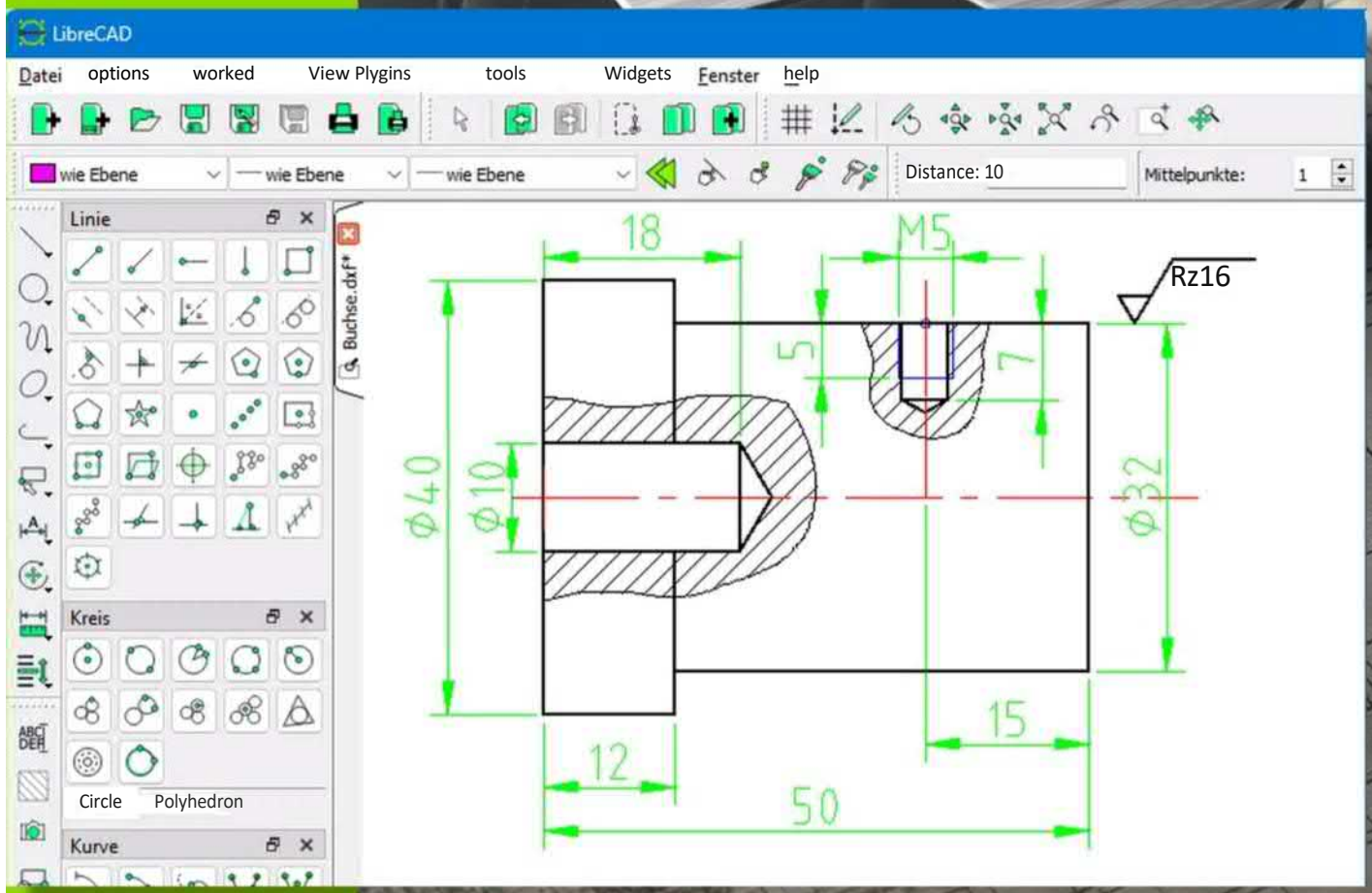


(Machine translation from original German LibreCAD book.)

LibreCAD

For beginners and advanced users

Unleash your creativity with LibreCAD



The easy-to-understand guide for precise and professional drawings, with lots of pictures and step-by-step instructions.

LibreCAD gibt es für:

- Windows
- macOS
- Linux

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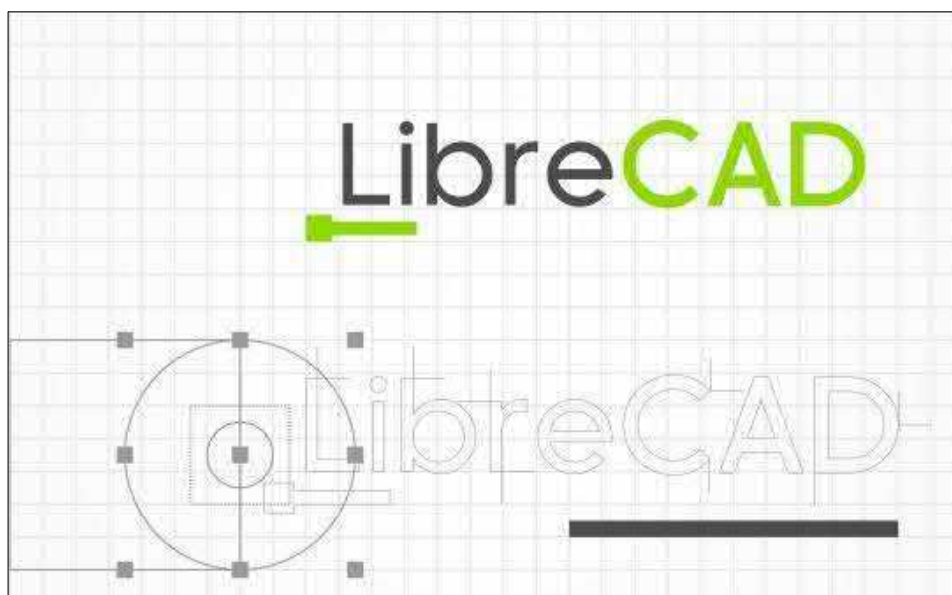
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The LibreCAD book for beginners and advanced users.

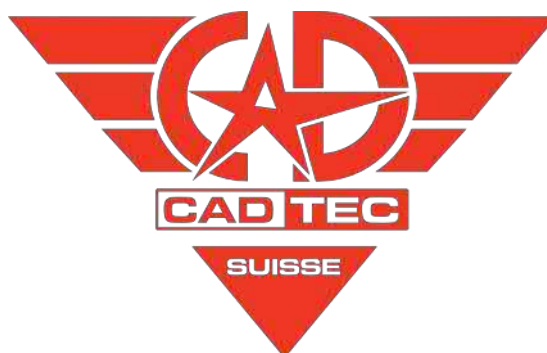
Unleash your creativity with LibreCAD.

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Preview copy Page 1-60

1 Contents

Table of contents

1 Contents	6
2 Foreword	12
2.1 One request	13
3 First things first!	14
4 What is a CAD program?	15
5 Entities, objects and elements	16
6 What is LibreCAD?	17
6.1 Download and links	17
6.2 LibreCAD is well suited for	18
6.3 The history of LibreCAD	18
6.4 The most important milestones in the development of LibreCAD	18
6.5 LibreCAD today	18
6.6 Is LibreCAD the right choice for you?	18
7 DXF and DWG file format	19
8 Basics	20
8.1 Coordinate systems	20
8.2 Angle	21
8.2.1 Conversion between Cartesian and polar	24
8.3 Repetition: Absolute and relative coordinates	24
8.4 Coordinate input	25
8.5 Views side view, elevation, floor plan	27
9 Pen assistant	28
10 Color	29
10.1 Line widths	30
11 Layers	31
11.1 Create level	32
11.1.1 Design level	32
12 Functions Brief overview	33
12.1 Vector-based drawings	33
12.2 Intuitive user interface	33
12.3 Various drawing tools	33
12.4 Layers	33

12.5 Catch functions, snapping (snap-in functions).....	33
12.6 Support for various file formats	33
12.7 Customizable user interface	33
13 Installation	34
14 Reset LibreCAD	35
15 User interface Quick overview	36
15.1 Coordinate display.....	37
15.2 Next action with the mouse.....	37
15.3 Zoom in, zoom out and pan-zoom.....	38
16 Basic settings.....	40
16.1 Program settings	40
16.2 Drawing settings.....	41
16.2.1 Register: Units and decimal places	41
16.2.2 Register: Dimensions.....	42
17 Further settings.....	44
17.1 Layers and line types.....	44
17.2 Level 0 (zero).....	47
17.3 Possible problems when printing line widths.....	48
18 Saving settings as a template.....	49
18.1 Standard template	50
19 Hotkeys / short keys.....	51
20 Drop-down or floating menus?.....	52
21 What are widgets?.....	53
22 Drawing tools and command line.....	54
22.1 Line tools.....	54
22.2 Circle tools.....	56
22.3 Arcs, ellipses and spline tools	58
22.4 Ellipse tools	59
22.5 Polyline tools	60
22.6 Catch functions, snapping options	61
22.7 Selection tools	64
22.8 Dimensioning tools.....	65
22.9 Editing tools.....	67
22.10 Query information.....	70
22.11 Text, hatching, dot.....	71
23 Calculator in the command line	72
24 librecad.alias (German command file).....	73

25 Procedure for creating drawings	76
26 Make your own first drawing	78
26.1 First of all, something important about the grid catch.....	78
26.2 So, draw this simple rectangle.	78
26.3 Another rectangle with Cartesian and polar coordinates.	79
27 Construct an oblique rectangle.	81
27.1 Attaching ventilation slots.....	84
28 Coordinates exercise 001	87
28.1 Control measure.....	89
29 Coordinates Exercise 002	90
30 Construct triangle	91
31 Semicircle and two lines, what is the radius?	93
31.1 The theorem of Thales	94
31.2 Constructive solution	95
31.3 Construction of a circle tangent.....	98
32 Tangents exercise	99
33 Prism control measure	100
33.1 Trim function	104
33.2 Delete	106
33.3 Draw a circle tangentially into the prism.....	107
33.4 Set control dimension C.....	108
34 Calculate 9-corner coordinates	109
34.1 Deactivate grid snap.....	110
34.2 N-corner.....	110
34.3 Display coordinates without having to dimension	111
35 Drawing: Belt drive and coordinates	112
35.1 Workpiece coordinates	115
35.2 Milling coordinates.....	115
36 Drawing: Planetary gearbox	117
36.1 Determine and draw pitch circle	119
36.2 Draw vertical axis	119
36.3 Draw first planet	119
36.4 Planet 2x circular pattern.....	120
36.5 Circle tangent to three planets	121
37 Drawing: Turned part	123
37.1 Trimming.....	125

Chapter 1 Contents

37.2 Mirror	126
38 Processing time of the runner from 2D drawing	128
38.1 Round off edges	131
38.2 Chamfer edges	131
38.3 Draw roughing paths.....	132
39 Draw cone	134
39.1.1 Draw axis.....	135
39.1.2 Draw contour	135
39.1.3 Round off edge.....	135
39.1.4 Parallel lines	135
40 Draw omega shape	137
40.1 Draw rectangle.....	137
40.2 Draw central axes.....	137
40.3 Creating circles and tangential lines	138
40.4 Draw angled lines.....	138
40.5 Tangential arcs.....	138
40.6 Center and opening angle	139
40.7 Designing cutter coordinates	140
41 Task: Draw drilling jig	144
41.1 Solution aid	145
41.1.1 45° axis, variant with coordinate input.....	145
41.1.2 45° axis, variant with functions.....	146
41.1.3 Bore D15	147
41.1.4 Draw guides	147
41.2 Infinitely extending auxiliary lines	148
41.3 Create sectional view	151
41.3.1 Copy drilldown	152
41.3.2 Change lines from invisible to visible.....	152
41.4 Hatching areas.....	153
41.4.1 Hatching restrictions	154
41.4.2 Break up block.....	154
41.5 Edit hatching.....	157
41.6 Add WHITE layer to layer list	158
41.7 Create hatching	159
42 Use blocks	160
43 Working with blocks	162
44 Integrate libraries, symbols, standard parts.....	163

44.1 Create your own standard part symbol (M8)	164
44.2 Central standard parts library.....	165
44.3 Side view of M8 thread	167
44.4 Route function in blocks.....	170
44.5 Remember this when printing filled areas	171
45 Bolt circle with first hole at 75°	172
46 Draw hook example.....	175
47 Jaw chuck plate with M8 thread.....	178
47.1 Proceed in segments when projecting	180
47.2 Alternative to projecting.....	180
47.3 Panel projection	180
47.4 Create sectional view	181
48 Download standard parts.....	182
49 Draw bushing from hand sketch	183
49.1 Create hatched breakout	186
49.2 Insert blind hole threaded hole as standard part.....	191
50 Create screw connection	196
50.1 Screw connection step by step.....	197
50.2 Complete template	202
51 Extend inclined bore	203
51.1 Extend crooked rectangle.....	203
52 Identify block in drawing.....	205
53 Plugins	206
53.1 Draw gear wheel	207
53.2 Points from ASCII file.....	208
53.3 Import ESRI shape data.....	209
53.4 Read PIC file	210
53.5 Plot of an equation.....	211
53.6 Same properties	212
53.7 Divide (Share).....	213
53.8 List objects	214
54 Create drawing header according to DIN	215
54.1 Draw drawing header according to ISO 7200 in LibreCAD	216
55 Print.....	218
55.1 Print preview window	218
55.2 Print to scale.....	219
55.3 Blank print preview	221

56 Tips and tricks with LibreCAD	222
56.1 Hatching trick.....	222
56.2 Accelerate LibreCAD.....	223
56.3 Relative object snap	223
56.4 Correct geometry with distance function.....	224
56.5 Spline offset or parallel?.....	226
56.6 Formatting multi-line text?	226
56.7 Where is the properties window?.....	226
56.8 Export settings.....	227
56.9 LibreCAD goes crazy.....	227
56.10 No numbers are displayed when dimensioning	228
56.11 The element disappears when drawing!	228
57 Useful links	229
57.1 Other CAD programs	229
57.2 CAM program	229
57.3 DWG, DXF converter	229
57.4 Standard parts, standard template and more	229
57.5 Miscellaneous.....	230
58 Contributors	231
58.1 Corrections, improvements, errors found	231
59 Index	232

2 Foreword

Grüezi, hello, hello,

LibreCAD is an exciting and user-friendly open source CAD program that is ideal for 2D drawings and designs. Whether you are an engineer, architect, designer or hobbyist, LibreCAD provides you with the key tools and features you need to create professional drawings.

This book or tutorial is designed to give you a comprehensive overview of LibreCAD and help you get the most out of this program. We will cover the basics of LibreCAD, such as the user interface, tools and functions, creating drawings and working with blocks and libraries. We will also cover more advanced topics such as panel projection, creating hatches and using variables.

This book is aimed at beginners and advanced users alike. If you are new to the world of CAD drawings, we will show you step by step how to create your first drawings and master the basics of LibreCAD. If you already have experience with LibreCAD or other CAD programs, we will help you improve your skills and learn new features and techniques.

We have designed this book to be easy to read and understand, with many examples and illustrations that will help you visualize the concepts and techniques. We have also made sure that this book has been updated with the latest version of LibreCAD so that you can learn about the latest features and tools.

We hope this book will help you improve your skills in LibreCAD and increase your creativity and productivity. Whether you are a professional designer or a hobbyist, we are confident that you will find useful information and techniques in this book that will help you achieve your goals.

Enjoy reading and learning!

2.1 One request...

We have endeavored to make this book as error-free as possible. However, if you notice any typographical errors or inaccuracies, we would be very grateful if you would report them to us. As this book is written by users for users, we are always striving to improve and update it to ensure that it is as useful and informative as possible. We hope that you will find this book a helpful resource for your work with LibreCAD and wish you every success with your projects!

This LibreCAD book is a living project and you will find on one of the last pages an e-mail address where you can send us improvements at any time.

"Merci vielmal!"

3 First things first!

The following points are advantageous if you already know them:

LM = left mouse button

RM = right mouse button

[Ctrl] the control key corresponds to **[Ctrl]** on a German keyboard.

RM-click opens a menu with the most recently used functions.

Ctrl + Z = Undo / **Ctrl + Y** = Redo.

Ctrl + C = Copy with reference point / **Ctrl + V** = Paste with target point.

Space bar activates the command line.

Polar coordinate input in the command line: **@30<45**

Angular input Horizontal is 0° **counterclockwise RM** or

Esc key cancels a function.

Zoom with mouse wheel / **Pan-Zoom** Hold down the mouse wheel.

Objects are **elements** are **entities**

See chapter 5 Entities, objects and elements

Calculator on/off: cal → Enter (command line) A

drawing template is the same as a **template**.

See chapter 18 Saving settings as a template

Standard parts are **blocks**, **symbols** are **blocks** → are **imported** files.

See chapter 42 Using blocks and 44.2 Central standard parts library

Standard parts path: for **library browser** see chapter 44.2 Central standard parts library

For functions that require a mass input, such as a parallel line, the distance can be entered at the top of the status line or **directly** in the **command line**.

4 What is a CAD program?

A 2D CAD (Computer-Aided Design) program is software that allows users to create, modify and save two-dimensional technical drawings, sketches, plans and diagrams. This type of software is often used in various industries, such as mechanical engineering, architecture, civil engineering, electrical engineering and many other areas where precise technical drawings are required.

2D CAD programs offer a variety of functions and tools to create and edit geometric shapes, lines, circles, arcs, curves and other elements. Users can combine these elements to create complex drawings that are true to scale and accurate. The most important functions of 2D CAD programs usually include

- Drawing tools: lines, circles, arcs, curves, polylines, splines, etc.
- Editing tools: move, rotate, scale, mirror, trim, extend, etc.
- Dimensioning and labeling tools: linear, radial, angular, diameter dimensions, texts, tolerances, etc.
- Layer management: Organize drawing elements in different layers for better control and visibility
- Hatches and patterns: Adding hatches and patterns for different materials and surfaces
- Import and export functions: Support of various file formats such as DWG, DXF, PDF, SVG etc.

Some examples of 2D CAD programs are AutoCAD, AutoCAD LT, DraftSight, TurboCAD, LibreCAD* and many others. These programs may be available as standalone software or as part of a more comprehensive CAD suite that also includes 3D modeling and other advanced features.

* Open source and **free of charge!**

5 Entities, objects and elements

In 2D CAD programs, different terms are used to describe the basic building blocks of drawings. Although these terms are often used interchangeably, there are subtle differences between them:

Entities are the most basic building blocks of a CAD drawing. They represent geometric shapes such as lines, arcs, circles, polygons and text. Entities have properties such as color, thickness and material.

Objects are groups of entities that can be combined and treated as a single unit. They can be used to create complex shapes or assemblies. Objects have properties such as name, description and layer.

Elements are a more abstract term for entities and objects. They can refer to individual entities, groups of entities or even entire drawings. Elements are often used in commands and functions of CAD programs.

To summarize:

- **Entities** are the basic geometric shapes.
- **Objects** are groups of entities that can be treated as a unit.
- **Elements** are a general term for entities and objects.

Here is an example to illustrate the differences:

Imagine you are drawing a house in a CAD program. The walls, the roof, the windows and the doors are all entities. You can group these entities into a "house" object. The house itself can then be used as an element in a parts list or other report.

Other differences:

- Entities always have a unique ID, whereas objects and elements do not necessarily have to.
- Entities can only belong to one object, while objects and elements can belong to several objects or groups.
- Entities are usually saved on the drawing level, while objects and elements can be saved in separate data structures.

The exact definitions and relationships between entities, objects and elements may vary depending on the CAD program. However, it is important to understand the basic differences in order to use CAD programs effectively.

6 What is LibreCAD?

LibreCAD is an open source CAD program that supports 2D drawings and designs. It is a powerful yet user-friendly tool that is available for various platforms such as Windows, MacOS and Linux. In this tutorial, we will cover the basics of LibreCAD's operating philosophy to help you get started with the software.

Main features:

- **Drawing tools:** lines, arcs, circles, polygons, splines, dimensions, labels and more
- **Modification tools:** move, copy, rotate, scale, mirror, trim, extend and more
- **Snap functions:** Object snap, end point snap, center point snap, intersection snap and more
- **Layers:** Organize your drawing in layers
- **Blocks:** Create reusable symbols and groups
- **Layouts:** Create multiple views of your drawing
- **DXF/DWG support:** Import and export of DXF and DWG files (AutoCAD format)
- **PDF and SVG export:** Export your drawings to PDF and SVG files
- **Multilingual:** Over 30 languages available
- **License:** GPLv2 (Free Software)

6.1 Download and links

Website: <https://librecad.org/>

Documentation: <https://dokuwiki.librecad.org/>

Wiki: <https://wiki.librecad.org/index.php>

Forum: <https://forum.librecad.org/>

6.2 LibreCAD is well suited for

- Technical drawings: Creating floor plans, sections, views and details
- Mechanical design: designing machines and components
- Electronics: Creating circuit board layouts
- Architecture: Designing buildings and spaces
- Landscaping: Creating garden plans
- Hobby and handicraft work: Creating designs for furniture, decoration and more

LibreCAD is an easy-to-use yet powerful 2D CAD software that is ideal for private and professional use.

6.3 The history of LibreCAD

LibreCAD is a free 2D CAD software that was created in 2011 from a fork of the open source software Qcad Community Edition. The original Qcad software was developed by RibbonSoft and was licensed under the GPLv3 license.

6.4 The most important milestones in the development of LibreCAD

2011: Foundation of the LibreCAD project

2013: Release of the first stable version 1.0

2015: Introduction of support for the DWG format

2017: Release of version 2.0 with numerous new functions

2020: Release of version 2.1 with further improvements

2023: Release of version 2.2 with support for 3D objects

6.5 LibreCAD today

LibreCAD is an active open source community with a constantly growing number of users and developers. The software is constantly being developed and improved. It is a free and open-source alternative to commercial 2D CAD programs such as AutoCAD and is ideal for private and professional use.

6.6 Is LibreCAD the right choice for you?

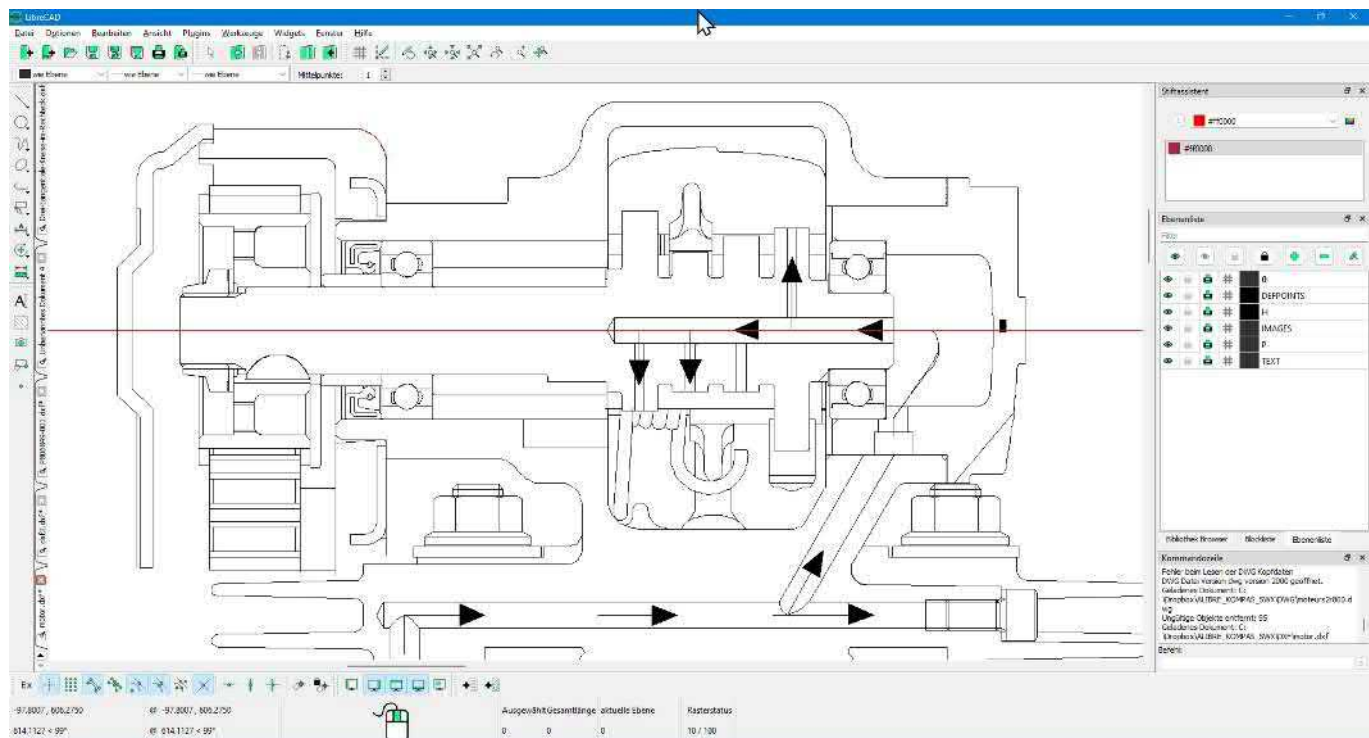
It depends on your individual needs and requirements. If you are looking for free, open source and cross-platform CAD software for creating two-dimensional drawings, LibreCAD is a good option and an excellent choice for users looking for CAD software without copy protection. LibreCAD is a **completely free and open source program licensed** under the GNU General Public License (GPL) v3.0. This means that you can download, use, modify and redistribute it freely.

7 DXF and DWG file format

DXF and DWG are the most important file formats in the 2D CAD sector. Both are vector graphic file formats. Vector graphics retain the same image quality at any scale and are therefore ideal for design purposes. In addition, the individual elements that make up a vector image can be easily edited and elements can be added and removed.

DXF (Data eXchange Format) was developed as a tool for exchanging drawings between different CAD and vector-based programs. It saves 2D vector images and can be used by almost all CAD programs as well as CNC and GIS (Geographic Information System) software.

Each element of the drawing is written in plain text or ASCII format, which contains all alphanumeric characters. Due to their file specifications, complex DXF drawings can be very large - from several to hundreds of megabytes. If such a document is to be transferred, it must either be split or compressed.



DWG (for Drawing) is a proprietary format and is only included in LibreCAD in the beta stage. If you need to import DWG files, we recommend the following DWG viewer/converter from **VariCAD** in chapter: 57 Useful links

This costs only USD 25 and is of excellent quality. It converts all DWG and DXF versions so that they can usually be read into LibreCAD without errors.

8 Basics

In order to use LibreCAD effectively, some concepts need to be understood. While a simple drawing or sketch can be created after the initial setup, for more complex drawings it is important to consider the elements of a complete drawing. This section provides an introduction to some of the concepts required to create a drawing, but is by no means exhaustive. The remainder contains a description of the tools used to configure, create and modify drawings. Further general examples and instructions can be found in the User Manual section.

LibreCAD supports two drawing perspectives: orthogonal and isometric projections. The orthogonal projection is the standard perspective for creating two-dimensional (2D) drawings. An isometric projection allows LibreCAD to display a three-dimensional object in two dimensions, sometimes referred to as "2.5D". Both projections use coordinates to localize drawing elements.

8.1 Coordinate systems

In 2D CAD programs (Computer Aided Design), geometric shapes such as points, lines, circles, arcs, etc. are created and edited on a two-dimensional plane. The position of these elements on the drawing plane is defined by a coordinate system consisting of an x-axis (horizontal) and a y-axis (vertical). Each point on this plane therefore has a unique address, which is represented by an ordered pair of coordinates (x, y).

Coordinate input is a method of precisely determining the position of geometric elements. Instead of manually drawing elements with the mouse, users can enter the exact coordinates where the element should be placed. This leads to greater accuracy and makes it possible to create complex geometric shapes.

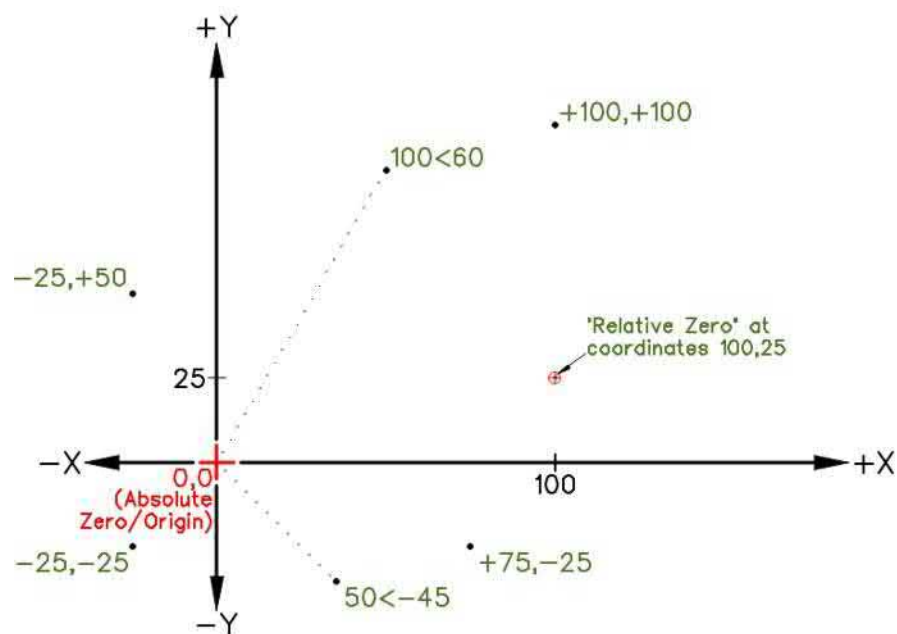
To use coordinate input in a 2D CAD program, users must usually first select the desired geometric element (e.g. a line or a circle). They can then enter the coordinates for the start point and the end point (or the center point and the radius in the case of a circle). This can be done either via a prompt in the command line of the program or by filling in fields in a dialog box.

It is important to adhere to the correct coordinate sequence (x, y) and to enter the coordinates in the unit of measurement supported by the program (e.g. millimetres, inches). Once the coordinates have been entered, the geometric element is automatically placed at the specified position on the drawing plane.

In summary, coordinate input in 2D CAD programs enables precise positioning of geometric elements on the drawing plane by entering coordinate values.

Understanding coordinate systems and the way coordinates work in LibreCAD is necessary to create accurate drawings. Points are used to describe an aspect of an object (e.g. the end of a line, the center of a circle, etc.) and can be accurately placed 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 coordinate system:

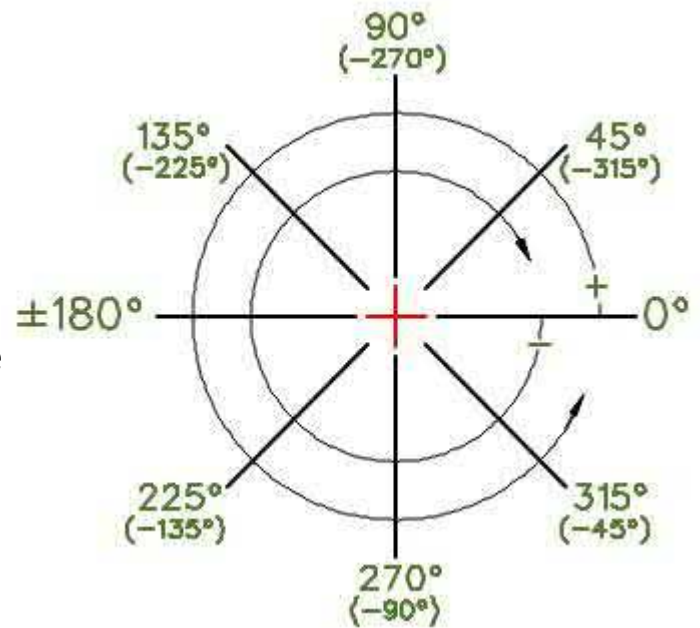
- a horizontal distance and a vertical distance from a reference point (**Cartesian**) or
- an angle and a distance from a reference point (**polar**).

8.2 Angle

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, starting at 0 degrees horizontally to the right of the origin or at

the 3 o'clock position. Angles that are entered as a positive value are measured counterclockwise. Angles that are entered as a negative value are measured clockwise.

When using absolute coordinates, whether Cartesian or polar, the points are entered in direct relation to the origin (0,0). To do this in LibreCAD, enter the desired point, e.g. **100.75** or **100<45**, as shown in the two images above.



The next point can also be placed relative to the previously placed point. The last point that was set when an object was created becomes a temporary reference for the next point. The newly set temporary reference point is the "Relative zero point" and the next coordinates can be entered relative to this point. To define the next point **relative** (often referred to as **incremental**) to the relative zero point, either Cartesian or polar, prefix the coordinates with "**@**". Points without the prefix **@** are always interpreted as absolute coordinates.

Cartesian coordinates are used to set a point 75 units to the right and 65 units above the previous point, take **@75,65**. In the example shown, the previous point was set 25 units horizontally and 35 units 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).

Relative coordinates or **incremental coordinates** can also be entered using the "**x.y**" format. This format corresponds to placing the "**@**" symbol in front of the coordinates. For example, if you enter the coordinates "**75..65**", this corresponds to "**@75,65**".

Please familiarize yourself with the possibility of entering Cartesian and polar coordinates. This will be used from time to time throughout the book.

Question: What do these following inputs generate when you use the function:

Lines → **2-point** takes? Command line: **0,0** ↵ **50,0** ↵ **@50<120** ↵ **0,0** ↵

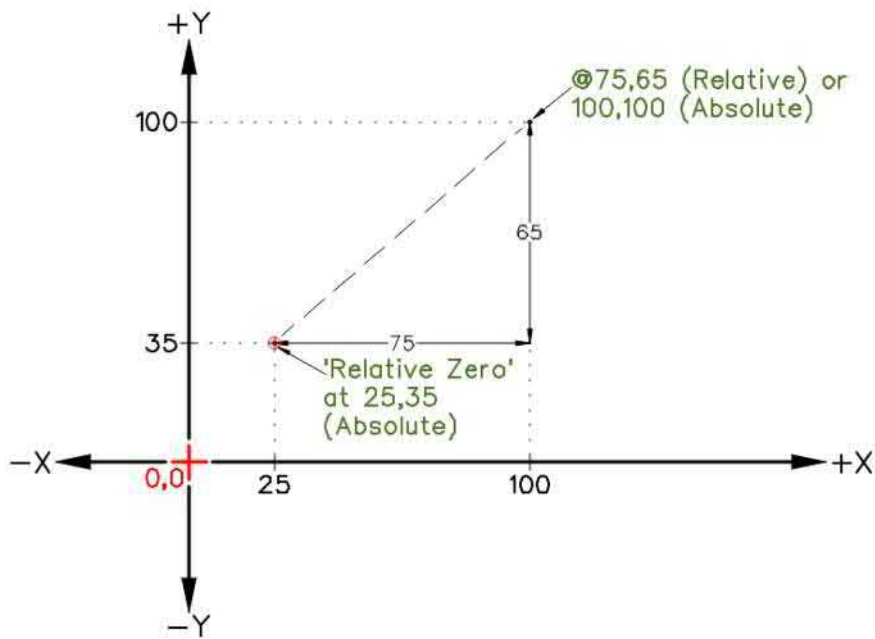


Figure 1: Absolute and relative Cartesian coordinates

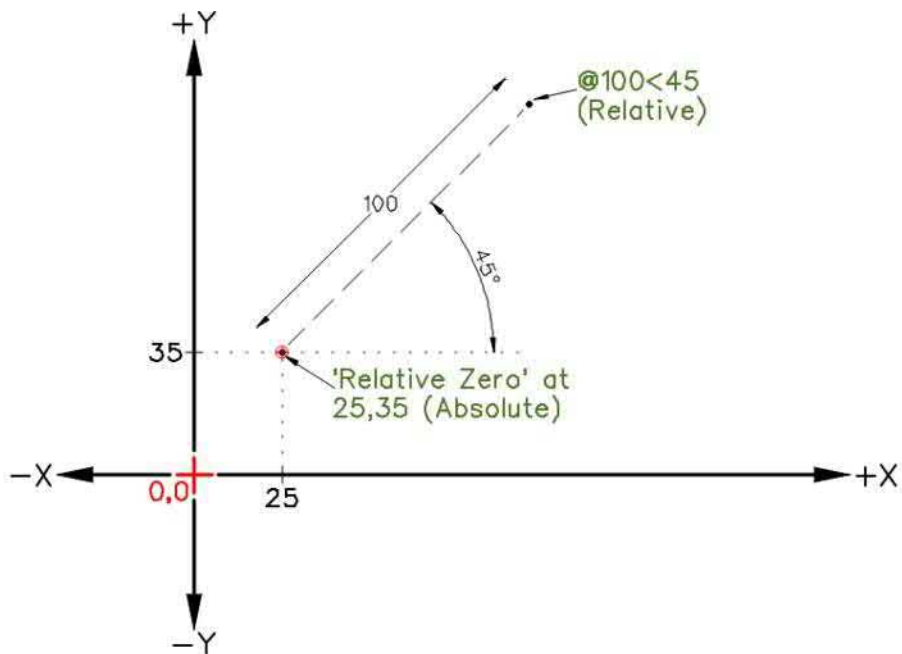


Figure 2: Absolute Cartesian and relative polar coordinates

Polar coordinates: To place a point 100 mm and 45 degrees from the last drawn point at 25.35 (absolute Cartesian), use "@100<45" (relative Polar).

8.2.1 Conversion between Cartesian and polar

Conversion: It is possible to convert between Cartesian and polar coordinates. The conversion from polar coordinates to Cartesian coordinates is carried out using the following formulas:

$$x = r * \cos(\theta)$$

$$y = r * \sin(\theta)$$

Cartesian coordinates are converted to polar coordinates using the following formulas:

$$r = \sqrt{x^2 + y^2}$$

$$\theta = \arctan(y/x)$$

To summarize, Cartesian and polar coordinates are two different ways of representing the position of a point in a two-dimensional plane. Cartesian coordinates use a distance from two axes, while polar coordinates use a distance and an angle from the pole and the polar axis. The choice of coordinate system depends on the specific application.

8.3 Repetition: Absolute and relative coordinates

Absolute coordinates: Absolute coordinates refer to a fixed coordinate system in which each point in the plane is represented by a unique pair of coordinates (x, y). The origin of the coordinate system is at a fixed position and all distances are measured from this origin.

Relative coordinates: Relative coordinates or incremental coordinates refer to the current position of the cursor or the last defined point. In this system, the position of a point is specified by the distance and direction from the current position. For example, if a point was defined at (10, 10) and the next point should be 5 units to the right and 3 units upwards, relative coordinates can be used to define the new point at (15, 13).

Differences:

1. **Reference point:** The main difference between absolute and relative coordinates is that absolute coordinates have a fixed reference point, while relative coordinates are based on the current position.
2. **Accuracy:** Absolute coordinates are generally more accurate than relative coordinates, as they are based on a fixed coordinate system and do not depend on the current position.
3. **Flexibility:** Relative coordinates are more flexible than absolute coordinates because they allow you to specify distances and directions from the current position instead of specifying the exact position of each point.
4. **Application:** Absolute coordinates are often used in applications where high accuracy is required, such as in mechanical engineering or architecture. Relative coordinates, on the other hand, are often used in applications where flexibility is more important than accuracy, e.g. in graphic design or image processing.

To summarize, absolute and relative coordinates are two different ways of representing the position of a point in a plane. Absolute coordinates refer to a fixed coordinate system, while relative coordinates are based on the current position.

8.4 Coordinate input

In LibreCAD, coordinates can be entered in various formats, depending on the user's requirements and the specific drawing. Basically, LibreCAD supports the input of Cartesian coordinates, polar coordinates and relative coordinates.

Cartesian: Cartesian coordinates can be entered in LibreCAD in the following form:

- **Absolute:** x,y , where x and y are the coordinate values. For example: **10,20**
- **Relative:** $@dx,dy$, where dx and dy are the displacement values from the **current** position are. For example: **@10,20**

Polar: Polar coordinates can be entered in LibreCAD in the following form:

- **Absolute:** $r<angle$, where r is the distance from the coordinate origin and $angle$ is the clockwise angle from the positive part of the x-axis. For example: **5<45**
- **Relative:** $@dx<angle$, where dx is the displacement value in the direction of the angle and $wangle$ is the clockwise angle from the positive part of the x-axis. For example: **@5<45**

It is important to note that in LibreCAD the units for the coordinate values are set to millimeters by default. However, users can select other units such as inches, feet, meters, etc. by changing the settings for the drawing.

In summary, LibreCAD supports various formats for coordinate input, including Cartesian, polar and relative coordinates. Users can select the format that best suits their requirements and their specific drawing.



Below are some examples of Cartesian and polar coordinate input in LibreCAD, both absolute and relative:

Cartesian coordinates:

- Absolute: To place a point at (5,3), enter "5,3" in the command line.
- Relative: To place a point 5 units to the right and 3 units up from the current point, enter "@5,3" in the command line.

Polar coordinates:

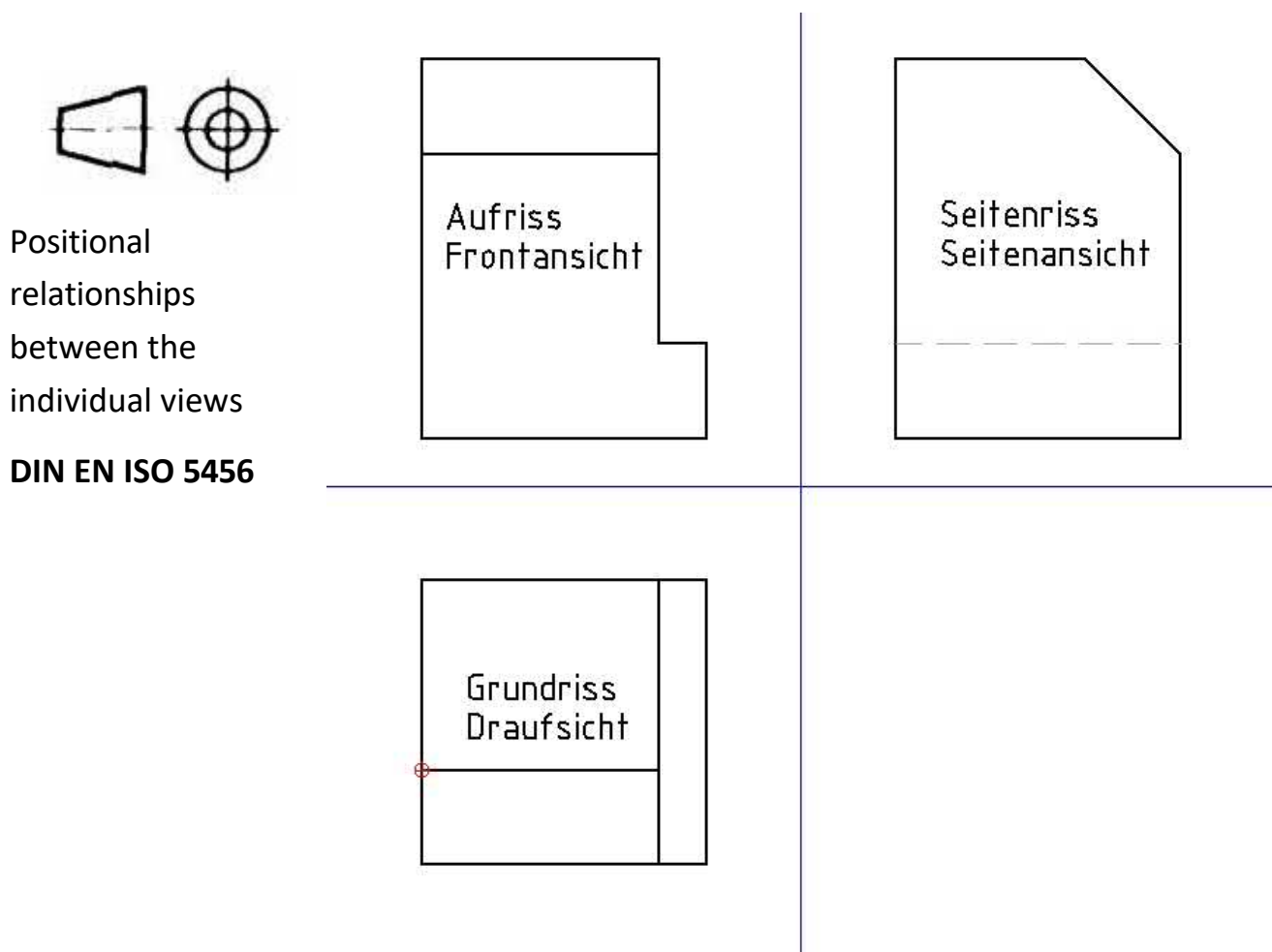
- Absolute: To place a point at (3<45) (i.e. 3 units from the origin and 45 degrees from the positive x-axis), enter "3<45" in the command line.
- Relative: To place a point 3 units away from the current point and 45 degrees clockwise from the positive x-axis, enter "@3<45" in the command line.

*Tip: If it is not possible to make the entry in the command line, you can use the **space bar** to set the input focus to these.*

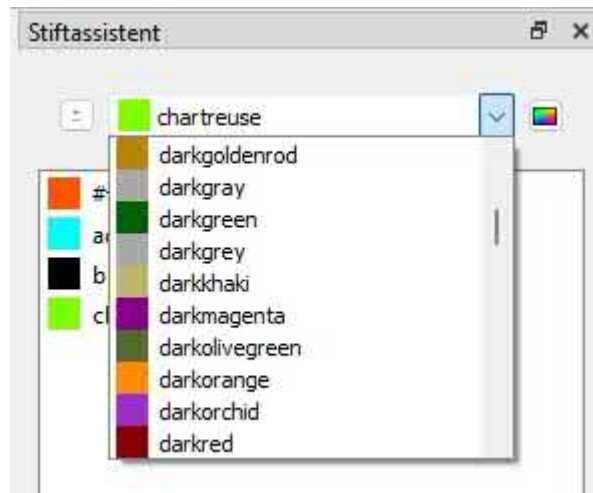
8.5 Views side view, elevation, floor plan

In 2D drawings, there are different views to represent an object from different perspectives. The most common views are

1. Floor plan (also known as plan view): This view shows the object from above, as if you were looking directly at it. It is often used to show the horizontal extent and arrangement of the elements of an object.
2. Elevation view (also known as front view): This view shows the object from the front, as if you were looking directly at it. It is often used to show the vertical extent and shape of the object.
3. Side view (side view or also referred to as profile view): This view shows the object from the side. It is used to show the depth and lateral extent of the object. There can be a left and a right side view, depending on which side the object is viewed from.



9 Pen assistant



As with many other aspects of drawing, the color, thickness and type of a line or circle are determined by drawing conventions or general practices. In LibreCAD, the three attributes are summarized under the term "pen":

- **Color** - LibreCAD has 16 standard colors, but supports the RGB color space (#000000 to #FFFFFF or 16,777,215 colors). The default color for objects is black.
- **Width** - The standard 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 in LibreCAD are "Point", "Stroke", "Split", "Center" and "Border".

The pen attributes can be defined for an individual element (using the "Properties" tool), for a group of selected elements (using the "Attribute" tool) or for a layer.

Note: As with objects, pens can also be applied to layers. For more information on setting the attributes of a layer, see Dock of the layer list or Layer list.

Perhaps someone can contribute more to the pen assistant?

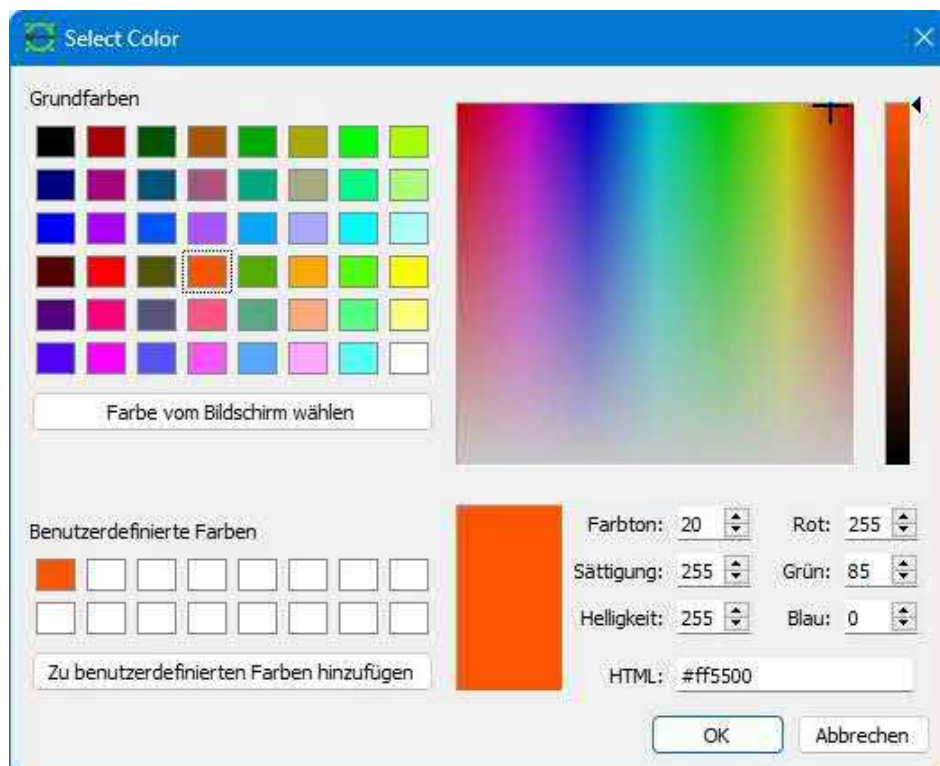
10 Color

The color for an element can be selected via the "Color" drop-down menu. The drop-down menu allows you to select the color "By layer", "By block", from the "Custom" color picker or quickly select from one of the 16 predefined colors:

If "By layer" is selected, the element is assigned the color that was defined for the layer (see above). If the selected color of the layer is subsequently changed, the color of the layer is assigned to all objects on the layer.

When editing a block, the color defined for the block is assigned to the added object if you select "By block". If the color of the block is subsequently changed, the color of the block is assigned to all elements of the block. If you select "Custom", you can choose from a palette of 36 colors and shades of grey or from colors defined by the user.

Custom colors are created by clicking the Add button and then selecting the hue and value with the color picker tool. Custom colors can be changed by right-clicking on a custom color and selecting a new hue and value. Custom colors can also be added.



10.1 Line widths

The standard line thickness of LibreCAD is 0.00 mm (hairline) and ranges up to 2.11 mm. The range includes lines with ISO standard width. The line widths vary depending on the intended use (outlines, hidden lines, sections, etc.) and the size of the drawing; thicker lines are used for larger drawings. In addition to the line widths, four further options are displayed in the drop-down list:

- **Unchanged:** Leaves the line width as previously selected.
- **By layer:** Adopts the line width set in the attributes of the layer.
- **After block:** Adopts the line width that was used when the block was created.
- **Default:** Resets the line width to the default value (0.00 mm). You can find examples of line thicknesses depending on the intended use under General line thicknesses in the appendix.

It seems that the representation of line widths in LibreCAD was a problem in the past, especially when it came to printing. In previous versions of LibreCAD, there were sometimes difficulties in displaying line widths accurately, especially when printing a project.

However, like many open source software projects, LibreCAD has an active developer community that is constantly working on improvements. It is therefore possible that these problems have been fixed in the meantime or that there are workarounds to circumvent them.

If you have specific issues with line width rendering in LibreCAD, I would recommend checking the latest version and making sure you have the latest updates installed. In many cases, a look at the official documentation or the user forum can also be helpful to find possible solutions or to get advice from other users who have had similar problems.

Note: Depending on the graphics card and resolution, line widths are sometimes displayed too thin or too wide. It is therefore important to test the settings and adjust them in 0.05 mm increments if necessary.

11 Layers

One of the most important functions of LibreCAD is the use of layers. Layers (also called layers in other programs) help with the organization of drawings by allowing the user to place and manage related elements. Traditional manual drawing used a similar approach. Layers were often drawn on separate transparent sheets. These sheets were then placed on top of each other to create the final drawing.

Whether in engineering, architecture, construction, manufacturing or other fields, layers have been used to represent various aspects of the drawing. Layers can be added to show centerlines or dimensions on engineering or manufacturing drawings, or to show various building systems on architectural drawings, such as exterior walls, partitions, electrical, HVAC, grid lines, etc. While a layer can contain multiple objects, each object in a drawing can only be associated with a single layer. Typically, objects with common functions or attributes are placed on the same layer. For example, all walls in a floor plan drawing would be placed on a layer named "Walls".

Layers have the additional advantage that all pen attributes can be assigned to a layer. Each element on this layer takes on the attributes that have been assigned to this layer. However, the attributes assigned by the layer can be overwritten for elements if required. In the example above, a line thickness can be applied to all elements on the "Walls" layer by changing the "Layer settings" for this layer.

Layers are an indispensable element in the CAD design process. Layers in DXF and DWG files are fully compatible with other CAD programs.

11.1 Create level

Layers are usually created to store elements with common attributes. Creating a layer is simple:

- Click on the Add layer icon
- Enter a layer name
- Optionally set the color, width and line type for the layer
- Click on Ok.



11.1.1 Design level

A **design level** is used to record geometric auxiliary construction lines:

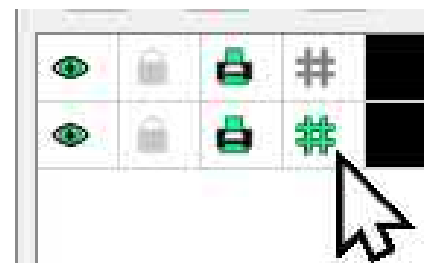
- A draft level **does not** appear on the printout.
- All lines of a design level are **unlimited in length**.
You can switch between draft and normal mode in **three** ways:

1. Click on the checkbox when creating or changing a layer

Design layer in the Layer settings dialog box.

2. Right-click on a named layer in the layer list and select **Switch construction layer**.

3. Click on the "#" symbol in the layer list →



12 Functions Brief overview

LibreCAD offers a wide range of functions that make it a powerful yet user-friendly tool for 2D CAD drawings:

12.1 Vector-based drawings

LibreCAD works with vector graphics consisting of geometric shapes such as lines, circles, arcs and curves. This makes it possible to create precise and scalable drawings that can be printed or exported in various sizes and resolutions without any loss of quality.

12.2 Intuitive user interface

The LibreCAD user interface is simple and clearly laid out, with a menu bar, toolbars, a drawing area and a status bar. This makes it easy to get started with the program, even for beginners in the CAD field.

12.3 Various drawing tools

LibreCAD offers an extensive range of drawing tools with which you can create lines, circles, ellipses, polygons, splines and other geometric shapes. You also have commands for copying, moving, rotating, mirroring and scaling objects.

12.4 Layers

LibreCAD supports layers to help you organize and manage your drawing. Layers can be shown and hidden, locked and renamed to simplify the drawing process and improve clarity.

12.5 Catch functions, snapping (snap-in functions)

LibreCAD has various snapping functions that help you to create precise drawings. These include end point, center point and intersection as well as a grid and orthogonal drawing.

12.6 Support for various file formats

LibreCAD uses the DXF format (Drawing Exchange Format) by default, which is compatible with other CAD programs. You can also export your drawings in other formats, such as SVG, PDF or PNG.

12.7 Customizable user interface

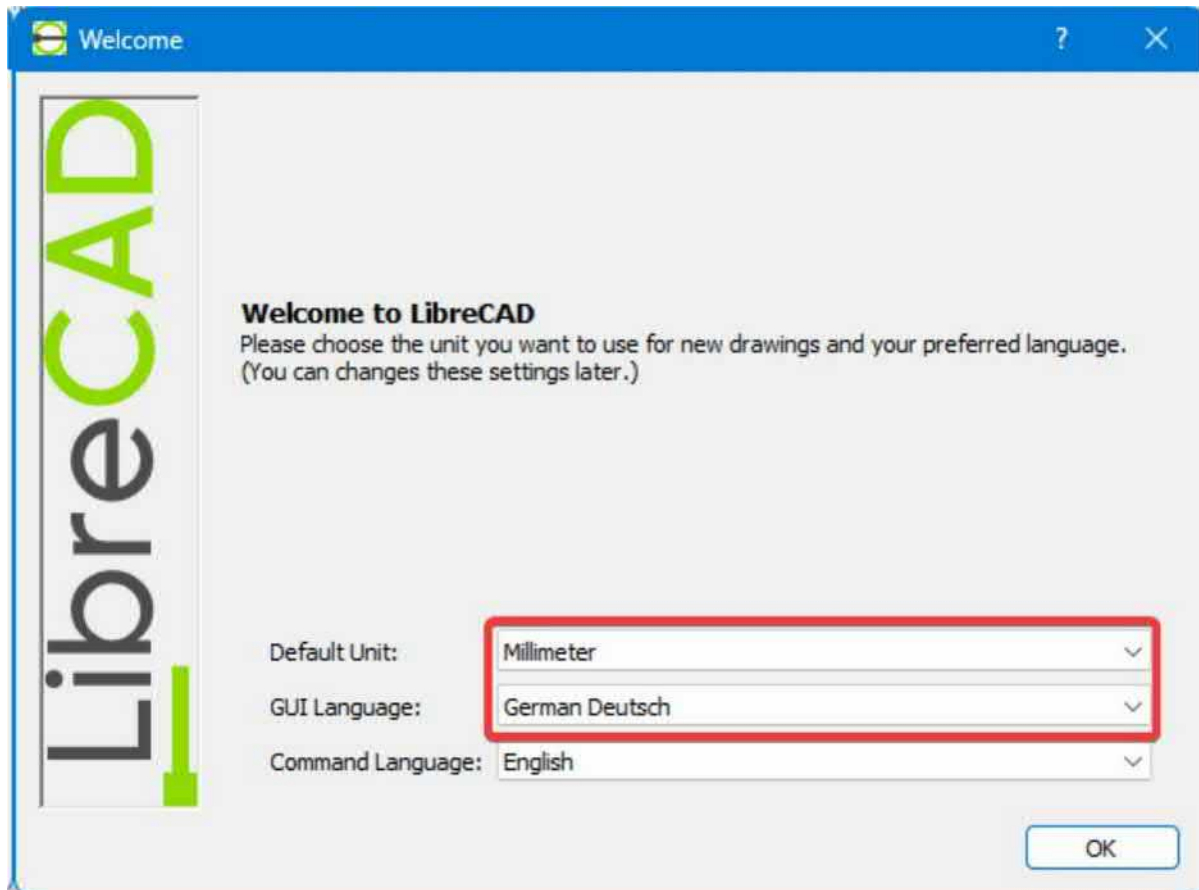
LibreCAD allows you to customize the user interface to your needs by adding, removing or rearranging toolbars and defining keyboard shortcuts for frequently used commands.

13 Installation

Download the appropriate setup file for your operating system from:

<https://www.librecad.org>

Execute the installation file:



Settings:

Default Unit: **Millimeter**

GUI Language: **German German**

Command Language: **English**

Note: In some Windows configurations, the Command Language: English is not set to English but to German.

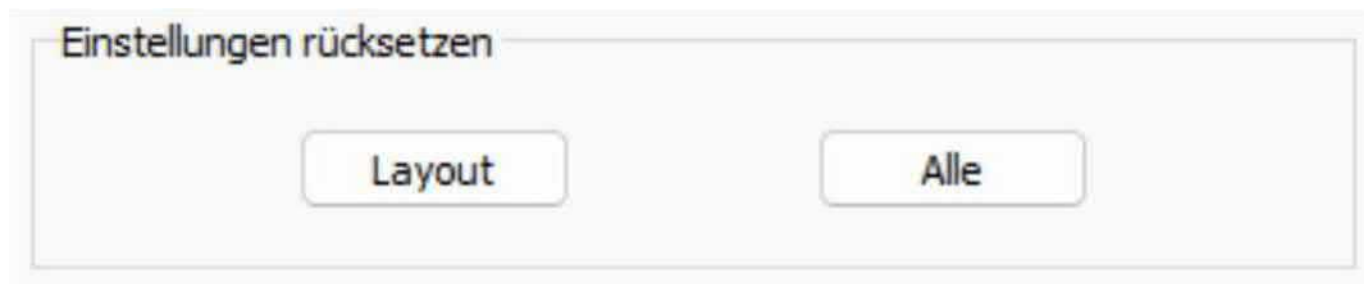
The German commands are described in Chapter 22 Drawing tools and command line.

14 Reset LibreCAD

To ensure that LibreCAD works optimally with this book, we recommend that you reset the software to its default values if you have already made changes or adjustments. To do this, you can perform the following steps:

Click on in the menu:

Options -> Application settings -> Tab: Standards



Layout: Resets the layout of the application window to the default configuration.

All: Resets the application to the default configuration. Window layout, color settings, user-defined menus and toolbars, etc. are all reset. The "Welcome" dialog box is displayed the next time the application is started.

It is important to note that the "**Reset settings**" function only affects the settings and adjustments of the program itself and has no effect on the drawings or files created by the user.

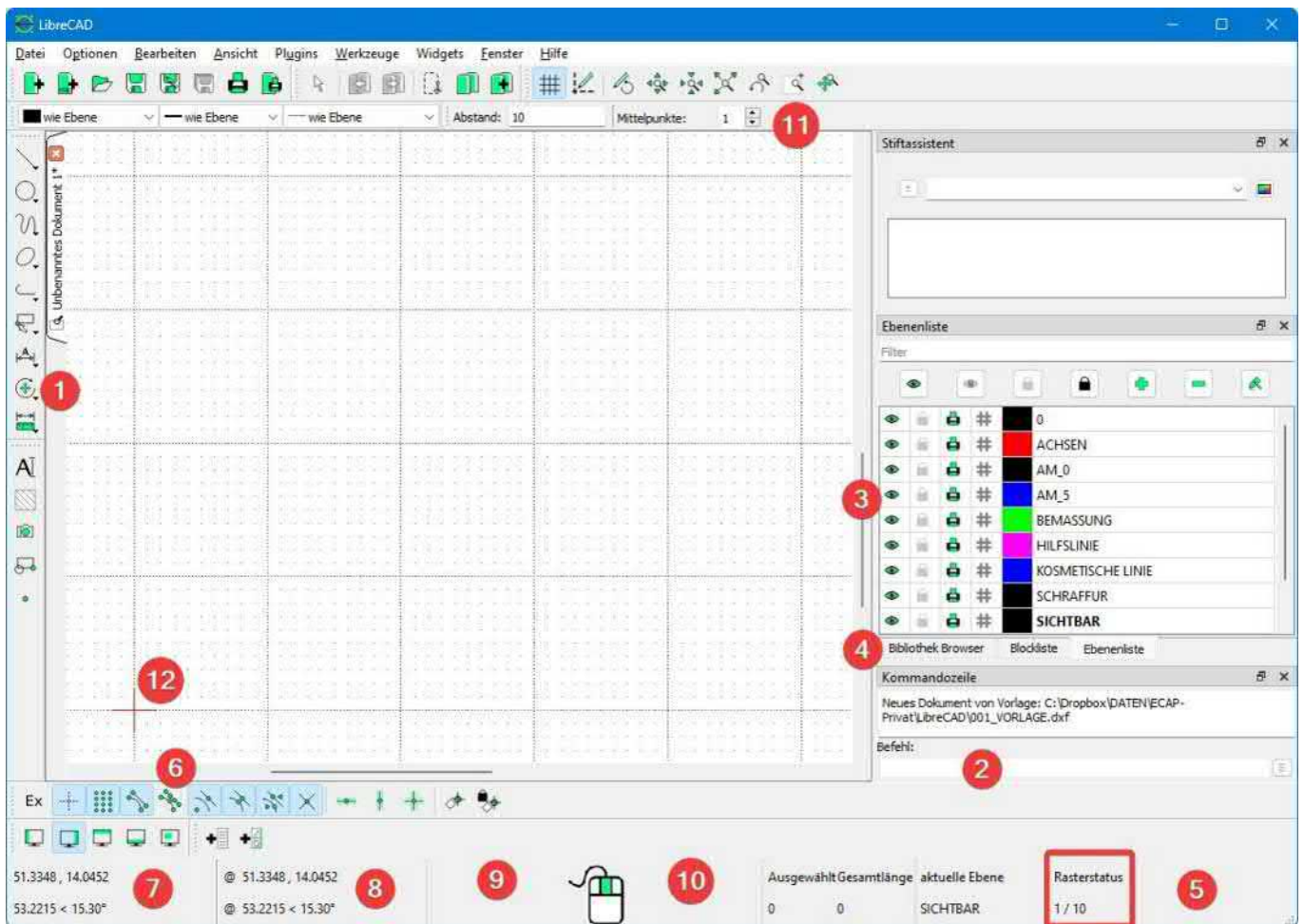
In summary, the "**Reset settings**" function in LibreCAD allows the user to restore the program's default settings by deleting all user-defined settings and customizations.

This is very useful if the user has made changes that they want to undo or if they want to reset the program to the original default settings.

After you have completed these steps, LibreCAD will be reset to its default values and should work optimally with this book.

15 User interface Quick overview

The LibreCAD user interface is clearly laid out and offers quick access to the most important functions.



LibreCAD user interface when reinstalling or resetting all settings.

- (1) CAD functions.
- (2) Command line, activate with [**Ctrl+M**] or [**Ctrl+M**].
- (3) Layer list (Layer) Show, hide, line types.
- (4) Switch between library browser, block list, layer list.
- (5) Grid status, dynamic background grid. *
- (6) Snap functions (especially the grid snap is well looked at in the book!).
- (7) See chapter 15.1 Coordinate display.
- (8) Relative coordinates in relation to the last point clicked.
- (9) Action for the left mouse button (LM) → Section 15.2 Next action with the mouse
- (10) Action for the right mouse button (RM) (usually canceling a function).
- (11) Upper status bar with entries corresponding to the function.
- (12) ZERO POINT of the drawing.

* The dynamic background grid in LibreCAD is a tool that helps you to create precise drawings. It is a grid of points that is displayed in the background of your drawing and serves as a reference point for positioning objects exactly.

Adjustable spacing: You can adjust the spacing between the grid points or lines to adapt the grid to your specific requirements. This can be done either globally for the entire drawing or locally for a specific area.

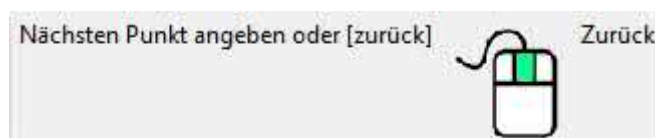
Dynamic display: The grid is displayed dynamically, i.e. it adjusts automatically when you zoom in or out in the drawing. This allows you to use the grid at different scales without having to adjust it manually each time.

15.1 Coordinate display



Displays the absolute coordinates (left) and the relative coordinates (right) of the crosshairs/cursor. The Cartesian coordinates are displayed at the top and the polar coordinates at the bottom. For more information on the type of coordinates, see **Coordinate systems**.

15.2 Next action with the mouse



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 symbol represent actions with the left mouse button (**LM**), those to the right represent actions with the right mouse button (**RM**).

Independence from the drawing: The dynamic background grid is not part of the actual drawing, but is only displayed as a visual cue. This means that it is not printed or exported and does not need to be shared with other users who may be editing the drawing. The dynamic background grid in LibreCAD is a useful tool to improve the accuracy and precision of your drawings and make the drawing process easier overall.

15.3 Zoom in, zoom out and pan-zoom

In LibreCAD you can enlarge, reduce and move the view of your drawing using the zoom and pan-zoom functions.

Zoom: The zoom function allows you to enlarge or reduce a specific area of your drawing. **Use the mouse wheel to zoom in and out.** To zoom in or out on a specific area, turn the mouse wheel. The zoom is cursor-centric.

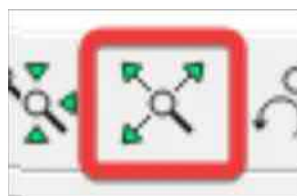
Pan-zoom: The pan-zoom function allows you to move the view of your drawing by holding down the mouse wheel and moving the mouse.

You can also combine the zoom and pan-zoom functions to quickly and easily navigate through your drawing. For example, you can zoom out to get an overview of the entire drawing and then use the pan-zoom function to navigate to a specific area that you want to take a closer look at. You can then zoom back in to examine the details of that area.

The zoom and pan functions can also be called up manually in the menu bar:

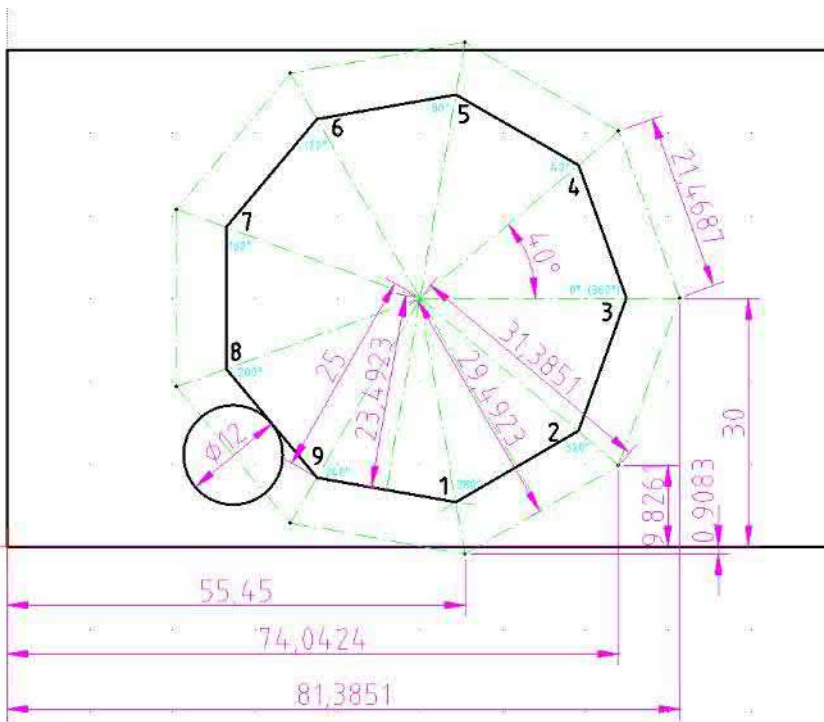
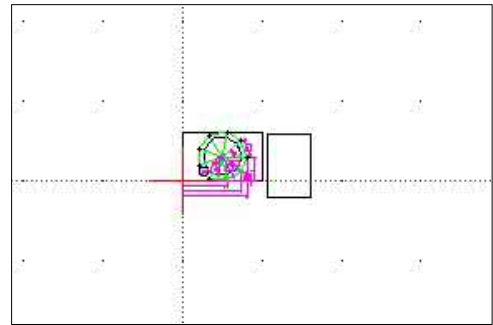


Auto zoom is a frequently used function for large drawings:



With this function, all drawing objects are optimally zoomed in on the screen...

If a drawing is displayed very small, simply click on the Autozoom function:



Zeitberechnung mit Fräser D12

Eine Seitenlänge berechnen:

Dazu muss ein Dreieck halbiert und die Länge der Ankathete (b) ermittelt werden:

$$b = c * \cos(\alpha) \rightarrow 25 * \cos(20) = 23.4923 \text{ mm}$$

Dann den Radius des Fräasers (6 mm) dazurechnen, ergibt: 29.4923 mm

Nun kann man die Gk.(a) berechnen:

$$a = b * \tan(\alpha) \rightarrow 29.4923 * \tan(20) = 10.7342 \text{ mm}$$

Hier noch die Hyp:

$$c = a / \cos(\beta) \rightarrow 10.7342 / \cos(70) = 31.385 \text{ mm}$$

Eine Seitenlänge fräsen ist: $2 * 10.7342 = 21.468 \text{ mm}$

Alle 9 Seiten = 193.2156 mm



"Zoom window" and "Last view" are two other useful functions in a CAD program that can help to control and optimize the view of the drawing.

With the "Zoom window" function, you can enlarge a specific area of the drawing by dragging a window around the desired area.

The "Last view" function allows you to quickly and easily return to the previous view of the drawing.

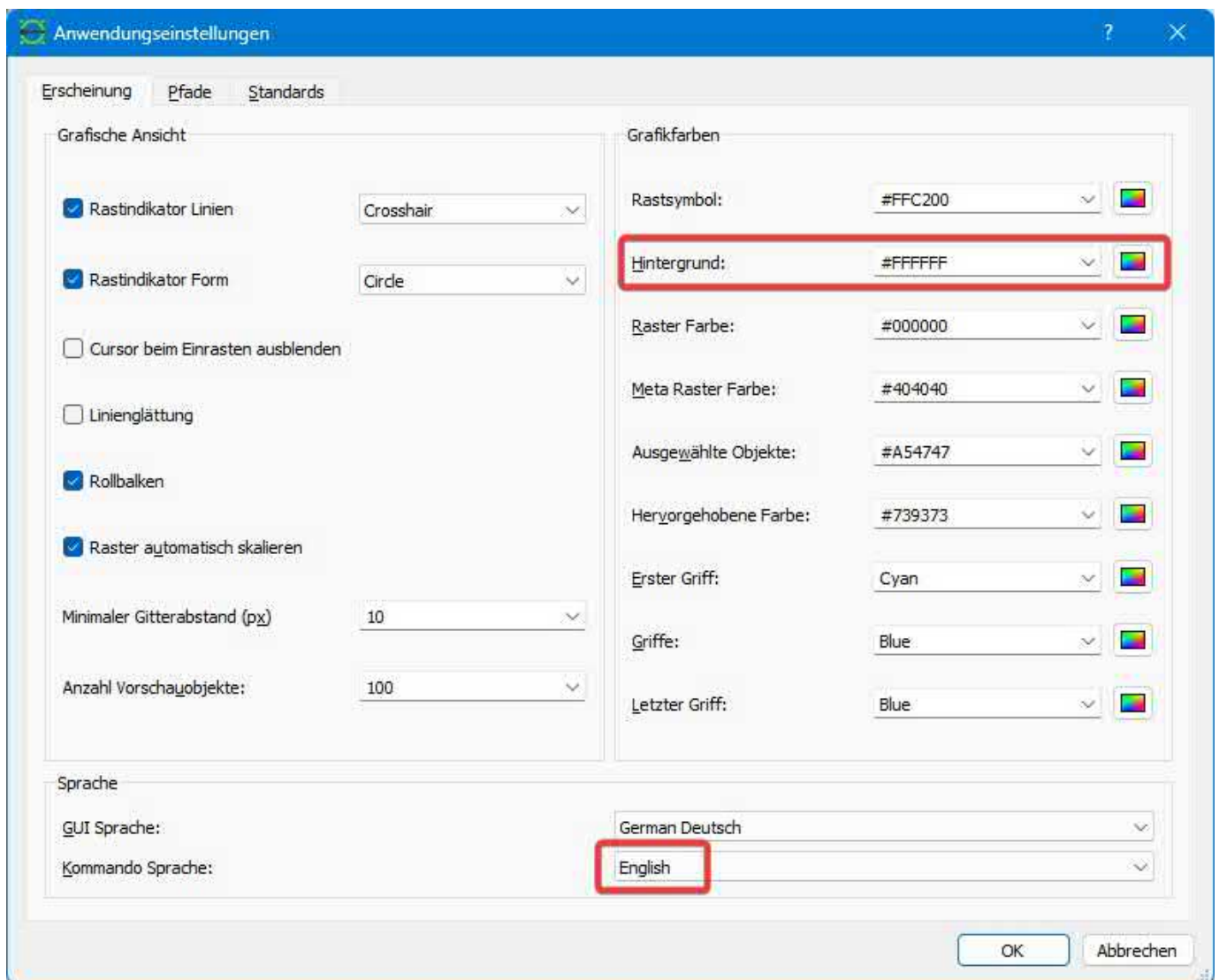
As a rule, however, most zoom functions are carried out directly with the mouse.

16 Basic settings

The following basic settings are also important for progressing through the book and are well suited for the mechanical area.

16.1 Program settings

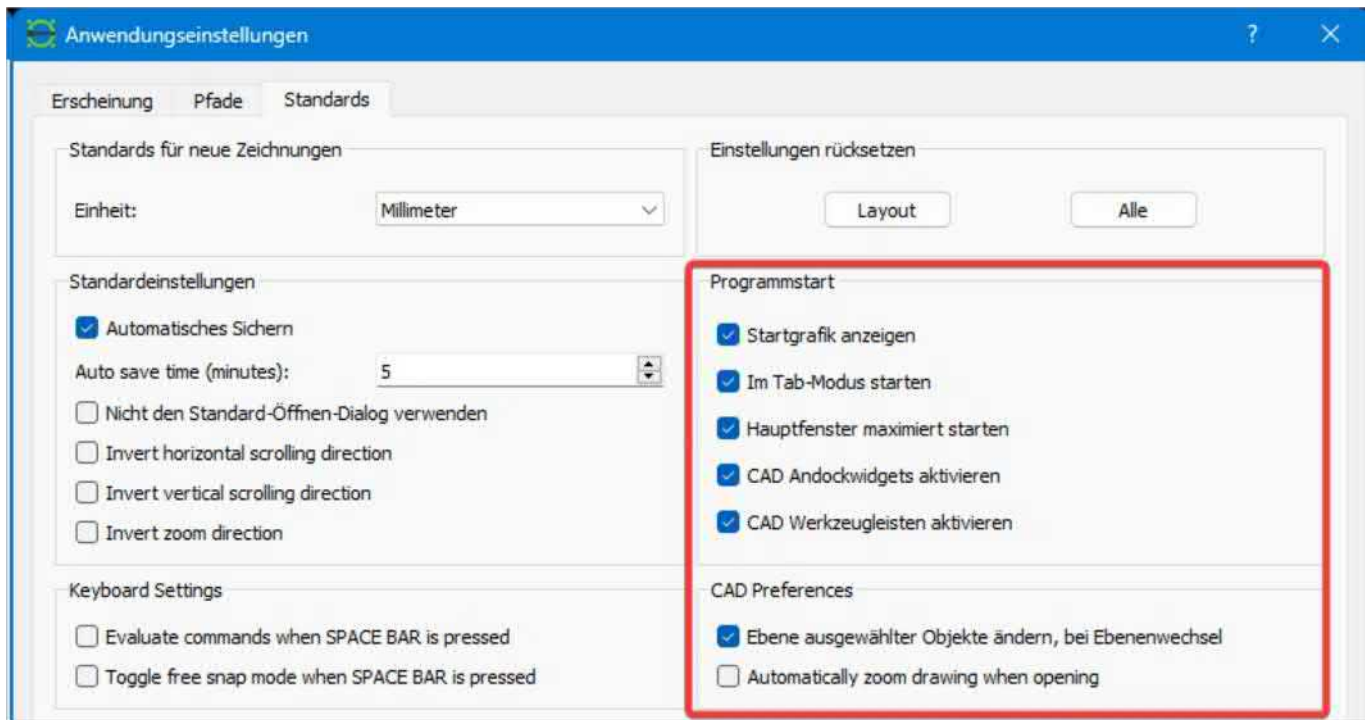
The following settings are made in the **Appearance** tab via **Options** -> **Application settings**:



The background color is usually white or black.

A white background is used in the book because it is more pleasant to read on a sheet of paper. In reality, the screen is often drawn against a **black background**.

In the **Standards** tab

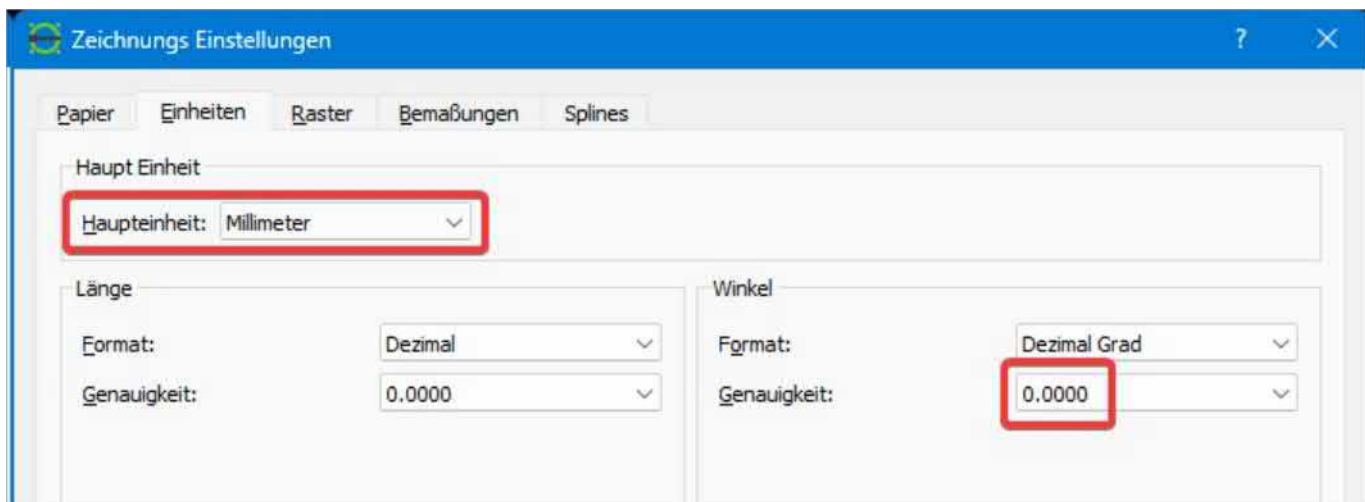


16.2 Drawing settings

Via **Options** -> **Drawing settings**

16.2.1 Register: **Units and decimal places**

When it comes to the accuracy of lengths, three to four decimal places are OK...

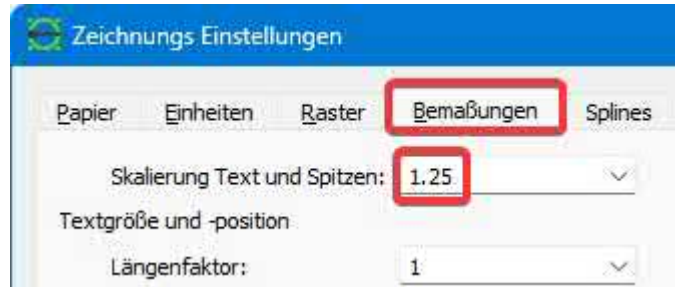


Main unit: **Millimeter**

Accuracy: **0.0000**

16.2.2 Register: Dimensions

The dimensioning is slightly too large in its original state. We adjust this by a factor of **1 to 1.25** instead of 2.5.

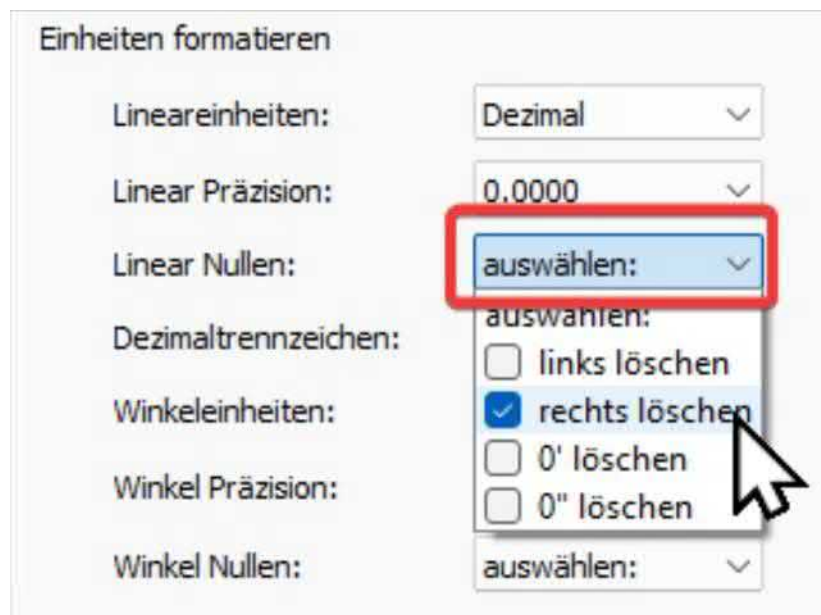


This is still easy to read and takes up less space on the sheet of paper.

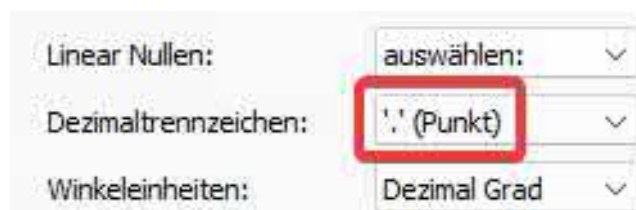
Setting the decimal places for linear dimensioning:

Linear precision: 0.0000

Linear zeros: Delete right



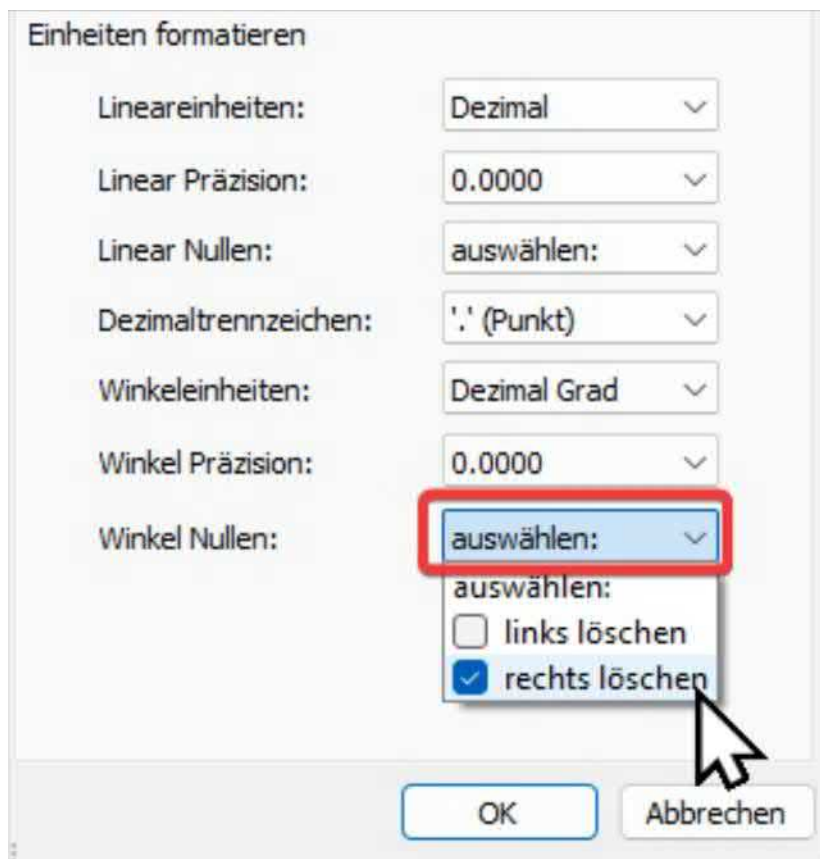
Decimal separator: "."



Setting the decimal places for angle dimensioning:

Angle precision: 0.0000

Angle zeros: delete right



Accept with **OK**.

The setting "**Delete zeros to the right**" for linear and angular dimensioning in LibreCAD refers to how many zeros should be displayed after the decimal point. If this setting is activated, all leading zeros after the decimal point are deleted.

Decimal point deleted. If it is deactivated, all zeros are displayed after the decimal point or the degree sign until the selected accuracy is reached.

For example, if you have a line with a length of 12.5000 mm and the accuracy is set to two decimal places:

- If "Delete zeros on right" is activated, the length is displayed as "12.5 mm".
- If "Delete zeros on right" is deactivated, it is displayed as "12.5000 mm".

This setting can be useful to improve the readability of masses and to remove unnecessary zeros, especially if there are many masses in a drawing.

17 Further settings

The most important thing when working with 2D CAD software is the layers. Without a layer structure, it is not possible to work with it properly. Layers can also be shown and hidden globally and switched on or off for printing on paper.

17.1 Layers and line types

In mechanical engineering, especially in 2D CAD, certain standards for line types, layers and colors have been established to ensure a uniform and clear representation of drawings. Although standards may vary by company, industry or country, there are some generally accepted guidelines. Here are some of the most common:

1. **Line types:** The line types can vary depending on the element in the drawing. Here are some commonly used ones:
 - **Extracted:** For visible edges, outlines and dimensions.
 - **Dashed:** For concealed edges or invisible lines.
 - **Semicolon:** For axes, intersecting lines or center lines.
2. **Layers:** Layers help to organize the drawing and show or hide certain elements. Here are some frequently used ones:
 - **0:** Standard layer, often for invisible elements or placeholders.
 - **NOTE:** For texts, dimensions and tolerances.
 - **DIMENSIONING:** For dimension lines and auxiliary lines.
 - **VISIBLE:** For visible and concealed geometry.
 - **INVISIBLE:** For concealed lines
 - **AXES:** For center lines and symmetry lines.
 - etc...

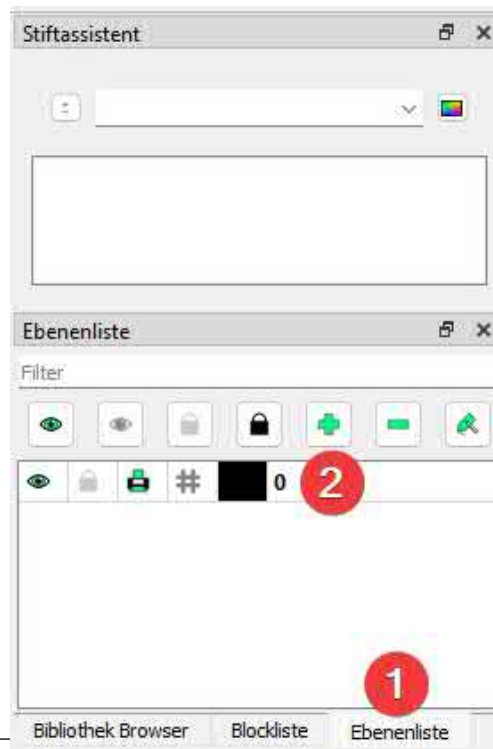
3. **Colors:** The following colors are generally used in mechanical engineering.

The color black is used for visible lines, dimensions and text, while white is the background.

However, other colors can be used for special purposes, e.g. magenta for corrections or revisions. Blue for cosmetic elements such as the thin thread line for threaded holes...

It is important to note that these standards may vary depending on the company or project. Therefore, it is always advisable to adhere to the specific guidelines that apply to your project or organization.

To define the lines, activate the **Layer list** tab (1) in the right-hand panel and click on the **[+]** button (2).



We agree on the following definitions:

Layer name	Color	Width	Line type
AXES	Red	0.13mm	Centerline (small)
DIMENSIONING	Green	0.13mm	continuous
WELFARE LINES	Magenta	0.00mm	continuous
COSMETIC LINE	Blue	0.13mm	continuous
HATCHING	Black	0.05mm	continuous
VISIBLE	Black	0.25mm	continuous
INVISIBLE	Gray	0.18mm	dashed (small)
WHITE	White (99.9%)	0.00mm	continuous

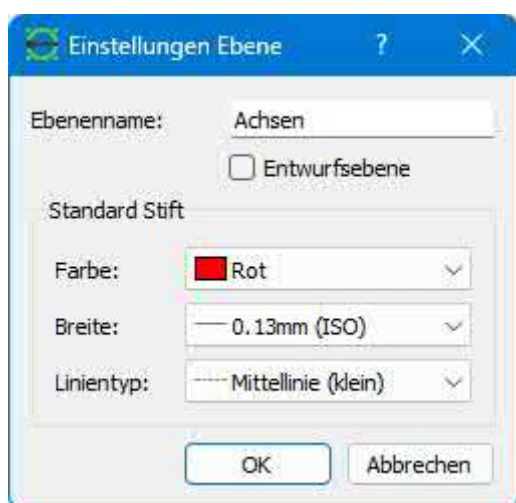


Chart 4: Axes

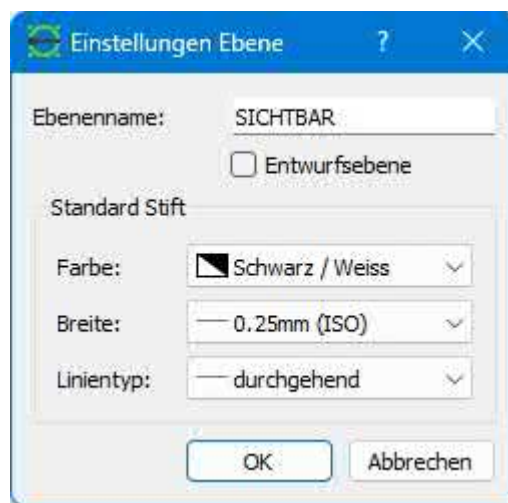


Figure 3: Visible lines

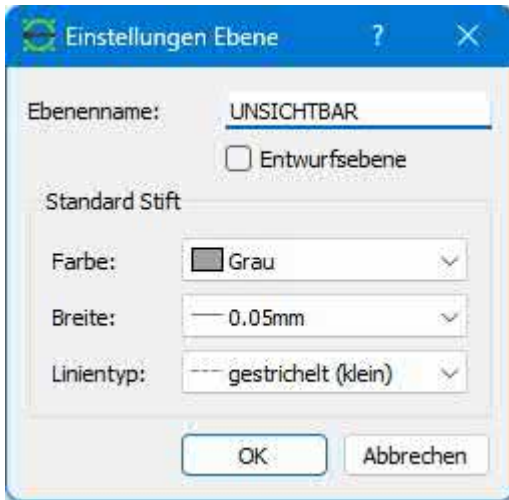


Figure 5: Invisible edges / lines

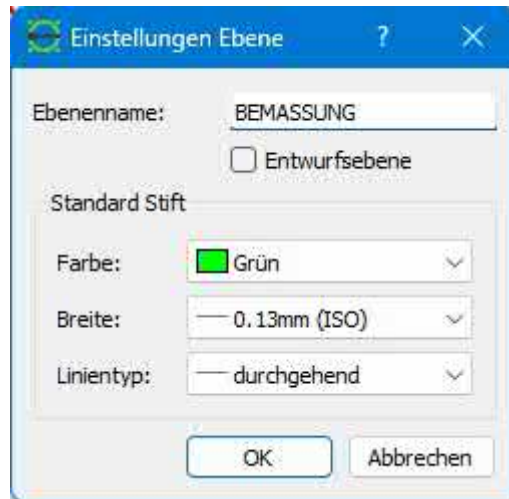


Diagram 6: Dimensions

Cosmetic lines can be used, for example, when drawing threads or tangential transitions. Auxiliary lines are simply auxiliary lines :-)

The color WHITE is a SPECIAL CASE.

To do this, quickly read chapter 41.6 Add WHITE layer to layer list The settings should look like this:



These settings guarantee us a consistent CAD image that we can use as a guide in this tutorial.

17.2 Level 0 (zero)

A "level" is the same as a "layer".

The "0" layer has no special meaning. It is the default layer for drawings that are not based on a template. In the context of blocks, it is equivalent to "no layer", similar to how color "By block" is equivalent to "no specific color" or linetype "By block" is equivalent to "no specific linetype".

In general, the "0" layer 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, the pen attributes (color, width, line type) also include "By layer" and "By block".

- Blocks with certain attributes (e.g. color set to blue, width set to 0.18 mm, etc.) retain these attributes when inserted into a drawing. To change one of the attributes, the block must be edited.
- Blocks whose attributes are set to "By layer" adopt the attributes of the layer into which they are inserted. The attributes of the block change when the attributes of the layer are changed.
- Blocks whose attributes are set to "By block" initially adopt the attributes assigned to the layer. The attributes can be changed using the Attribute tool.

So it's worth mentioning again, because it's important:

According to information from LibreCAD, the layer "0" may generally only be used when creating blocks and should be the only layer in a drawing for a block.

Are there people who can say more about this and have a concrete example?

17.3 Possible problems when printing line widths

The problem with outputting technical drawings with different line widths on different printers is that the actual line widths on the paper print may differ from the values specified in the drawing. This can lead to inconsistencies and ambiguities in the drawing, especially if the line widths are used to differentiate between different elements or to represent hierarchies in the drawing.

The causes of these deviations are manifold and can be due to various factors, such as

1. Printer resolution: The resolution of the printer can influence the accuracy of the line widths. A lower resolution can result in lines being printed wider or thinner than they should be.
2. Printer settings: Various printer settings, such as print quality, color mode or paper type, can affect the display of line widths.
3. Drivers and software: The printer drivers and software used can influence the line widths by interpreting and processing the drawing data in different ways.
4. Calibration: Incorrect or inaccurate calibration of the printer can lead to deviations in the line widths.

To minimize these problems and ensure consistent representation of line widths in technical drawings, the following measures can be taken:

1. Use of standardized line widths: The use of standardized line widths, e.g. according to DIN 1356 or ISO 128-23, ensures that the line widths in the drawing are uniform and clearly defined.
2. Printer calibration: Regular calibration of the printer ensures that it reproduces the line widths correctly. Special calibration tools or test prints can be used for this.
- 3. And readjust the line thicknesses in all templates!**

By implementing these measures, deviations in the line widths of technical drawings can be reduced and consistent presentation on different printers can be guaranteed.

18 Save settings as a template

So-called templates offer several advantages in a CAD program:

Standardization: Templates can be used to ensure that all drawings in a project or company are uniform. You can define standards for fonts, layer structures, line types, units of measurement and much more.

Time saving: Instead of having to start from scratch with every new drawing, you can start with a template that already contains basic settings and elements. This can include, for example, a ready-made layout, a title block or frequently used objects.

Reduction of errors: By using a template, errors can be reduced that can occur due to manual input of repeated information. This leads to greater accuracy in the drawings.

Efficiency: Templates can also contain certain settings that optimize the performance of the CAD program. This can increase the working speed and improve efficiency.

Professional look: A well-designed template can help your drawings make a professional impression. This is especially important if you are sharing drawings with clients, contractors or other external parties.

Under **File** → **Save as**.

Let's save the file with the name
001_VORLAGE.DXF.

Every time you create a new drawing, you could select **File** -> **New from Template** → **001_VORLAGE.DXF...**



... or it is better to create a so-called **standard template**, as described in the following chapter.

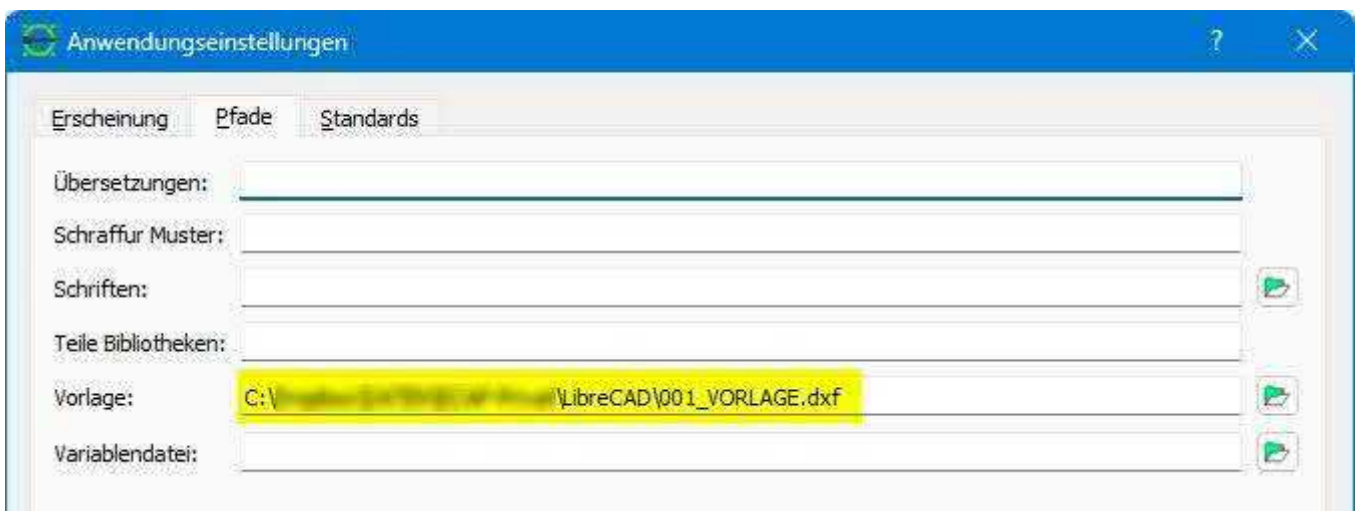
18.1 Standard template

Simply put, templates are nothing more than a simple drawing file that contains, in particular, various settings and components that can be used repeatedly. These settings can be as simple as the paper definition and unit specifications, but can also include layers, block devices and drawing elements like any other drawing.

However, these settings are only limited to the drawing itself. Templates cannot change application preferences or other global settings in LibreCAD.

LibreCAD contains a standard template, "**empty.dxf**", which is installed with the application and is located in the resource directory. It is initially used as the default template for all new drawings. A custom template can be used instead of the default template by specifying the new template in the application preferences.

In the menu, select **Options** → **Application settings** and click on the "Paths" tab. Enter the path and the file name (...**001_VORLAGE.DXF**) of the **template** to be used as the default template. The specified template is used instead of the LibreCAD default template when the application is started and for new drawings:



The download link for the standard template used in the book can be found in the chapter 57.4 Standard parts, standard template and more

Tip: We recommend that you store the standard template and the standard parts library in a central location that can be accessed from all computers if you are working with several computers.

19 Hotkeys / short keys

The Ctrl **key** corresponds to the Ctrl **key** on the German layout.

A + Enter	Bow
Pos 1 / Home	Sequence: Move objects to the top
Image up / PgUp	Sequence: Move objects one to the top
Picture down / Pg Dn	Sequence: Move objects one down
End / End	Sequence: Move objects all the way to the bottom
Ctrl + A	Select all
Ctrl + C	Copy to clipboard
Ctrl + D	Switches to or from draft mode. In draft mode, hatchings are invisible and only bounding boxes for images and text are displayed. Drawings are displayed faster, especially on slow computers and good for large drawings.
Ctrl + G	Show or hide grid
Ctrl + I	Show/hide status bar
Ctrl + K	Deselect all selected
Ctrl + O	Open file
Ctrl + P	Print
Ctrl + R	Reverse direction
Ctrl + V	Paste from clipboard
Ctrl + Z	Undo
Ctrl + Y	Redo
Del / Del	Delete selected objects
Ctrl + S	Save
Ctrl + P	Print
F5	Redraw
F11	Enlarge to full screen

For more information, see chapter 22 Drawing tools and command line

20 Drop-down or floating menus?

Most LibreCAD commands are accessible via the menu bar.

Floating menus are menus that are not fixed to a specific position in the user interface, but float freely and can be placed anywhere on the screen. The advantage of floating menus is that they are flexible and can be adapted to any working environment. For example, they can be dragged to the edge of the screen to create more space for the drawing area or placed near the workspace to allow quick access to frequently used commands.

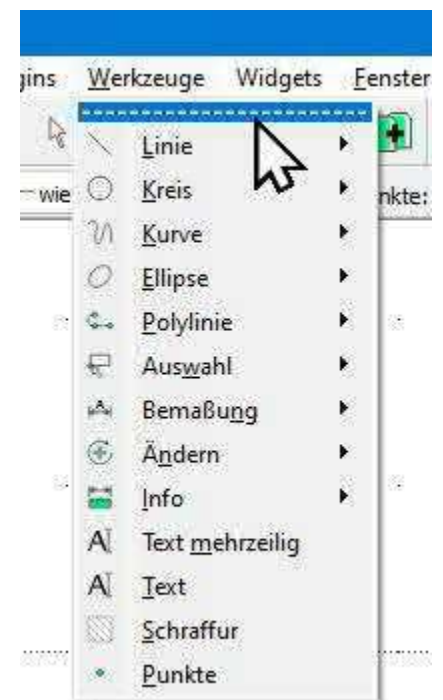
Drop-down menus are menus that hang down from the menu bar at the top of the window. They are normally called up by clicking on a menu button in the menu bar. Drop-down menus usually contain a wide range of commands and functions that are relevant for the entire application.

Drop-down menus provide access to file and print operations, application and drawing configurations, drawing and editing tools and other functions.

The drop-down menus can be converted into a floating menu.

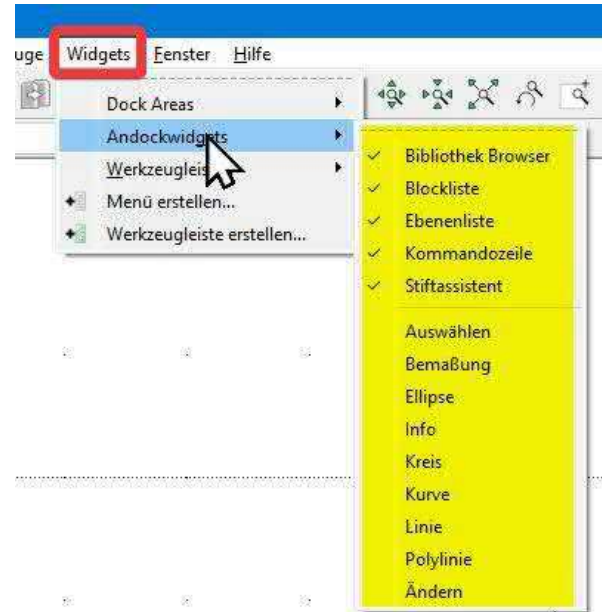
Each drop-down menu has a **dashed line at the top**. If you click on the dashed line and drag it, you can "tear off" the menu and move it to another location on the screen. postpone.

If you click on the item in the top right-hand corner of the floating menu, it will close.



21 What are widgets?

In LibreCAD, widgets are graphical user interface elements that provide certain functions and tools of the program. These widgets can be displayed in the **Widgets** → **Docking widgets** menu and provide a quick and easy way to access frequently used commands and functions.



Widgets can be customized and configured in LibreCAD to optimize your workflow. You can add or remove widgets, change their size and position and even create custom widgets.

To add or remove a widget in LibreCAD, right-click on an empty space in the toolbar or in the menu and select "**Docking widgets**" from the context menu. A dialog box appears in which you can select the available widgets and add or remove them.

Docking widgets provide a quick and easy way to access frequently used commands and functions in LibreCAD. By customizing and configuring widgets, you can optimize your workflow and draw and edit more efficiently.

Info: As everyone can position their own widgets as they wish, it would be complicated to explain where to find the functions in the rest of this book. **Therefore, this book refers to working WITHOUT widgets.**

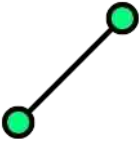
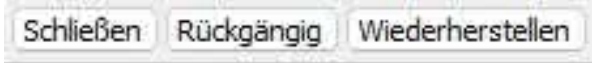
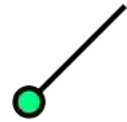
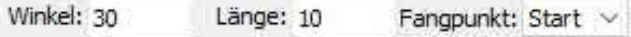


22 Drawing tools and command line


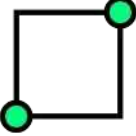
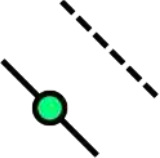





The drawing tools are used to create and change elements such as lines, circles etc. in a drawing. Commands can be selected from the menu or toolbars or entered via the command line. Right-click anywhere in the drawing window to bring up a pop-up menu that displays the history of commands called from the menus or toolbars, with the most recent command at the top of the list. **Note that the command line input is not displayed.**


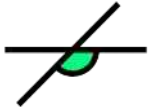

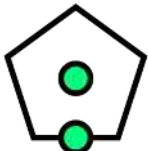
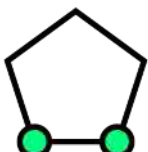
Some drawing tools require additional parameters, which are displayed in the **upper status bar** or toolbar as "**Tool options**". This toolbar should always be activated. If the tool options are not displayed, select **Widgets → Toolbars →** and activate **Tool options**. If you are using the command line, the same tool options are available via the toolbar or the command line. The tools that have options are listed in the table below.

Info: Commands *in italics* apply to the English version.

22.1 Line tools

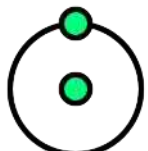
Action	Icon	Command	Remark
2 points		<i>l, li, line</i> <i>l, li, line</i>	Draw a line between two assigned points. Tool options: 
Angle			Line from a point with an angle and a length. Tool options: 
Horizontal		<i>hor, horizontal</i>	Horizontal line over center, end of line and length. Tool options: 

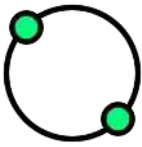

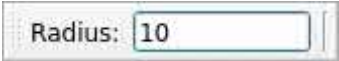
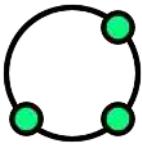
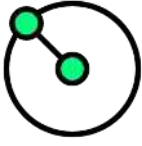
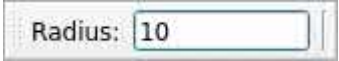
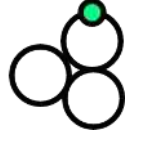
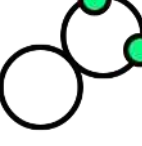

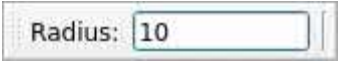

Vertical		Ver, vertical	Vertical line over center, end of line and length. Tool options: Length: <input type="text" value="10"/> Snap Point: <input type="text" value="Start"/>
Rectangle		rec, right rect, rectangle	rectangle by assigning the points of two diagonally opposite corners.
Parallel through point		Pp, ptp	A certain number of lines parallel to a selected existing line through an assigned point. Tool options: Number: <input type="text" value="1"/>
Parallel		o, pa, offset, parallel	A number of lines, parallel to a selected existing line, with a distance between the lines. Distance <input type="text" value="10"/> Number: <input type="text" value="1"/>
Angle halves de		bi, bisect	A certain number of lines that bisect two angled lines. Length: <input type="text" value="10"/> Number: <input type="text" value="1"/>
Tangent (P,K)		tanpc, tangentpc	Line from an assigned point tangential to an existing circle.
Tangent (K,K)			Line tangent to two existing circles.
Tangent at right angles			Line that is tangent to an existing circle and perpendicular to an existing line.

Right-angled		ortho, perp	A line of a certain length that runs perpendicular to an existing line and whose center lies at a certain point. <input type="text" value="Length: 10"/>
Relative angle			Line with a certain length and a certain angle in relation to an existing line. <input type="text" value="Angle: 30"/> <input type="text" value="Length: 10"/>
Polygon (center, corner)		poly2 pl, polyline	N-corner with a certain number of sides by determining the center point and the vertex. <input type="text" value="Number: 3"/>
Polygon (Cen, Tan)			N-corner with a certain number of sides by assigning the two points to one side. <input type="text" value="Number: 3"/>
Polygon (2 corners)		poly2, polygon2v	N-corner with a certain number of sides by assigning the two points to one side. <input type="text" value="Number: 3"/>

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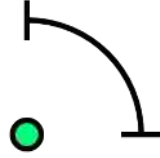

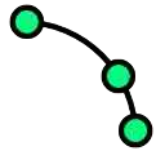
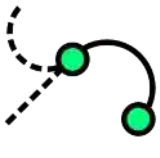
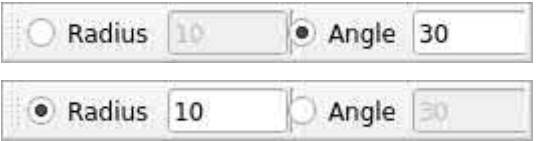


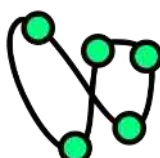

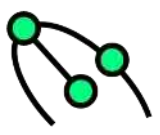
22.2 Circle tools


Action	Icon	Command	Remark
Center, Point		kr, circle <i>ci, circle</i>	Circle with a free radius, and center.

2 points		c2, circle2	Circle with a certain diameter, over you two opposite points on the circumference of the circle.
2 points, radius			Draw a circle with two points on the circumference and with a certain radius: 
3 points		c3, circle3	Circle with three points on the circumference.
Center point, radius		cc, circlecr	Circle with a certain radius whose center is a certain point on the circumference of the circle. 
Tangential, 2 circles, 1 point			Draw a circle that is tangential to two existing circles and define a center point to determine the radius.
Tangential, 2 points			Circle tangential to an existing circle via the diameter and two points on the circumference.
Tangential, 2 circles, radius			Circle tangential to two existing circles with a certain radius. 
Tangential, 3 circles		ct3, tan3	Circle tangential to three existing circles and/or lines.

22.3 Arcs, ellipses and spline tools

Arcs, splines, ellipses


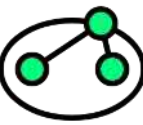
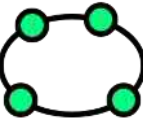
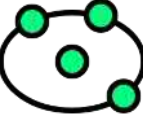
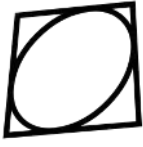
Action	Icon	Command	Remark
Center, point, angle		ci, circle	Circular arc with a radius through a center point and a point on the circumference, indicating the direction of rotation: 
3 points		a, ar, arc	Draw an arc by assigning three points on the circumference of the arc that define the start position, a point on the circumference and the end position of the arc.
Tangential arc			Draw an arc tangential to the end of an existing line segment with a defined radius or angle (deg). 
Spline		spl, spline	Draw an open or closed spline (curve) by assigning control points and a specific degree of freedom (1 - 3). 
Spline through points		stp, spline2	Draw an open or closed spline (curve) by defining points on the spline. 
Elliptical arc (axis)			Elliptical arc over main axis, second axis, start and end angles.

Freehand line		fhl, free fhl, free	Draw a non-geometric line.
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22.4 Ellipse tools



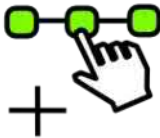
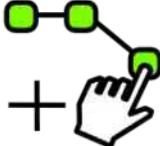
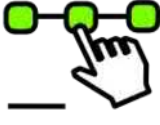
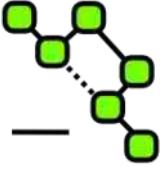
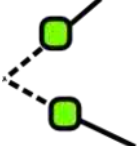
Ellipses



Action	Icon	Command	Remark
Ellipse (axis)			Draw an ellipse by assigning a center point, a point on the circumference of the main access point and a point on the circumference of the secondary access point.
Ellipse Focal point			Draw an ellipse by assigning two focal points and a point on the circumference.
Ellipse 4 point			Draw an ellipse that has four points on its circumference.
Ellipse center and 3 points			Draw an ellipse by defining a center and three points on the circumference of the circle.
Ellipse within 4 lines		ellipsis inscription d	Draw an ellipse that is bounded by four existing non-parallel line segments.

...

22.5 Polyline tools

Polylines are contiguous elements as a block.

Action	Icon	Command	Remark
Polyline		pl, polyline	<p>Draw an ellipse by assigning a center point, a point on the circumference of the main access point and a point on the circumference of the secondary access point.</p>  <p>The options include line and for curved segments: tangential, tan radius and angle.</p>
Add node			Add a node to an existing polyline. (Use "Snap to element" to place the new node on the segment).
Attach node			Add one or more segments to an existing polyline by selecting the polyline and adding a new node end point.
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 so that they intersect in a new node.



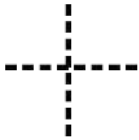
<p>Create equidistant polylines</p>			<p>Draw a certain number of polylines parallel to a selected existing polyline with a certain distance between the lines.</p> <p style="text-align: center;"> <input type="text" value="Spacing: 10"/> <input type="text" value="Copies: 1"/> </p>
<p>Create polyline from existing segments</p>			<p>Create a polyline from two or more existing separate line or arc segments that form a continuous line.</p>

...

22.6 Catch functions, snapping options

Snap functions offer the option of selecting precise positions with the mouse. Various snapping tools are available with which the user can snap to different positions on objects or the drawing area. These functions are located in the **lower** status bar.

Objects Catching, snatching, snapping

Action	Icon	Command	Remark
<p>Exclusive fishing mode</p>			<p>On: Only one snapping mode is permitted. Off: Several catch modes are permitted The catch modes are saved in every state.</p>
<p>Imaginary center</p>			<p>Enables the "imaginary" snapping of a center point without an existing line. Click on any two points. The center point of the line is snapped.</p>
<p>Free</p>		<p>os, sf</p>	<p>Allows the crosshairs to move freely while other capture modes are activated.</p>

58 Contributors

There are many people who have contributed to the LibreCAD User Guide. Some contributions came via the LibreCAD forum, the wiki or the source code.

Some of those who have contributed directly or indirectly are: André

Hurler

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

58.1 Corrections, improvements, errors found

Please contact us at the following e-mail address: cadcam@etik.com

59 Index

Sorted alphabetically:

3

3Dfindit..... 229

4

4MCAD..... 153, 229

9

9-corner..... 109

A

Beveling..... 69

Absolute coordinates..... 24

Alias configuration file..... 73

Alibre Atom3D..... 153, 229

Alibre Design EXP..... 229

Change..... 67

Docking widgets..... 53

Views..... 27

Application settings..... 227

Equidistant..... 61, 140

Attribute..... 69

Break open..... 70, 104, 130

Tear down..... 27

Unraveling..... 104f., 130

Hide..... 150

Breakout line..... 190

Aligned..... 65

Fix alignment..... 180

Autozoom..... 38, 101

B

Command file..... 73

Command line..... 14, 54, 72, 79, 100

Dimensions..... 65

accelerate..... 223

Library..... 170

Library Browser..... 14, 162

Libraries..... 163

Block..... 104, 160, 170

Break up block..... 154

Blocks..... 14, 154, 160

Block filter..... 205

Blocklist..... 161f., 170, 205

Block technology..... 203

Drilling..... 229

C

cal..... 14

CAM..... 229

Camotics..... 229

Ctrl..... 14

D

DIN EN ISO 5456..... 27

Divide..... 213

Top view..... 27

Turning..... 67, 229

Drop-down menus..... 52

Print..... 171, 218, 221

Print preview..... 218, 221

Diameter..... 66

DWG..... 19, 229

DXF..... 19, 229

DXF converter..... 229

E

Level "0"..... 47

Hide level..... 201

Levels..... 31

Layers and line types..... 44

Level list..... 202

Edit..... 67

Own standard parts library..... 191

Properties..... 69, 212

Properties window..... 226

Fade in..... 150

Insert..... 14

Export settings..... 227

Elements..... 14, 16

Ellipses..... 59

Elliptical arc..... 58

End point..... 62

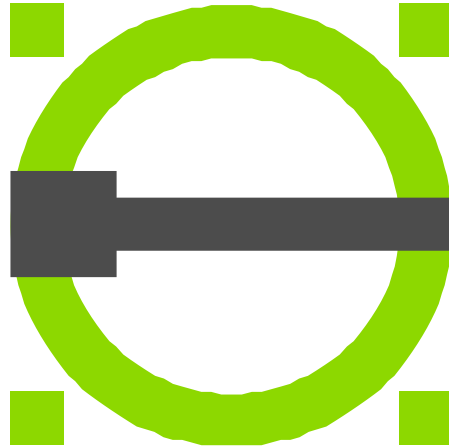
Distance point to point	70, 115, 142	Horizontal.....	66
Entity	170	Hypotenuse.....	94
Entities	14, 16	I	
Design level	32	Imaginary center.....	61
Design mode	51	Incremental coordinates	22, 24
Exclusive fishing mode	61	Instances	163
Explode	70, 154	K	
F		Cartesian coordinates	22
Catch	61	Plain text calculator.....	230
Catch functions	61, 101	Knots	60
Color selection dialog	158	collinear	141
Colors	44	Commands and orders.....	73
Chamfer.....	69	Compas 3D.....	229
Filter line	205	Configuration file.....	73
Flowchart	230	Converter	229
Milling.....	229	Coordinates.....	143
Milling cutter	141	Coordinate input	20, 26
Cutter coordinates.....	140	Coordinate systems.....	21
Milling coordinates.....	115, 142	Copy	14, 67
Free	61	Circle tools.....	56
Free catch.....	193	Circular arc	58
freehand.....	183	Shorten/lengthen.....	141
Freehand line	59	L	
Front view	27	Location relationships	27
G		Layer.....	31
Counterclockwise	14	Layers.....	44
Threaded hole	184	Space bar.....	14, 26, 79
Grid	51	librecad.alias	73
Same properties	212	LibreDWG.....	230
Equation	211	Linear	65
Global guides.....	179	Line tools.....	54
Engraving.....	229	Line widths	48
Floor plan	27	Line thickness.....	31
GUI configuration	227	left mouse button	14
H		Linux.....	17
Semicircle	94	Bolt circle	172
Half section.....	76	Delete.....	70, 106
Hand sketch.....	183	M	
Auxiliary construction.....	203	MacOS.....	17
WELFARE LINE	155	Printing to scale.....	219
Guide lines	147, 150	Mouse wheel.....	38
Arrow	66	MegaCAD	229
HKEY_CURRENT_USER	227		

Measure	115	Radius	66, 69
Center	62	Grid	51, 62
Modify	67	Grid snap	78
Modify	67	Deactivate grid snap	110
MText	71	right mouse button	14
Sample	67, 172	Right-angled	56
N		Right-angled triangle	94
N-corner	56	Redo	14
Decimal places	42f.	regedit	227
NC-Corrector	229	Order	51, 67
NCnetic CNC	229	Relative coordinates	22, 24
Neuneck	109	Lock relative zero point	63
Standard part levels	202	Relative object snap	223
Standard parts library	165	Reverse direction	68
Standard parts	14, 163, 182, 229	Rotation pattern	172
Standard parts path	14	Rotate	67
O		Rounds	69
Objects	14, 16	S	
List objects	214	Blind hole threaded hole	191
List objects	205	Theorem of Thales	94
Offset	68	Switching claw	76
Restrict orthogonally	80	Snap	61
P		Cutting	69
Pan-Zoom	14, 38	Intersection	63
Parallel	68	Sectional drawing	151
Parallel	55, 115	Hatching	153
Parallelograms	94	Hatched breakout	186
Plan view	27	Hatching	71, 155
Plugin	207f., 211	Edit hatching	157
Plugins	206	Hatching trick	222
Polar coordinate input	14	Oblique rectangle	81
Polar coordinates	23	Screw catalog	182
Polyline	60	Title block	215
Polyline	61, 155	Black/white mode	171
Polylines	60	Floating menus	52
Pop-up menu	54	Side view	27
Profile view	27	Side view	27
Projection lines	180	SimplexCAM	229
Projecting	180	Scaling	67
Item	71	Scaling factor	166
Points from ASCII file	208	Snapping	61
R		Mirror	67, 185
		Spline	58
		Sqrt	100

Standard template	50, 229	Vector graphics.....	19
Status bar	54	Extend	68
Foundation attributes	31	Move	67
Routes	69, 170, 224	Vertical	66
Ctrl.....	14	Viewer	229
Symbol libraries	163	Front view	27
Symbols	14	Default settings	227
Toolbars	54	Template	14
T		Complete template	202
Panel projection	180	Templates.....	49f.
Tangent	55	W	
Tangents.....	99	Web service	182
Tangential.....	57	Workpiece coordinates	115
Tangential bend	58	Widgets	53
Pocket calculator	14, 72, 100	Windows	17
Share	213	Angle	66
Template	14	Angle between two lines	70
Templates.....	49	Angle inputs.....	14
Text.....	71	Angle bisector.....	55
Formatting text.....	226	Z	
Text lines	216	Gear wheel	207
Thales	94	Drawing settings.....	227
Trapezoids	94	Drawing head	215
Disconnect.....	69	Drawing tools	54
Trimming.....	68, 104	Central standard parts library	165
U		Center	62
Transmission	212	Circular pattern	172
Perimeter line.....	190	Zoom	14
Undo.....	14	ZW3DCAD and ZW3DCAM	229
V		#	
VariCAD	19, 229	#ffffe	158



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