: ("Windows", "Fusion"). More information can be found at http://doc.qt.io/qt-5/stylesheet.html and http://blog.qt.io/blog/2012/10/30/cleaning-up-styles-in-qt5-and-adding-fusion/

Any style of a "Style Sheet" can be modified, but it is recommended that the "Fusion" style (Qt 5) be used as a base, because it is intended to be a cross platform style. After setting a style sheet you can edit it & save and then switch back to LibreCAD and use "Reload Style Sheet" (Ctrl+T).

Tips:

- Start with the example below to obtain and understanding of how style sheet affect the appearance of LibreCAD.

- Look at the examples for "Color" and "Gradient" in the list of property types.

- Check out the widget specific examples at: http://doc.qt.io/qt-5/stylesheet-examples.html

Example

A style sheet can be as simple as:

```
QMenu { font-size: 16px; }
```

Or as a more complex example, save the following text as alpha.qss or alpha.txt, and then load the file with Options -> Widget Options -> Style Sheet:

```
/* alpha.qss v.01 a modification of the Fusion style */

/* Layer List */
QTableView
{
    selection-background-color: #ccffcc;
    selection-color: Blue;
    font-size: 16px;
    font-family: "Arial";
}

QMenu
```

(continues on next page)

```
{
    padding: 4px;
    font-size: 16px;
}

QMenu::item
{
    padding: 2px 25px 2px 20px;
    border: 1px solid transparent; /* reserve space for selection border */
}

QMenu::item:selected
{
    border-color: darkblue;
    background: rgba(240, 255, 255, 150);
}

QMenu::icon:selected
{
    border-color: darkblue;
    background: rgba(255, 255, 255, 255);
}

QToolBar
{
    background-color: rgb(230, 230, 230);
    spacing: 3px;
    padding: 4px;
}

QToolButton
{
    background-color: #eeeeee;
    border-style: outset;
    border-width: 2px;
    border-radius: 2px;
    border-color: beige;
    font: 12px;
    padding: 2px;
}

QToolButton:checked
{
    border-color: grey;
    background-color: qlineargradient(x1: 0, y1: 0, x2: 0, y2: 1,
                                      stop: 0 #dadbde, stop: 1 #f6f7fa);
}

QToolButton:hover
{
    border-color: grey;
    background-color: qlineargradient(x1: 0, y1: 0, x2: 0, y2: 1,
```

```
                                        stop: 0 #dadbde, stop: 1 #f6f7fa);
}

QStatusBar { background-color: azure;}

QMenuBar { background-color: #fefefe; }

QTextEdit { background-color: honeydew; }

QToolTip { background-color: white; }
```

LibreCAD v2.2.0 - User Manual

This is an **interim release** *of the of the* **LibreCAD User Manual** *and is subject to change.*

The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

FOUR

USER GUIDES

The **User Guide** brings together many other parts of the user manual and will show how to use a variety tools to create and modify drawings. The user guides provides examples of tasks performed with LibreCAD. The tasks are generic in the sense that they can be applied to any drawing. The examples provided in the guides build on the previous examples and when complete will provide a finished drawing that uses many of LibreCAD's features. Be sure to read through the *Reference* section, particularly the *Fundamentals*, the *Drawing Tools* and the *Snapping* sections to obtain an understanding of the basic operation of LibreCAD and its various tools.

The example drawings in the **User Guides** use LibreCAD's default configuration with the exception of the drawing window's background color. It has been changed to light grey (#e6e6e6) in **Options -> Application Preferences -> Appearances** tab, "Background" color under "Graphic Colors". The **Exercises** provide the opportunity to use the examples to create the sample drawing used in the **User Guide**.

LibreCAD v2.2.0 - User Manual

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4.1 Setting up a Drawing

As with other software; word processors, spreadsheets, etc, there are many ways a user can to setup or configure preferences for a new document. With LibreCAD, that document is a drawing. Some of the preferences for a drawing will be governed by drafting conventions, some will be determined by organizational requirements, and others might just be personal preferences. The drawing's configuration is largely determined by the *Drawing Preferences*.

Under normal circumstances, after the initial installation and configuration of LibreCAD, little if any additional configuration needs to done before creating a drawing. However, there are a many drawing parameters that can be changed to suit the user's requirements and the drawing's final appearance. The majority of these settings can be left as the defaults as LibreCAD's defaults reflect normal drafting conventions and practices, such as the "Text Height" of 2.5 mm / .10" / 3/32", etc. However, these and many other preferences and drawing elements, such as fonts, layer, line thickness and type, pen colors, etc. can be set to suit users' requirements.

Some of the key considerations for setting up the drawing include:

- Drawing units
- Page size
- Scale and dimensioning format
- Layers and "pens"

4.1.1 Drawing Unit

During the initial setup, the default *unit of measure* was set (defaulting to *millimeter*). The unit of measure should be set to the unit most frequently used as it is used for all new drawings and doesn't need to be changed. If necessary, the default unit of measure can be changed in the *Application Preferences* or overridden for a single drawing in the *Drawing Preferences*.

4.1.2 Scale and Dimensioning

Setting the scale of a drawing is the easy part, drawings should be created *full-scale* (1:1)! The zooming abilities of LibreCAD will make the whole drawing fit in the drawing window or magnify sections to view fine detail. On the other hand, when producing output the drawing will need to be adjusted in size to fit the *page size*. Generally, output is a printed page, but it can also be a pdf, or *exported to another image format*.

As an example, for an annotated drawing of a floor plan to appear correctly when the drawing is reduced in size, or *scaled down*, to print on an "A4" page (210 x 297 mm), the dimension text would need to be proportional to the floor plan itself prior to being reduced to fit the page. Trying to determine the dimension text size for scaling such a large object would be tedious at best, but a feature of LibreCAD makes it simple. The "General Scale" is used to adjust the dimension text, arrows and related parameters to the sizes suitable for the required page size. Both the page size, and the dimensioning parameters - including the "General Scale" - are configured in the *Drawing Preferences*.

There are two tabs in **Drawing Preferences**, "Paper" and "Dimensions", that require attention for a new drawing. Specifically the items that need to be addressed are:

1. On the "Paper" tab: Selecting a paper size and orientation will determine the *print scale* used for final output. The print scale is used to determine the "General Scale".

2. On the "Dimensions" tab: Setting the "General Scale" will adjust the dimension text and related parameters to suit the page size.

The paper format, orientation, and margins to be used is an important to consideration when setting the drawing preferences. The page size is entirely up to the user to determine based on what is available (depending on the printer or printing service that is being used). While it can be done at anytime, establishing the page size sooner than later will help determine the "General Scale".

Tip - Determining Scale

If multiple options are available for the paper size, setting the paper size *after drawing the object but prior to dimensioning the drawing* will help determine the print scale, the "General scale", and subsequently the appropriate line spacing for dimensions. See *Dimensioning* for additional information.

While any scale can be used when printing a drawing there are commonly used scales for different types of drawings. Refer to *Scales* in the appendix for some examples.

Be sure to allow room on the drawing for dimension lines and text when determining the print scale. More details on printing a drawing are found in the *Printing Guide*.

4.1.3 Layers

Using layers to organize drawings and assign *pen* attributes to the entities on the layer is an important concept. Refer to *Layers* in **Fundamentals** and the *Layer List Dock* under **Dock Widgets** in the **Reference** section for additional details on using layers.

In LibreCAD, layers are managed using the **Layer List Dock**. Use the **Layer List Dock** to add and remove, show and hide and modify the layer's attributes. Creating a layer is simple:

1. Click the **Add a layer** icon .

2. Specify a *Layer Name*.

3. Optionally specify the Pen Color, Width and Line Type for the layer.

4. Click **Ok**.

The first layer added defaults to the layer name "noname", but the name can be replace with any alpha-numeric text. Additional layers add will adopt the name of the currently selected layer and append a sequential number, but can also be renamed.

Tip - Using Layers

Layer **0** is a special layer and should not be used for general drawing purposes. Create at least one additional layer for the drawing.

Prefix the layer name with a sequential number when naming the layer to help sort the layers in the dock list. Layers are listed in alpha-numeric order, e.g. 1a, 1b, 2a, 2b, etc.

Use "Construction Layer" to create reference geometry to help align other drawing entities.

Layers that have been completed can be *locked*. Locking layers prevents accidental changes and can improve the performance when working with very large complex drawings.

Hiding layers while drawing reduces the *visual complexity* and makes it easier to focus on the current drawing efforts.

4.1.4 Templates

Having to set the unit, page size, layers, etc. each time a new drawing is created can be avoided by using *templates*. Templates are *prototype* drawings that provide a method to save a drawing's configuration so it does not need to be defined each time a new drawing is started. A template's configuration can include the settings defined in the **Drawing Preferences**, such as the page size, unit of measure, and the dimensioning format. Templates can also include layers and other drawing elements such as a border and / or a title block. These settings and drawing elements are inherited by a new drawings created from the template.

When LibreCAD is launched it creates a new drawing using a *default template*. The default template is also used when a new drawing is created within LibreCAD by selecting **File -> New** or clicking the **New** icon . The new document is initially called "unnamed document 1". Any addition new drawings created while LibreCAD is still open will be number sequentially; "unnamed document 1", "unnamed document 2", and so on. New drawings inherit the template drawing contents, but do not take the template's name as a file name. Users are prompted for a file name when saving the drawing.

User-defined templates can be created to suit various purposes such as different page sizes or orientation, unit of measure, etc. User-defined templates can be used by selecting **File -> New From Template** or clicking the **New from Template** icon and then selecting the desired template file. See "Creating Templates" below.

Creating Templates

Templates are created starting with a new drawing, setting the desired *Drawing Preferences*, and adding any required drawing elements (e.g. layers, borders, etc). To create a template, starting with a blank drawing:

1. Set the the drawing preferences by selecting **Options** from the menu bar and then **Current Drawing Preferences**.

    1. On the "Paper" tab set the paper size, orientation and margins as desired.

    2. Select the "Units" tab and set the "Main Drawing Unit". (Note the "Length" and "Angle" settings configure the *status bar* format and not the dimensioning format that appear on the drawing.)

    3. Click the "Dimensions" tab and adjust the values.

    4. Check the remaining tab, "Grid" and "Splines" and adjust those settings if necessary.

    5. Click "OK" when done.

2. Add any layers as needed. See *Layers* for details on using layers and setting the attributes.

3. Add any other drawing elements (borders, title boxes, etc) as required.

4. Save the template as a "dxf", e.g. "myTemplate.dxf", to any location with read / write permissions.

LibreCAD supports the use of multiple templates. A user that plans on creating similar drawings may require only a default template created to suit their needs. Users that create several different types of drawings may desire multiple templates. For example, templates can be created for each paper size available and / or for each paper orientation.

Default Template

LibreCAD includes a default template, "empty.dxf", that is installed with the application and found in the resources directory. It is used initially as the default template for all new drawings. A user-defined template can be used instead of the default by specifying the new template in the *Application Preferences*. From the **Options** menu select **Application Preferences** and click on the "Paths" tab. Enter the path and filename of the template to be used as the default. The specified template is used instead of the default LibreCAD template when the application is launched and for new drawings.

LibreCAD v2.2.0 - User Manual

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4.2 Creating and Editing Entities

Drawing geometric shapes, or *entities* is familiar to many people that have used drawing and paint programs. As with many programs, there are a multitude of ways to generate the desired results. LibreCAD is the same - there are many ways to obtain the same results in a completed drawing - far too many to be able document an example for each one in a User Guide. For users it is important know what tools are available and how to apply them.

This section of the **User Guide** brings together many other parts of the LibreCAD manual and will show how to use a variety tools to *create* and *modify* drawings. Be sure to read through the **Reference** section, particularly the *Fundamentals*, the *Drawing Tools* and the *Snapping* sections to obtain an understanding of the basic operation of LibreCAD and its various tools. It is also helpful to understand the different *View* tools and commands.

Note: The examples below use both the the command line and the mouse for input. In cases where the command line is used and additional input is required because of a **Tool Option** (see **Drawing Tools**) see the *note* below the command text. Where the mouse is the primary form of input, the instructions are provided in a numbered list.

4.2.1 Creating Entities

Creating new geometric entities all use similar methods. Generally, creating an entity requires placement of a minimum of two points (Well, unless the entity being drawn is a point), or a single point and an additional parameter such as length or radius. In some case the same result can be obtained using several different methods. For example, drawing a vertical line can be done via keyboard entry using the *command line* to select a tool and then define two points with the *absolute* or *relative Cartesian* coordinates:

Using absolute Cartesian coordinates:

```
li
10,10
10,110
k
```

The first set of coordinates place the starting point of the line at 10 units horizontally and 10 units vertically from the origin (0,0). The second set places the end point of the line at 10 units horizontally and 110 units vertically from the origin (0,0), resulting in a vertical line 100 units long.

Or, using relative Cartesian coordinates:

```
li
10,10
@0,100
k
```

The "@" designates *relative coordinates*. The *@0,100* relative Cartesian coordinates places the next point 0 units horizontally (X axis) and 100 units vertically (Y axis) from the point previously placed at 10,10.

It can also be done with relative *polar* coordinates:

```
li
10,10
@100<90
k
```

The "<" designates a *polar coordinate*. The *@100<90* polar coordinate places the next point 100 units 90 degrees from the X axis of the point previously placed at 10,10.

The same line can also be drawn using just the mouse:

1. Enabled **Snap on Grid** .

2. Select the **2 Points** line tool .

3. Click at *10,10* (use the *Status Bar* to locate the absolute coordinates)

4. Click at *10,110*

5. Press [Esc] or right-click twice to exit the command.

Other tools can also be used achieve the same result:

```
ver
10,10
```

Note: Ensure the *Length* is set to "100" and the *Snap Point:* is "Start" in the "Tool Options" text box.

All the above examples create a line of the same length with the same start and end points. These examples help illustrate the multiple ways to achieve the same result. The use of a particular tool may be determined by the next operation that allows the drawing to be quickly extended. A good understanding of the available tools allows the user to select the appropriate tool for the current operation.

While the above example always start at *10,10*, the initial point can be placed anywhere:

```
li
10,110
10,10
k
```

Drawings are generally created with multiple lines segments with the end of one segment being the start of another. Lines can be drawn with multiple connected segments by using the end of the current segment as the starting point and defining the end point of the next segment. Further, additional points can be defined using any of the methods previously shown:

```
li
10,110
10,10
@125,0
@50<90
k
```
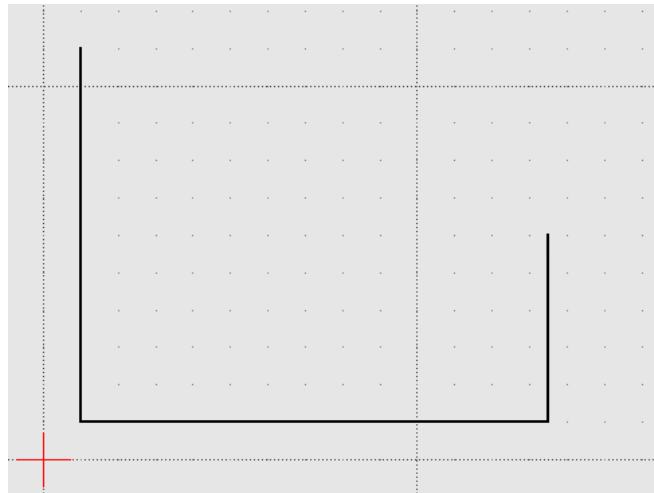

Fig. 1: Example #1

Using the mouse, or another pointing device, along with the snapping tools provide an alternative to using the command line for creating drawings. As with the command line, there are multiple line tools that can be used to produce the same result. For example, adding the to the the previous example, a horizontal line can be added using various methods:

1. Click the **2 Points** line tool icon .

2. Enable the **Snap on Endpoints** and click close to the end of the line at *absolute coordinate* 10,110.

3. Drag the mouse to the right and, with the **Snap on Grid** enabled, place a point at *absolute coordinate* 85,110. Clicking the mouse should result in a horizontal line ending at precisely at 85,110.

4. Press [Esc] or right-click to exit the command.

Hint: It may be necessary to "Zoom In" to ensure the grid is at the correct resolution to be able to accurately place a line's start or end point.

A combination of the **2 Points** line tool the **Restrict Horizontal** tool can also be use as an alternative to **Snap on Grid** .

Alternatively, a horizontal line can be added:

1. Click the **Horizontal** line tool icon . On the "Tool Options" tool bar specify a length of 75 units and the "Snap Point" at the "Start".

2. With the **Snap on Endpoints** enabled click close to the end of the line at *absolute coordinate* 10,110. Clicking the mouse should result in a horizontal line ending at 85,110.

3. Press [Esc] or right-click to exit the command.
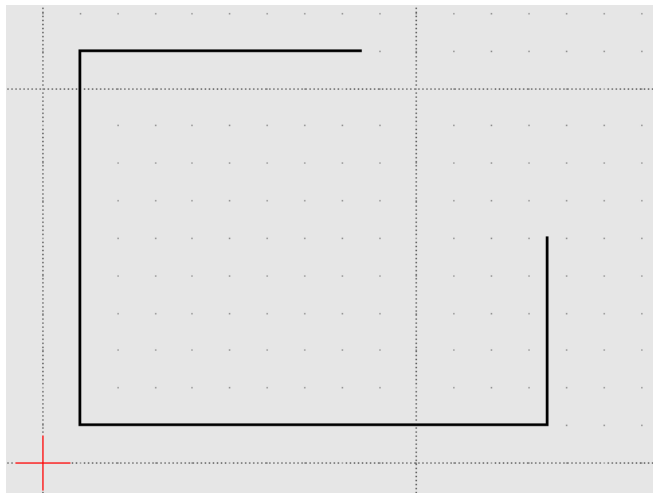
Fig. 2: Example #2

Being that the end points of the existing lines have been established, the outline can be completed quickly with the addition of a line:

1. Click the **2 Points** line tool icon .

2. Enable the **Snap on Endpoints** and click close to the end of the line at *absolute coordinate* 85,110.

3. Drag the mouse to the right and down and place a point close to 135,60. Clicking the mouse should result in a line angled down and to the right, closing the object's outline.

4. Press [Esc] or right-click twice to exit the command.

Whichever of the above methods is used, the result should be similar to:

Another option is to draw the line at the desired angle:

1. Click the **Angle** line tool icon . On the "Tool Options" tool bar specify an *Angle* of "135", a *Length* of "80" units and the *Snap Point* at the "Start".

4.2. Creating and Editing Entities 75
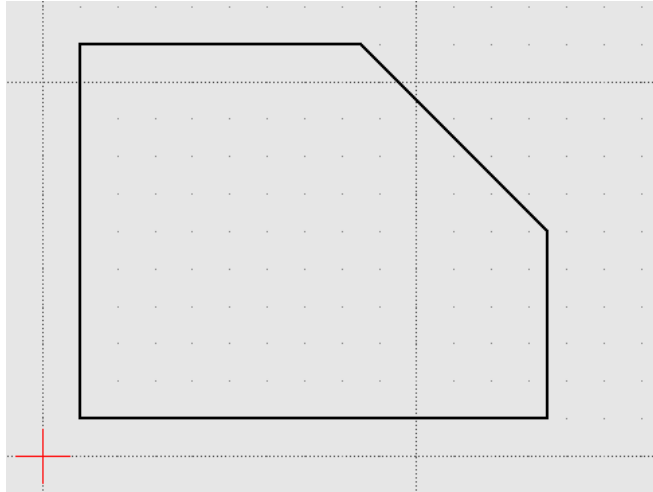
Fig. 3: Example #3

2. With the **Snap on Endpoints** enabled, click close to the end of the line at *absolute coordinate* 135,60. Clicking the mouse should result in a line angled up and to the left.

3. Press [Esc] to exit the command.

This option will result in a image similar to what is shown above, but with the diagonal line being a bit too long. The line can be *trimmed* to suit (see "*Modifying Entities*" below):
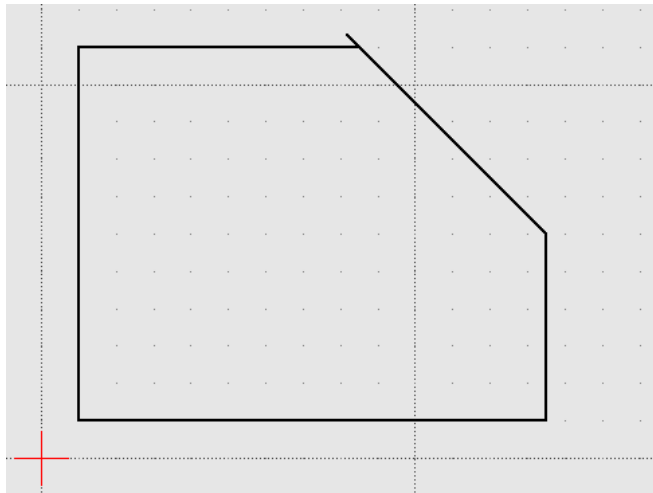
Fig. 4: Example #3a

Alternate Approach

All of the above examples create the object by drawing individual lines. A completely different approach is to start with a *rectangle*:

```
rec
10,10
@125,100
k
```
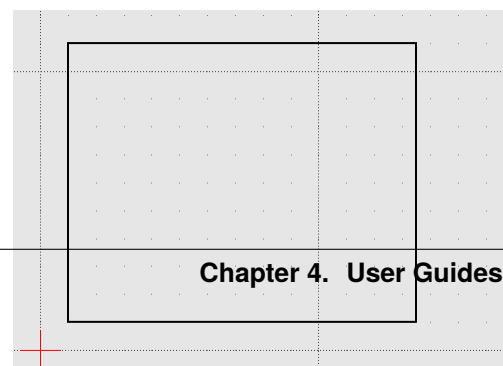
And then modify it with the "Bevel" tool (**see below**).

Circles can be added in a similar fashion. It can be drawn with the "Center, Point" tool by specifying the coordinates for the centre of the circle and of a point on the circumference:

```
ci
50,70
@0,20
k
```

A circle of a given size can also be drawn with a known radius:

1. Click the *Circle, Radius* tool icon . On the "Tool Options" tool bar specify a *Radius* of "40".

2. With the **Snap on Grid** enabled, place the centre of the circle at *absolute coordinate* 50,70.

3. Press [Esc] or right-click to exit the command.

The drawing should now appear as:
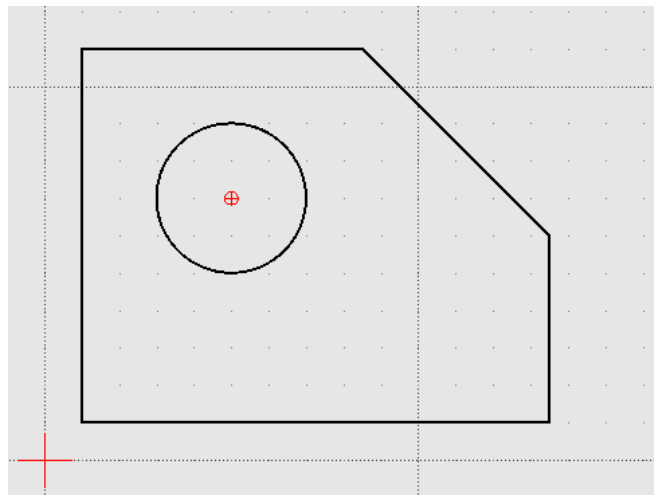
Fig. 6: Example #4

4.2.2 Modifying Entities

There are a variety of tools that can be used to edit and modify existing entities. The tools are found in the **Tools -> Modify** menu or as a drawing tool *dock widget*. These tools allow entities, depending on the type, to be moved, rotated, scaled, mirrored, lengths increased or decreased, divided (i.e. split), etc. A complete list and descriptions of the tools can be found in the *Drawing Tools - Modify* reference section.

Alternate Approach

Continuing with the alternate example - starting with a rectangle - the shape can be modified as required with the **Bevel** (or "chamfer") tool. Its operation is similar to the fillet tool:

1. Click the **Bevel** tool icon . Ensure with "Trim" is checked, and "Length 1" and "Length 2" is set at "50" in the "Tool Options".
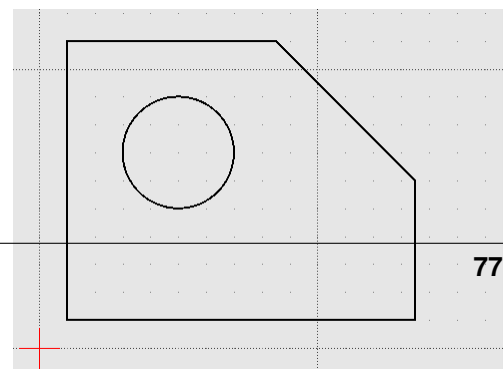
2. As prompted on the Status Bar, select the first entity (the top horizontal line of the rectangle), and then

3. the second entity (the right vertical line of the rectangle).

4. Press [Esc] or right-click to exit the command.

A rounded corner can be added to the drawing's the lower left corner with the **Fillet** tool:

1. Click the **Fillet** tool icon . Ensure with "Trim" is checked and "Radius" is set at "10" in the "Tool Options".

2. As prompted in the "Command Line" dock, and on the Status Bar, select the first entity (the bottom horizontal line of the rectangle), and then

3. Select the second entity (the left vertical line of the rectangle).

4. Press [Esc] or right-click to exit the command.

The drawing should appear as:
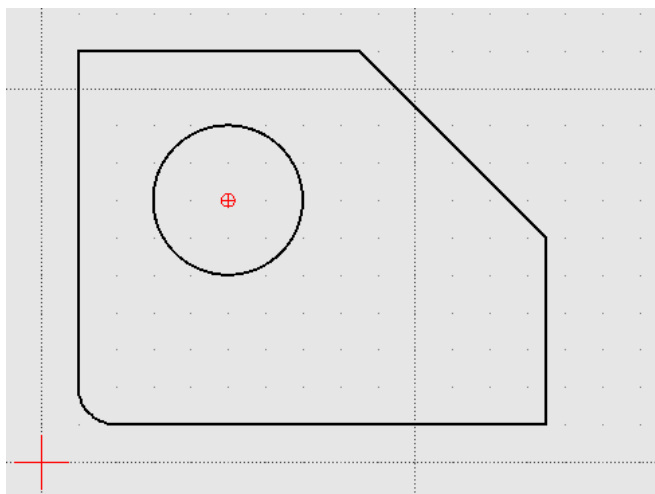
Fig. 8: Example #5

Example 3a above left a diagonal line that is too long. The length can be easily trimmed:

1. Click the "Trim" icon

2. Click the top horizontal line. This line is the "limiting entity" that determines where the second line is going to be trimmed to.

3. Click the line to be trimmed, the "entity to trim" anywhere along the line that is to be kept (below the top horizontal line).

4. Press [Esc] or right-click to exit the command.

Important: These examples do not provide an example of every tool available in LibreCAD, but is intended to show the basic operation of some of the drawing and modifications tools. Most of the other drawing and modifying tools work in a similar manner. Being familiar with the *Drawing* and *Modify* tools in the **Reference** section will help determine what tool can be used in a particular situation.

These examples also illustrate that there are multiple ways to achieve the same result using a variety of methods. There is no one best method. The particular method used may depend on the state of the drawing and how existing entities can be used to build on, or perhaps it is a simple matter of using a preferred drawing / modifying tool.

Hint:

Working on *circle* and *arc* with the `Trim` command can be confusing, and not working in the way we wish. At certain constellations it might work, at others it doesn't.

Circles and *arcs* have running directions, usually counterclockwise from starting to ending point. For a *circle* start/end are both at 0 degree (center in +X direction).

It's generally easier to cut them up with `Divide` at the desired points and `Delete` the unwanted parts.

Changing Attributes and Properties

As shown in the *Entities* section in **Fundamentals**, an entity consists of "Pens" (color, width, line type) and "Layers". These *attributes* can be changed using one of two *Modify* tools:

- **Attributes**: allows the "Pen" or "Layers" to be modified for one or more entities.

- **Properties**: allows the "Pen", "Layers" or geometry *of a single entity* to be modified.

Both tools operate in a similar fashion and for similar purposes, but there are a couple of key differences. The **Attributes** tool allows a change to the attributes to be applied to *one or more selected entities* while the **Properties** tool can only be used for a *single entity*. In addition, the **Properties** tool allows the *geometry* to be edited. The geometry of an entity will vary be the type of entity. For example a line's geometry consist of the X and Y coordinates of the endpoints, while a circles geometry consists of the X/Y coordinates of the centre of the circle and its radius.

Layers and Pens

Change an entity's layer is similar with both the **Attributes** and **Properties** tools.

Using the **Attributes** to change an entity's layer:

1. Select the entity (or entities) to be moved to a different layer.

2. From the menu select **Tools -> Modify -> Attributes**, or click the **Attributes** icon .

3. In the *Attributes* dialog, select the desired layer from the drop-down *Layer* selection box.

4. Click **Ok**.

Hint: Entities can also be moved from one layer to another by selecting one or more entities and then selecting the new *destination* layer in the **Layer List** dock. To use this method the *Modify layer of selected entities, at layer activation* option on the **Application Preferences Defaults** tab must be enabled.

In a similar manner the color, width and/or line type can be changed:

1. Select the entity (or entities) to be moved to a different layer.

2. From the menu select **Tools -> Modify -> Attributes**, or click the **Attributes** icon .

3. In the *Attributes* dialog, select the desired pen attribute from the drop-down *Color, Width* and/or *Line type* selection box.

4. Click **Ok**.

The **Properties** tool operates in a similar manner, but the tool need to be selected *before* selecting an entity:

1. From the menu select **Tools -> Modify -> Properties**, or click the **Properties** icon .

2. Select the entity.

---

3. In the *Properties* dialog, select the desired layer or pen attribute from the appropriate selection box.

4. Click **Ok**.

Geometry (Properties)

The **Properties** tool also allows the *geometry* of an entity to be changed. The geometry is the information used to describe the entity. The geometry available depends on the type of entity, for example:
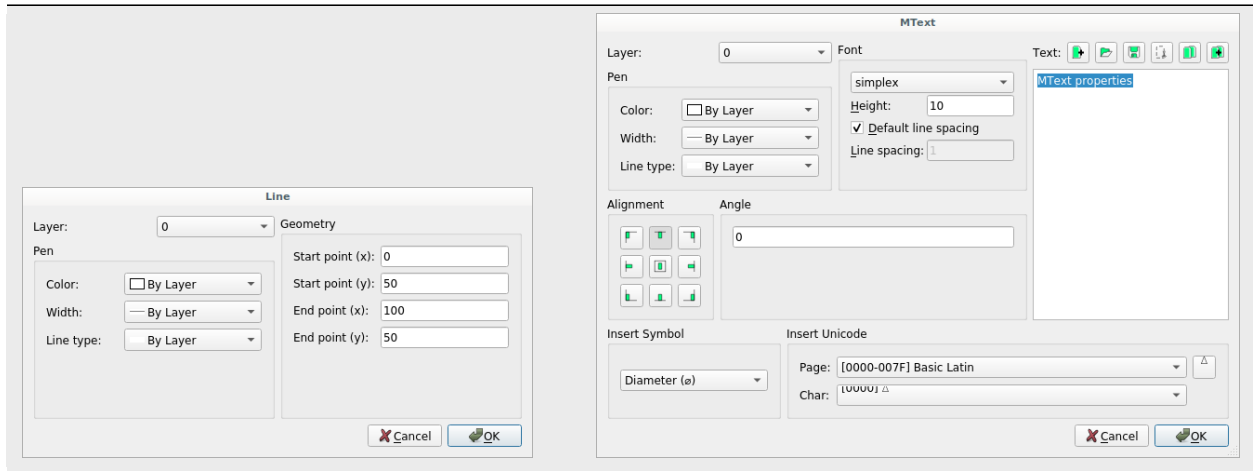
Some entities, such as a polyline, have limited properties available that can be changed. Other entities, such as Text, have many properties that can be changes (including the text itself).

Also, the properties of a specific entity type, e.g. line, does not vary even if the specific tool used to create the entity varies. A line drawn with the **2 Point** line tool will have the same properties as a line drawn with **Angle** tool.

Entity Type	Drawing Tool	Properties
Line	2 points, Angle, Horizontal, etc	Start and end point X/Y coordinates
Circle	"Centre, Point", 2 Points, etc	Center point X/Y coordinates, radius
Curve	"Center, Point, Angles", 3 Points, etc	Center point X/Y coordinates, radius, start/end angle
Ellipse	Ellipse (Axis), Ellipse Foci Point, etc	Center point X/Y coordinates, major/minor axis, rotation, start/end angle
Polyline	Polyline, Rectangle	Open or closed
Text	Text	Text, font, text height/angle/width factor, alignment, special characters
MText	MText	Text, font, text height/angle/line spacing, alignment, special characters
Dimension	Aligned, Linear, etc	Label, special symbol
Blocks	Block List, Library Browser	Insertion point X/Y coordinates, scale X/Y, rotation angle, array rows / columns and spacing

4.2.3 Drawing Views

LibreCAD supports two drawing views to represent three-dimensional objects:

- Orthogonal: Uses multiple two-dimensional views.

- Isometric: Represents a three-dimensional view in two dimensions.

Orthogonal View

Orthogonal drawings use multiple views to provide a complete description of an object. Each view of an orthogonal drawing is parallel to the axes. Normally a front (the most complex view), top and right view are added to the drawing. Additional views; back, left and bottom can be added if necessary.

Orthogonal drafting beyond the scope of the **LibreCAD User Manual**, but additional resources and examples are available in LibreCAD's wiki or elsewhere on the web.

4.2. Creating and Editing Entities
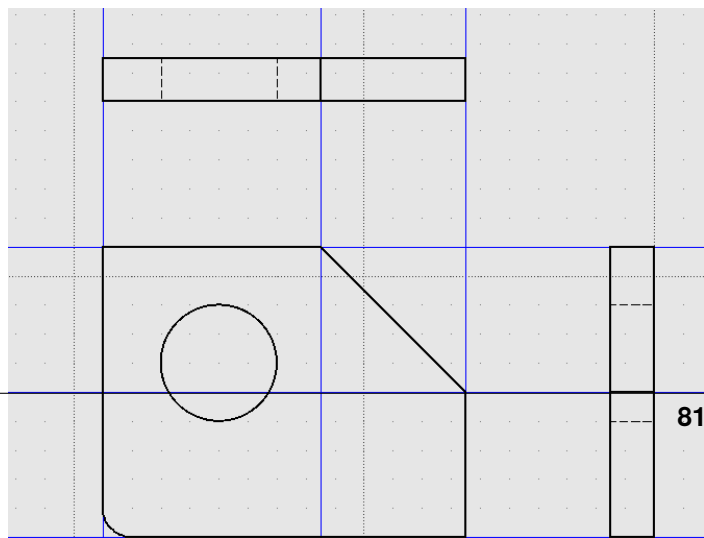
Isometric View

Isometric drawings represent an object on three axes in a two-dimensional drawing. Isometric drawings uses the same tools for creating, modifying and changing attributes as an orthogonal drawings, but on a grid configuration specific for an isometric perspective. Set the grid to suit an isometric perspective on the **Grid** tab of *Drawing Preferences*. Selecting "Isometric Grid" displays the grid on three axes (X,Y,Z) allowing 3 dimensional drawings to be drawn in a 2D view.

In addition to setting the grid for isometric drawings, the "Snap indicator lines" on the **Appearance** tab of *Application Preferences* can be set to "Isometric" to assist in with locating entities on the grid. The "Isometric" crosshairs can be configured to Left, Top or Right to further aid in locating points on the grid.

Isometric drafting does require some techniques unique to isometric projection and is beyond the scope of the **LibreCAD User Manual**, but additional resources and examples are available in LibreCAD's wiki or elsewhere on the web.

LibreCAD v2.2.0 - User Manual

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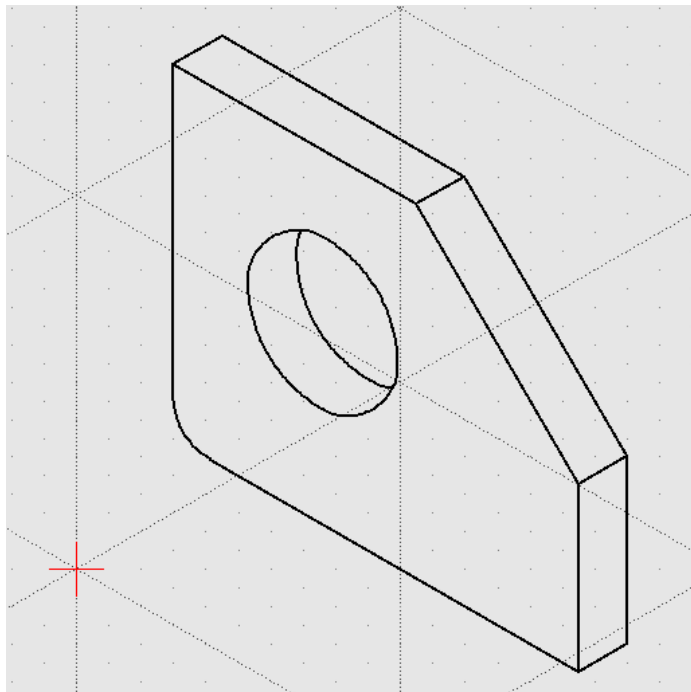
Fig. 10: Doohickey - Isometric View

4.3 The Command Line

LibreCAD's command line offers users an alternative to using the mouse to select tools and draw. Using the keyboard to select tools and enter coordinates can provide greater speed and accuracy when creating drawings. The command line provides other useful features not available using the mouse:

- Multi-command input
- A calculator

The command line is displayed in its own *dock widget* and consists of three components

- Command prompt.
- Command output window.

- Button that displays a drop-down menu that includes:

    - Toggle *Keycode Mode* off or on.

    - Load a Command file.

    - Paste commands.

4.3.1 Command Line Functions

The commands available to use are shown in the *Drawing Tools* reference. The command line is activated by in a variety of ways:

1. Start typing any command (e.g. li, rect, etc.) and then [Enter] or the [Space bar].

2. Press the [Space bar], type any command, and then [Enter] or the [Space bar].

3. Press [Ctrl + M], any command, and then [Enter] or the [Space bar].

4. Press [Shift], any command, and then [Enter] or the [Space bar].

5. Press [F1], any command, and then [Enter] or the [Space bar].

6. With the *Keycode Mode* on, type a **two letter** command (e.g. li, ci); pressing [Enter] is not required.

When the command line is activated, the prompt above the input text-box (initially showing "Command:" in black text) turns blue. After typing a command, the prompt will indicate the next input required, such as coordinates or the next action available.

Pressing [Esc] will cancel the current action and pressing it a second time will cancel the operation. For example, when using the **2 Points** line tool, the first prompt shows "Specify first point" and the second "Specify next point". After drawing at least two segments of a line, the next prompt reads "Specify next point or [close/undo]". LibreCAD is expecting another set of coordinates to be entered, or the shape (with a minimum of two segments) can be closed or the last actions can be reversed. close or undo can be entered on the command line, or by clicking on buttons on the *Tool Options* toolbar.

For example, to draw a square using the **2 Points** line tool from the command line:

```
Command:
li
Specify first point
0,0
Specify next point
@10,0
Specify next point or [close/undo]
@0,10
Specify next point or [close/undo]
@-10,0
close
```

Tip - Using the Command Line

- In addition to the comma separated coordinates, *relative coordinates* can also be entered using the format *X..Y*, i.e. typing 10..20 is equivalent to @10,20. For users who prefer entering numbers using the numeric keypad, using the two decimals is faster than typing the comma.

- The *Keycode Mode* permits the use of **two letter** commands and eliminates the need to press [Enter] after typing the command.

- Pressing the [Space-bar] is an alternative to pressing [Enter] after each command.

- Typing c or u followed by [Enter] can be used instead of typing close or undo.

- *Tab completion* can be used on the command line when entering commands. Typing a partial command such as cir followed by [Tab] will complete the command to circle. If the partial text is not unique to a single command, the command output will show all the possible commands starting with the text provided. For example, typing c and pressing [Tab] will list circle, circle2, circle3, circlecr, and cut in the command output.

- Coordinates can be expressed in arithmetic expressions. See *Calculator* section or *Calculator* appendix.

The available commands are shown in the *Tools* reference. Many of the commands have multiple forms. For example the *2 Points* line tools can be selected on the command line by typing l, li, or line. Many tools display the **Tool Options** toolbar when selected. Some tools will also provide command line prompts in addition to the **Tool Options**. For example the "Parallel" line tools displays:

- a command prompt: Specify Distance <10> or select entity or [through]

- a **Tool Options** toolbar: Distance 10 Number: 1
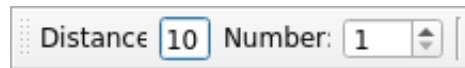
Either can be used can be used to enter new values. The current value on the command line is displayed in angle brackets as shown above. To change the value from the command line, type the value and press [Enter].

The command output window displays the command history, error messages, and other output (see **Calculator** below). The text in the output window can be copied simply by highlighting it. The text is automatically copied to the clipboard and can be pasted into another document. The output window can be cleared of all text by typing clear in the command line.

4.3.2 Multi-Command Input

Command input can be combined and entered on a single line by separating the commands and other input with semicolons. Entering li;0,0;10..0;0..10;-10..0;c;k on the command line will draw a 10×10 square. A list of commands, separated by a newline (return), can be copied and pasted using **Paste multiple commands** from the command line button (lower right corner of the **Command Line Dock**).

Command input can also be loaded from text files. Entering the commands and other input into a text file separating each with a newline. For example, create a text file and enter the following commands:

```
li
0,0
@10,0
@0,10
@-10,0
c
k
```

Save the file as multiCmd.txt. In LibreCAD select "Load Command File" from the the drop-down menu by clicking the command line button. Locate the file and click the **Open** button. The above commands will draw a 10×10 square.

Multi-command input can be assigned to a variable and variables can also contain other variables (they are read recursively):

```
a=ci;0,0;10
b=ci;10,0;10
c=\a;\b;kill
\c
```

Enter each line of the text above on the command line. When \c is entered, two overlapping circles with a radius of 10 units are drawn. The \ character is an escape character that allows the command line to interpret the variable name as an action. In the above example, \c expands to ci;0,0;10;ci;10,0;10;kill.

A "variable file" can be set to load at startup via *Application Preferences* **-> Paths -> Variable File**. Save the first three line of the above example to a text file and configure the path to the text file. Restart LibreCAD and when \c is entered at the command line the two circles are drawn.

Note: The # character defines a comment, so everything following this will not be considered by LibreCAD.

4.3.3 Command Aliases

As previously noted many of the commands in LibreCAD have multiple forms. The long *untranslated* form is the native command and the short forms are *aliases* to the long form. For example, l and li are aliases to line. The aliases are defined in the librecad.alias configuration file. The format of the configuration file is <alias>[Tab]<command-untranslated>. The default aliases for the **2 Points** line appears as:

```
...
l   line
li  line
...
```

Aliases can be added or modified to suit users' preferences. The file is found in the following locations:

- **Linux**: $HOME/.local/share/LibreCAD/LibreCAD/librecad.alias
- **Windows**: C:\Users\ *{UserName}*\AppData\Local\LibreCAD\librecad.alias
- **macOS**: $HOME/Library/Application Support/LibreCAD/librecad.alias

---

Important: Only change the alias and *not* the long *untranslated* form.

4.3.4 Calculator

LibreCAD includes a built-in calculator that uses the command line interface. Typing cal on the command line toggles the *calculator mode* on and off. With the calculator mode on, math expressions typed on the command line will display the results in the output window, e.g. typing 1+1 displays 1+1 = 2 in the output window. Some other examples are:

```
sqrt(3^2 + 4^2) = 5
sin(pi/6) = 0.5
6^5 = 7776
```

If the cal mode is *off*, entering a math expression will result in an error message such as Unknown command: 1+1.

A complete list of operators and functions can be found in the *appendix*.

Note:

The constant *Pi* is defined in LibreCAD as 3.14159265359.

It can be used as `_pi` or `pi` in arithmetic expressions.

Trigonometric functions use *radians*. (*radians = degrees*pi/180*)

To provide angles in *degree* use affix d; e.g., `sin(90d)`.

More details are in the *appendix*.

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4.4 The Console Tool

LibreCAD's console tool gives users certain functionality without having to open the actual GUI version of LibreCAD. To use it, users need to specify options in their console: `librecad [options] ....` Running `librecad --help` or `librecad -h` is a basic option that should show a list of all possible options. Running `librecad --debug <level>` or `librecad -d <level>` allows launchin LibreCAD in various debug levels from 0 being in no debug level whatsoever and 6 being the debugging level.

4.4.1 DXF2PDF Tool

LibreCAD allows convertion of DXF drawings to PDF in console without any need to run the GUI. To do so, users can run `librecad dxf2pdf [options] ...` in their command line or terminal. This tool has its own set of options and its own help option which can be obtained by running *librecad dxf2pdf –help`* or `librecad dxf2pdf -h` (or simply `librecad dxf2pdf` with no options at all).

In a nutshell, users can use it to convert a DXF file or multiple DXF files to a single multi-page PDF or multiple PDFs. It can be done without specifying any options other than the input filename: `librecad dxf2pdf file_name.dxf`. The output file is saved in the executable folder under the name of `file_name.pdf`. If multiple (N) DXF files are selecred (`librecad dxf2pdf file_name1.dxf file_name2.dxf ... file_nameN.dxf`) then N PDF files are made. It is also possible to convert DXF files to PDF and put the results in a single multipage PDF `outfile.pdf` as follows: `librecad dxf2pdf -o outfile.pdf file_name1.dxf file_name2.dxf ... file_nameN.dxf`. The same approach can be used if there is a single DXF file input but the outpit filename needs specification.

There are other options available which can be obtained by running `librecad dxf2pdf --help` and at this stage include:

```
-h, --help                Displays help.
-v, --version             Displays version information.
-a, --fit                 Auto fit and center drawing to page.
-c, --center              Auto center drawing on page.
-k, --grayscale           Print grayscale.
-m, --monochrome          Print monochrome (black/white).
-p, --paper <WxH>         Paper size (Width x Height) in mm.
-r, --resolution <integer> Output resolution (DPI).
-s, --scale <double>      Output scale. E.g.: 0.01 (for 1:100 scale).
-f, --margins <L,T,R,B>   Paper margins in mm (integer or float).
-z, --pages <HxV>         Print on multiple pages (Horiz. x Vert.).
```

(continues on next page)

```
-o, --outfile <file>        Output PDF file.
-t, --directory <path>      Target output directory.
```

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4.5 Blocks

Blocks are reusable drawings of commonly used objects such as bolts, furniture, electronic components, title blocks, etc. Inserted blocks, also called *instances* or *inserts*, are composed of geometric shapes (lines, arcs, etc.), but can also include text and dimensions. Blocks can be created in the current drawing, inserted from the block library, or imported from a separate drawing file, Once inserted blocks can be used repeatedly within the drawing.
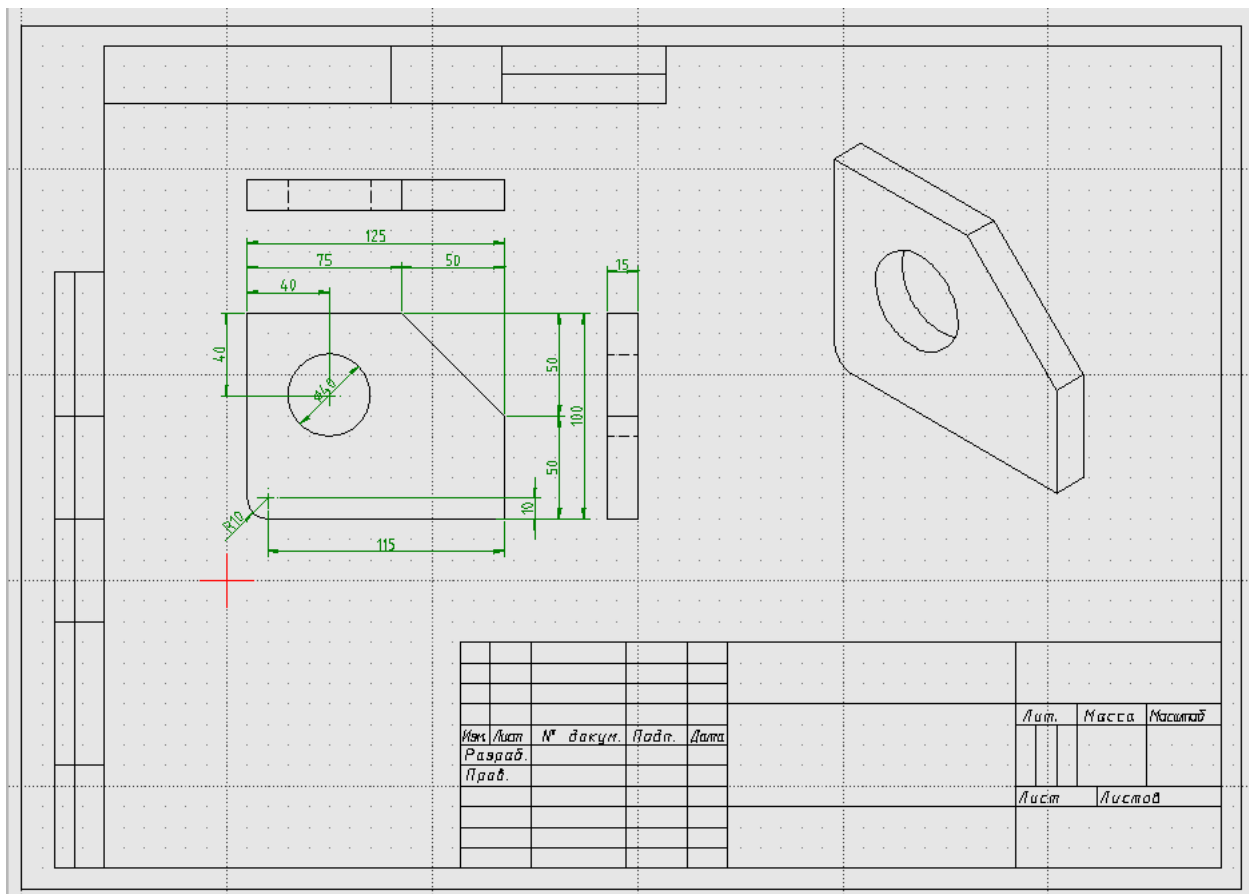
Fig. 11: Example of the Doohickey drawing with blocks, including the page border from the **Library Browser** ("sheets -> A4H") and a block imported from a drawing file (isometric view).

4.5.1 Working with Blocks

There are two dock widgets for managing blocks. The *Block List* dock for managing blocks *within the current drawing* and the *Library Browser* dock that displays a *collection of blocks* in the library. Both widgets allow users to insert blocks into the current drawing, but there are a couple of important differences:

- Blocks inserted from the **Block List**:
    - A block can be placed at a different locations, scale, and/or rotation angle, and also be placed in an array.
    - Blocks inserted from the same block in the **Block List** is called an *instance*. If a block is inserted from the same block multiple times all *instances* of the block are linked. **Changes to one instance of the block will be reflected in all instances of that block.**
- Blocks inserted from the **Library Browser**:
    - Each block can be placed at a different location, rotation angle and/or scale.
    - Blocks inserted multiple times will create a new instance of the block in the **Block List** each time it is inserted. After the initial insertion, the blocks will numbered sequentially (e.g. *BlockName*, *BlockName-0*, *BlockName-1*, . . .) The inserted blocks will be independent of each other.

Blocks can also be imported from a separate file and will appear in the **Block List**. Drawing files imported multiple times will create a new instances in the **Block List** and be numbered sequentially as with blocks inserted from the **Library Browser**.

4.5.2 Using the Block List

Creating a Block

Blocks can be created in the current drawing for use within the drawing. There are two ways to create a block:

- From an existing object:
    1. Ensure the all entities for the object to be made into a block are on layer "0".
    2. Select all the entities that make up the object.
    3. Click on the **Create Block** icon .
    4. Specify the reference point for the block. The reference point is used to locate the object when it is inserted into the drawing.
    5. Provide a unique name for the new block and click **OK**. The new block will appear in the **Block List**.
    6. The original object can be deleted from layer "0". The new block remains available in the **Block List**.
- From an empty block:
    1. Click on the **Add an empty block** icon . Provide a unique name for the new block and click **OK**. The new empty block will appear in the **Block List**.
    2. Select the new block in the **Block List** and click the **Edit the active block in a separate window** icon .
    3. Ensure layer "0" is selected and draw the object. The drawing's origin (0, 0) will become the reference point for the block.
    4. Close the new block's drawing window and the new block will be saved.

Important: Layer "0" has a special significance. It is the default layer for new drawings. More importantly in the context of blocks, it is equivalent to "no layer", similar in the way that *color "By Block"* is equal to "no specified color"

---

or *line type "By Block"* is equal to "no specified line type". **Generally layer "0" should only be used when creating blocks and should be the only layer in a drawing for a block.**

Pay particular attention to the *Attributes* when creating blocks. In addition to the specific attributes, pen attributes (Color, Width, Line Type) also include "By Layer" and "By Block".

- Blocks with specific attributes (e.g. color set to blue, width set to 0.18 mm, etc) will retain those attributes when inserted into a drawing. The block needs to be edited to change any of the attributes.

- Blocks with the attributes set to "By Layer" will adopt the attributes of the layer they are inserted in to. The block's attributes will change if the layer's attributes are changed.

- Blocks with the attributes set to "By Block" will initially adopt the attributes assigned to the layer. The attributes can be changed with the **Attribute** tool.

Inserting Blocks

Blocks can be inserted from the **Block List** or from the **Library Browser** (see *below*). When inserting a block from the **Block List**, the **Block List Tool Options** toolbar is displayed:

- Block can be rotated by the specified *Angle* and scaled by the *Factor*.

- A pattern of blocks can be created by specifying the number of columns and rows in the *Array* and *Spacing* between the columns and rows.

To insert a single block:

1. Select a layer for the inserted blocks.

2. Select a block in the **Block List**.

3. Click on the **Insert the active block** icon .

4. Set the rotation angle in the *Angle* field and the scale factor in the *Factor* field as required. (See *Angles* in **Fundamentals** and "Scale" in *Modify* tools for more information.) Note: Leave the *Array* values as "1" and *Spacing* values remain as "1.0".

5. Place the block at the desired location within the drawing.

6. Adjust the rotation angle and scale as needed and place additional copies of the block, or press [Esc] to exit the command.

Transformations (rotation, scale) and an array can be combined in a single block insertion. The the scale and spacing is applied to the individual blocks within the array, however the rotation angle is applied to the entire array.

To insert an array of blocks:

1. Select a layer for the inserted blocks.

2. Select a block in the **Block List**.

3. Click on the **Insert the active block** icon .

4. Set the rotation angle and scale factor as needed.

5. Set the numbers of columns and rows in *Array* to create the required pattern.

6. Set the *column spacing*. This is the distance between the insertion point of two blocks in adjacent columns.

7. Set the *row spacing*. This is the distance between the insertion point of two blocks in adjacent rows.

8. Place the block at the desired location within the drawing. The insertion point of the array is the insertion point of the lower-left item in the array (at 0 degrees rotation).

9. Adjust the options as needed and place additional copies of the block, or press [Esc] to exit the command.

Note: When the Angle, Scale, Array size and Spacing are all applied to an array of blocks, the array is created as follows:

- the scale, number of columns / rows and spacing is applied to the blocks in the array, and then
- the entire array is rotated around the lower left corner of the array.
- The blocks are rotated as the array is rotated and the spacing remains as defined even if the block entities are scaled.

Using an array will treat all blocks in the array as a *single block instance*. Selecting one entity of the array will select the all blocks in the array. To keep instances separate, insert multiple instances from the block list or create additional copies with the "Move / Copy" tool or normal edit commands (cut, copy, paste).

Editing a Block

1. Select a block in the **Block List** and click the **Edit the active block in a separate window** icon .

2. Edit the block as necessary.

3. Close the block's drawing window and the block will be saved and all instances of the block will be updated in the current drawing.

Saving Blocks

Blocks can be saved to a separate file and used in other drawings or added to a user-defined library. To save the block:

1. Select a block in the **Block List**.

2. Click the **Save the active block to a file** icon .

3. Select a file location, specify a file name and click **Save**.

Recommendation

When saving blocks to be added to the library it is recommended that the block's entities be placed on **layer "0"** and layer "0" is the *only layer* in the drawing. Any additional layers in the block's drawing will be added to the new drawing when the block is inserted. The additional layers may have unintended consequences.

4.5.3 Using the Library Browser

LibreCAD includes several categories of blocks in its own library; algorithm, elektro, plan/air_water, plan/architect, etc. A user-defined library can also be configured in LibreCAD.

Inserting Blocks

Recommendation

When using blocks from the library, insert a *single insert* from the **Library Browser** and then insert subsequent *instances* from the **Block List**. Inserting the block from the **Block List** retains the link between instances of the same block insert. If a block is edited from the **Block List**, all instances of the block will show the changes.

Only insert multiple *inserts* of a block from the **Library Browser** if they are to be independent.

To use blocks from the *Block Library*, select the block from the tree view and insert it in the drawing. Blocks inserted from the library can be rotated and scaled through the **Library Browser Tool Option** toolbar:

The rotation angle and the scale factor behave as they do for a block inserted from the **Block List**.

To insert a block:

1. Select a layer for the inserted blocks.

2. Select a block in the **Library Browser**.

3. Click on the **Insert** button.

4. Set the rotation angle and scale as needed.

5. Place the block at the desired location within the drawing.

6. Adjust the options as needed and place additional copies of the block or press [Esc] to exit the command.

Adding a User-defined Library

Additional parts and libraries can be added for blocks created by users, libraries downloaded from the LibreCAD wiki (https://wiki.librecad.org/index.php?title=Part_Libraries) or from other internet resources. LibreCAD can be configured to show user-defined blocks in the library browser *in addition* to the blocks included with LibreCAD.

The easiest method of installation, which does not require or Linux Root privileges or Windows Administrator access, is to create a new directory such as "Parts_Library" in the home directory or "Documents" folder. The path to this directory would be something similar to "/home/*{Username}*/Parts_Library/" or "C:\Users\ *{Username}*\Documents\Parts_Library\ ". Blocks and libraries can then be placed under the parent "Parts_Library" directory. The sub-directories will create categories that will appear in the tree view of the **Library Browser**.

Important: Do not place blocks directly in the parent directory of the parts library. Blocks must be placed in sub-directories to the parent libraries directory to appear in the **Library Browser**.

To include the new blocks in the **Library Browser** tree view, edit LibreCAD's *Application Preferences* to add the path to the directory or folder with the user-defined blocks. From the menus, select **Options -> Application Preferences** and select the **Paths** tab. Type the full path to the part library, e.g. /home/*{Username}*/PartsLibrary/ or C:\Users\ *{Username}*\Documents\PartsLibrary\ , into the text-box labelled "Part Libraries" and click "OK". Click the **Rebuild** button on the **Library Browser** dock and the new libraries will appear in the tree view.

4.5.4 Importing Blocks

Blocks can also be inserted from as a file from other sources. With an open drawing:

1. Select **File -> Import -> Block**.

2. Locate and select the block drawing file and click "Open".

3. Set the rotation angle and scale as needed.

4. Place the block at the desired location within the drawing.

5. Adjust the options as needed and place additional copies of the block or press [Esc] to exit the command.

As with inserting a block from the **Library Browser**, importing a block multiple times will create a new instance of the block in the **Block List** each time it is imported, each with a sequentially numbered block name after the initial block insert. When using blocks from external files the same layer criteria applies (i.e. layer "0", ...)

4.5.5 "Exploding" Blocks

Once a block has been inserted into a drawing, it is treated a single entity and can only manipulated (moved, scaled, etc.) as a single entity. Select any one entity within the block selects the entire block. However, blocks can be disassembled, or *exploded*, and once exploded the individual entities making up the block can be edited. To explode a block:

1. Select a block.

2. Click on the **Explode** icon .

Inserted Block   "Exploded" Block
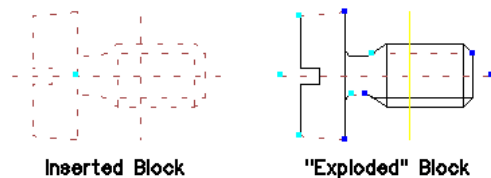
Note: Once a block has been exploded it is no longer a block and cannot be treated as a single entity. If it is necessary to recreate the block, refer to **Creating a Block** above.

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4.6 Annotating a Drawing

A drawing in and of itself provides an image of what the object might look like, but it doesn't provide a complete description of the object. Knowing the size of the object is also very important. The size of the drawn object is shown with measurements, *dimensioning*, and other textual information to describe the drawn object. Together, dimensions and text is called *annotating* a drawing.

Dimensioning an object on a drawing provides the information necessary to be able to interpret the object and ultimately create it, whether it be a building or a *doohickey*. Other textual information in the form of notes, symbols, call-outs, etc. provide further details for the object drawn.

4.6.1 Dimensioning

Dimensions are used to define length, width, height, and/or angle of a line entity, the diameter of circle entity, or radius of arc entity. A drawing's dimensions must be:

- Accurate
- Legible
- Complete

Types of Dimensions

LibreCAD supports the following types of dimensions:

Type	Description
Aligned	Places the dimension parallel to a line between the two endpoints of the dimension.
Linear	Dimension between two points from any angle of interest. The default is 0 (horizontal) and is changed via **Tool Options** toolbar.
Horizontal	Horizontal Dimension between two points.
Vertical	Vertical Dimension between two points.
Radial	Radius of an arc.
Diametric	Diameter of a circle.
Angular	Angle between two lines or linear parts of objects.
Leader	Not a dimension per se, but used for notes in drawings.

Complete descriptions of the *dimensioning tools and related options* are found in **Drawing Tools** of the **Reference** section.

Dimensioning Appearance

A dimension consists of several parts:
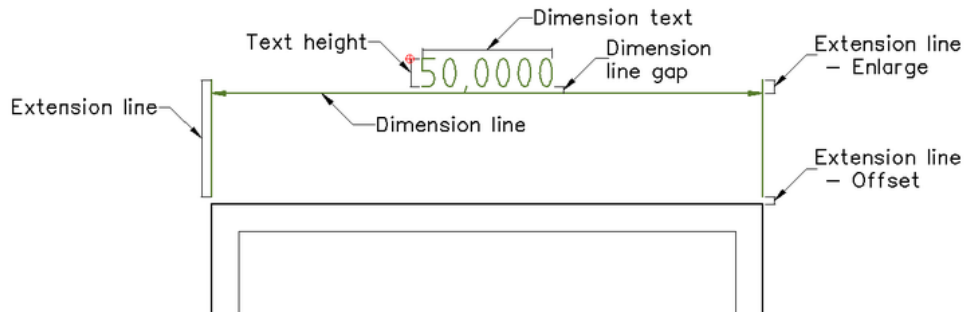
The appearance of the dimensions are configured in the *Dimensions* tab in the **Drawing Preferences**. As with many other aspects of a *drawing's setup*, there are some generally accepted values for dimensioning:

Dimension Component	mm	Decimal Inch	Fractional Inch
Dimension text height	2.5	.100	3/32
Dimension line gap	1.5	.0625	1/16
Extension line - Offset	1.5	.0625	1/16
Extension line - Enlarge	3	.125	1/8
Arrow size	3	.125	1/8

Note - Adjusting Dimensions for Printing

The size of each dimension component: "Text Height", "Arrow size", etc. should be set to the desired "real world" size in the configuration. That is to say if the desired text height is 2.5 mm when printed, the "Text Height" should remain set as 2.5 mm. If the drawing is printed full scale (1:1) the dimension text will appear correctly. However if the drawing is scaled up or down the "General Scale" needs to be adjusted accordingly. The "General Scale" is set to the *inverse* of the printing scale. For example, if the printed scale is determined to be 1:4, the "General Scale" should be set to 4 (4:1).

The minimum spacing between dimension lines needs to be scaled with the drawing. For example, if the drawing is 1:10, the spacing will need to be adjusted to 60 mm between dimension lines and 100 mm from the entity.

Additional information can be found in the *Drawing Setup* and *Printing* guides.

Tip - Rules for Dimensioning

A few rules will help to ensure dimensions are accurate, legible and complete:

- There should be only one way to interpret any one dimension.

- Dimension and extension lines should not cross.

- Extension lines and entity lines should not overlap.

- Provide space between dimensions to ensure legibility.

- The view that best shows an entity is the view that should be dimensioned.

- Each entity on the drawing should be dimensioned and dimensioned only once.

- There should be no need to calculate or scale a dimension of an entity.

- A dimension should be referenced to a logical origin point.

- When there are multiple lines of dimensions, the longer dimensions are to be placed outside of shorter ones.

- Except for large circles and arcs, all dimensions should be placed outside the part and spaced 10mm / 3/8" from the entity.

- Dimension circles with diameters and arcs with radiuses.

- Center lines or center marks should be used on all circles and arcs.

- Extended a circle's or arc's center lines and use as extensions line when possible.

- Multiple lines of dimensions are spaced uniformly with a minimum of 6mm / 1/4" between dimension lines (see note above).

- Use arrow heads or slash marks at the end of the dimension lines.

To improve legiblity, make corrections, or make adjustments after scaling a drawing, dimension labels, dimension lines and extension lines can be repositioned using *Entity Handles*.

Dimensioning drawings is beyond the scope of the **LibreCAD User Manual**, but additional resources and examples are available in LibreCAD's wiki or elsewhere on the web.

Dimensioning Example

Good Dimensioning   Bad Dimensioning
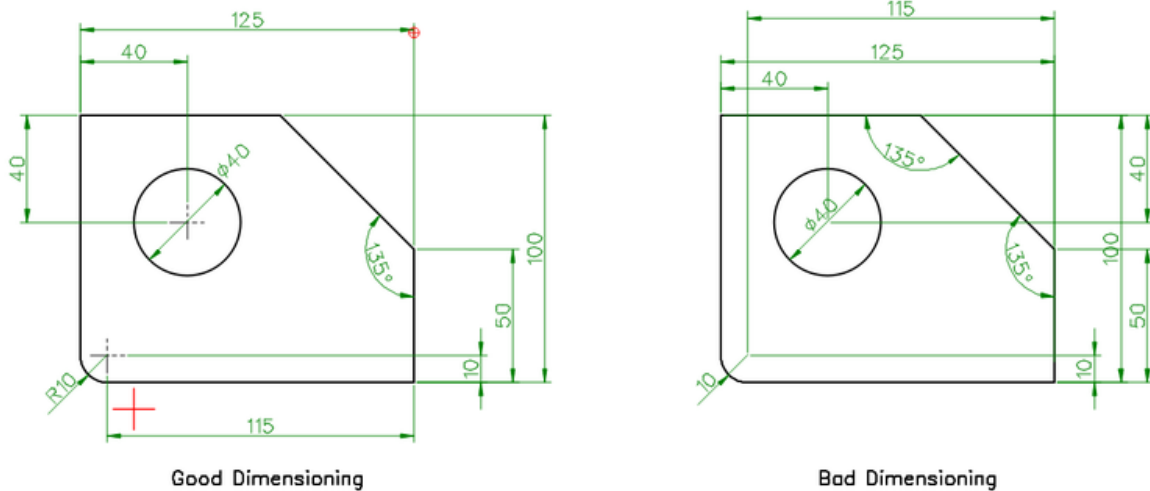
Hint:

Modifying a dimension label prohibits the automatic adjustment of the value when the dimension is changed.

There is the placeholder <> for dimension values, that can be used to keep this functionality.

Thus to add a unit to a dimension, you can modify the label via `Properties` and enter the label text `<> mm`. This will show the label as `100 mm`, and when the dimension is lengthen, it may show `120 mm` automatically.

4.6.2 Leaders

While leaders do not dimension an entity, they are closely related to dimensioning as they are important for annotating and adding clarity to entities. Leaders provide the ability to place pointers to identify a specific area of interest when adding a note and linking it to a particulate object. Leaders take their settings from the *dimension settings* in **Drawing Preferences**.

This leader points to
the centre of the circle.
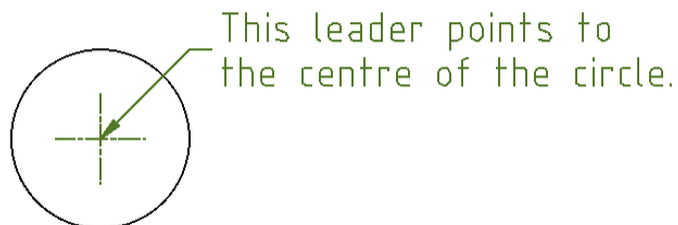
4.6.3 Text

Adding text to a drawing provides addition information; build notes, drawing title and related details, and so forth. Text can be added using either of the two types of text tools:

"Text":   "MText" (multi-line):

Both tools proved several options for the appearance and placement of text, however a couple are unique to the single-line **Text** tool as shown below:

Option	Description	Text	MText
Font Settings:			
• Font	Select font for text	X	X
• Height	Set font height	X	X
• Angle	Places text at specified *angle*	X	X
• Oblique	*Inactive*	X	X
• Width factor		X	X
• Default line spacing	Use default line spacing for specified font	X	X
• Line spacing		X	X
Alignment:			
	Place text aligned to *handle*: • top, left/center/right • middle, left/center/right • baseline, left/center/right • bottom, left/center/right	X X X X	X X X
• *Fit*	Places text between specified points while maintaining set height	X	X
• *Aligned*	Places text between specified points while maintaining width to height ratio (scales text)	X	X
• *Middle*	Places text with equidistant above and below, left and right of text as defined by text box	X	X
Insert symbol	Insert predefined symbol (Diameter, Degree, Plus / Minus, At, Hash, Dollar, Copyright, Registered, Paragraph, Pi, Pound, Yen, Times, Division)	X	X
Insert Unicode:			
• Page	Select Unicode page to select character from	X	X
• Char	Select character to insert into text	X	X
• **Insert** button	Click button to insert Unicode character into text input field	X	X

LibreCAD v2.2.0 - User Manual

This is an **interim release** *of the of the* **LibreCAD User Manual** *and is subject to change.*

The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

4.7 Completing and Printing

Completed drawings are normally printed to scale and include a border and title block. However, drawings without a border can be useful for drafts, sketches, exporting to CNC software or a bitmapped image, and so on. In addition to supporting a large variety of paper sizes; ISO, ANSI, Arch (Architectural), custom, etc., LibreCAD also has the ability to print drawings across several pages using *tiled printing*, allowing a drawing to be printed on smaller paper and assembled into a large format print.

Completing a drawing and making it ready for printing includes a few steps:

- Finalizing the page size and drawing scale,
- adjusting the dimensions and spacing, and
- adding page border and title block as required.

Tip - Using Border and Title Blocks

Adding a border provides a finished look and title blocks include important information for the drawing. Title blocks vary widely in content and layout, but can include:

- Drawing name
- Project name
- Scale
- Drawing perspective
- Date
- Revision
- Page number
- Paper size
- Name of organization
- Name of draftsperson, approvers, etc
- Others as required

4.7.1 Print Preview Window

LibreCAD's **Print Preview** displays the drawing on a virtual page and allows adjustments to the layout prior to committing the drawing to paper. To switch to the print preview window, click the **Print Preview** icon or select **File -> Print Preview**:
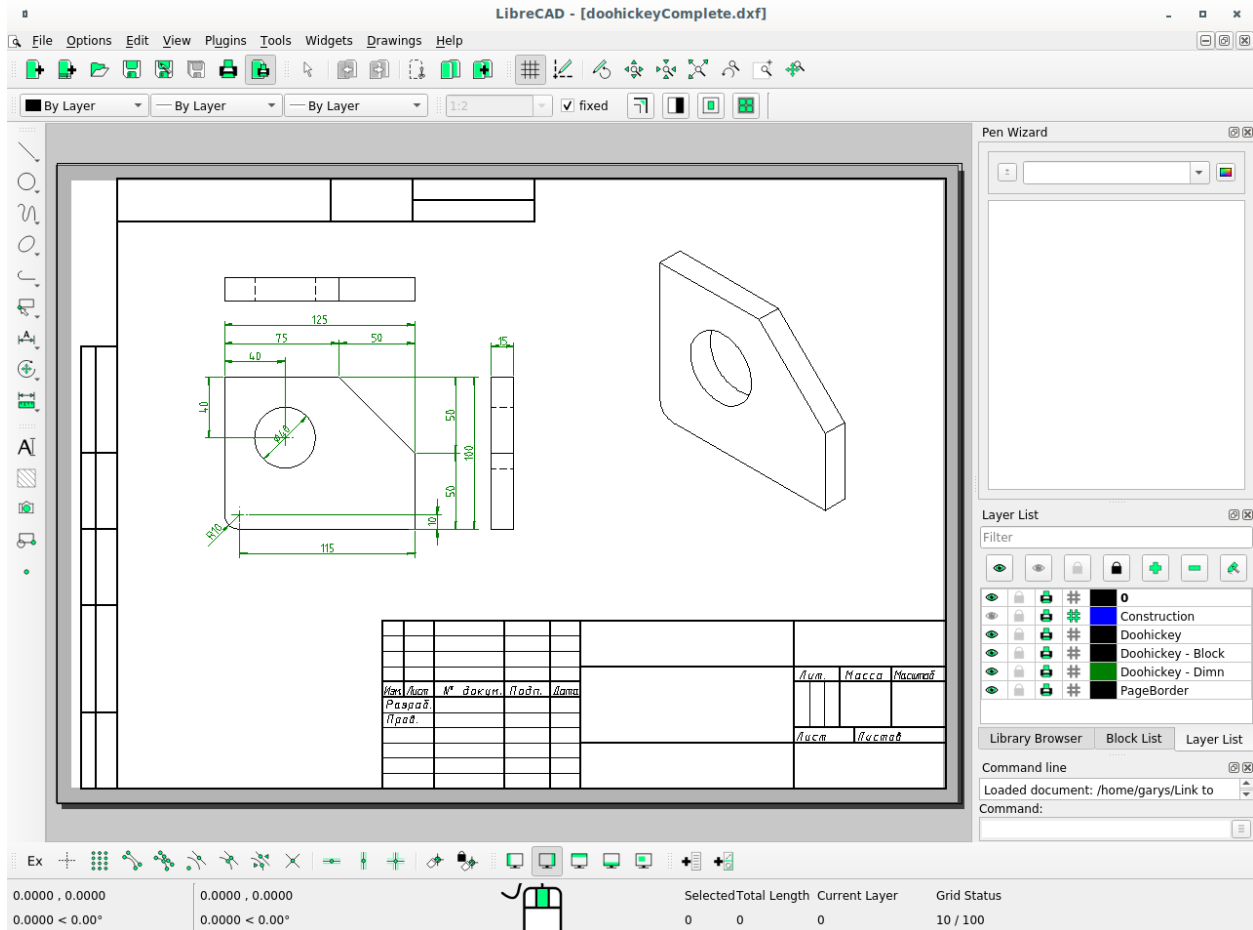
Fig. 12: LibreCAD's Print Preview window displaying the *Doohickey* drawing

Opening the print preview window also displays the **Print Preview Tool Options** toolbar:

More details for *Print Preview Tool Options* can be found in the **Reference / Toolbars / Tool Options** section.

The print preview window shows the virtual page as a white rectangle with a black border and a drop shadow. The black border represents the paper format and orientation as it is configured on the "Paper" tab of the *Drawing Preferences*. If margins are specified on the "Paper" tab, they are shown as a dark grey border.

4.7.2 Printing

While a drawing can be printed directly by clicking the **Print** icon or selecting **File -> Print**, the recommended approach is to print drawings from the **Print Preview** window:

Starting with the drawing open in the drawing window:

1. Switch to print preview window by clicking the **Print Preview** icon .

2. Open the **Drawing Preferences** (**Options -> Current Drawing Preferences**) and check the paper layout ("Format", "Orientation" and "Margins") for the current drawing: e.g. A4, Landscape, Top, Bottom Left and Right Margins each at 10. Adjust the layout if necessary and click **OK**.

3. Click the **Fit to Page** icon on the toolbar. This will ensure the drawing is scaled to fit the current paper format and centered on the page.

4. Click the **Print** icon or select **File -> Print**.

5. Select the printer on the *Print* dialogue and confirm the properties by clicking the **Properties** icon. Adjust the properties if necessary and then click the **Print** button.

Tiled Printing

Tiled printing provides the ability to print a scaled drawing that is larger than a printer's available paper format. Tiled printing is useful where a drawing needs to be printed at a fixed scale that would otherwise require a large format printer. The drawing is printed across several pages that can be assembled into a large single document.

Tile printing uses "Format" and "Orientation" found on the "Paper" tab of the *Drawing Preferences*. It lays out the pages horizontally and vertically in a grid pattern to form a single large format page to accomodate a scaled drawing. For example, a portrait A4 page with 2 pages horizontally and 1 vertically would result in a page 420 x 297 mm, or a landscape A4 in a 2 x 2 pattern would be a page of 594 x 420 mm. Note that both examples use a margin of 0. If margins are defined, the assembled document is reduced by the margin widths on the common edges of the paper where they are assembled.

To print a tiled document, starting with the drawing open in the drawing window:

1. Switch to print preview window by clicking the **Print Preview** icon .

2. Click the **Fit to Page** icon on the toolbar. This will ensure the drawing displayed correctly in the print preview. Note that "fixed" needs to be *unchecked*.

3. Open the **Drawing Preferences** (**Options -> Current Drawing Preferences**) and check the paper layout ("Format", "Orientation" and "Margins") for the current drawing: e.g. A4, Landscape, Top, Bottom Left and Right Margins each at 10. Adjust the values if necessary and click **OK**. (The "Number of Pages" will be calculated below.)

4. Adjust the scale as desired. Change the print scale by:

    1. Selecting a predefined scale ratio from the drop-down, or type the required scale ratio in the text-box and press [Enter].

    2. Lock the print scale by placing a checkmark in the "fixed" checkbox.

5. Click the **Calculate number of pages...** icon on the toolbar. The print preview will show a grid of pages with the drawing placed in the center. The *Number of pages* on the "Paper" tab will be updated to reflect the number of pages shown in the print preview.

6. Adjust the position of the drawing image on the page by repositioning the paper by dragging it as needed (the drawing stays centered on the screen). Click and hold anywhere in the drawing space and drag the paper to the desired position. Pressing [Shift] allows only *horizontal* movements of paper and pressing [Ctrl] allows only *vertical* movements.

7. If necessary, adjust the dimensions "General Scale" and dimension line placement. Refer to *Adjusting Dimensions for Printing* in **Dimensioning and Text** for details.

8. Click the **Print** icon or select **File -> Print**.

9. Select the printer on the *Print* dialogue and confirm the properties by clicking the **Properties** icon. Adjust the properties if necessary and then click the **Print** button.

In case when a page has the margins (margins > 0) the print preview takes on a special look in that the margins between a neighboring pages aren't shown. It makes possible to represent the printable areas of all pages as one whole area and to show an undivided drawing. Or, in other words, the print preview looks like the assembled drawing.

Tip - Setting the Margins

Set the margins to the printer's minimum margins when using tiled printing to maximize the assemble print size.

Next example shows the print preview (left) and the output of tiled printing with the margins (right):
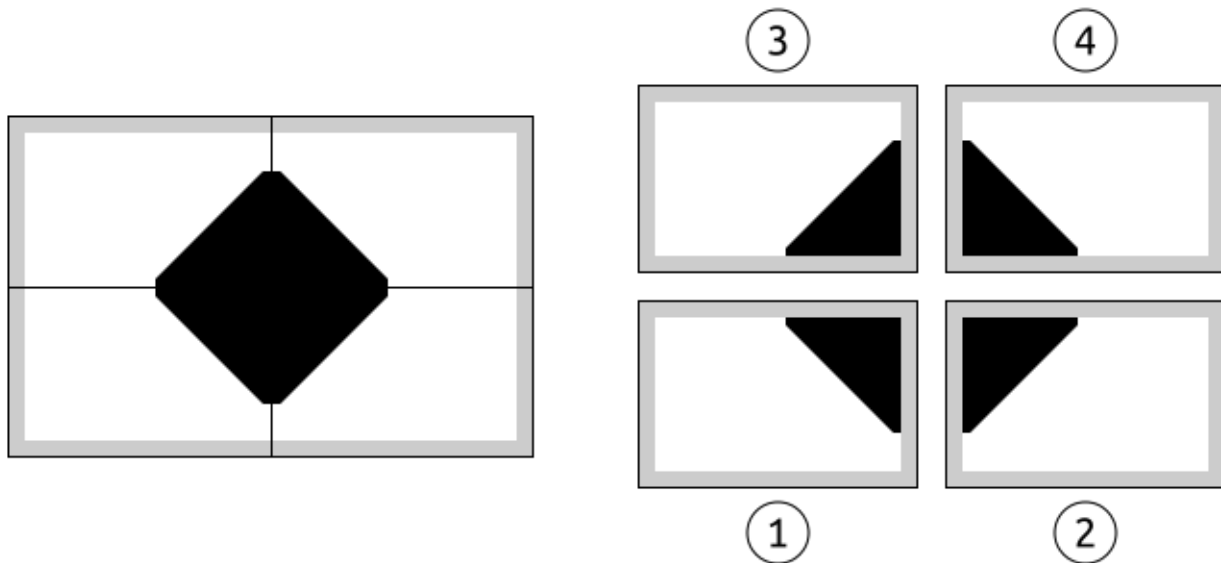
The sequence of the output is from bottom left page to top right page. In the picture above the order of the output is marked by numbers.

4.7.3 Completing and Printing a Drawing

A finished drawing will include a border / title block and be scaled to suit the paper format and orientation. See *Scale and Dimensioning* in the **Drawing Setup** for more details on determining the suitable scale for a drawing.

Important: These steps assume a drawing that is drawn to full-scale (1:1).

Starting with the drawing open in the drawing window:

1. Switch to print preview window by clicking the **Print Preview** icon .

2. Click the **Fit to Page** icon on the toolbar. This will ensure the drawing displayed correctly in the print preview. Note that "fixed" needs to be *unchecked*.

---

3. Open the **Drawing Preferences** (**Options -> Current Drawing Preferences**) and check the paper layout ("Format", "Orientation" and "Margins") for the current drawing: e.g. A4, Landscape, Top, Bottom Left and Right Margins each at 10. Adjust the layout if necessary and click **OK**.

4. Click the **Fit to Page** icon on the toolbar. This will ensure the drawing is scaled to fit the current paper format and centered on the page. Note the scale in the text-box of the print preview toolbar.

5. Adjust the scale using a *smaller* ratio than the "Fit to Page" scale. For example use "1:2" if "1:1.18" is the calculated scale. Change the print scale by:

   1. Selecting a predefined scale ratio from the drop-down, or type the required scale ratio in the text-box and press [Enter].

   2. Lock the print scale by placing a checkmark in the "fixed" checkbox.

6. Adjust the position of the drawing image on the page by:

   1. Clicking the **Center to Page** icon , or

   2. repositioning the paper by dragging it as needed (the drawing stays centered on the screen). Click and hold anywhere in the drawing space and drag the paper to the desired position. Pressing [Shift] allows only *horizontal* movements of paper and pressing [Ctrl] allows only *vertical* movements.

7. Close the print preview window by clicking the **Print Preview** icon .

8. If necessary, adjust the dimensions "General Scale" and dimension line placement. Refer to *Adjusting Dimensions for Printing* in **Dimensioning and Text** for details.

9. If using a border / title page block, insert the block from the **Library Browser**:

   1. Insert a block from *sheets* that matches the paper format and orientation, e.g. "A4H" for a landscape A4 page.

   2. Set the blocks scale "Factor" suit the scale determined above, e.g. "2" for a drawing scale of 1:2. See **Inserting Blocks** in *Using the Library Browser* for details.

   3. "Explode" the border block and add/modify text using the "Properties" tool. Refer to the *Modify* tools.

10. Switch to the print preview window and check the completed drawing. If necessary, click "Zoom All" and then "Fit to Page" to align the drawing to the paper.

11. Click the **Print** icon or select **File -> Print**.

12. Select the printer on the *Print* dialogue and confirm the properties by clicking the **Properties** icon. Adjust the properties if necessary and then click the **Print** button.

LibreCAD v2.2.0 - User Manual

This is an **interim release** *of the of the* **LibreCAD User Manual** *and is subject to change.*

The manual is based on LibreCAD v2.2.0-rc1 with a few additions. If you are using another version of LibreCAD, your mileage may vary.

APPENDICES

5.1 Online Resources

LibreCAD main website: https://librecad.org/

5.1.1 Download Installation Packages

There are 2 types of installation packages depending on code maturity: first the official release, then the last build. A build is an interim state of LibreCAD development with additional improvements yet not mature enough to provide a new release. Note that a build may be less stable than an official release.

Operating System	URL
MS Windows: - GitHub	https://GitHub.com/LibreCAD/LibreCAD/releases
OSX: - GitHub	https://GitHub.com/LibreCAD/LibreCAD/releases
Linux Distributions: - AppImage (runs on most Linux) - Debian/Ubuntu (Stable) - Debian/Ubuntu (UnStable) - Arch Linux - Fedora - Gentoo - OpenSUSE	https://GitHub.com/LibreCAD/LibreCAD/releases https://packages.debian.org/stable/graphics/librecad https://packages.debian.org/unstable/graphics/librecad https://www.archlinux.org/packages/community/x86_64/librecad/ https://apps.fedoraproject.org/packages/librecad https://packages.gentoo.org/packages/media-gfx/librecad https://software.opensuse.org/package/librecad

5.1.2 Getting Help

There are many other resources for getting help with LibreCAD:

- LibreCAD forum: http://forum.librecad.org/
- LibreCAD Wiki: https://dokuwiki.librecad.org/
    - FAQ: https://dokuwiki.librecad.org/doku.php/usage:faq
    - Tutorials: https://dokuwiki.librecad.org/doku.php/usage:tutorials
    - Part Libraries: https://wiki.librecad.org/index.php/Part_Libraries

5.1.3 Reporting Bugs

You can help with the improving LibreCAD. Please report any bugs at LibreCAD's GitHub Issues page: https://GitHub.com/LibreCAD/LibreCAD/issues

5.2 Line Attributes

5.2.1 Common Line Widths

The following table provides suggested line widths for ISO A4/A3/A2 or ANSI A/B/C paper sizes:

Line Weights	Pen Size (mm)	Purpose	Recommended
Extra Thin	0.00, 0.05, 0.09	Hidden lines Hatching Reference line	0.00 mm
Thin	**0.13**, 0.15, **0.18**, 0.20, **0.25**	Outlines Center lines Dimension lines Leader and extension Phantom lines Grid lines Text	0.18 mm
Medium	0.30, **0.35**, 0.40, **0.50**	Hidden lines Text normal (0.30 mm) Text - sub-headings (0.50 mm) Visible object outlines	0.35 mm
Thick	**0.70**	Cutting lines Match lines Section lines Text - titles/major headings Viewing planes	0.70 mm
Extra Thick	**1.00**	Title sheet border	

Note: Pen sizes shown in **bold** are ISO standard sizes.

5.2.2 Line Types

Continuous	
Dot	
Dot (tiny)	
Dot (small)	
Dot (large)	
Dash	
Dash (tiny)	
Dash (small)	
Dash (large)	
Dash Dot	
Dash Dot (tiny)	
Dash Dot (small)	
Dash Dot (large)	
Divide	
Divide (tiny)	
Divide (small)	
Divide (large)	
Center	
Center (tiny)	
Center (small)	
Center (large)	
Border	
Border (tiny)	
Border (small)	
Border (large)	
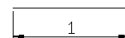
Image can be printed 1:1 using scale shown at bottom.

5.3 Hatches

Unless shown otherwise, hatch patterns are shown at 1:1 scale (1x)

angle	ansi31 4x	ar−b816c
ar−b816	ar−b88	ar−brelm
ar−brstd	ar−brstd	ar−conc
arcs	ar−hbone	ar−parq1
ar−roof	ar−rshke	box
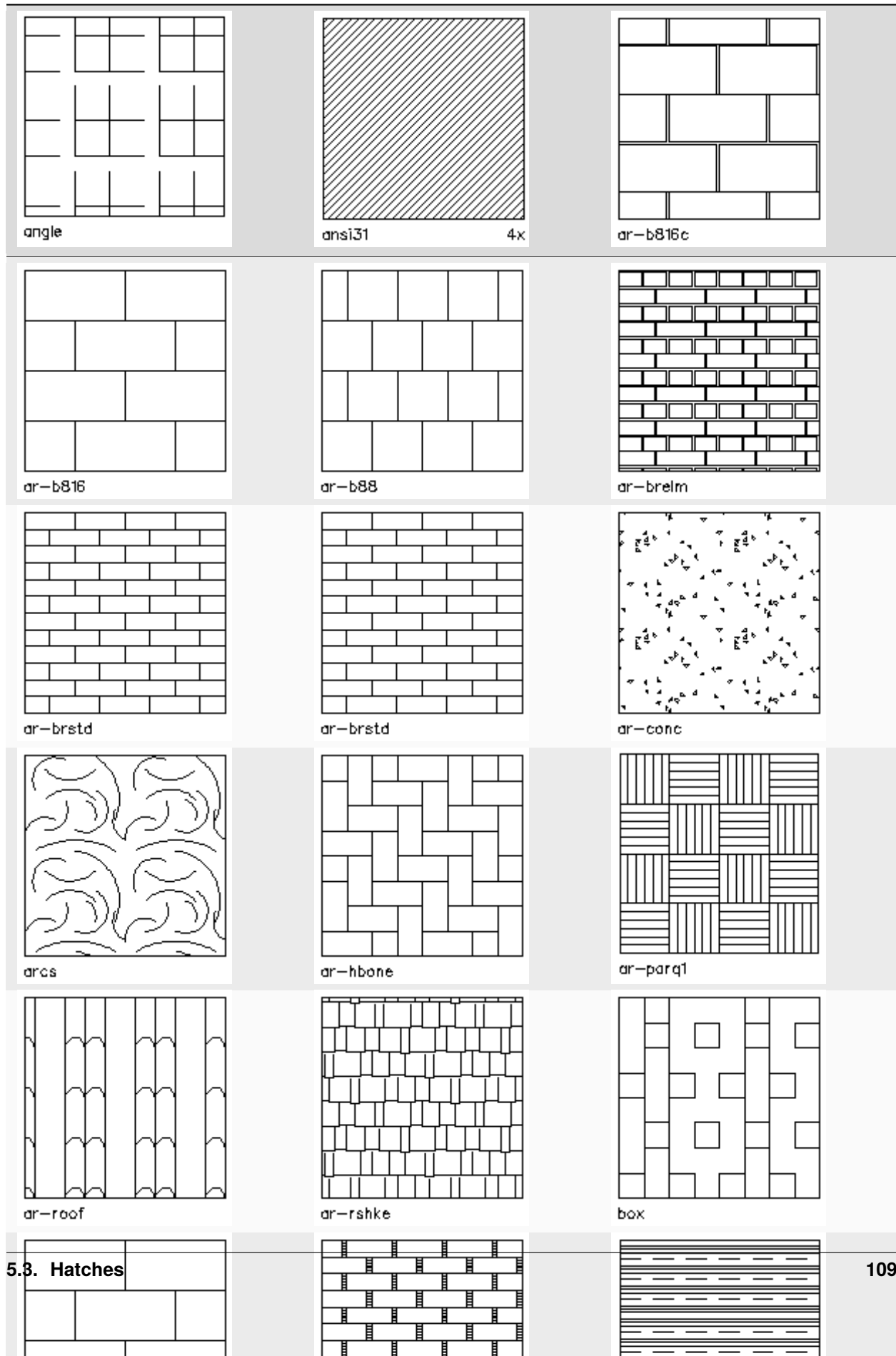
5.4 Fonts

Font Name	Font Filename	Font Type	Technical (*)	Sample
amiri-regular	amiri-regular.lff	Serif		abcABC123!@#
azomix_i	azomix_i.lff	Sans Serif	X	abcABC123!@#
azomix	azomix.lff	Sans Serif	X	abcABC123!@#
cursive	cursive.lff	Sans Serif	X	abcABC123!@#
cyrillic_ii	cyrillic_ii.lff	Sans Serif	X	abcABC123!@#
gothgbt	gothgbt.lff	Blackletter / Gothic		abcABC123!@#
gothgrt	gothgrt.lff	Blackletter / Gothic		abcABC123!@#
gothitt	gothitt.lff	Blackletter / Gothic		abcABC123!@#
greekc	greekc.lff	Greek Lettering		αβγΑΒΓ123!@#
greekcs	greekcs.lff	Greek Lettering		αβγΑΒΓ123!@#
greek_ol	greek_ol.lff	Greek Lettering		abcABC123!@#
greekp	greekp.lff	Greek Lettering	X	ΑΒΓΑΒΓ123!@#
greeks	greeks.lff	Greek Lettering	X	αβγΑΒΓ123!@#
iso3098_i	iso3098.lff	Sans Serif	X	abcABC123!@#
iso3098	iso3098_i.lff	Sans Serif	X	abcABC123!@#
iso	iso.lff	Sans Serif	X	abcABC123!@#
italicc	italicc.lff	Serif		abcABC123!@#
italiccs	italiccs.lff	Serif		abcABC123!@#
italict	italict.lff	Serif		abcABC123!@#
kochigothic	kochigothic.lff	Sans Serif		abcABC123!@#
kochimincho	kochimincho.lff	Serif		abcABC123!@#
kst32b	kst32b.lff	Sans Serif	X	abcABC123!@#
lc_opengost-ar	lc_opengost-ar.lff	Sans Serif	X	abcABC123!@#
lc_opengost-br	lc_opengost-br.lff	Sans Serif	X	abcABC123!@#
OpenGostTypeA-Regular	OpenGostTypeA-Regular.lff	Sans Serif	X	abcABC123!@#
OpenGostTypeB-Regular	OpenGostTypeB-Regular.lff	Sans Serif	X	abcABC123!@#
romanc	romanc.lff	Serif		abcABC123!@#
romancs	romancs.lff	Serif		abcABC123!@#
romand	romand.lff	Sans Serif		abcABC123!@#
romanp	romanp.lff	Serif		abcABC123!@#
romansi	romansi.lff	Sans Serif	X	abcABC123!@#
romans	romans.lff	Sans Serif	X	abcABC123!@#
romant	romant.lff	Serif		abcABC123!@#
scriptc	scriptc.lff	Formal Script		abcABC123!@#
scripts	scripts.lff	Formal Script		abcABC123!@#
simplex	simplex.lff	Sans Serif	X	abcABC123!@#
standard	standard.lff	Sans Serif	X	abcABC123!@#
syastro	syastro.lff	Symbol		☿☊☌♀♀123!@#
symap	symap.lff	Symbol		↑⚡☉○○△123!@#
symath	symath.lff	Symbol		←↓∂ℵ′|123!@#
symbol	symbol.lff	Symbol		αβχΑΒΧ123⊾#
symbol_misc1	symbol_misc1.lff	Symbol		○□▲|\⟍⌐⌐◊/◊

continues on next page

Table 1 – continued from previous page

Font Name	Font Filename	Font Type	Technical (*)	Sample
symbol_misc2	symbol_misc2.lff	Symbol		◊◊◊,,,.,..
symeteo	symeteo.lff	Symbol		|\∖⋯123!@#
symusic	symusic.lff	Symbol		⌐⌐⌐123!@#
unicode	unicode.lff	Sans Serif	X	abcABC123!@#

*: These fonts are composed of single lines and are suitable for technical drawings.

ASCII character set render in each font:

Font Name	ASCII Text
amiri-regular	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwzyz |||~
azomix	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz {|}~
azomix__i	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz {|}~
cursive	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz {|}~
cyrillic__ii	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
gothgbt	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
gothgrt	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
gothitt	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
greekc	!;#$%&'()*+,->/ 0123456789 :;<=>?@ ΑΒΓΔΕΖΗΘΙΚΛΜΝΞΟΠΡΣΤΥΦΧΨΩ« [\] ^_` αβγδεζηθικλμνξοπρστυφχψω« |||~
greekcs	!;#$%&'()*+,->/ 0123456789 :;<=>?@ ΑΒΓΔΕΖΗΘΙΚΛΜΝΞΟΠΡΣΤΥΦΧΨΩ« [\] ^_` αβγδεζηθικλμνξοπρστυφχψω« |||~
greek__ol	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\ ⋯ abcdefghijklmnopqrstuvwxyz {;}
greekp	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ΑΒΓΔΕΖΗΘΙΚΛΜΝΞΟΠΡΣΤΥΦΧΨΩ« [\] ^_` ΑΒΓΔΕΖΗΘΙΚΛΜΝΞΟΠΡΣΤΥΦΧΨΩ« |||~
greeks	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ΑΒΓΔΕΖΗΘΙΚΛΜΝΞΟΠΡΣΤΥΦΧΨΩ« [\] ^_` αβγδεζηθικλμνξοπρστυφχψω« |||~
iso3098__i	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\]_, abcdefghijklmnopqrstuvwxyz {|}~
iso3098	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\]_, abcdefghijklmnopqrstuvwxyz |||~
iso	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
italicc	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
italiccs	!;#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz ||.
italict	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
kochigothic	!"$,,&'()×+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ⋯ abcdefghijklmnopqrstuvwxyz {;}
kochinincho	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz {|}~
ksf32b	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefgh1jklmnopqrstuvwxyz {|}~
lc__opengost-ar	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
lc__opengost-br	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
OpenGostTypeA-Regular	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
OpenGostTypeB-Regular	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
romanc	!;#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||.
romancs	!;#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||.
romand	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||.
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romansi	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz {|}~
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simplex	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
standard	!"#$%&'()*+,->/ 0123456789 :;<=>?@ ABCDEFGHIJKLMNOPQRSTUVWXYZ [\] ^_` abcdefghijklmnopqrstuvwxyz |||~
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5.5 Common Drawing Scales

5.5.1 Architect's Scale (SI)

Drawing Scale	Common Use
1:1	Mockups / Samples / Small details
1:2	Construction details
1:5	Construction details
1:10	Construction details / Wall sections
1:20	Building sections
1:50	Building sections / Floor plans / Elevations
1:100	Floor plans / Elevations
1:200	Floor plans / Elevations / Site plans
1:500	Site plans
1:1000	Area plans

5.5.2 Architect's Scale (Imperial)

Drawing Scale	Ratio	Common Use
Full scale	1:1	Mockups / Samples / Small details
3=1-0	1:4	Small details
1 1⁄2=1-0	1:8	Small details
1=1-0	1:12	Small details / Construction details
"3⁄4=1-0"	1:16	Construction details / Wall sections
"1⁄2=1-0"	1:24	Building sections
3⁄8=1-0	1:32	Wall sections / Building sections
1⁄4=1-0	1:48	Building sections / Floor plans / Elevations
"3⁄16=1-0"	1:64	Floor plans / Elevations
"1⁄8=1-0"	1:96	Floor plans / Elevations / Site plans
"3⁄32=10"	1:128	
"1⁄16=1-0"	1:192	Site plan

5.5.3 Engineer's scale (Imperial)

Drawing Scale	Ratio	Common Use
1=10-0	120	Details
1=20-0	240	Details / Working plans
1=30-0	360	Working plans
1=40-0	480	Working plans
1=50-0	600	Working plans
1=60-0	720	Working plans
1=100-0	1200	Area plans

5.6 Languages

LibreCAD supprts these languages to varying degrees:

Albanian	German	Portuguese (Brazil)
Arabic	Greek, Modern (1453-)	Romanian
Basque	Hindi	Russian
Catalan; Valencian	Hungarian	Slovak
Chinese (China, Simplified)	Indonesian	Slovenian
Chinese (Taiwan, Traditional)	Italian	Spanish (Argentina)
Czech	Japanese	Spanish (Mexican)
Danish	Korean	Spanish; Castilian
Dutch; Flemish	Latvian	Swedish
English (Australia)	Macedonian	Tamil
Estonian	Norwegian	Turkish
Finnish	Panjabi; Punjabi	Ukraine
French	Polish	
Galician	Portuguese	

Note: For more information about translations, to see the current status or to contribute, refer to https://translate.librecad.org/projects/librecad/

5.7 Systems of Measurement

LibreCAD supports a large number of measurement systems/units:

Unit	Abbr.	System	Equivalences	Uses (*)
Inch	in	Imperial	3 barleycorns	Widely used in the United States, Canada, and the United Kingdom.
Foot	ft	Imperial	Size 46 (EU) shoe size	Still widely used in the United States, Canada, and the United Kingdom.
Mile	mi	Imperial	0.2756 of a Dutch mile	Used way too often in other than a ''international mile''.
Millimeter	mm	SI	More than 25 in an in	Vanilli's favorite
Centimeter	cm	SI		
Meter	m	SI	nm	
Kilometer	km	SI		Sigh, SI units are not funny…
Microinch	µin	Imperial		
Mil	mil	Imperial	1/1000 of an inch	Replaced with ''thou'' (but not ''you'').
Yard	yd	imperial	3 foots	Measure ale!
Angstrom	å		one ten-billionth of a meter	Zoom to a parsec - I dare you.
Nanometer	nm	SI	1 E9 m	the UoM formerly known as the ''millimicrometre'' and ''mµ''.
Micron	µm	SI		Really is a ''micrometer'', but not to be confused with a ''micrometer screw gauge''.
Decimeter	dm	SI	0.1 meter	
Decameter	dam	SI	10 meters	''dam'', don't be juvenile.
Hectometer	hm	SI	100 meters	Should be funny, but why?
Gigameter	Gm	SI	1,000,000,000 meters	That's one beeellion meters.
Astro	au, ua or AU	astronomical unit		also a ''a South Korean boy band''.
Lightyear	ly	astronomical unit	63241 au	First name ''Buzz''.
Parsecs	pc	astronomical unit	206264.806247096 astronomical units	er, big. What else can one say? Made popular by James T.

*: Obtained from Wikipedia and Wiktionary and wildly interpreted by the author of this page.

5.8 Calculator Operators and Functions

Typing `cal` at the command line will toggle the calculator mode on and off.

Hint:

Trigonometric functions expect angles in *radians*, they are identifiable by the function parameter `(a)` in the table! (*radians = degrees*pi/180*)

This is not very comfortable for humans, but there is a simple solution to pass angles in degrees.

Append the affix `d` to the degree value; thus, `sin(90d)` is a valid expression and will return the *sinus* of 90°.

As the calculator accepts arithmetic expressions and constants, this term will work too: `sin(90*pi/180)`

The part `90*pi/180` will be internally calculated and results in `1.57079632679`, which is 90° in *radians*.

And of course, arithmetic expressions can be used for other function parameters too, e.g. to convert imperial to metric values and vice versa.

Functions

Name	Function / Operation	Usage
+	addition	x+y
-	subtraction	x-y
*	multiplication	x*y
/	division	x/y
^	raise x to the power of y	x^y
sin	sine function	sin(a)
cos	cosine function	cos(a)
tan	tangens function	tan(a)
asin	arcus sine function	asin(a)
acos	arcus cosine function	acos(a)
atan	arcus tangens function	atan(a)
sinh	hyperbolic sine function	sinh(a)
cosh	hyperbolic cosine	cosh(a)
tanh	hyperbolic tangens function	tanh(a)
asinh	hyperbolic arcus sine function	asinh(a)
acosh	hyperbolic arcus tangens function	acosh(a)
atanh	hyperbolic arcur tangens function	atanh(a)
log2	logarithm to the base 2	log2(x)
log10	logarithm to the base 10	log10(x)
log	logarithm to base e (2.71828...)	log(x)
ln	logarithm to base e (2.71828...)	ln(x)
exp	e raised to the power of x	exp(x)
sqrt	square root of a value	sqrt(x)
sign	sign function -1 if x<0; 1 if x>0	sign(x)
rint	round to nearest integer	rint(x)
abs	absolute value, negatives become positive	abs(x)
min	min of all arguments	min(x, y, ...n)
max	max of all arguments	max(x, y, ...n)
sum	sum of all arguments	sum(x, y, ...n)
avg	mean value of all arguments	avg(x, y, ...n)

Constsants

Name	Alias	Constant	Value
_pi	pi	The number	3.14159265358979323846643
_e		Euler's number	2.718281828459045235360287

Hint: Behind the scenes LibreCAD uses the *muParser library* for the command line and many input boxes. So the above information describes basically the features of *muParser*, which are implemented in LibreCAD.

For advanced users or the courious ones, you can read more about *muParser* on the inventors website:

Reference: https://beltoforion.de/en/muparser/index.php#idStart

But, be aware, that LibreCAD probably does not use the latest version of *muParser*. Also, it is not fully implemented. The *muParser* library has much more capabillities than LibreCAD uses.

5.9 Building from Source

Building from the source code allows users to run the cutting-edge version of LibreCAD that includes bug fixes and possibly new features. LibreCAD can be compiled to run on multiple operating systems; Linux, Microsoft Windows, and macOS. The process differs depending on the OS.

Important: The instructions are as complete as possible to provide the necessary steps to allow any user to build LibreCAD from the source code, however some understanding of the operating system and installation of the required tools and dependencies is required. The instructions are intended for users that want to try the cutting-edge version of LibreCAD and are **not** intended to replace the instructions for building packages or contributing to the LibreCAD project.

If you are a developer and want to contribute to LibreCAD see the *Contributing* section in the **Appendices**.

The tools and dependencies required to build LibreCAD are:

- C++ compiler and related utilities
- Qt development framework
- Boost C++ source library
- muParser math expression parser library
- git distributed version control system (optional)

5.9.1 Download the Source Code

The source code is hosted on GitHub and is common to all three operating systems. It can be download as a "zip" archive or cloned using "git". These instructions use the download option.

Go to the LibreCAD GitHub page to download the source code. On the **<> Code** tab, click on the "Clone or Download" button and then click "Download ZIP". Save the zip file; *LibreCAD-master.zip*.

Cloning is suggested if users want to build LibreCAD more frequently as the source code is updated. More information about GitHub, git tools, and creating a local source code repository can be found on LibreCAD's Developer's wiki.

5.9.2 Building on Linux

Note: These instructions are for building LibreCAD on **Debian** and other derivatives. Further instructions for building LibreCAD on other Linux distributions (openSUSE, Red Hat, FreeBSD) and generic Unix can be found in LibreCAD's GitHub Developer's wiki in the **Build from source** section.

Install Tools and Dependencies

Install the required tools and libraries (compiler, Qt, boost, muparser, etc.):

```
$ sudo apt install g++ gcc make qtbase5-dev libqt5svg5-dev qttools5-dev\
  qtchooser qttools5-dev-tools libboost-dev libmuparser-dev librsvg2-bin\
  libfreetype6-dev libicu-dev pkg-config
```

You also have to either install the qt5-default package; `apt install qt5-default`, or use qtchooser prior to running qmake; `qtchooser -qt5`.

Build LibreCAD

Create a development directory for the source code and related libraries; e.g `~/dev/`. Extract the contents of the source code zip file, *LibreCAD-master.zip*, to the development directory. When complete, a complete copy of the source code will found in the *~/dev/LibreCAD-master* directory. Compiled LibreCAD with the following commands:

```
$ cd ~/dev/LibreCAD-master/
$ qmake -r
$ make -j4
```

If the build is successful an executable is created, *~/dev/LibreCAD-master/unix/librecad*, and LibreCAD can be launched by typing:

```
$ ./unix/librecad &
```

5.9.3 Building on Windows

Building LibreCAD on Windows is a little more involved and requires a few additional steps. Please read these instructions carefully.

Note: Detailed instructions for building LibreCAD on Windows, including instructions for building LibreCAD in **Visual Studio 2013** and newer, can be found on the GitHub Developers wiki (https://github.com/LibreCAD/LibreCAD/wiki) in the **Build from source** section.

Install Tools and Dependencies

Qt Framework

The Windows version of Qt includes both the required framework library and the compiler. Download the *open source* version of the **Qt Online Installer** from Qt download <https://www.qt.io/download>. Launch the installer accepting the defaults. Install Qt to the default path prompted by the installer, e.g. *C:\Qt*. On the *Select Components* page expand the tree view under the most recent version of Qt, e.g. *Qt 5.12.3*, and select the latest version of the **MinGW** compiler; *MinGW 7.3.0 (32-bit or 64 bit as required)*. No other components are needed.

Boost

Download the current release of the boost library "zip" file from Boost downloads <https://www.boost.org/users/download/>. Click on the link to download the current Windows library, e.g. *boost_1_70_0.zip* and save the file.

muParser

muParser is not required to build LibreCAD on Windows as the library is now included with the LibreCAD source code.

Build LibreCAD in Qt Creator

Create a development folder for the source code and related libraries, e.g *C:\dev*. Extract the contents of the source code zip file, "LibreCAD-master.zip". When complete, a copy of the source code will found in the *C:\dev\LibreCAD-master* folder.

Extract the boost library the files to the development folder. Note the folder name the boost library was extracted to, e.g. *C:\dev\boost_1_70_0*.

> *After* extracting the LibreCAD source code, open the *custom.pro* file in *.\LibreCAD-master\librecad\src* under the development folder and add the following two lines (**note the forward slashes in the path.**):

```
BOOST_DIR = C:/dev/boost_1_70_0/
BOOST_LIBDIR = C:/dev/boost_1_70_0/
```

After completing the required edit, launch Qt Creator (**Start -> All Programs -> Qt -> Qt Creator**) and open the *librecad.pro* project file from the LibreCAD source folder (**File -> Open File or Project** and go to *C:\dev\LibreCAD-master*). The project should open to **Configure Project**. Ensure a "kit", e.g. *Desktop Qt 5.12.3 MinGW 32-bit* is checked and click the **Configure Project** button. It will take a few moments for the project to open and parse.

Click the **Project** icon on the left side of the Qt Creator window. Disable the "Shadow build" option in the *Debug*, *Profile* and *Release* build configurations. Each build configuration can be selected from the drop down below **Build Settings**. Save the project (**File -> Save All**).

With the configuration complete, run the build process in Qt Creator by clicking the **Build** icon on the lower left side. If the build is successful an executable is created: .\LibreCAD-master\windows\librecad.exe.

Important: Several *Dynamic-link libraries (DLL)* are required to run LibreCAD. The DLLs are found in the C:\Qt\5.12.3\mingw73_32\bin folder (or mingw73_64) and need to be copied to the same directory as the executable (or included in the path). The DLLs are:

- libgcc_s_dw2-1.dll
- libstdc++-6.dll
- libwinpthread-1.dll
- Qt5Core.dll
- Qt5Gui.dll
- Qt5PrintSupport.dll
- Qt5Svg.dll
- Qt5Widgets.dll

Once the DLLs have been copied to the executable folder, LibreCAD can be launched by Clicking **Start -> Run** and typing:

```
> C:\dev\LibreCAD-master\windows\librecad.exe
```

5.9.4 Building on macOS

There are a few options for compiling LibreCAD for the macOS. This method uses the **LLVM Compiler Infrastructure** and the **Homebrew Package Manager**. Refer to the Github Developer's wiki for additional methods.

Install Tools and Dependencies

From the command prompt, download and install Homebrew with:

```
/bin/bash -c "$(curl -fsSL https://raw.githubusercontent.com/Homebrew/install/master/
↪install.sh)"
```

Further details are available on the Homebrew website.

Install the latest versions of boost and qt5 with Homebrew:

```
brew install boost qt5
```

Build LibreCAD

Create a development directory for the source code and related libraries; e.g ~/dev/. Extract the contents of the source code zip file, *LibreCAD-master.zip*, to the development directory. When complete, a complete copy of the source code will found in the *~/dev/LibreCAD-master* directory. Change to the source code directory with:

```
cd ~/dev/LibreCAD-master/
```

Next build LibreCAD by running qmake. Note that during the installation step above, `brew install qt5`, the binaries have not been linked to */usr/local*. In order to use qmake, use the full path:

```
/usr/local/Cellar/qt5/5.7.0/bin/qmake librecad.pro -r -spec macx-clang
```

To compile the application, at the prompt run:

```
make -j4
```

If the previous steps were successful, an application named **LibreCAD.app** will be in the build directory. Run it at the command prompt by typing either:

```
open LibreCAD.app, or
LibreCAD.app/Contents/MacOS/LibreCAD
```

Note: *The macOS build section been revised, but not tested.* Please provide any feedback on the build process on the LibreCAD forum: http://forum.librecad.org/Help-wanted-to-build-on-MacOS-td5717273.html

5.10 Contributing

5.10.1 Code

If you are a developer, outstanding in C++, or a beginner we can use all the help. With a bit of determination, the IRC channel, Zulip chat and the Libre-CAD-dev mailing list, we can help you get started and making progress. The source code is hosted on GitHub (https://github.com/LibreCAD/LibreCAD). A good starting point for developers is the LibreCAD Developer's wiki on GitHub (https://github.com/LibreCAD/LibreCAD/wiki/Git-and-GitHub).

The *Online Resource* section provides links to further information on downloading and building LibreCAD.

5.10.2 Documentation

Developers are usually bad in making documentation, so if you like writing documentation your involvement in developing the **User Manual** would be greatly appreciated.

The documentation is hosted by Read the Docs. The most recent version of LibreCAD's User Manual can be found at https://librecad.readthedocs.io/. As with the source code, the *documentation source* is hosted GitHub (https://github.com/LibreCAD/docs). The documentation is created using a markup language called **reStructuredText** (Refer to http://docutils.sourceforge.net/rst.html). For those interested in contributing to the documentation, refer to LibreCAD Documentation wiki on github (https://github.com/LibreCAD/docs/wiki).

5.10.3 Translation

Help localize LibreCAD, we currently support over 30 languages. Our translation server is at https://translate.librecad.org/. You can contribute to a language already started or or request the creation of a translation for a new language in the forum in the **LibreCAD-translate** section.

5.11 License

5.11.1 LibreCAD and the GPLv2

LibreCAD is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License version 2 (GPLv2) as published by the Free Software Foundation.

This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

The GPLv2 can also be found in the *licenses* folder as *gpl-2.0.txt*.

- <https://github.com/LibreCAD/LibreCAD/tree/master/licenses>
- <http://www.gnu.org/licenses/old-licenses/gpl-2.0.txt>

History

QCAD community edition version 2.0.5.0 was released under the GPLv2. > Copyright (C) 2001-2003 RibbonSoft. All rights reserved.

- <http://www.ribbonsoft.com>

LibreCAD is a fork of QCAD community edition version 2.0.5.0; therefore, any modifications of or additions to the program are also released under the GPLv2.

The project's first commit on Github was Aug 16, 2010.

- <http://librecad.org/>
- <https://github.com/LibreCAD/LibreCAD>

Rights holders of the code

Some of the following may be composites of multiple copyright notices.

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- 2010, 2011, 2012, 2015 R. van Twisk (librecad@rvt.dds.nl)
- 2011-2015 Dongxu Li (dongxuli2011@gmail.com)
- 2011-2015 José F. Soriano (rallazz@gmail.com)
- 2014 cgrzemba (cgrzemba@opencsw.org)
- 2014 Pavel Krejcir (pavel@pamsoft.cz)
- 2014-2015 Christian Luginbühl (dinkel@pimprecords.com)
- 2015 A. Stebich (librecad@mail.lordofbikes.de)
- 2015-2016 ravas (github.com/r-a-v-a-s)

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Honorable mention

These are known contributors whose copyright notice could not be found.

- Claude Sylvain | <csylvain@electro-technica.com>
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A list of contributors to the LibreCAD repository can be found at: <https://github.com/LibreCAD/LibreCAD/graphs/contributors>

5.11.2 Licenses of bundled content

The LibreCAD source code is released as a whole under the GPLv2. The following refers to the licenses used by these resources, when they were released independently from LibreCAD.

The SVG icons in *librecad/res/icons* were all released under the CC0 license.

- <https://github.com/LibreCAD/Resources>
- <https://creativecommons.org/publicdomain/zero/1.0/>

libdxfrw was released under GPLv2 "or any later version".

- <https://sourceforge.net/projects/libdxfrw/>

muParser was released under the MIT license.

- <http://beltoforion.de/article.php?a=muparser&hl=en&p=licence>

Shapelib was released under a "MIT Style" license or LGPLv2.

- <https://github.com/LibreCAD/LibreCAD/tree/master/plugins/importshp/shapelib>

The AZOmix fonts were released under the KST32B version 2.0 license.

- <https://github.com/LibreCAD/LibreCAD/blob/master/licenses/KST32B_v2.txt>
- <http://domisan.sakura.ne.jp/article/cadfont/cadfont.html>

The Amiri font was released under the Open Font License.

- <http://www.amirifont.org/>
- <https://github.com/khaledhosny/amiri-font/blob/0.108/OFL.txt>

The Hershey Fonts were originally created by Dr. A. V. Hershey while working at the U. S. National Bureau of Standards. The format of the Font data in this distribution was originally created by:

James Hurt Cognition, Inc. 900 Technology Park Drive Billerica, MA 01821