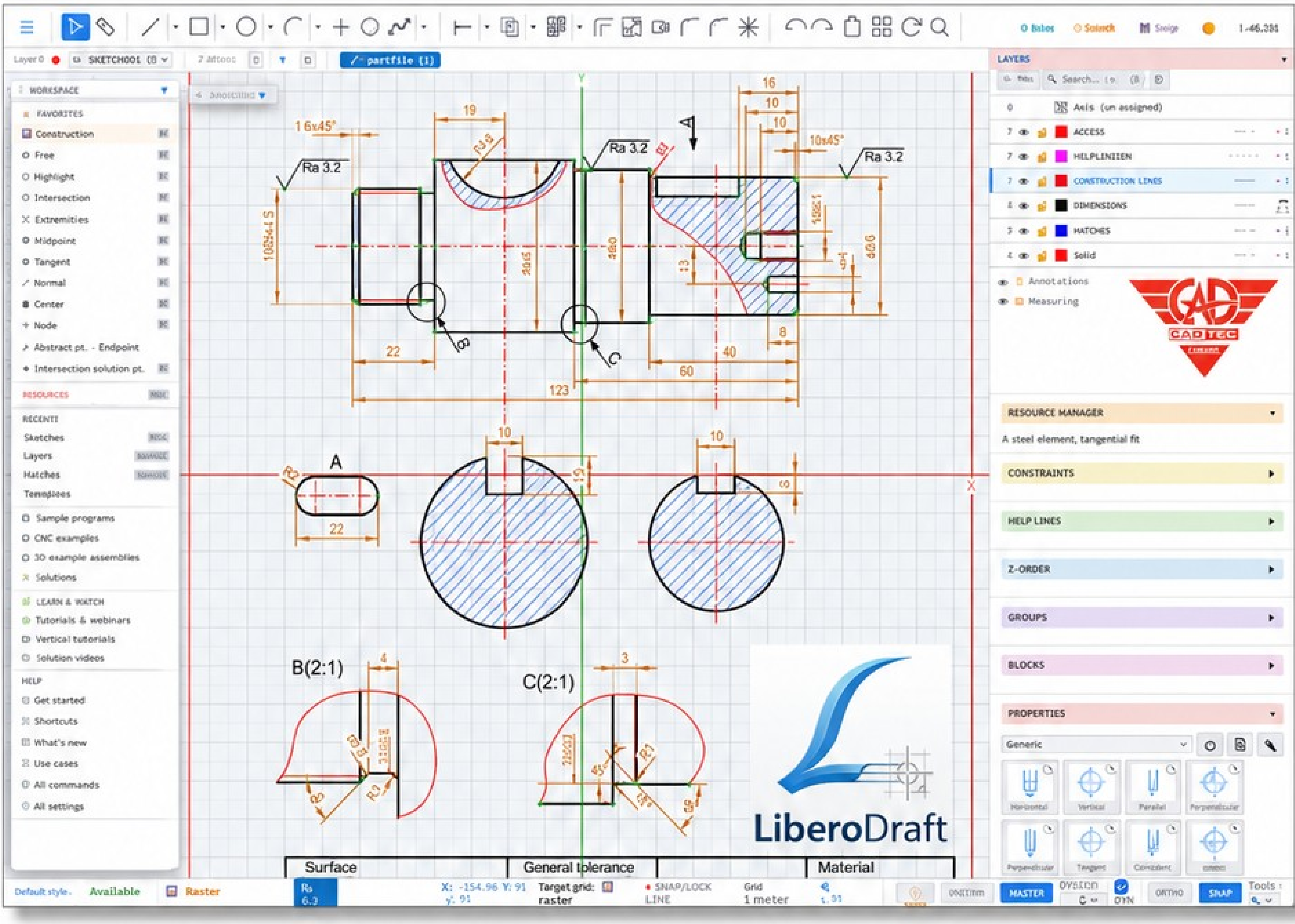
The CAD book for LiberoDraft for beginners and advanced users.

LiberoDraft

2D CAD directly in the browser

- Precise 2D drawings.
- Intuitive operation
- Layers, groups & blocks
- Snapping & constraints
- DXF / SVG import & export
- No subscription - no installation



Powerful. Flexible. Save locally.

Your data stays on your computer, NAS or in Dropbox.

Windows | macOS | Linux | Store locally | NAS | Dropbox

Cloud CAD

Program from the cloud
Data stays local

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Notes:

The 2D CAD book for beginners and advanced users.

Unleash your creativity with LiberoDraft.

The easy-to-follow guide to precise and professional drawings, with plenty of pictures and step-by-step instructions. :-)

https://www.cadtec.ch

1.3.35

This book and tutorial was created by:

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1 Contents

Inhaltsverzeichnis

2 Foreword

Grüezi, good day, hello,

For some time now we have been seeing a trend towards simple, fast 2D CAD programs that start up within a second in any web browser and on any PC or tablet. People simply no longer want to boot up an entire 3D CAD program every time they need a simple drawing, open a 3D project just to draw a plain plate or bracket including threads, then trigger a 2D derivation from it, insert a drawing frame, define the views, and so on... That usually takes far too long, and it does not run on a tablet or iPad at all. **LiberoDraft can do all of that** and combines simplicity, speed, ease of use, independence and **zero cost** etc. all under one roof :-)

LiberoDraft is an exciting and user-friendly CAD program that is ideally suited to 2D drawings and designs. Whether you are an engineer, architect, designer or hobbyist, LiberoDraft gives you the most important tools and functions you need to create professional drawings.

This book, or tutorial, is intended to give you a comprehensive overview of LiberoDraft and to help you get the most out of the program. We will cover the basics of LiberoDraft, such as the user interface, the tools and functions, creating drawings, and working with layers, groups, blocks and libraries.

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 LiberoDraft. If you already have experience with LiberoDraft or other CAD programs, we will help you improve your skills and get to know new functions 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 to the latest version of LiberoDraft, so that you can get to know the newest functions and tools.

This book will help you improve your skills in LiberoDraft 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 reach your goals.
→ Enjoy reading and learning :-)

2.1 One request...

We have done our best to make this book as free of errors as possible. If you do notice typing errors or inaccuracies, we would be very grateful if you would report them to us.

Since this book is being written by users for users, we are always keen to improve and update it so that it is as useful and informative as possible.

We hope you will find this book a helpful resource for your work with LiberoDraft, and we wish you every success with your projects!

This LiberoDraft book is a living project, and you will find an e-mail address on one of the last pages where you can send us improvements at any time, and you are very welcome to do so.

«Many thanks!»

3 Why 2D CAD is making a quiet comeback

It is a quiet development, but one we are asked about more and more often...: for some time now we have been seeing a measurable trend back towards easy-to-use 2D CAD programs. At first glance that may be surprising -> after all, 3D was considered the measure of all things for years. But anyone who does **not** design every day knows the situation only too well: you quickly need a precise sketch. A connecting plate, a sheet metal blank, a simple yet accurate workshop drawing, a dimensioned floor plan etc. **And what for?** For a few lines, arcs and dimensions you boot up a complete 3D CAD system, wait for the license server, create a project, choose a work plane, create a sketch in part mode -> and after ten minutes of administrative overhead you have not drawn a single line.

That is exactly where the return to 2D comes from: **many tasks are two-dimensional by their very nature.** Laser and plasma cutting, gaskets, front panels, layout plans, schematic sketches, detail drawings for the workshop -> none of that needs solid bodies, feature trees or assembly management. It needs precision, dimensioning, hatching and a clean printout at the right scale. A 3D system can of course do that too -> but it is like driving an articulated lorry to the letterbox.

3.1 3.1 LiberoDraft: 2D CAD that is simply there, in the browser

LiberoDraft grew out of this idea -> a semi-parametric 2D CAD that runs entirely in the browser. No installation, no license server, no account: open the page, start drawing. What sounds like a compromise is technically the opposite. LiberoDraft draws via WebGL and keeps its scene in graphics buffers instead of rebuilding it for every frame. Even drawings with several thousand elements and hundreds of blocks therefore stay fluid -> zooming and panning feel like a native program.

And "easy to use" should not be confused with "simplistic". Under the hood a fully-fledged CAD is at work:

- **Precise drawing** with lines, circles, arcs, ellipses, rectangles, polylines, splines and polygons -> plus fillets and chamfers with selectable trim behavior, offset, trim, break, split, mirror, rectangular and polar arrays.
- **A well thought-out snap system** with endpoint, midpoint, intersection, center, tangent, element and object snap tracking (AutoTrack): points align with existing geometry as if by themselves, entirely without construction lines.

- **Experimental constraints** such as horizontal, vertical, parallel, perpendicular or concentric capture the intent of the design -> semi-parametric in the best sense: as much intelligence as necessary, as little administration as possible. Welded endpoints stay connected, even if you later extend or rotate an element.
- **Standard-compliant dimensioning** that stays persistent and editable: linear, aligned, radius, diameter, angular, arc length, chain and baseline dimensions -> with tolerances, driving and driven dimensions, and settings that can be defined for every single dimension. Alongside the mechanical engineering style, construction and timber construction dimensioning with slash terminators is available.
- **Annotation** with text, remarks, leaders, item balloons, datum features and feature control frames to ISO 1101.
- **Automatic hatching**: one click inside a closed area is enough, islands are detected automatically. The seed point stays visible and can be dragged into another chamber; if the surrounding geometry changes, the area adapts immediately.
- **Layers, blocks with parts list data, standard parts and groups** -> the block manager shows the nesting as a tree, hides individual blocks and evaluates quantity or structured BOMs.
- **Paper space with viewports**: model and sheet kept separate, an individual scale per viewport, and dimensions may point through the viewport onto the drawing. The print dialog has scale, paper format and line widths firmly under control, including storable print profiles.
- **DXF and SVG import/export** for smooth exchange with the rest of the CAD world -> with adaptable profiles for customer-specific layer mappings. Blocks remain real blocks, together with their own dimensions and hatches.
- **Images as an underlay**: scanned plans or cadastral extracts can be inserted, fitted to scale and dimensioned directly on top.

All of this can be operated exactly as you would wish: a command palette finds every function at the press of a key, frequently used tools are dragged and dropped into a personal favorites bar, and anyone who prefers typing to clicking enters coordinates, lengths and angles directly in the floating input field -> cartesian or polar. Wherever a function needs several steps, a hint flying along with the mouse pointer guides you through the sequence. A gizmo allows the selection to be moved, rotated and scaled with numerical precision. A light and a dark appearance are available, and the interface comes in German and English -> further languages will follow.

3.2 Your data stays your data

One point that matters a great deal to us: LiberoDraft loads **no external scripts**, uses **no CDN** and includes **no third-party libraries**. Drawings are stored locally as files -> **there is no obligation to store data in the cloud and nobody reads along.** What you design is nobody else's business. In times when even simple tools demand a subscription and a user account, that is deliberately old-fashioned -> and exactly as intended.

3.3 Free of charge -> on a donation basis

LiberoDraft is developed by CADTEC (Schweiz) GmbH and published on a voluntary donation basis. And that without any small print: **commercial use is expressly permitted as well** -> whether in the workshop, in the design office or on site at the customer's, no license fees apply. Anyone who uses LiberoDraft professionally and would like to support its further development does so with a voluntary donation. The program is currently being polished as a **release candidate** and is being developed continuously -> anyone trying it out today is already working with the full range of functions and can look forward to what is still to come.

So the articulated lorry can stay in the garage. For your next precise sketch a browser is enough:

→ **liberodraft.com**

LiberoDraft -> semi-parametric 2D CAD in the browser. By CADTEC (Schweiz) GmbH.

4 First things first!

It is an advantage to know the following points right from the start:

LMB = **L**eft **m**ouse button

RMB = **R**ight **m**ouse button

MW = **M**ouse **w**heel

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

RMB 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.

Spacebar activates the command line.

Angle entries are always **counter-clockwise,** with horizontal always being 0°.

RMB or **the [Esc] key** cancels a function.

Zooming with the **MW** / **pan-zoom**: **hold the MW** pressed.

Selection methods: if a window is dragged in a drawing from **left to right**, only elements lying **completely** inside the area are selected. From **right to left**, all **crossed** elements are selected as well.

With functions that require a dimension to be entered, such as a parallel, the distance can be entered directly in the floating window.

4.1 Cancel: Esc, Redraw, Zoom to fit

Pressing Esc ends ALL running functions at once and switches back to the select tool. That also applies when several things are open at the same time — for example a drawing function that has been started, an active window zoom and an existing selection.

"Redraw" and "Zoom to fit" have the same effect. So if you ever no longer know what state the program is in: press Esc and you start again cleanly.

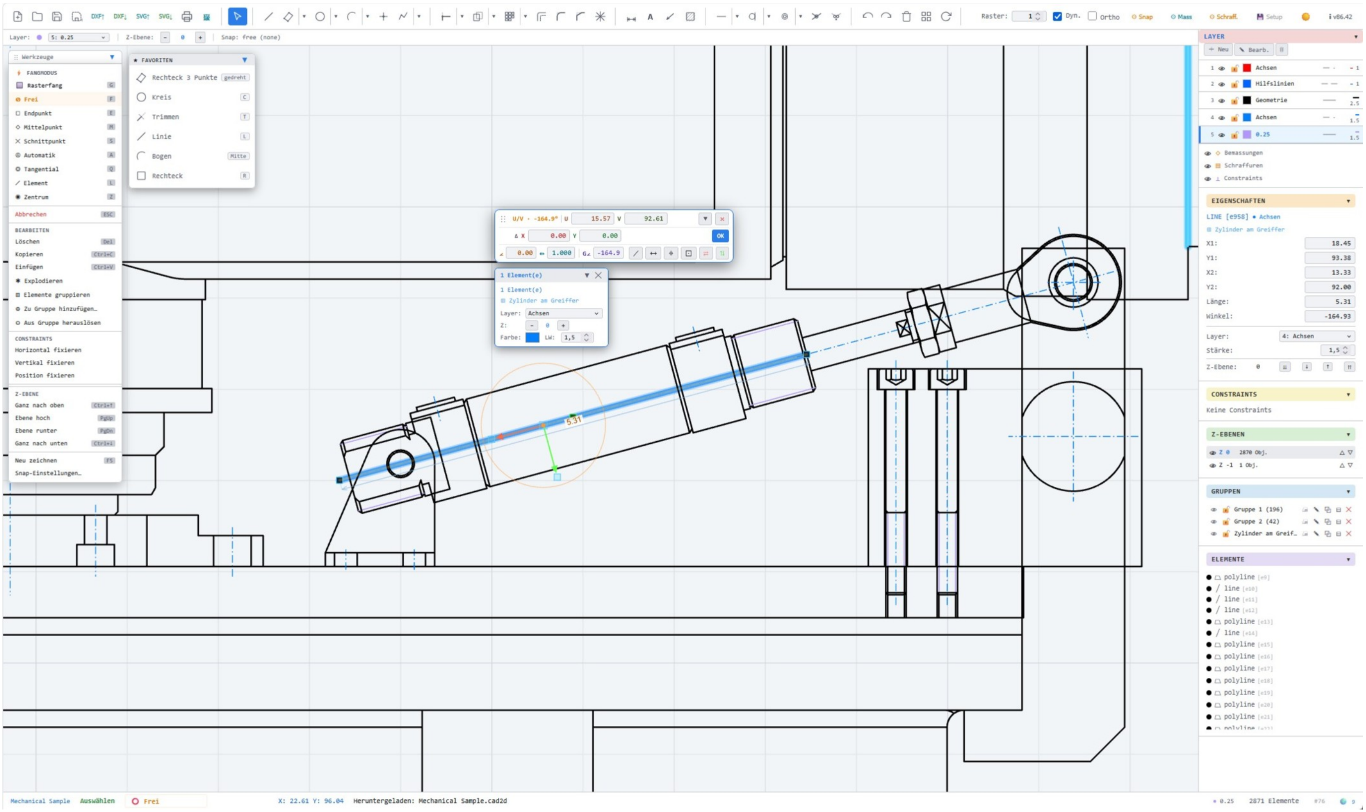
Cancelling changes nothing about the geometry — it only ends whatever is currently in progress.

If a window is open, for example the info or license display, Esc closes that window first.

INFO:

As already mentioned above, this book is being written by several LiberoDraft users. Depending on the country CH, DE, AT, US, GBR ... the **CTRL** key corresponds to the **STRG** key and vice versa :-) → [**CTRL**]+[**F**] = [**STRG**]+[**F**]

5 What is LiberoDraft?



LiberoDraft is a simple CAD program that supports 2D drawings and designs. It is a powerful yet user-friendly tool that is available in any browser on Windows, macOS and Linux. In this tutorial we will cover the basics of the operating philosophy of **LiberoDraft** to make getting started with the software easier for you.

Summary:

- **Drawing tools**: lines, arcs, circles, polygons, splines, dimensions, annotations and more
- **Modify tools**: move, copy, rotate, scale, mirror, trim, extend and more
- **Snap functions**: object snap, endpoint snap, midpoint snap, intersection snap automatic snap with track point and more
- **Layers**: organize your drawing in layers, groups, blocks, isolated islands and more.
- **Groups**: create reusable symbols, groups and blocks
- **DXF/SVG support:** import and export of DXF and SVG files
- **Multilingual**: German, English. Further languages are planned...

5.1 LiberoDraft is well suited to:

- Technical drawings: creating floor plans, sections, views and details
- Mechanical design: designing machines and components
- Electronics: creating printed circuit board layouts
- Architecture: designing buildings and rooms
- Landscaping: creating garden plans
- Hobby and craft work: creating designs for furniture, decoration and more

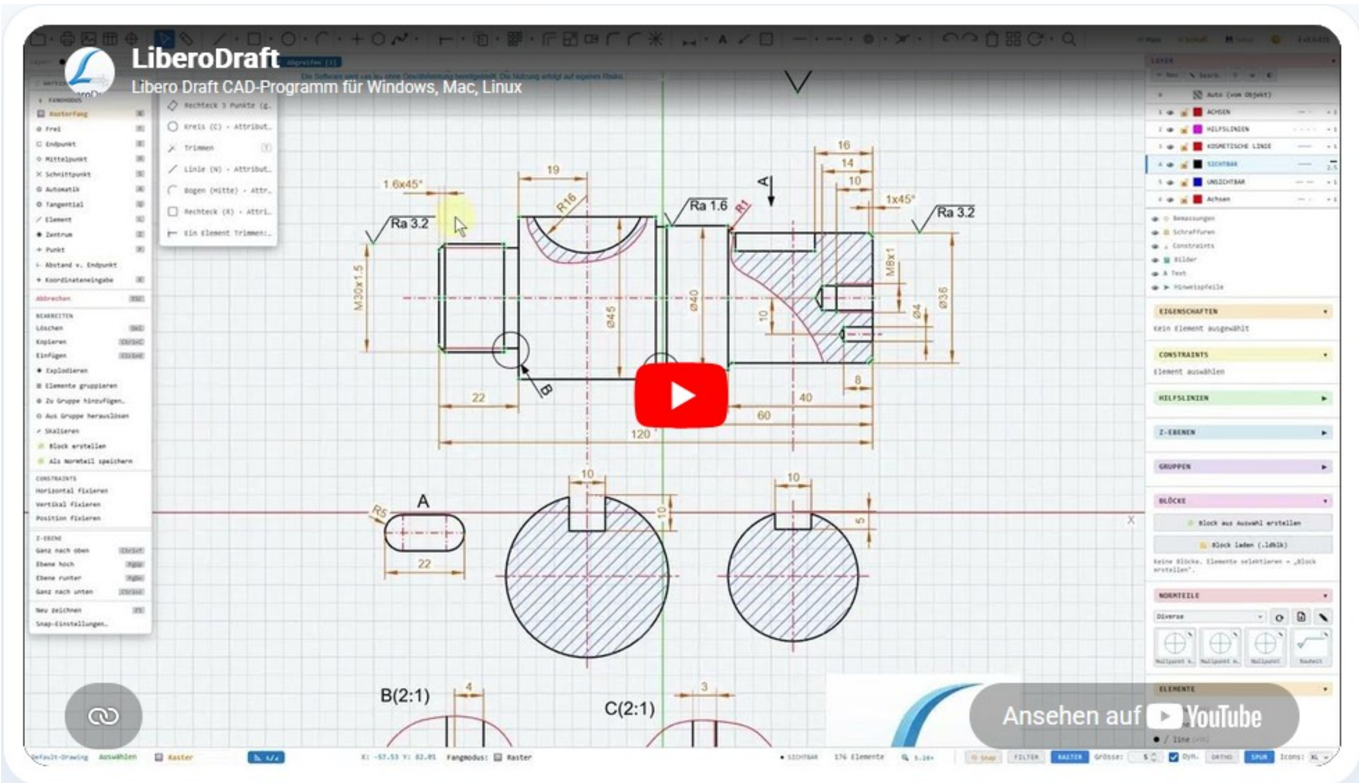
LiberoDraft is used in machine and equipment construction (in electrical engineering, among other things, for printed circuit board designs), in the building industry (structural and civil engineering, interior design), in urban and landscape design and much more, and by hobby technicians.

5.2 The LiberoDraft license model

LiberoDraft is a **free, donation-based online CAD program.** That means you may use it freely. All your data stays on your computer and is **not stored in the cloud. You have 100 % sovereignty over your data.**

5.3 A short video to start with

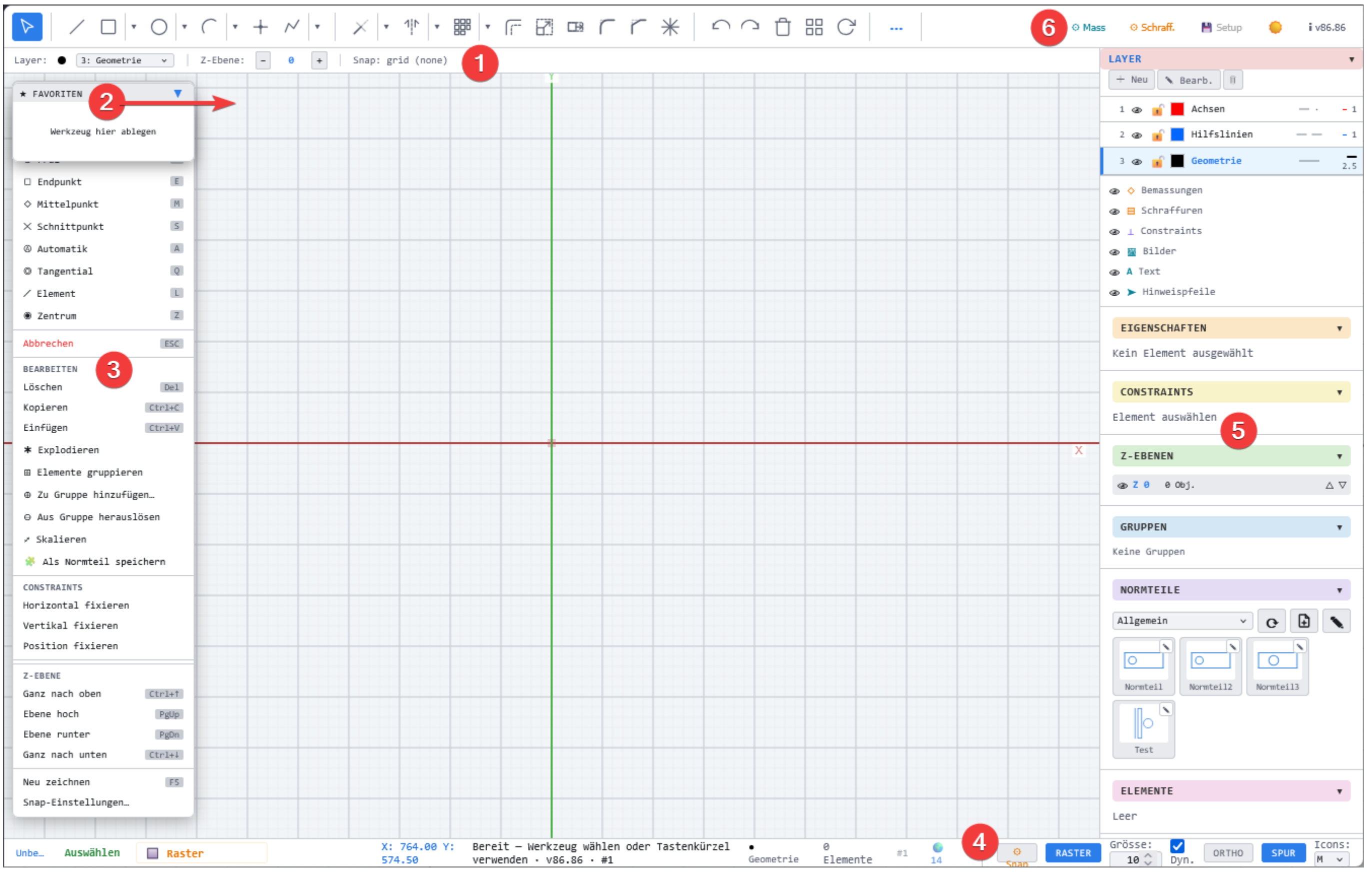
Here is a short training video to get a first impression:



YouTube: https://www.youtube.com/watch?v=4CA9-OL2Xik

6 The interface

A short overview of the user interface



1. **Toolbar** and edit functions. Some icons can be expanded with the small triangle, whereupon further functions appear. Directly below it is the **cursor HUD bar** with further information and functions.

2. Frequently used functions can be dragged with drag and drop and dropped into the favorites window. The window can be moved → .

3. Snap functions, clipboard, explode, group elements, scale and so on.
   The hotkeys are shown in the gray boxes.

4. Adjustable snap, grid, ortho and tracking lines as well as the size of the tool icons.

5. The right-hand panel holds all the "administrative" CAD functions (expandable and collapsible via the small black triangles ► ▼):
   • layer and group management
   • display of the object properties
   • Z levels for restacking elements
   • block and standard part management
   • element list

6. Further program settings, setup and switching between day and night mode.

7 Functions at a glance

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

7.1 Vector-based drawings

LiberoDraft works with vector graphics that consist 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 at different sizes and resolutions without any loss of quality.

7.2 Intuitive user interface

The LiberoDraft user interface is designed to be simple and clear, with a menu bar, toolbars, a drawing area and a status bar. This makes getting started with the program easy, even for CAD beginners.

7.3 Various drawing tools

LiberoDraft offers an extensive range of drawing tools with which you can create lines, circles, ellipses, polygons, splines and other geometric shapes. In addition, commands are available for copying, patterning, moving, rotating, mirroring and scaling objects.

7.4 Layers

LiberoDraft supports layers with which you can organize and manage your drawing. Layers can be shown and hidden, locked and renamed in order to simplify the drawing process and improve clarity.

7.5 Snap functions, snapping

LiberoDraft has various snap functions that help you create precise drawings. These include endpoint, midpoint, tangent point, intersection and 10 further snaps as well as a grid and orthogonal drawing.

7.6 Support for various file formats

By default LiberoDraft uses its own file format based on JSON, which you can read and modify dynamically! It also supports the DXF format (**D**rawing e**X**change **F**ormat), which is compatible with other CAD programs, for import and export. You can also export your drawings to other formats such as SVG, PDF or PNG.

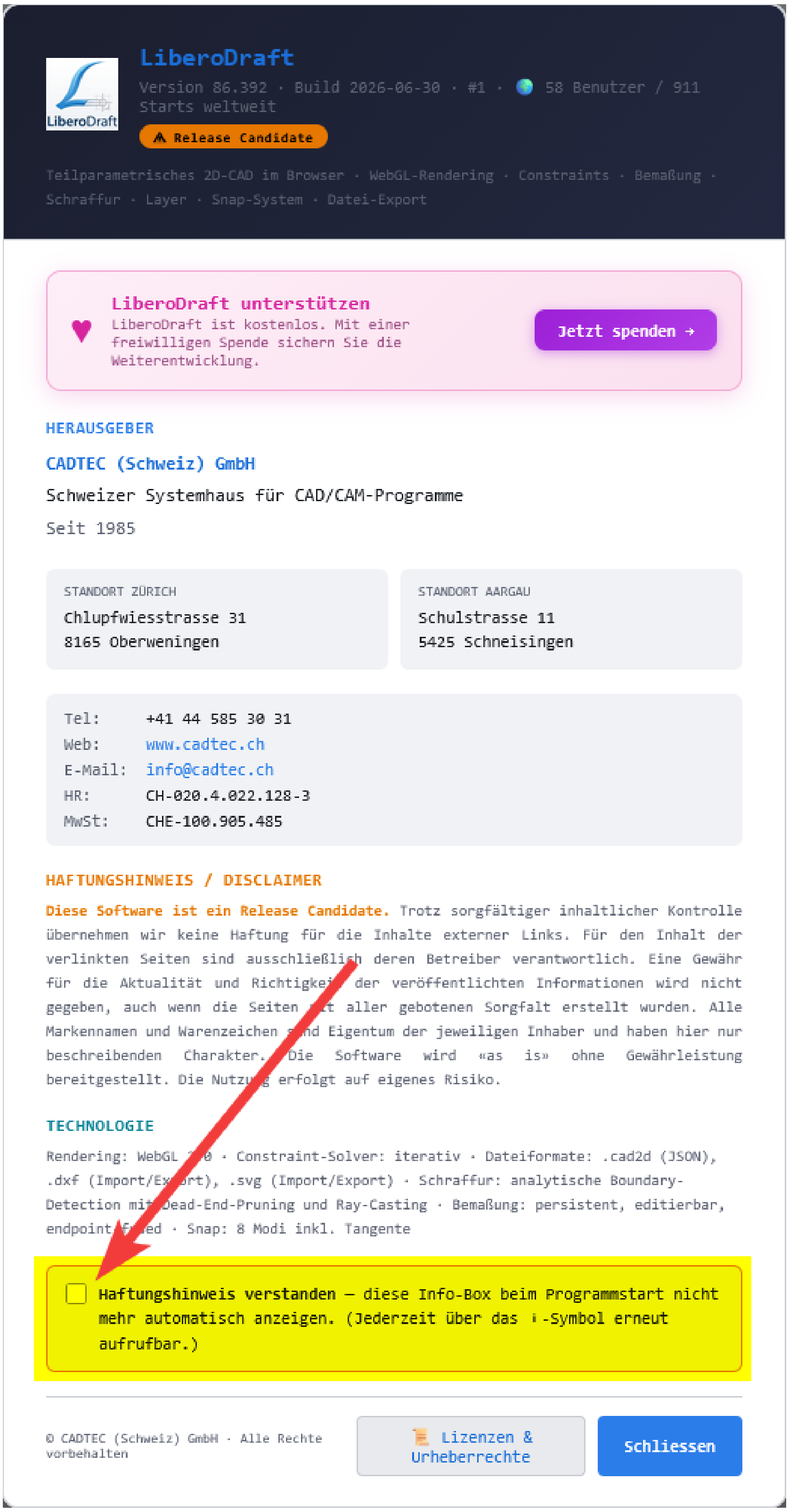
7.7 Customizable user interface

LiberoDraft lets you put together your favorite functions freely.

8 LiberoDraft CAD operating information

8.1 Starting the program for the first time

The CAD program starts with a notice box and a test drawing as long as you have not ticked the check box for the disclaimer.



Info: if you work at a low screen resolution or with enlarged screen fonts, you have to scroll down in this dialog, because it may not be displayed completely.

Click the check box and then click Close.

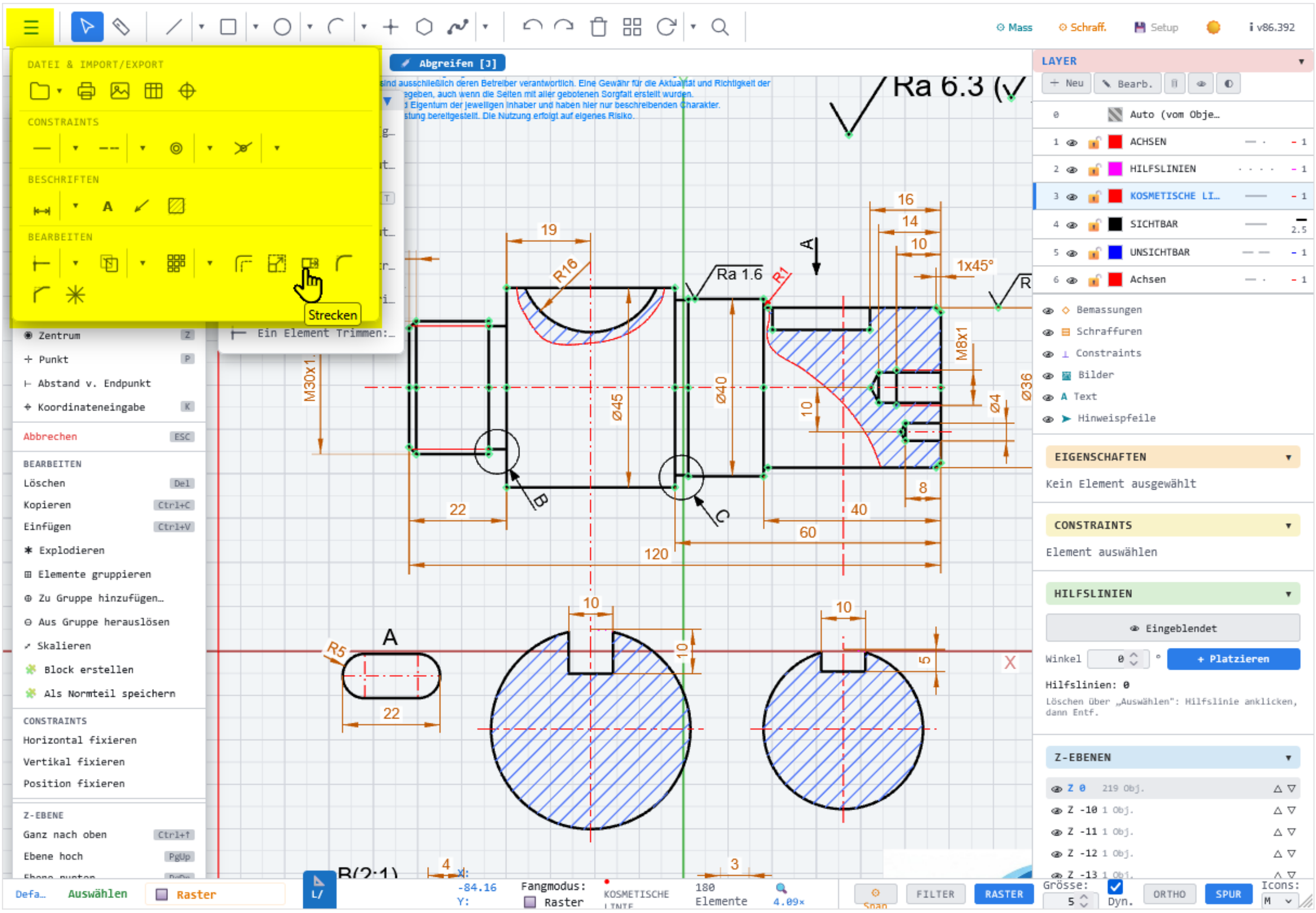
Tip: before you do so, we would be delighted to receive a voluntary donation towards the further development of the program.

You thereby support us in a fiercely contested market and will continue to benefit in the future from a clean 2D CAD program for everyone. :-)

8.2 On a small screen or tablet

If you work with a tablet, a small screen or scaled operating system texts, the function bar may have too little space.

In that case the familiar **hamburger menu** is shown so that you can still reach every function:



A blank drawing is opened with **Ctrl+N** or **File → New drawing**.

The left-hand panel can be collapsed so that you get more drawing area on a small screen.

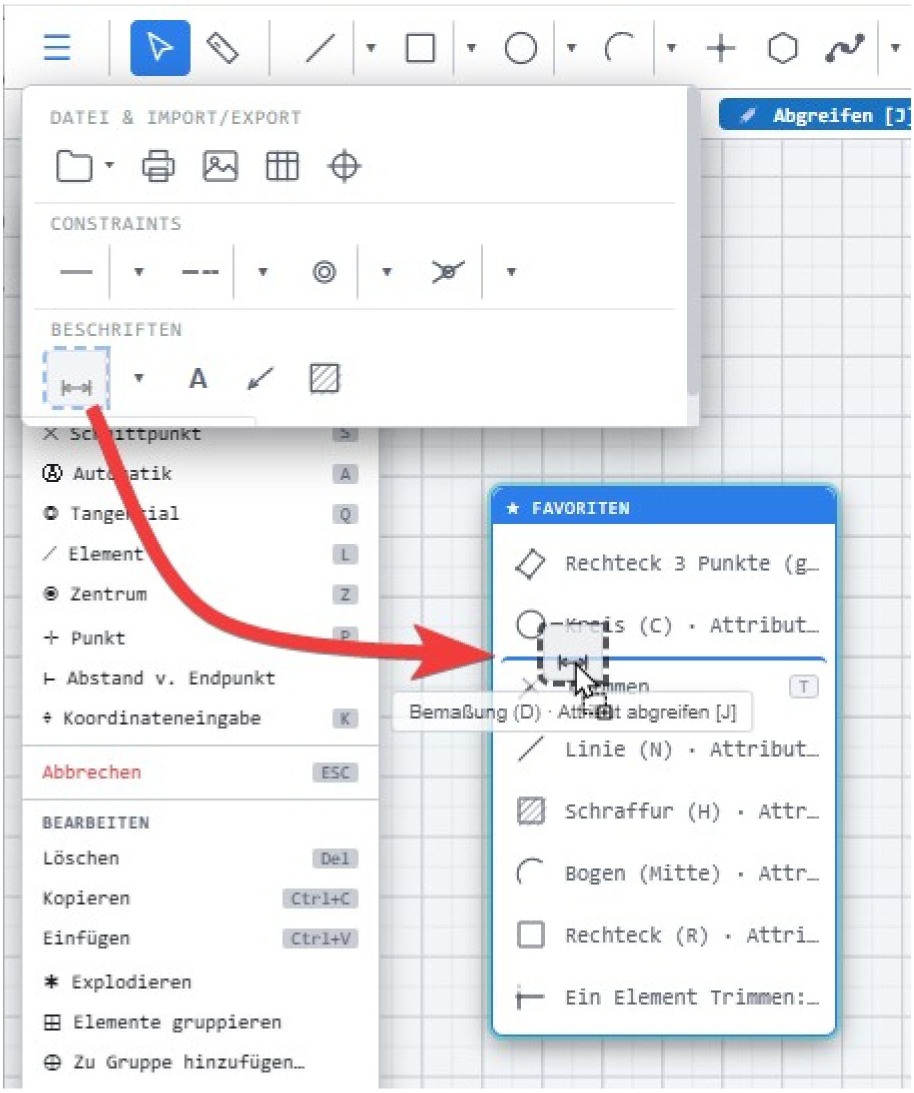
The width of the right-hand panel can be adjusted, and you can scroll inside it with the mouse.

Related terms: iPad, laptop, notebook, small screen, resolution

8.3 Favorites menu

If functions you use a lot are located in the hamburger menu, they can be dragged into the favorites menu with drag and drop so that you can access them directly.

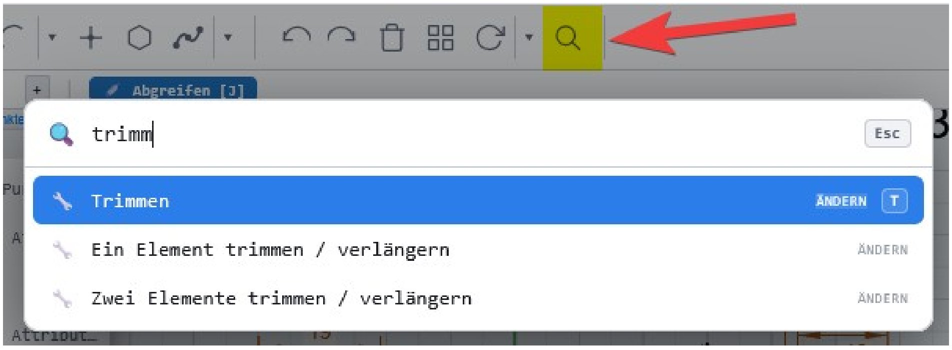
If the favorites menu is covered by the hamburger menu, the favorites menu can be moved.



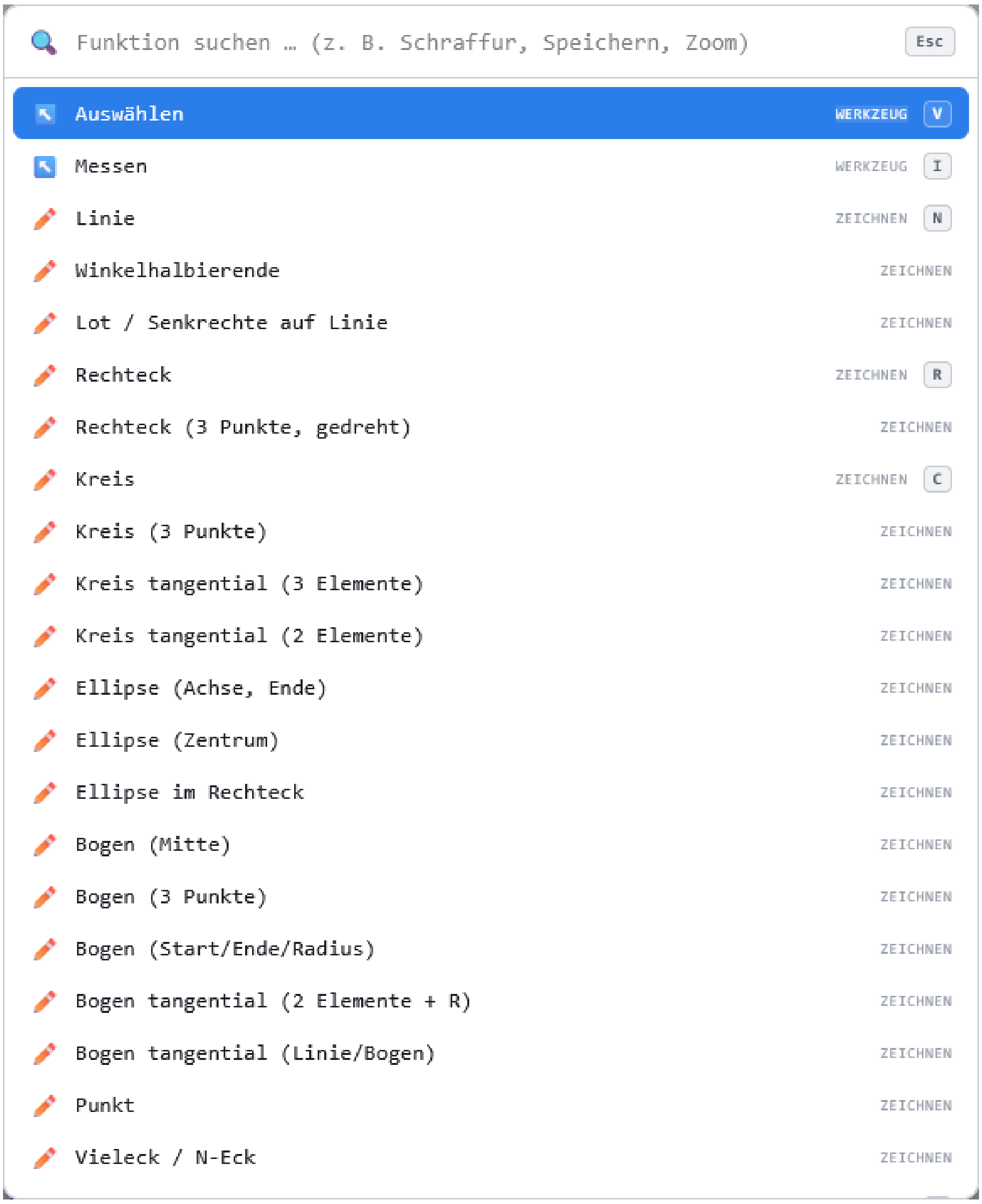
Related terms: preferred functions, custom menu, user interface

8.4 Search functions

Thanks to the command search in the menu you will find the most important functions, settings and commands in no time:

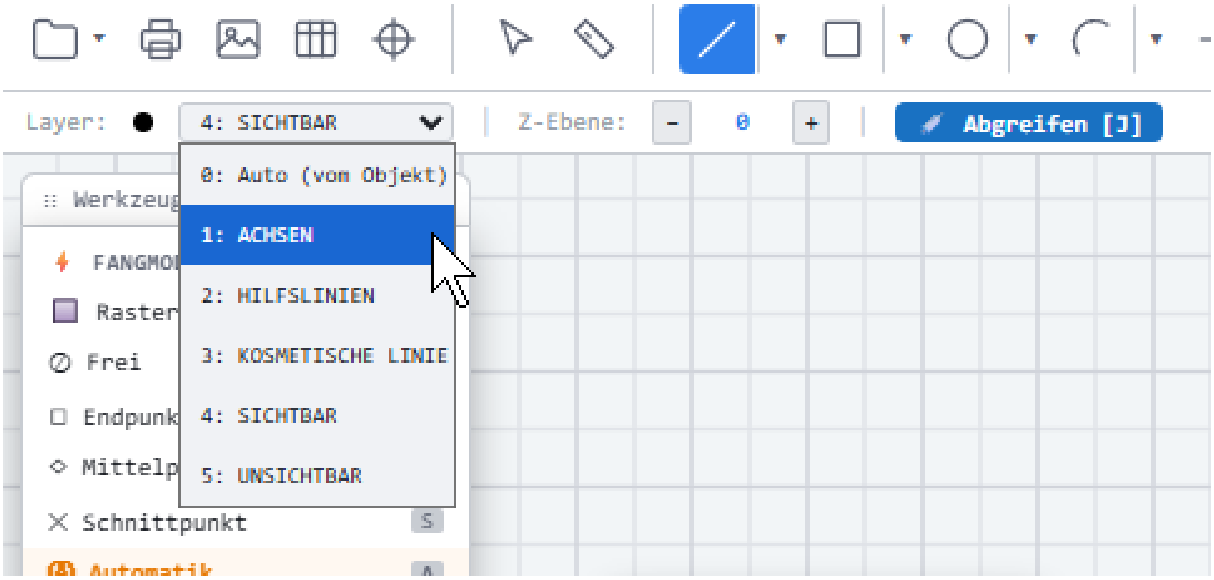


The search runs live while you type the term you are looking for…



8.5 Drawing your first element

As an example, draw a **line**:



The layer on which the line is to be placed is selected in the pull-down menu at the top left.
Alternatively the layer can also be selected in the panel on the right.

Golden tip: if the layer list contains many dozens or even hundreds of layers, the (blue) Pick up [J] button can save you a lot of work. **It can always be called up, even while you are in the middle of a drawing function.**

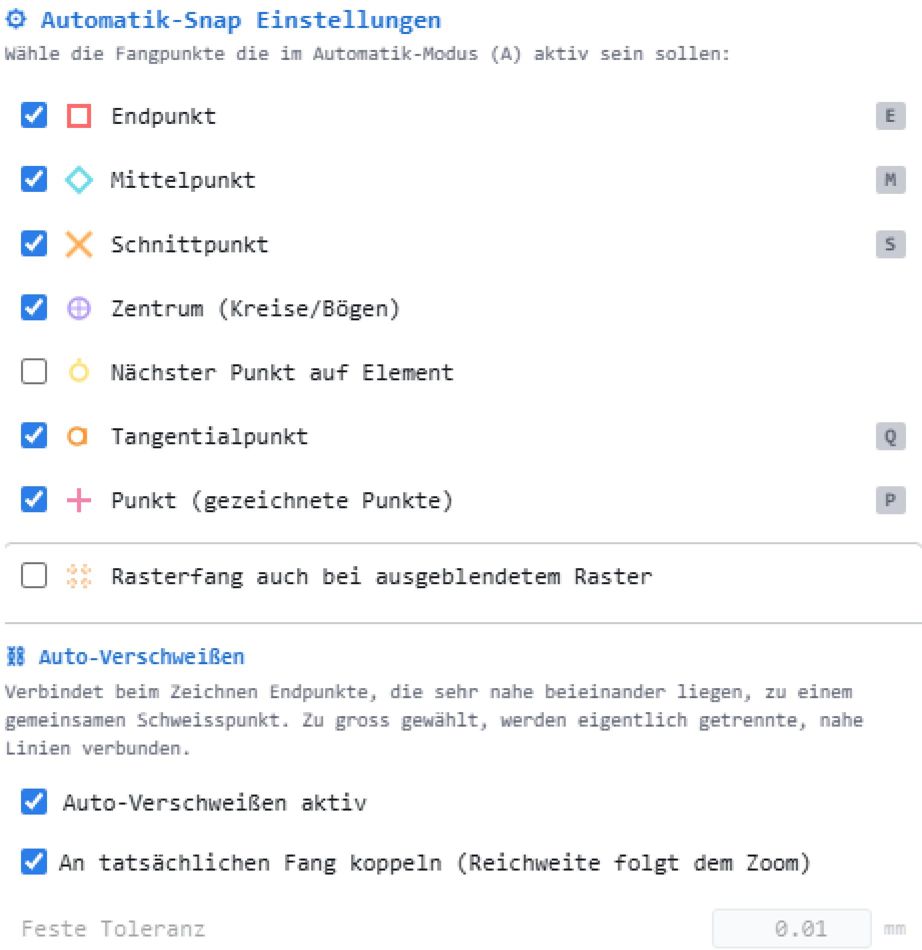
With it you pick up the layer attributes directly from other drawing elements. They are transferred immediately to the element you are currently drawing with the mouse.

8.6 The most important information for your first drawing

At the bottom right in the status bar you can influence the general behavior of the program and set it up to suit your needs:

The factory defaults are already quite good. The most important ones are described briefly below.

The **Snap button** configures the **automatic snap [A]**:



8.6.1 Couple to the actual snap (range follows the zoom)

If this option is active, auto-weld does not use the fixed tolerance but the current **snap range** -> that is, exactly the radius within which the cursor snaps to points while drawing.

This range depends on the zoom: zoomed far in it is very small and precise, zoomed out correspondingly larger. What gets welded is therefore exactly what visibly snapped while drawing -> what you snap is what gets joined.

Recommendation: switch it on if you want to rely on the visible snap while drawing. Switch it off and use the fixed tolerance instead if welding should be independent of the zoom and reproducible -> for example with scale-critical drawings that are edited at different zoom levels.

The **Filter button** is needed when only certain elements may be selected. This function only becomes important as the drawing grows. Not just yet, though. It is described in more detail further on in the book.

The **Grid button** shows or hides the grid.

Grid size: 5 mm — if this value is changed, the adjacent **Dyn.** check box switches off. In that case the grid always snaps fixed to the grid value that has been set (2 mm):

The factory default is the **dynamic grid,** which adjusts itself optimally depending on the zoom level and steps through the following values: 1, 5, 10, 50, 100… mm

That is extremely practical, since you do not constantly have to change the grid spacing by hand when you design a drawing at different zoom levels.

The **Ortho button** forces drawing exclusively **horizontally** and **vertically**.

8.7 TRACK button -> object snap tracking (AutoTrack)

Object snap tracking helps you place new points exactly **in alignment** with existing geometry -> without having to draw construction lines.

While drawing, hover briefly with the cursor over a snap point (for example an endpoint or midpoint) and it is captured as a **track point** and marked with a small cross (+). Invisible horizontal and vertical guide lines run through every track point: as soon as the cursor approaches such a line, the line appears dashed and the cursor snaps onto it. If two track points have been captured, the **intersection** of their guide lines snaps exactly as well -> this lets you place a point that is aligned horizontally with one reference point and vertically with the other, for example.

Hovering again over a track point that has already been captured removes it.

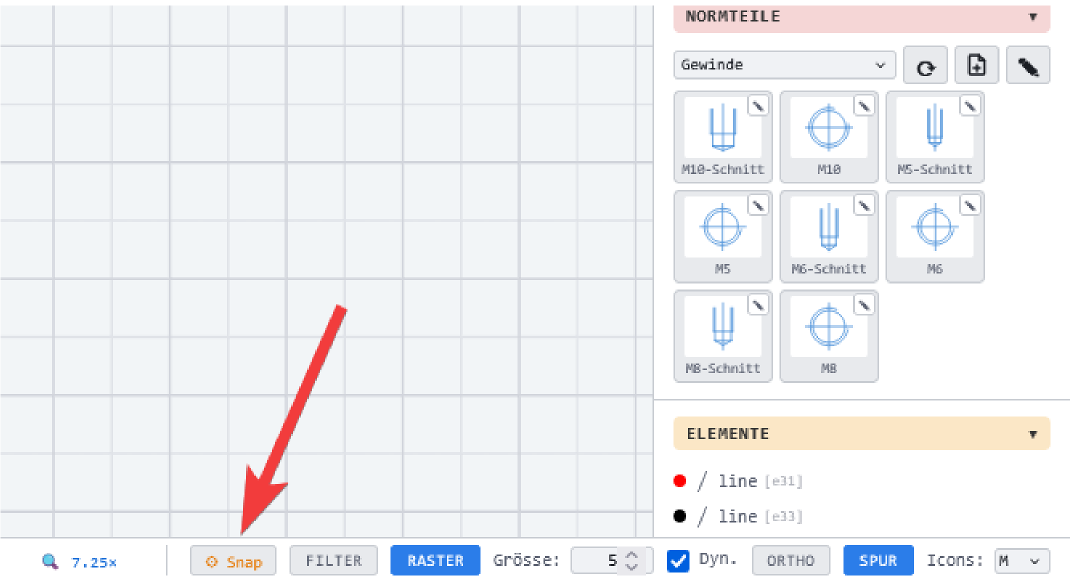
Keyboard shortcuts: Shift+T sets a temporary track point directly at the current cursor position; Shift+M snaps the midpoint between two points. The function is switched on and off with the **TRACK** button in the status bar.

Related terms: pilot lines

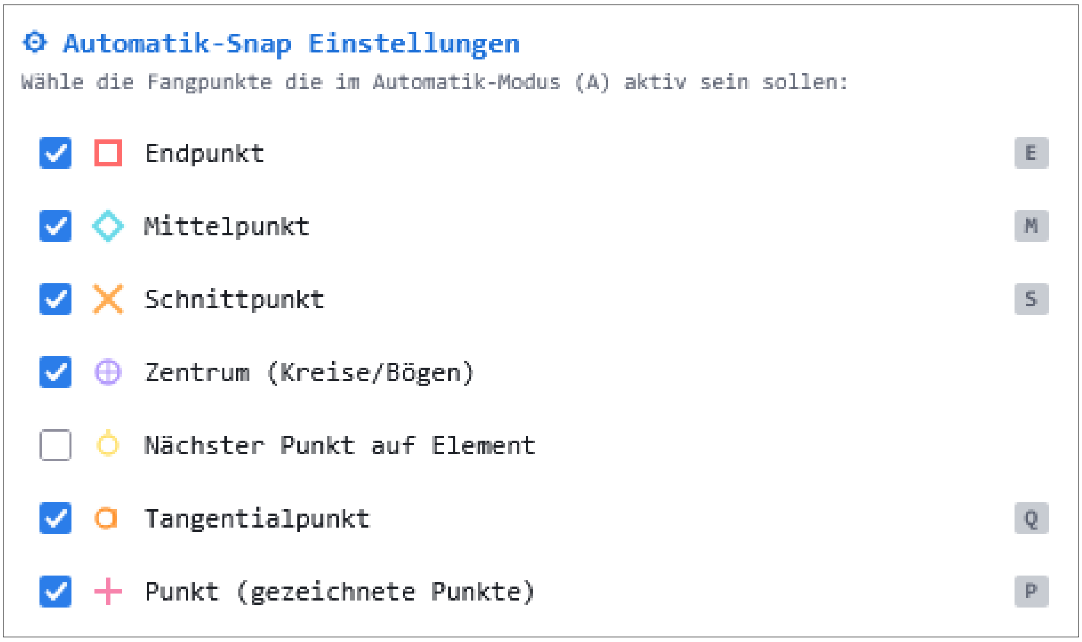
8.8 Automatic snap

When you draw your first lines, the CAD program is in grid mode, so that you can snap the origin precisely, for example.

After that it is a good idea to grab the snap points directly with the hotkeys:
[E] = endpoint
[S] = intersection
[Z] = center of a circle or arc
[F] = free
[G] = grid or raster
[Q] = automatic snap



The automatic snap is called up in the status bar.

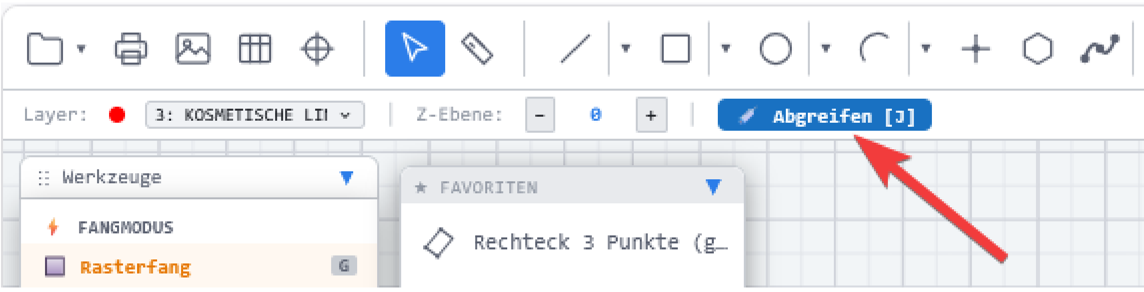


Here you define which points the automatic snap should catch.

8.9 Picking up attributes

This valuable and widely usable function is often overlooked.

The **Pick up [J]** function picks up the attributes of other drawing elements directly in the drawing while you draw:

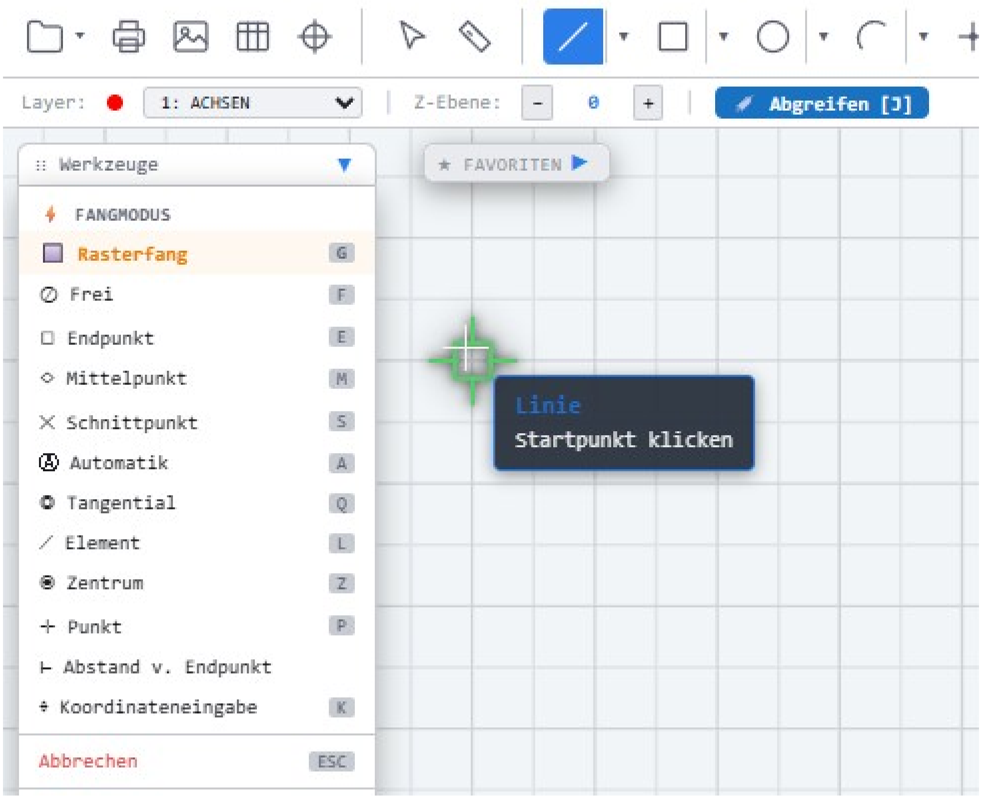


This brilliant function speeds up the entire design and modification process enormously.

8.10 So now draw your first element

Open **File → New drawing** in the menu.
Choose the **Line** function.

The **Grid** snap mode is active automatically and can be switched over directly with the hotkeys ([**F1**] shows an overview) . Alternatively you can also switch the snap functions in the left-hand **Tools** panel:

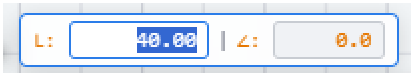


So go ahead and draw away, and watch how the snap functions latch onto all the relevant points automatically.

Use the [**ESC**] key to interrupt a function, or click the **right mouse button** to see further options.

INFO:

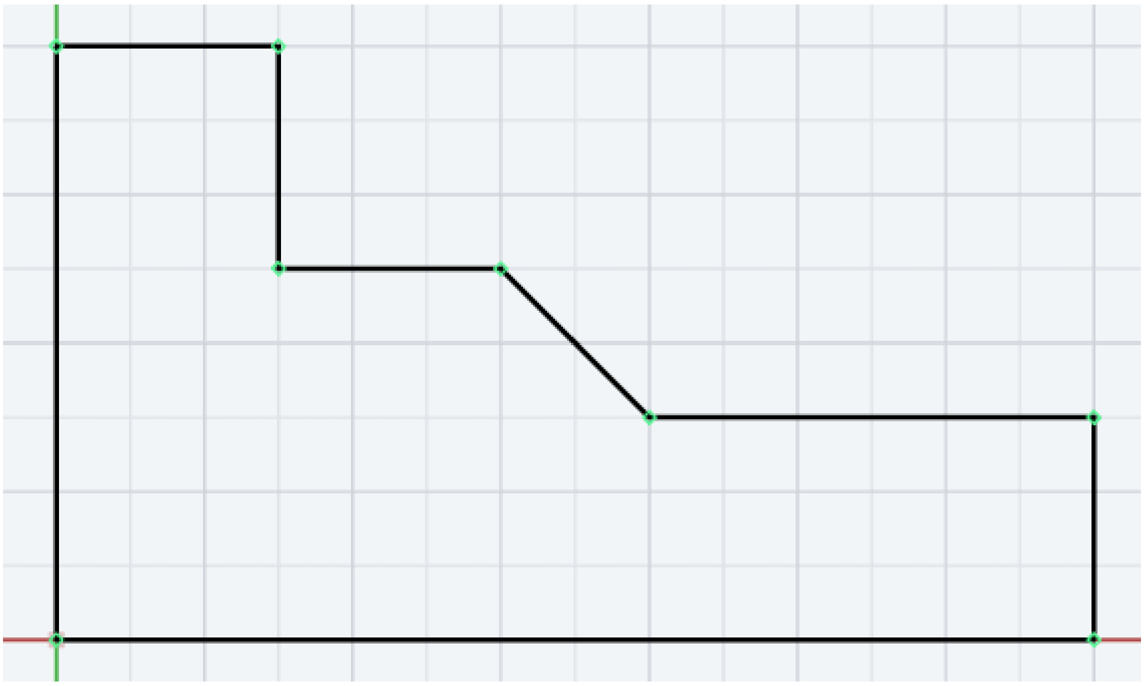
When you draw elements, a neat dimension menu always flies along in which you can enter the values you want. When **drawing a line**, for example, the **length** and the **angle** of the line are entered:



The [**spacebar**] switches over to X and Y coordinates:

This makes you completely flexible in the drawing process! Excellent!

You can simply draw a shape like this on the grid:

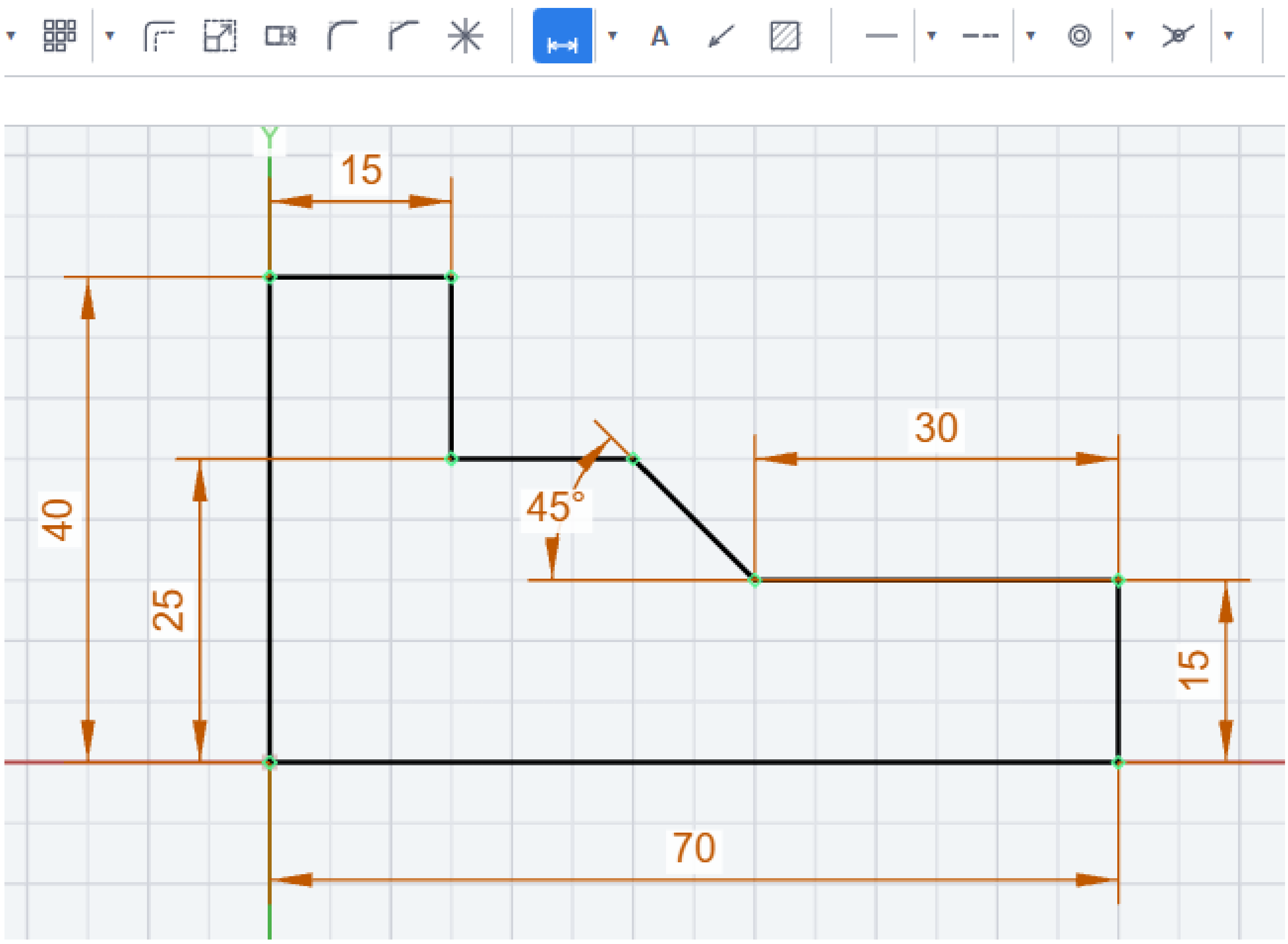


Then place a few dimensions with the dimension function.

For your information: there is only one dimension function, and it can dimension everything.

It is best to switch straight away to the **Automatic** snap mode with hotkey [**Q**].

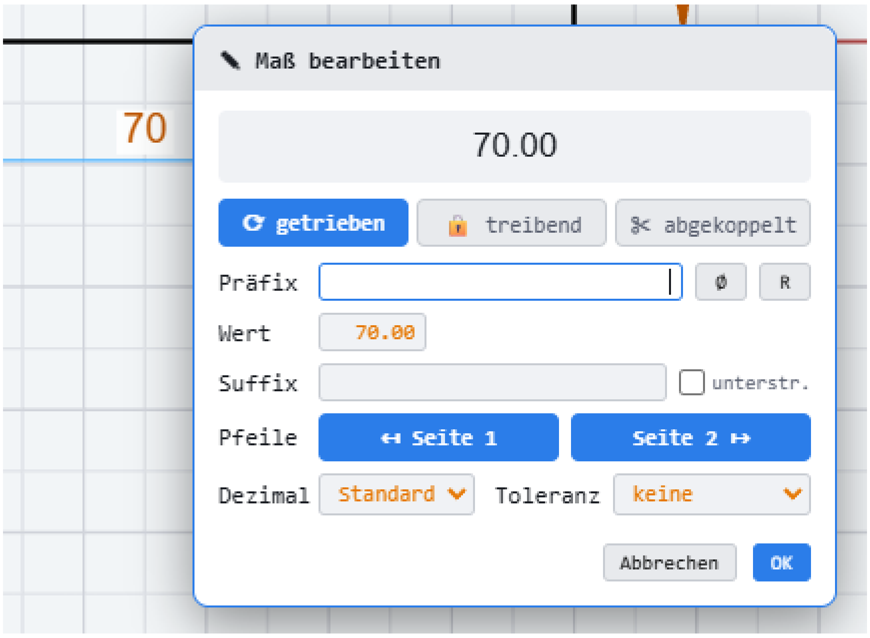
Simply dimension the elements like this:



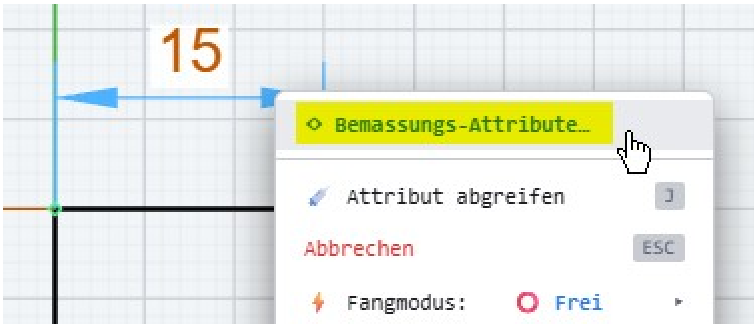
Go back to select mode with [**ESC**].

A **double-click** on a dimension shows this dialog→

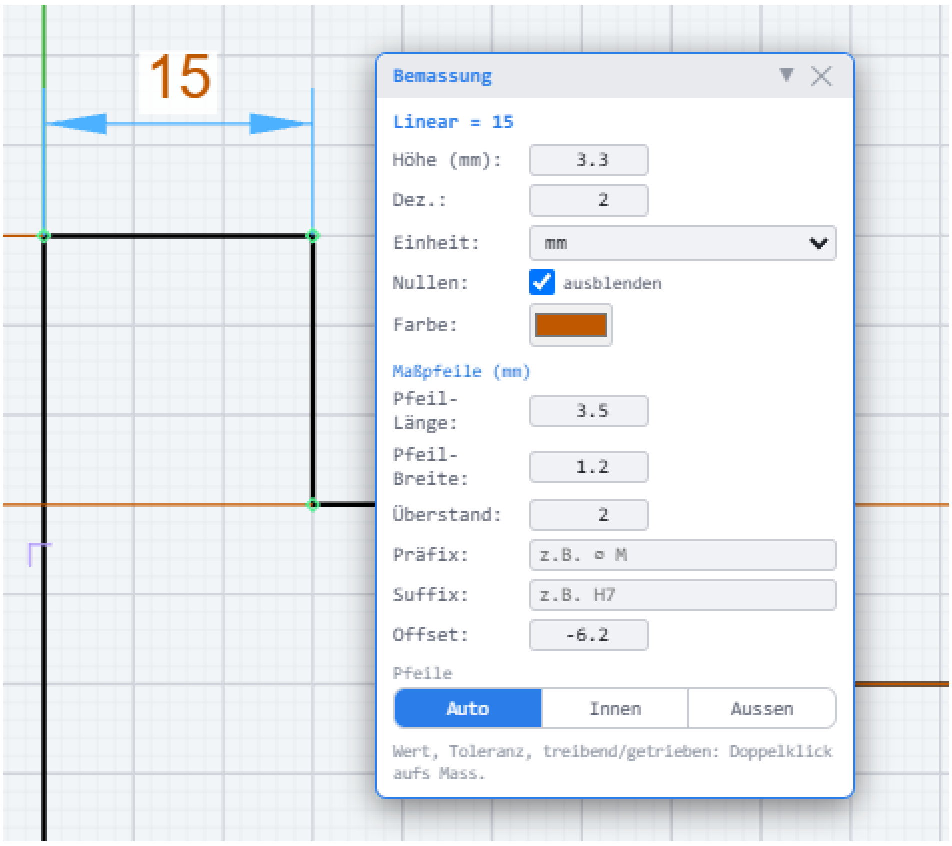
Here dimensions can be adjusted individually.



A **right-click** on a dimension shows the following menu:



It lets you set the dimension attributes individually for each dimension:

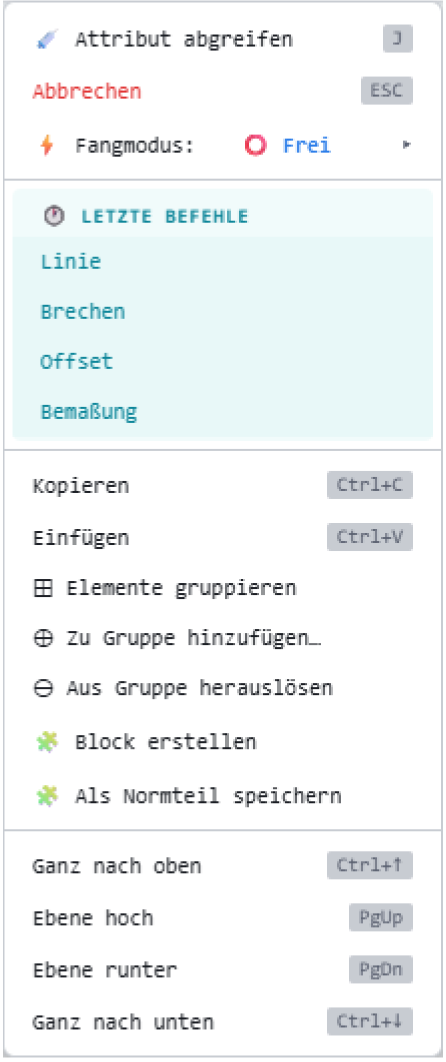


Further on in the book there are many example drawings for you to copy.

8.11 Right-click shows the most recent functions

While designing you often have to change functions, choose others and go back again, which can require a lot of clicks.

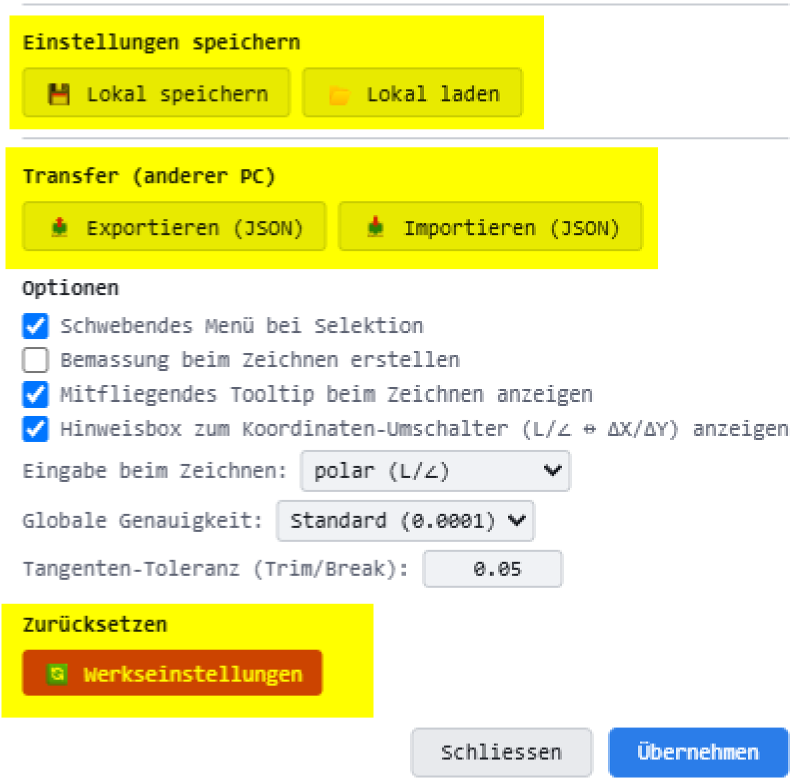
That is why the right mouse button menu contains the most recent commands under **RECENT COMMANDS**, in the order in which you called them up. The function used last is therefore always at the top.



9 Factory defaults

If you have once "messed up" the settings in the CAD program, you can reset the CAD program to the factory defaults.

In the menu at the top right, click **Setup**:



Save settings

Transfer settings

With these functions you can **save** and **load** your settings, and also **transfer** them to a new computer.

9.1 Calling up the hotkeys with [F1]

Tastaturkürzel — Schliessen (Esc)

WERKZEUGE

V	Auswählen
I	Messen
N	Linie
R	Rechteck
C	Kreis
Y	Polylinie
D	Bemassung
X	Text
T	Trimmen
B	Brechen
U	Runden / Fillet
O	Offset
Ctrl+M	Verschieben

OBJEKTFANG

G	Raster
F	Frei
E	Endpunkt
M	Mittelpunkt
S	Schnittpunkt
Z	Zentrum
L	Element / Kante
P	Punkt
Q	Automatik
K	Koordinateneingabe (Startpunkt)

AUTOTRACK

Shift+T	Temporärer Spurpunkt
Shift+M	Mitte zwischen 2 Punkten

DATEI

Ctrl+N	Neue Zeichnung
Ctrl+O	Öffnen
Ctrl+S	Speichern
Ctrl+Shift+S	Speichern unter
Ctrl+P	Drucken

BEARBEITEN

Ctrl+Z	Rückgängig
Ctrl+Y	Wiederholen
Ctrl+X	Ausschneiden
Ctrl+C	Kopieren
Ctrl+V	Einfügen
Ctrl+A	Alles auswählen
Entf / ⌫	Löschen
Esc	Abbrechen

ANSICHT & ANZEIGE

W	Fenster-Zoom
A	Zoom einpassen / Alles zeigen
H	Herauszoomen
F5	Neu zeichnen
Ctrl+F	Funktion suchen
F1	Tastaturkürzel ein / aus
Ctrl+Shift+G	Gizmo ein / aus

STAPEL (Z-REIHENFOLGE)

Bild ↑	Eine Ebene nach oben
Bild ↓	Eine Ebene nach unten
Ctrl+↑	Ganz nach oben
Ctrl+↓	Ganz nach unten

BEIM ZEICHNEN / EINGABEFELD

Enter	Bestätigen
Tab	Feld wechseln
Leertaste	Polar / kartesisch umschalten
K	Startpunkt per Koordinate

NORMTEIL PLATZIEREN

R	Drehen +90°
Shift+R	Drehen -90°
H	Horizontal spiegeln
V	Vertikal spiegeln
+ / -	Vergrössern / Verkleinern
B	Basispunkt setzen

DRUCKVORSCHAU

H	Hochformat
Q	Querformat
Esc	Schliessen

Einzeltasten wirken ohne aktives Eingabefeld; Objektfang-Tasten (F/E/M/S/Z/Q/L/P/A/G) auch im fliegenden Eingabefeld. Die Gruppen «Normteil platzieren» und «Druckvorschau» gelten nur im jeweiligen Modus. **Maus:** Rad = Zoom · mittlere Taste oder Alt+Ziehen = Verschieben · Rechtsklick = Kontextmenü.

Object snap keys (F/E/M/S/Z/Q/L/P/A/G) also work in the floating input field.
The groups «Place standard part» and «Print preview» apply only in the respective mode.
Mouse: wheel = zoom · middle button or Alt+drag = pan · right-click = context menu.

10 Starting the program

Start LiberoDraft directly from here:
https://www.LiberoDraft.com/ → **start LiberoDraft now**

10.1 Resetting LiberoDraft

To make sure that LiberoDraft works optimally with this book, we recommend that you reset the software to the factory defaults.
In the menu at the top right, click:

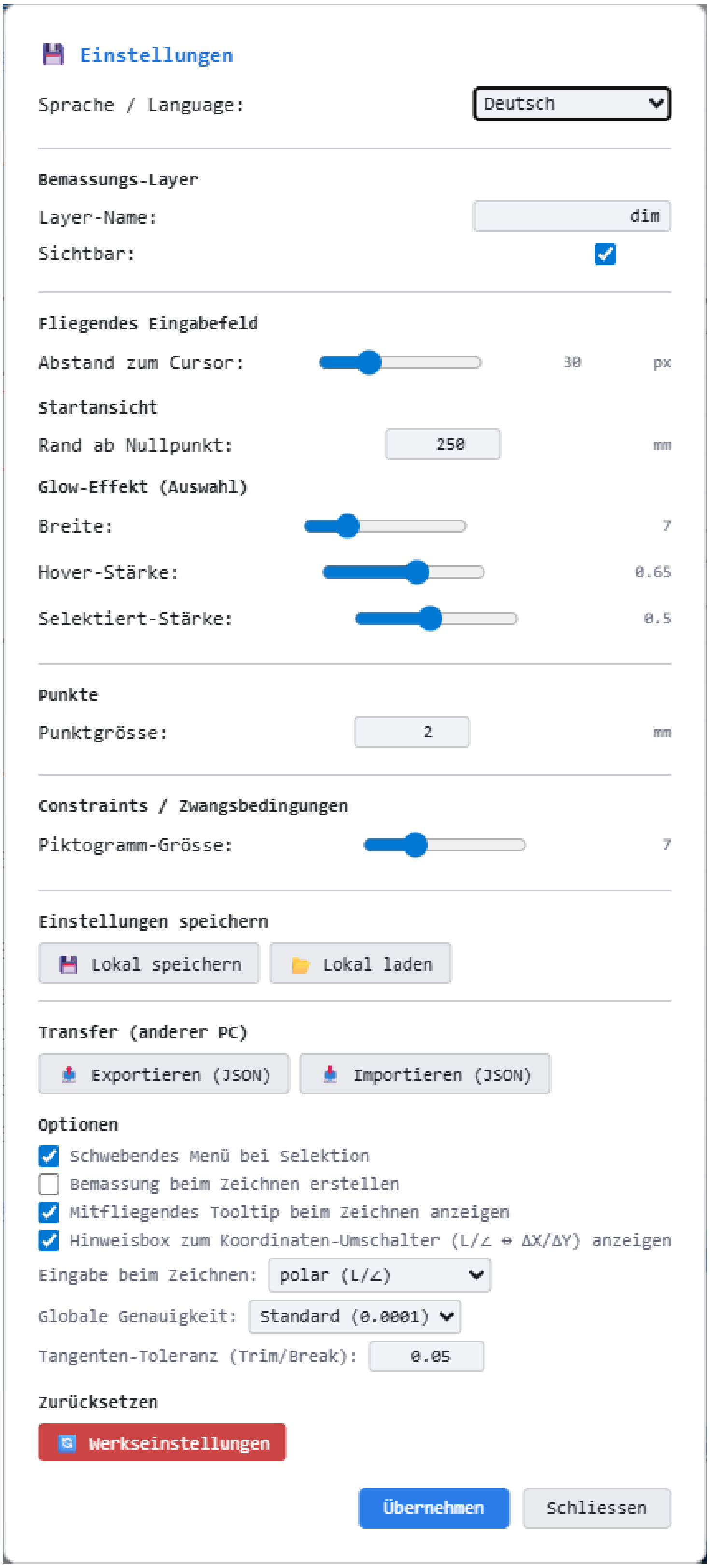
Setup -> **Factory defaults**

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

Moving settings: the settings can be transferred from one PC to another PC with the two transfer buttons.

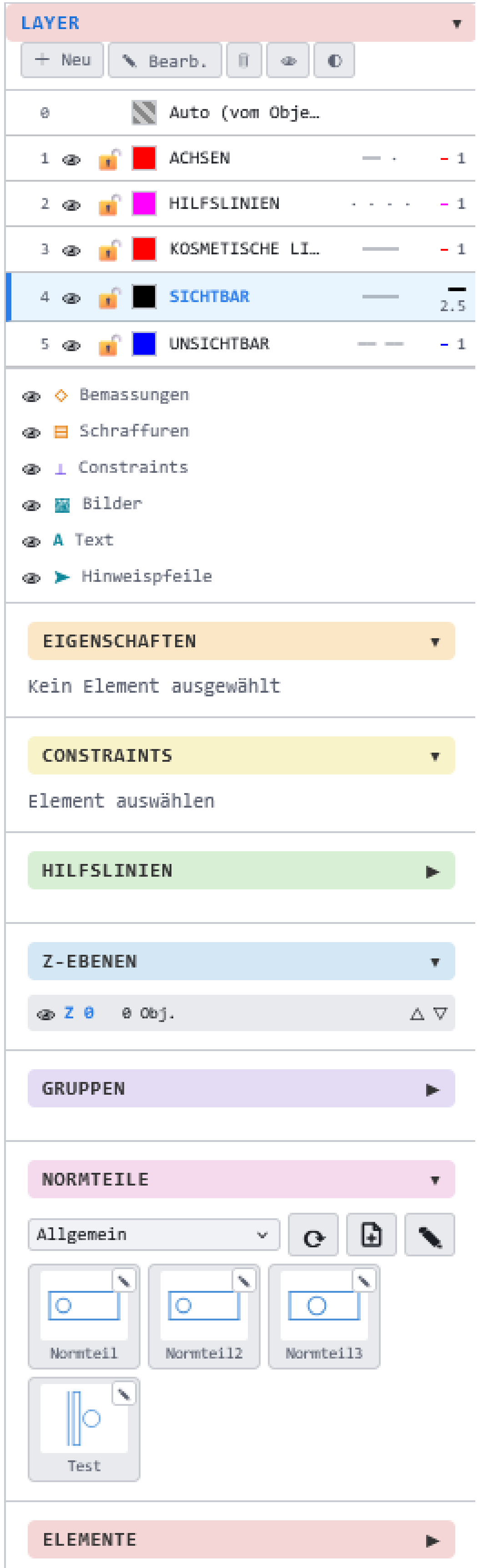
In summary, the "**Factory defaults**" function in LiberoDraft allows the user to restore the program's default settings by deleting all user-defined settings and customizations.

This is useful when the user has made changes that they would like to undo.



11 Program panel on the right

The right-hand panel in LiberoDraft serves as the central management and information bar for the drawing. Among other things, it displays layers, properties, constraints, Z levels, groups, standard parts and elements. With groups, this panel is particularly helpful for keeping an overview of the existing groups in the drawing and managing their contents more easily.



Layer functions are described further on in the Layers chapter.

Element toggles: quickly showing and hiding element groups. A click on the "eye" hides and shows whole element groups. If a drawing is saved with groups switched off, this is remembered when it is loaded again.

Element properties for selected elements.

Constraints and dependencies (experimental)

Guides are infinite and important.

Z levels for bringing forward, for example, the outline of a filled area.

Group functions are described further on in the Groups chapter.

Standard part management for managing symbols and reusable blocks.

11.1 Layers

One of the most important functions of LiberoDraft is the use of layers. They help to organize drawings by allowing the user to place and manage related elements together.

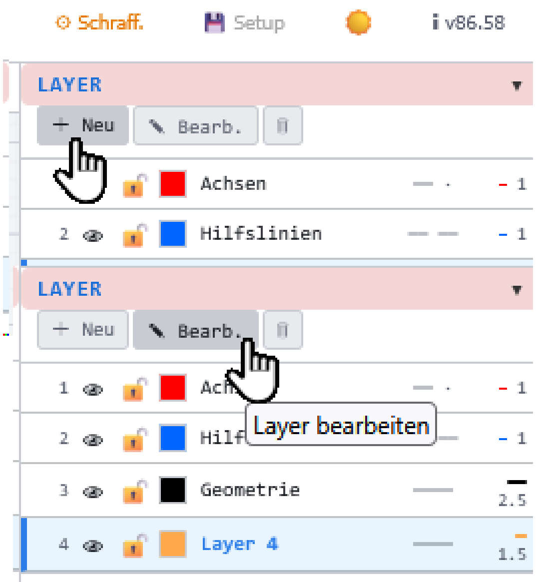
Traditional manual drawing used a similar approach. The layers were often drawn on separate transparent sheets. These sheets were then laid on top of each other to create the final drawing.

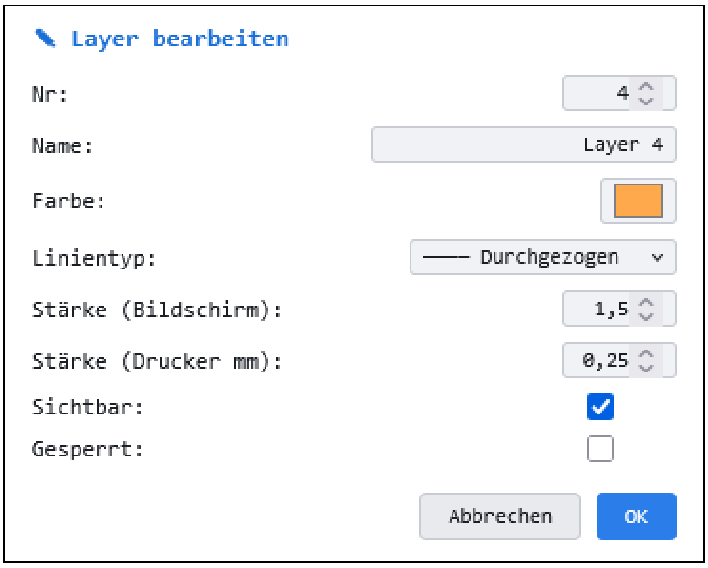
Whether in engineering, architecture, construction, manufacturing or other fields, layers were used to represent different aspects of the drawing. Layers can be added to show center lines or dimensions on technical or manufacturing drawings, or to show different building systems on architectural drawings, such as exterior walls, partition walls, electrical, HVAC, grid lines and so on.

Layers have the additional advantage that all pen attributes can be assigned to a layer. Every element on that layer adopts the attributes assigned to that layer. The attributes assigned by the layer can, however, be overridden for individual elements when required.

Layers are normally created in order to store line types with attributes. Creating a layer is easy:

- Click on **+ New**
- A new "Layer 4" is appended automatically.
- Click on **Edit**
- Define the parameters you want
- Click on **>Ok<**.





11.2 Element toggles

In the right-hand panel, various special drawing elements are clearly grouped into their own sections. These include dimensions, hatches, constraints, images, text and leaders. This division helps you keep an overview even in larger drawings and to control specific element types in a targeted way.

The eye symbol switches the visibility of the respective elements on or off. This lets you temporarily hide dimensions or hatches, for example, without deleting them from the drawing. That is particularly useful if you want to concentrate on the pure geometry while designing.

The individual sections therefore act as quick management for additional drawing information. Texts, images, notes and geometric dependencies stay organized separately and can be displayed, checked or edited selectively when needed. This makes working with more complex CAD drawings in LiberoDraft considerably clearer.

11.3 Properties

The **Properties** section shows the most important data of the currently selected drawing element. For a line these are, for example, the coordinates of the start and end point, the length and the angle. These values can be checked directly and changed if required. Elements can therefore be edited not only with the mouse, but also very precisely via numerical values.

In addition, organizational properties such as **layer**, **line width** and **Z level** can be adjusted here. The layer determines which drawing level the element belongs to. The width influences how the line is displayed. The Z level determines whether an element lies in front of or behind other elements.

The properties panel is therefore an important tool for precise drawing and for subsequently editing CAD elements in LiberoDraft.

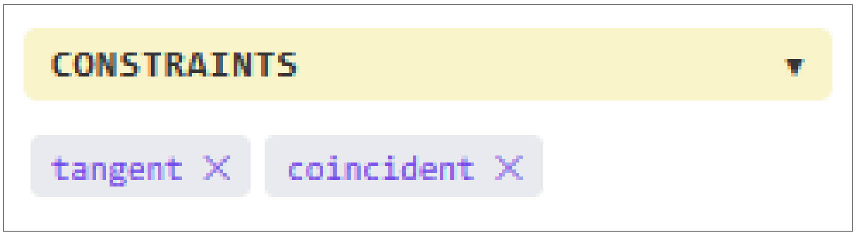
11.4 CONSTRAINTS

Constraints in the right-hand panel

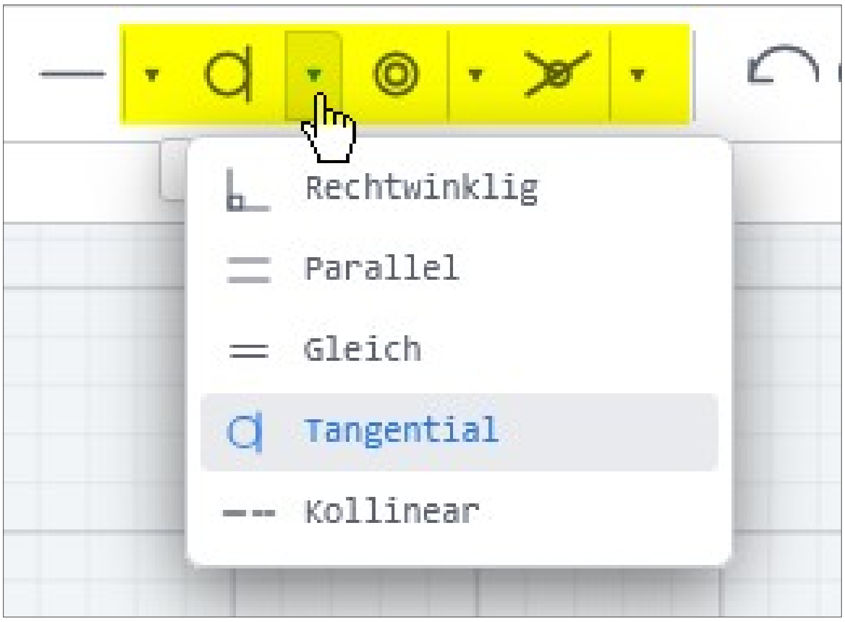
The **Constraints** section shows the geometric relationships of the selected element. Constraints ensure that certain properties or dependencies between drawing elements are preserved. The design therefore stays controlled and geometrically correct even when changes are made.

Examples of such dependencies are **tangent** and **coincident**. A tangential relationship makes two elements meet in a touching manner, for example a line on a circle. A coincident relationship means that points or elements lie exactly on top of each other or are joined together.

The constraints shown in the panel can be checked directly and removed if required. This makes it easy to see why an element behaves in a certain way or why it cannot be moved freely. Constraints are therefore particularly helpful for building precise and stable CAD geometry.



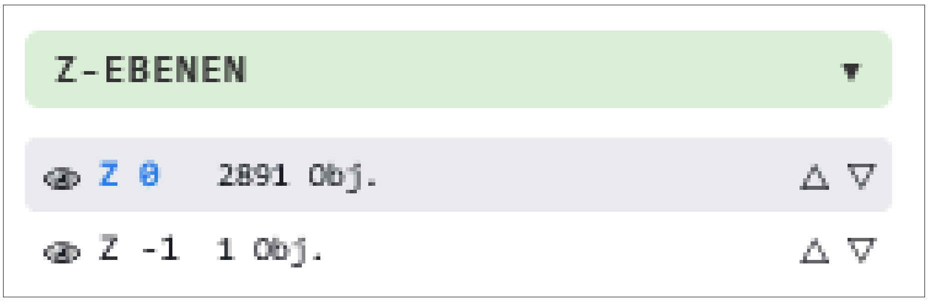
The active constraints (relationships) assigned to the selected element are shown here. A constraint can be removed selectively with the X*.*



Available constraints

11.5 Z levels in the right-hand panel

The Z levels section shows on which depth or display levels the drawing elements are located. This function is particularly important when elements overlap in the drawing. The Z level determines which object visually lies in front of another object and which one is shown behind it.



In the list you see the entries Z 0 and Z -1, for example. Behind them, the number of objects on that Z level is displayed. In the example shown, 2891 objects are on level Z 0 and 1 object is on level Z -1. This immediately shows how the drawing is organized spatially, or rather in its display order.

In many cases Z level 0 is the normal working level for most drawing elements. Elements with a lower Z level, for example Z -1, can lie behind other objects. That is helpful when images, background elements, hatches or auxiliary geometry should not appear in the foreground.

The eye symbol switches the visibility of an entire Z level on or off. This lets you quickly hide certain display levels without deleting the objects they contain. That is particularly useful in extensive drawings when you briefly want to check only certain areas or object types.

The difference between layers and Z levels is important:

A layer serves above all to organize a drawing by discipline or subject, for example for geometry, dimensions, center lines or auxiliary constructions. The Z level, on the other hand, governs the display order. It determines whether an element appears in front of or behind another element.

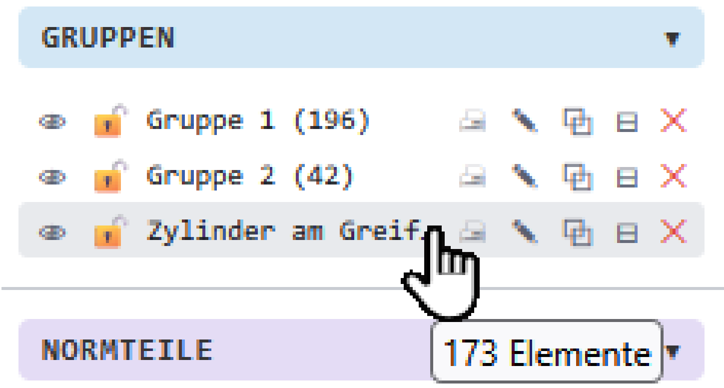
Z levels therefore help to improve the overview in complex drawings and to control overlapping objects in a targeted way. This function is very helpful especially with imported drawings, images, hatches or large assemblies, because the visible order of the elements can be checked and adjusted quickly.

11.6 Groups

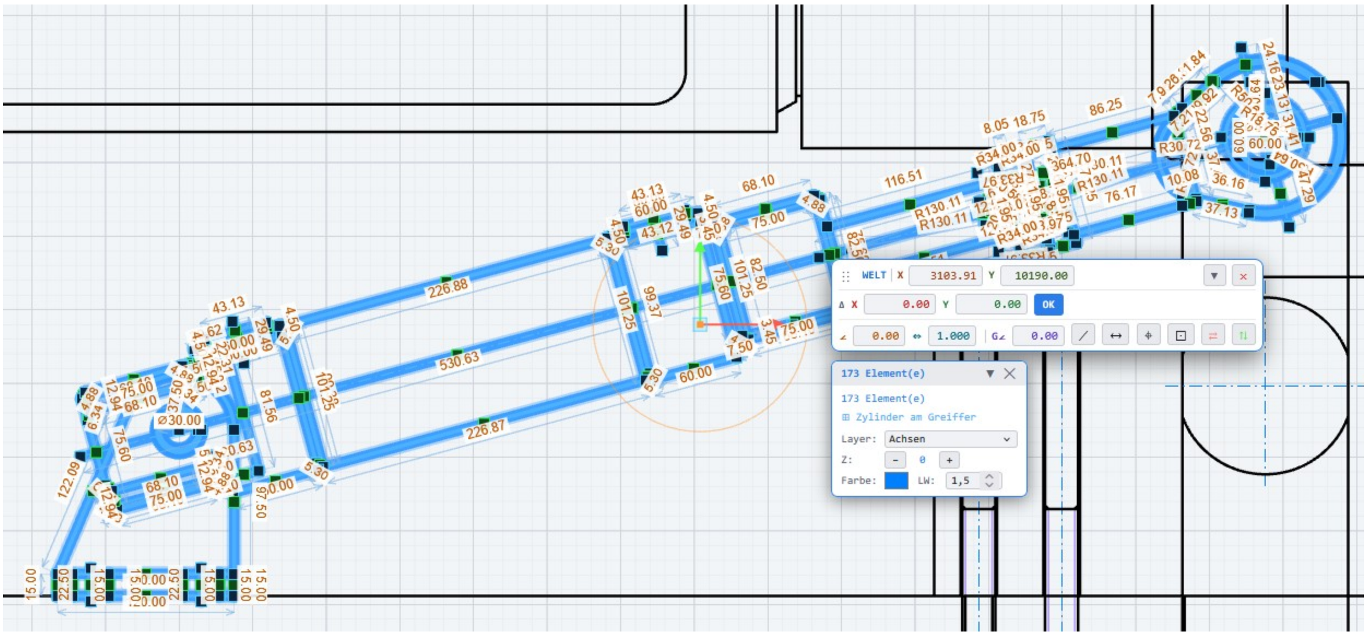
The **Groups** section lists all existing groups of the drawing clearly. Every group has a name and shows the number of elements it contains in brackets. You can therefore see immediately how many lines, circles, texts or other objects an assembly consists of.

The symbols next to each group let you carry out important functions directly. A group can be shown or hidden, locked against unintentional editing, selected, edited, copied or deleted. This makes it possible to structure even extensive drawings with many individual parts more effectively.

Groups are particularly suitable for recurring or related drawing areas, for example standard parts, assemblies, connections or complete design units. If a group is selected, the elements it contains can be moved, copied or edited together, without having to mark every element individually. That saves time and reduces errors when editing complex CAD drawings.



Groups let you combine several drawing elements into a single unit. That is particularly practical when related lines, circles, texts or symbols are to be moved, copied, scaled or managed together. In LiberoDraft, elements can be grouped, added to an existing group or removed from a group again. The drawing therefore stays clear, and recurring design parts can be edited more quickly.



On request, selected elements are dimensioned automatically and can be edited via a dedicated LiberoDraft gizmo.

11.7 The "Blocks" section in the right-hand panel

Detailed description of the functions, as of version 1.3.22

11.8 1. What a block is

A **block** is a reusable symbol or component: a welded part, a connection detail, a circuit symbol, a screw. LiberoDraft cleanly separates two things here:

- The **block definition** contains the geometry once — in block-local coordinates whose origin is the insertion point. The definition also includes the parts-list data (designation, article number, material and so on).
- An **instance** is a single occurrence of that block in the drawing. It only remembers which definition it points to, where it is located, and how it is rotated and scaled.

That gives the biggest advantage: a hundred instances take up hardly more space than one, and **a change to the definition affects all instances at once**. Blocks may be nested inside one another — a block can contain other blocks; only not itself.

11.9 2. Structure of the section

The "Blocks" section is located in the right-hand panel between "Groups" and "Standard parts". It consists of five buttons with the block list below them:

Schaltfläche	Wirkung
Block aus Auswahl erstellen	macht aus den markierten Elementen eine neue Blockdefinition
Block laden (.ldblk)	liest eine Blockdatei ein und stellt sie in dieser Zeichnung bereit
Stückliste (BOM)	öffnet die Stückliste aller platzierten Blöcke
Blockverwaltung	öffnet das grosse Verwaltungsfenster mit Baumansicht und Suche
Zusatzprogramme	öffnet die Zusatzprogramme von liberodraft.com (Zahnrad-, Verschraubungsgenerator und weitere)

As long as no block definition exists, the list shows the note "No blocks. Select elements → create block".

11.10 3. Creating a block

First mark the elements that are to belong to the block, then click **Create block from selection**. A small input field asks for the name; press Enter and the block is created.

The insertion point — the point at which the block later hangs on the mouse — is determined as follows:

1. If you have set your own pivot on the **Gizmo** , that pivot applies. *
2. Otherwise the **gizmo** center applies, provided the gizmo is visible.
3. Otherwise LiberoDraft takes the center of the selection.

So if you want to place the insertion point deliberately on a corner or the center of a hole, set the **gizmo** pivot there before creating the block.

Dimensions and hatches travel into the block: those explicitly marked in every case, and in addition all dimensions whose reference elements lie completely within the selection. A dimensioned detail therefore stays complete when it is inserted.

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* You will find further information about the **gizmo** in the chapter:
Chapter 32 Topic: Gizmo Page (126)

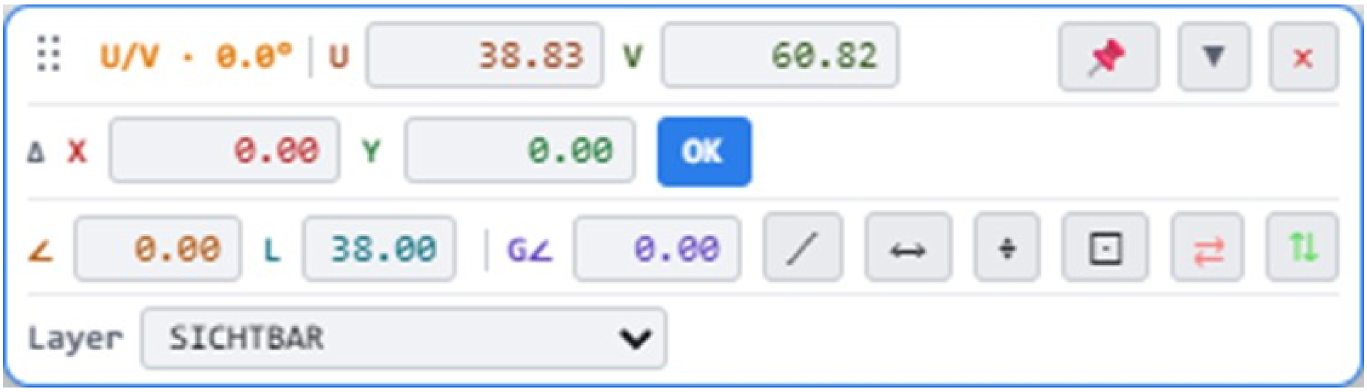


Figure: the LiberoDraft gizmo.

11.11 4. The block list

Every block definition occupies one row in the list. If a block contains further blocks, the row can be expanded via the **triangle** on the left — the sub-blocks appear indented below it.

The structure of a row from left to right:

Element	Bedeutung
▸ / ▾	Unterblöcke auf- und zuklappen (nur bei verschachtelten Blöcken)
👁 / 🚫	Block ein- oder ausblenden
🧩 Name	Klick auf den Namen startet das Einfügen
n×	Stückzahl (siehe unten)
+	Einfügen an der Maus
📐	Block-Geometrie im Editor bearbeiten
✏️	Stücklisten-Daten bearbeiten
💾	Block als Datei exportieren (.ldblk)
🗑	Blockdefinition löschen (nur wenn ungenutzt)

The way things are counted follows a deliberate rule: at the top level the number shows **how many times the block is placed in the drawing**; for an indented sub-block, on the other hand, it shows the **multiplicity within the parent block**. A block that is both contained in another block and placed independently in the drawing additionally appears at the top level, so that it never becomes unreachable.

Two aids to orientation: move the mouse over a row and all instances of that block are highlighted in the drawing. Conversely, if you mark an instance in the drawing, the corresponding row stands out with a blue bar.

11.12 5. Inserting a block

A click on the name or on **+** hangs the block on the mouse — similar to pasting from the clipboard. While placing it:

- **Click** sets an instance; you can set several one after another.
- **R** rotates, **H** and **V** mirror horizontally and vertically respectively.
- **+ / −** scale.
- **ESC** or a **right-click** finishes placing.

Mirroring is done via a negative scaling in exactly one axis. LiberoDraft cleanly separates magnitude and sign here: radii, text heights and arrows keep their size, directions of rotation reverse correctly — and **texts stay legible**, just as when mirroring individual texts.

11.13 6. Hiding blocks

The **eye symbol** hides an entire block definition. It then behaves like a hidden layer: the instances are not drawn, not snapped to, not clickable and do not bound any hatch — not even when they are contained in another block. That is well suited, for example, to making all screws disappear temporarily for a printout. The drawing is of course saved complete.

11.14 7. The block editor

The editor edits the **geometry of the definition** and therefore all instances at once. After it starts, the drawing area switches into the block: your drawing is set aside together with its own undo history, and you see only the block geometry — in local coordinates, with the **orange origin as the insertion point**.

A banner at the top edge of the screen indicates that the editor is active, names the block and offers **Save** and **Cancel**:

- **Save** adopts the geometry, increments the block version and updates all instances — including those in other blocks.
- **Cancel** discards all changes.

In the editor all drawing tools are available to you, including dimensioning and hatching. A block cannot contain itself; LiberoDraft refuses to insert the block currently being edited and displays a note.

11.15 8. BOM data (✎)

Here you enter the commercial and technical details for the **definition** — they therefore apply to all instances:

Block name · designation · article number · standard · material · unit (default "pcs") · weight · supplier · price · remark.

In addition there is the switch **"List as a single part in the BOM"** (switched on by default). It decides how a block is treated that itself contains blocks: switched on, it counts as *one* assembly; switched off, it is broken down and its individual parts appear in the parts list. For checking, the head of the window shows how many times the block is placed in the drawing.

11.16 9. The parts list

The parts list evaluates all placed blocks — including the blocks inside blocks, with the correct multiplication. It knows two views:

- **Quantity BOM** — flat, every item with its total quantity; identical article numbers are combined.
- **Structured BOM** — hierarchical, with item numbers such as 1, 1.1, 1.2, corresponding to the structure of the assemblies.

The columns are No., Qty, Unit, Designation, Article number, Standard, Material and Remark. Blocks without stored data are listed in gray with the note "no BOM data" — so you see immediately where something is still missing.

At the foot of the window there are two output paths:

- **Insert into drawing** places the parts list as a table in the drawing.
- **Export CSV** writes `stueckliste.csv` — semicolon-separated and with a UTF-8 marker, so that Excel displays the umlauts correctly.

To match this, the toolbar offers the **item balloon**: a circle with the parts-list item number that you attach to a block instance. The number comes from the same evaluation and therefore stays consistent with the parts list.

11.17 10. The block manager

The management window is the big overview. It shows all definitions as a **tree** — including unused ones that are not placed anywhere — and can be moved freely by its title bar; the position is retained for next time. The drawing underneath stays operable; the window only closes via "Close" or ✕.

- **Search field** with a hit counter; with the **Parts-list data** check box, the article number, material, standard, supplier, weight, price, unit and remark are searched as well. **Show all** clears the search again.
- The buttons at the bottom: **Insert · Show · Edit · BOM data · Replace with … · Rename · Export · Delete**.

Delete only works on unused definitions — and tells you the reason if it does not work: either the block is still placed *n* times in the drawing, or it is contained in another block, which is named. The reason appears in the window itself, not only in the status bar.

11.18 11. What the context menu additionally offers

With the right mouse button on a block instance you reach four further commands:

- **Edit block** — opens the editor for the definition of this instance.
- **BOM data...** — takes you straight to the data sheet of this definition.
- **Split off as a variant** — the way out for the classic case "a hundred identical blocks, but three of them should be different": the selected instances receive a copy of the definition under the name "Name (2)", all the others stay untouched. The parts list separates them automatically, and an existing article number receives a suffix so that they are not combined again after all. With exactly one new variant, the editor opens right away.
- **Replace with another block...** — swaps the selected instances for a different definition; position and rotation are retained.

11.19 12. Passing blocks on: export and import (.ldblk)

With you write a definition as an `.ldblk` file. **All nested sub-blocks** are taken along, as well as the layers used, with name, color, line type and line width — the file is therefore complete in itself.

Load block reads such a file back in, assigns the layers by their names and makes the block available in the current drawing. An existing block with the same name is **never overwritten**; instead LiberoDraft asks briefly. This way you gradually build up your own library of symbols and parts that you can use across drawings and workstations.

11.20 13. Practical tips

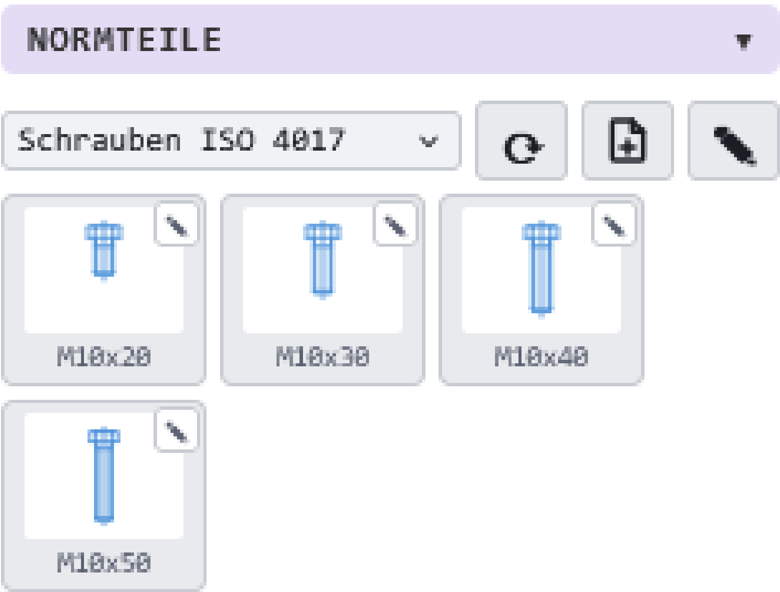
- Before creating the block, set the **gizmo** pivot * to the point at which the block is later to be "picked up" — that saves rework every time you place it.
- Maintain the parts-list data right when you create the block; the gray "no BOM data" row in the parts list reliably shows you where something is still missing.
- For assemblies it is worth looking at the switch "List as a single part" — it decides whether the parts list shows the assembly or its individual parts.
- Never change a definition when in fact only individual instances are meant to be different: that is what **Split off as a variant** is for.
- Hiding is a working aid for the view — the saved drawing always contains everything.

* You will find further information about the **gizmo** in the chapter:

Chapter 32 Topic: Gizmo Page (126)

11.21 Standard parts in the right-hand panel

The Standard parts section in LiberoDraft is used for quickly choosing and managing frequently used standard components. These include, for example, screws, nuts, washers or other recurring drawing elements that do not have to be constructed anew every time. In the example shown, the category "Screws ISO 4017" is selected.



Standard part management, server-based as well as local.

The selection list lets you switch between different standard part groups. This makes it possible to choose suitable components specifically by standard or component type. The tiles displayed below show the available variants within that group, for example M10x20, M10x30, M10x40 or M10x50. Every tile contains a small preview so that the desired standard part can be recognized quickly.

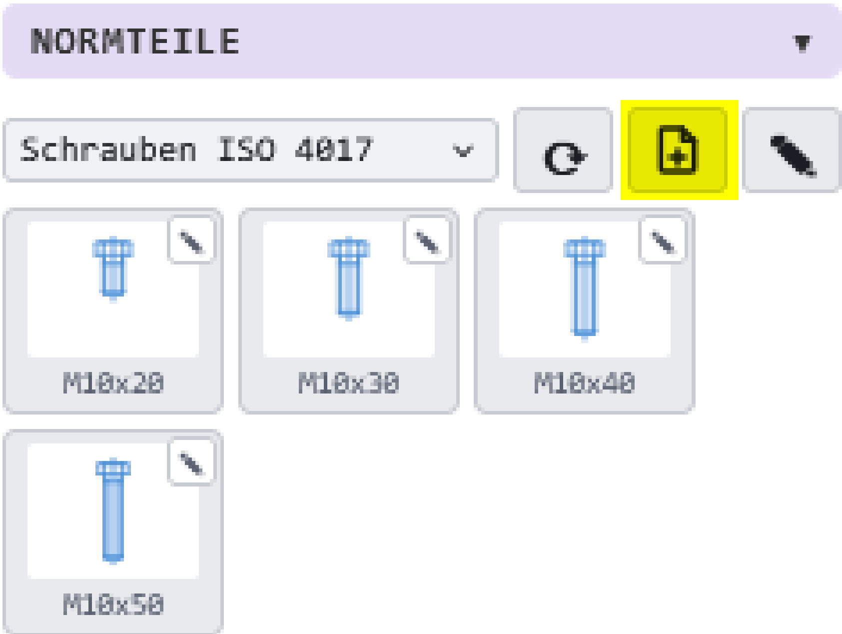
Standard parts can be inserted into the drawing directly from this section. That is particularly practical when the same or similar components are needed again and again in technical drawings. Instead of building up a screw manually from lines, circles or symbols, it is taken from the standard part library as a ready-made element. That saves time and ensures a uniform appearance.

The symbols in the upper area are used to manage the standard parts. Depending on the function, standard part groups can be updated, new standard parts added or existing entries edited. The library can therefore be adapted to your own requirements. Your own frequently used drawing elements can likewise be organized as reusable components.

This function is particularly helpful with recurring designs, assembly drawings, steel construction or mechanical engineering drawings. Screws, fasteners or other standard parts only have to be prepared cleanly once and are then available at any time. Drawings are therefore created faster, stay clearer and retain a consistent graphical quality.

The Standard parts section therefore makes LiberoDraft more than just a simple 2D drawing program; it also supports a structured and efficient way of working. Recurring components are stored centrally, found quickly and can be taken over into new drawings without much effort.

11.21.1 Local standard parts



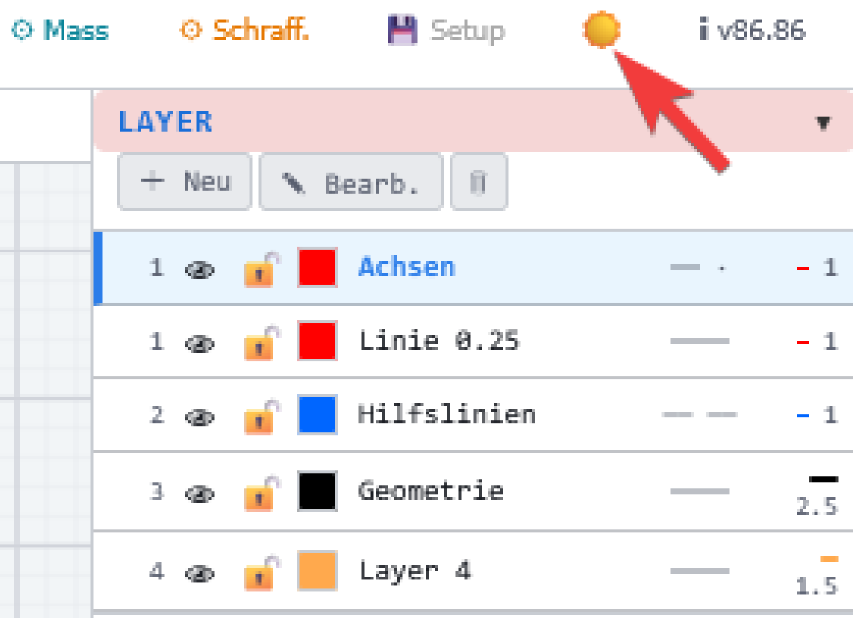
With the "Insert local standard part" function you can insert your own standard parts into a drawing. The standard part database is being extended continuously.

12 Basic settings

The following basic settings are likewise important in order to follow along in the book, and they are well suited to mechanical work.

12.1 Day/night switching

LiberoDraft has a practical day/night switch with which the appearance of the program interface can be adapted to the working environment. The symbol with the sun or moon switches between the light and the dark view. In day mode the interface appears light and high in contrast, while night mode works with a dark background.

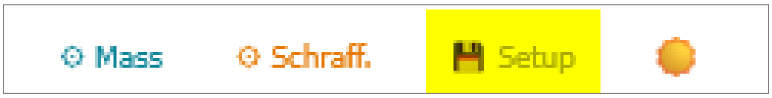


Day mode is particularly suitable for bright workplaces, daylight or presentations on screen. The controls, symbols and texts are clearly visible on a light background. This view corresponds to the classic appearance of many programs and feels familiar and clear.

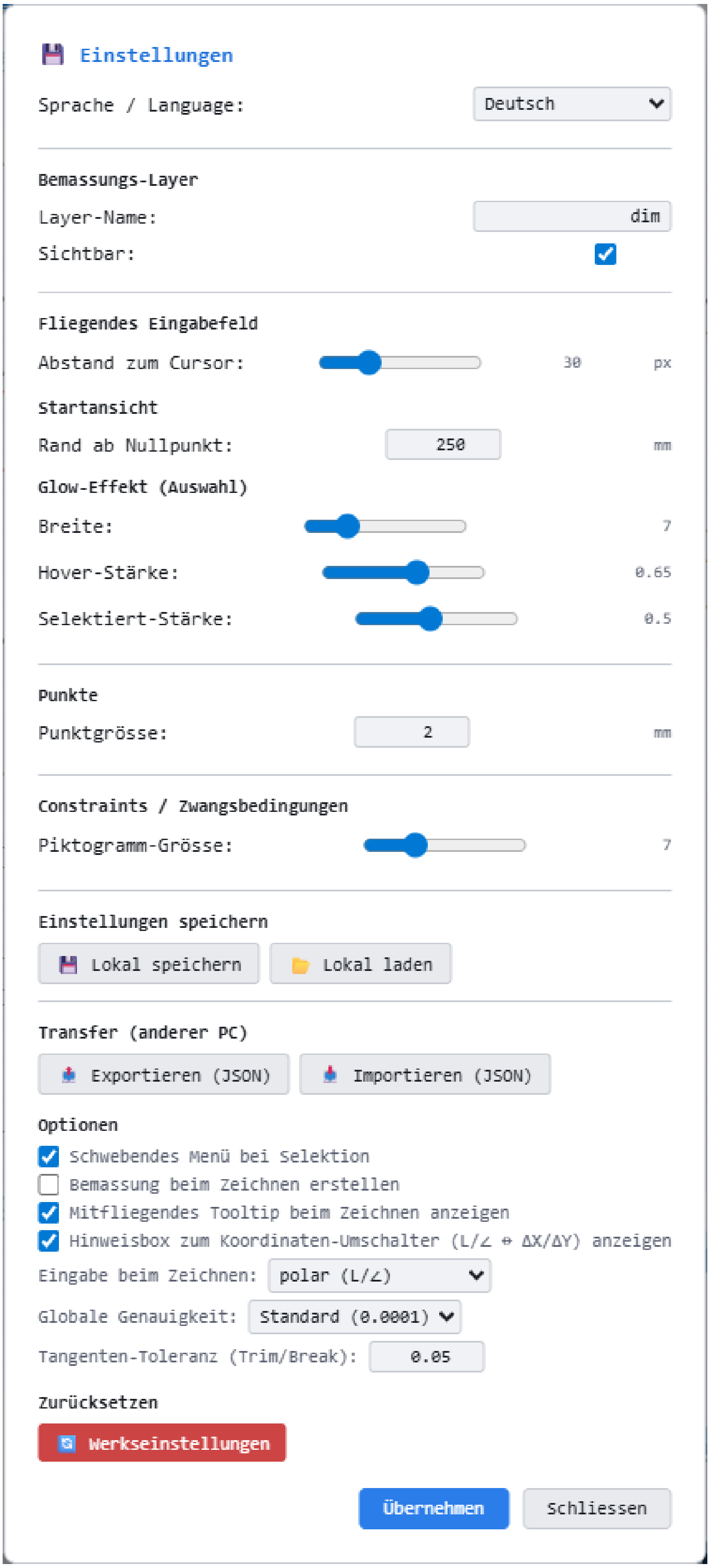
Night mode is above all pleasant during longer periods of screen work. The dark interface reduces the brightness of the screen and can therefore relieve the eyes, especially in darker rooms or when working in the evening. Tools, menus and status information remain clearly recognizable, but are shown on a dark background.

The switch affects above all the LiberoDraft user interface, that is toolbars, buttons, panel areas and display elements. Every user can therefore decide for themselves which appearance is more pleasant for the current working situation.

13 Setup and settings



Via the dialog:



Setup-Settings you adapt LiberoDraft to your personal way of working. Among other things, you define here how dimensions are managed, how strongly selected elements are highlighted and with which precision the program calculates. All changes only take effect when you click **Apply** at the bottom; **Close** exits the dialog.

13.1 Dimension layer

In this section you control on which layer the dimensions of your drawing are stored. The **layer name** field contains the name Mass by default. A separate layer for dimensions has the advantage that you can show and hide them independently of the actual geometry. The **Visible** check box determines whether the dimensions are currently displayed. If the box is ticked, they appear in the drawing; if you clear it, all dimensions are hidden without being deleted.

13.2 Floating input field

The floating input field is the small input field that appears directly next to the mouse pointer while drawing and in which you can type values such as lengths or angles. The **Distance to cursor** slider defines how far this field is displayed from the mouse pointer – 30 pixels in the example. A larger distance prevents the field from covering the drawing directly under the cursor; a smaller distance keeps input and cursor closer together.

13.3 Glow effect (selection)

So that you can see at a glance which elements you are currently touching or have selected, LiberoDraft highlights them with a glowing outline (glow effect). Three sliders control its appearance:

- **Width** determines how wide the glowing outline is drawn overall (7 pixels in the example).
- **Hover strength** defines how intensely an element glows as soon as you merely move the mouse pointer over it without having clicked it (1 in the example).
- **Selected strength** governs the glow of an element that is actually selected (0.9 in the example).

With these values you can make the highlighting more discreet or more conspicuous, depending on what is pleasant for your eyes and your screen.

13.4 Constraints

Constraints (geometric conditions) are rules with which you fix geometry geometrically – for example that two lines should be parallel or that a point lies on another line. Such conditions are shown in the drawing by small symbols (pictograms). The **Pictogram size** slider sets how large these symbols are displayed (7 in the example). On large or high-resolution screens a higher value can improve legibility.

13.5 Save settings

In this section you save your settings directly on the current device. With **Save locally** you store the current state in the browser or on the PC, so that it is available again at the next start. With **Load locally** you bring these stored settings back – for example if you have changed something in the meantime and would like to return to the saved state.

13.6 Transfer (another PC)

If you would like to transfer your settings to another computer, use this section. With **Export (JSON)** you save all settings in a JSON file that you can then copy to another device. There you read the file back in with **Import (JSON)**. This means you do not have to set up your preferred settings by hand at every workstation. (JSON is simply a standardized file format for exchanging such data – you do not need to worry about its content.)

13.7 Options

Here you find general settings for the behavior of the program:

- **Floating menu on selection:** if this check box is ticked, LiberoDraft displays a context-sensitive menu directly next to the selection when an element is selected, through which you can quickly reach suitable actions.

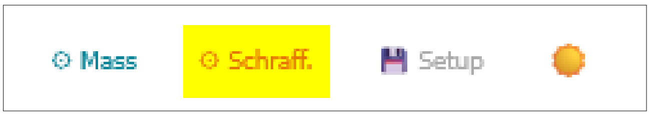
- **Create dimensions while drawing**: creates the finished dimension for every element drawn, directly while drawing. If this function is deactivated, the dimension can be added at any time with the dimension functions.
- **Show the floating tooltip while drawing:** if this check box is ticked, LiberoDraft displays a context-sensitive tooltip directly next to the selection when an element is selected, in which you see help on using the functions.
- **Show the notice box for the coordinate switch …:** displays a temporary notice box above the button at the bottom of the status bar. Particularly well suited to beginners.
- **Input while drawing:** defines the type of coordinate entry — polar via length and angle, or cartesian via the X and Y coordinates.
- **Global precision:** this selection field defines the precision with which the program calculates and processes values. The default setting Standard (0.0001) is suitable for most applications.
- **Tangent tolerance (trim/break):** this value (0.05 in the example) determines how generously the program treats tangential transitions when trimming and splitting (trim/break). It defines up to which deviation two elements still count as "touching".

13.8 Reset

The button **Factory defaults**, highlighted in red, resets all settings to the delivery state. That is helpful if you have "messed things up" and simply want to start again from scratch. Please note that your individual customizations are lost in the process.

Here is the description block for the Hatch settings dialog, in the same style and structure as before.

14 Hatching



With hatching you fill a closed area with a pattern (for example a set of lines) or with a solid fill. In LiberoDraft the function normally detects the boundary and any islands **automatically**: you simply click inside the area you want. This chapter describes the workflow and then explains every individual setting.

14.1 Hatching an area

Activate the hatch tool. The status bar shows the note *"click inside a closed area"*. Then proceed as follows:

- Use Hatch (hatch settings) to set the pattern and the options you want.
- With the hatch tool, click **inside the area** that is to be filled – not on a line, but in the free space between the lines.
- LiberoDraft determines the enclosing boundary, leaves closed islands free and creates the hatch.

The hatch is an element in its own right. It lies on the layer that has been set, can be selected, moved, copied and deleted again, and it is output when printing.

14.2 Automatic area and island detection

With automatic detection, LiberoDraft traces the nearest closed boundary starting from the click point and then determines all closed elements lying completely inside it as **islands** (holes). Islands are left free provided the "Exclude islands" option is active. For this to work reliably, the boundaries should be cleanly closed – ideally welded at the corners. Small gaps are bridged by the tolerance.

The detection deliberately treats a few special cases in a particular way:

1. **Features on the boundary (notches):** a feature that is connected to the outer boundary at two places (for example a groove or a circle protruding beyond the boundary) is treated as a notch in the boundary – the hatch leaves that place free.
2. **Spur and center lines:** an open line that runs into an inner shape via a single web does not tie that shape to the boundary. Such "bridges" are ignored and the inner shape remains an island in its own right.
3. **Overlapping islands:** if several islands intersect (for example a circle and a rectangle), they are combined into one common hole and cleanly left free – also with the SOLID full-area hatch.

4. **Point markers:** very small circles (up to the point marker radius that has been set) count as construction or point markers and are skipped during detection.

14.3 Manual area definition

If the automatic detection does not recognize an area or island for once – for example with heavily nested or deliberately open boundaries – switch to **Manual** under "Area definition" in the hatch settings:

- Click the boundary elements that form the outline (and the islands) one after another. The snap mode changes to FREE while doing so.
- Confirm with **Enter** or "✔ Hatch" – the area is formed from the selected elements and filled.
- With **ESC** or "✕ Cancel" you discard the manual selection.

14.4 The hatch settings

You open the dialog with the ⚙ **Hatch** button. All changes take effect **immediately** – there is no need to confirm them separately, OK merely closes the dialog.

14.4.1Hatch patterns

LiberoDraft comes with the five most important basic patterns:

Pattern	Type	Description
ANSI31	Lines	Parallel set of lines at 45°, line spacing 5 mm. The classic steel/general hatch and the default.
ANSI32	Lines	Like ANSI31, but tighter (line spacing 2.5 mm). For small areas or a denser appearance.
ANSI37	Lines	Parallel set of lines at 135° (opposite direction to ANSI31). Practical for visually distinguishing adjacent components.
CROSS	Cross	Cross-hatch made of two sets of lines crossing at right angles.
SOLID	Solid	Opaque solid fill of the area instead of individual lines. Islands stay excluded.

The pattern properties (angle, spacing, color, line width, line type) are **global**: if you change the angle of ANSI31, for example, the appearance of every area hatched with ANSI31 in the drawing changes immediately.

14.4.2Overview of all settings

Setting	Meaning
Pattern	Selects one of the hatch patterns described above. The default

Setting	Meaning
	is ANSI31.
Angle	Inclination of the set of lines in degrees (only with line and cross patterns). 0° = horizontal, 90° = vertical. With CROSS the angle applies to the first set; the second one is perpendicular to it.
Spacing	Distance between two adjacent hatch lines in millimeters. Smaller values produce a denser hatch. Has no meaning with SOLID.
Color	Line or fill color of the hatch. The default is the CADTEC orange.
Line width	Display width of the hatch lines in pixels. Purely visual; it does not change the geometry.
Line type	Appearance of the hatch lines: solid, dashed or dotted.
Layer	Layer on which new hatches are placed. This allows hatches to be shown, hidden or protected together.
Z level	Drawing order (stacking height). Negative values place the hatch BEHIND the boundary lines, so that the geometry stays visible on top. Default −1.
Tolerance	Snap tolerance of the area detection in millimeters (default 0.5). It determines at which distance two boundary ends still count as belonging together, and whether a line merely touches a curve (tangentially) or passes through it. Larger values close small gaps in the boundary; values that are too large, however, can merge closely adjacent boundaries.
Point markers r ≤	Small solid circles up to this radius are treated as point or construction markers and ignored during area detection (not traced around). This keeps the detection from "stumbling" over a center point lying on an edge. 0 = consider all circles. Default 1.0.
Opacity	Transparency of all hatches from 10 % (strongly translucent) to 100 % (fully opaque). Takes effect immediately on the whole drawing and is useful for keeping the geometry underneath visible.
Exclude islands	If the option is active, closed elements inside the area (holes, cut-outs, components) are automatically left free as islands. Switched off, the entire area is hatched continuously.
Area definition	Switches between automatic detection (click inside the area)

Setting	Meaning
	and manual selection of the boundary elements (see the sections below).

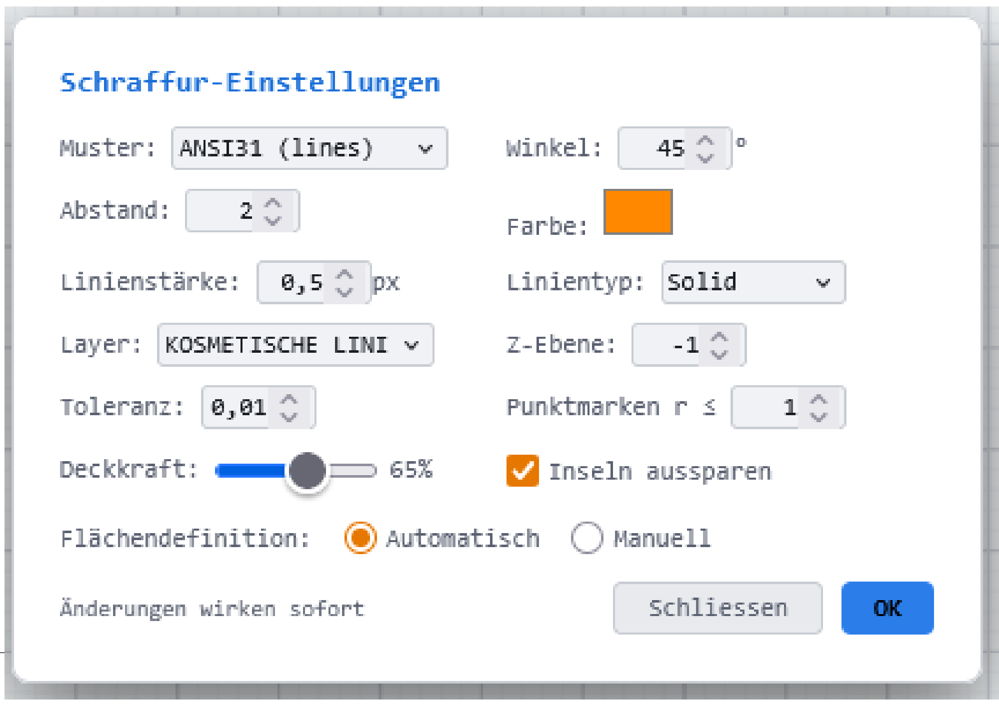
14.5 Background: line hatch versus solid hatch

Line patterns (ANSI31, ANSI32, ANSI37, CROSS) are drawn as individual lines clipped to the boundary and islands. They are the classic technical representation and cope with almost any boundary shape.

The **SOLID** pattern, on the other hand, fills the area opaquely. To do this the area is internally split into triangles, with the islands left free as real holes (the grid or the geometry underneath shows through). Overlapping islands are first merged into one common outline so that the fill stays complete and correct in that case too.

14.6 Tips

1. Close boundaries cleanly and weld the corners – that is the safest basis for automatic detection.
2. If a click does not react, click a little further inside the free area (not too close to an edge or an island).
3. If the automatic detection does not recognize a narrow gap in the boundary, try increasing the tolerance in small steps.
4. If the geometry underneath is to stay visible, reduce the opacity or leave the Z level at a negative value.
5. For heavily nested geometry, manual area definition is often quicker than repairing the boundary.



The **Hatch settings** dialog defines how areas in your drawing are filled with a hatch. A hatch is the regular pattern of lines or areas with which you mark section faces, materials or particular regions. A note at the bottom edge points out: **changes apply immediately** – your adjustments therefore become visible in the drawing right away, without you having to confirm them first. With **OK** you adopt the settings finally; with Close you leave the dialog.

14.7 Pattern and angle

The **Pattern** selection field determines the type of fill. The SOLID setting produces a continuous full-area fill; alongside it, further patterns such as slanted lines are normally available. With the **Angle** field you rotate the pattern by the number of degrees given (0° in the example). With a line pattern this defines the direction in which the hatch lines run.

14.8 Spacing

The **Spacing** field determines how far apart the individual lines of a pattern are. A smaller value produces a denser hatch, a larger value a more widely spaced one. With a solid fill (SOLID) this value has no effect.

14.9 Color

With the colored **Color** field you choose the shade in which the hatch is displayed. Clicking it opens the color picker, so that you can color the hatch to match your drawing or to represent particular materials.

14.10 Line width and line type

These two settings concern the appearance of the hatch lines. With **Line width** you define in pixels how thick the lines are drawn. With **Line type** you choose whether the lines are drawn solid or, for example, dashed.

14.11 Layer and Z level

In the **Layer** selection field you determine on which layer the hatch is placed – in the example on the Guides layer. Assigning it to a layer lets you show and hide hatches selectively or edit them together. With the **Z level** (-1 in the example) you control the stacking order: it defines whether the hatch lies in front of or behind other elements. A negative value makes the hatch move into the background so that the actual geometry stays visible above it.

14.12 Tolerance

The **Tolerance** value (0.5 in the example) determines how precisely the program detects the boundaries of an area. A slightly higher tolerance helps to find a closed area even when the boundaries do not meet exactly. If the tolerance is too low, small gaps may not be bridged and the area cannot be filled.

14.13 Opacity

With the **Opacity** slider you set how opaque the hatch appears (100 % in the example). At full opacity the hatch is completely opaque; if you reduce the value it becomes increasingly transparent, so that elements underneath show through. That is useful when the hatch is meant to mark an area while the geometry underneath should remain recognizable.

14.14 Exclude islands

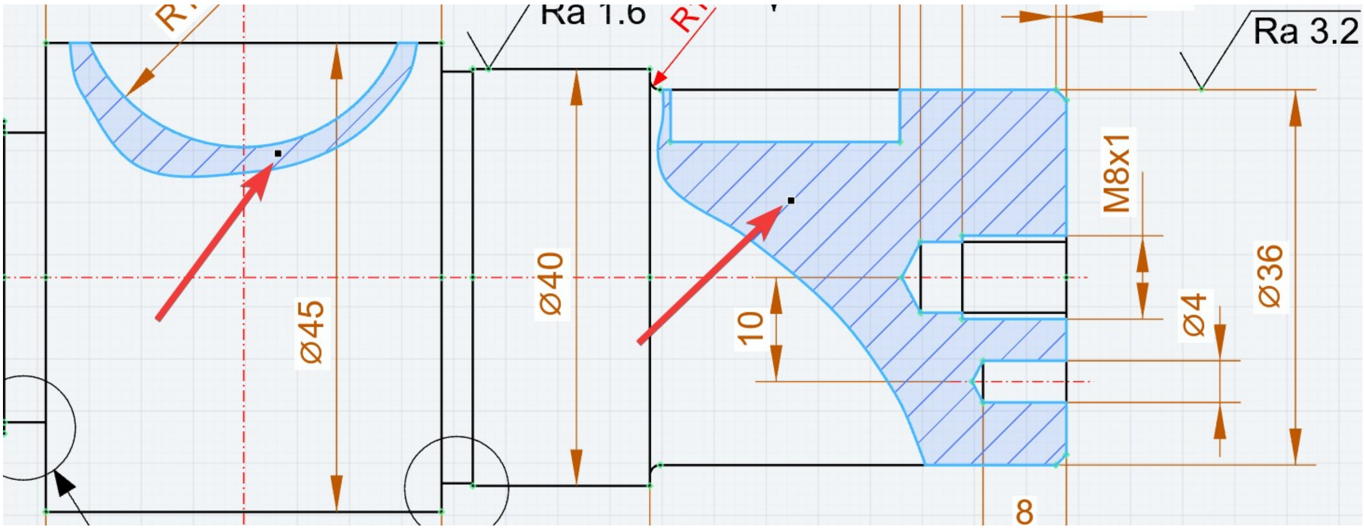
If the **Exclude islands** check box is ticked, the hatch leaves so-called islands free – that is, closed areas lying inside the area to be filled (for example a hole inside a larger area). These inner areas then remain unfilled, which corresponds to the usual representation in technical drawings.

14.15 Area definition

Here you define how the area to be hatched is determined. With the **Automatic** option, LiberoDraft determines the area itself, and you merely click inside a closed region. With the **Manual** option you define the boundary of the area yourself by deliberately selecting the boundary elements. For most cases the automatic detection is the fastest and most convenient choice.

14.16 The seed point of a hatch

If you select an existing hatch, a small black square appears: the seed point. It shows the place from which LiberoDraft searched for the area to be filled.



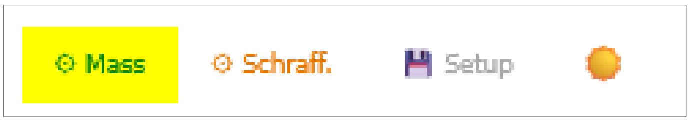
[FIGURE: selected hatches with a visible seed point]

That is more than a display: drag the seed point into another chamber and the area is searched again there — the hatch therefore moves into the other region without you having to delete it and set it anew. With the Ctrl key held down, dragging creates a second hatch instead.

If the surrounding geometry changes — you drag a line straight through it, for example — the hatch area adapts immediately.

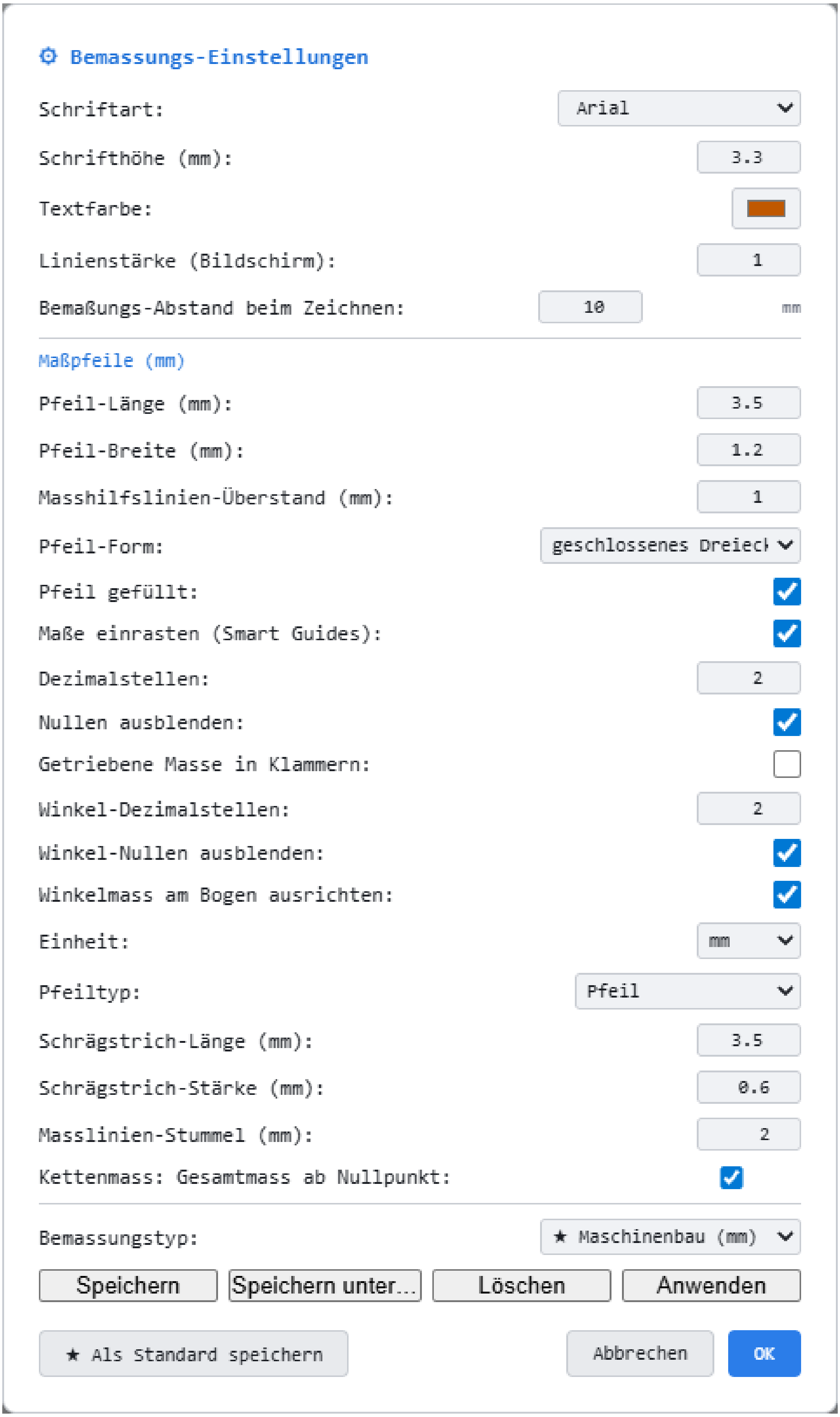
Hatching also works across blocks: for the area search, blocks are broken down, so a hatch can end on a block edge.

Let us now turn to the dimension settings, which are called up at the top right of the window as follows:



This dialog has grown over time and is therefore shown on the next page.

15 Dimension settings



In the **Dimension settings** dialog you determine the appearance of the dimension values in your drawing. Here you define in which font, size, color and precision the dimensions are displayed and in which unit measurements are given. With **OK** you adopt your changes; with **Cancel** you discard them and close the dialog without changing anything.

We will not go into every single setting here, but refer you to the tooltip that is displayed when you move the mouse pointer over an element and that describes its function. Simply keep the mouse pointer on an element for a second to display the tooltip with the explanation.

15.1 Font

The Font selection field lets you choose the typeface in which the dimension values are written – Arial in the example. A plain, easily legible font is customary in technical drawings and ensures that the values stay clearly recognizable even at a small size.

15.2 Font size

With the Font size field you define how large the dimension values are displayed (12 in the example). The small arrows on the right let you increase or decrease the value step by step; alternatively you type it in directly. Choose the size so that the values look neither too small nor too obtrusive in relation to the drawing.

15.3 Text color

The colored Text color field determines the color in which the dimension values appear. Clicking it opens the color picker. This lets you set the dimensions off from the actual geometry by color, for example, so that they are clearly distinguishable from the rest of the drawing content.

15.4 Decimal places

With the Decimal places field you define how many decimal places are shown for the dimension values (2 in the example). A value of 2 shows a length as 25.40, whereas a value of 0 shows it as 25. Choose the number to suit the precision you need: more decimal places increase the precision of the display, but with many dimensions they can also make the drawing look busier.

15.5 Hide zeros

If it is active, trailing zeros after the decimal point are dropped:

Value	off	on
40,00	`40,00`	`40`
40,50	`40,50`	`40,5`
40,55	`40,55`	`40,55`

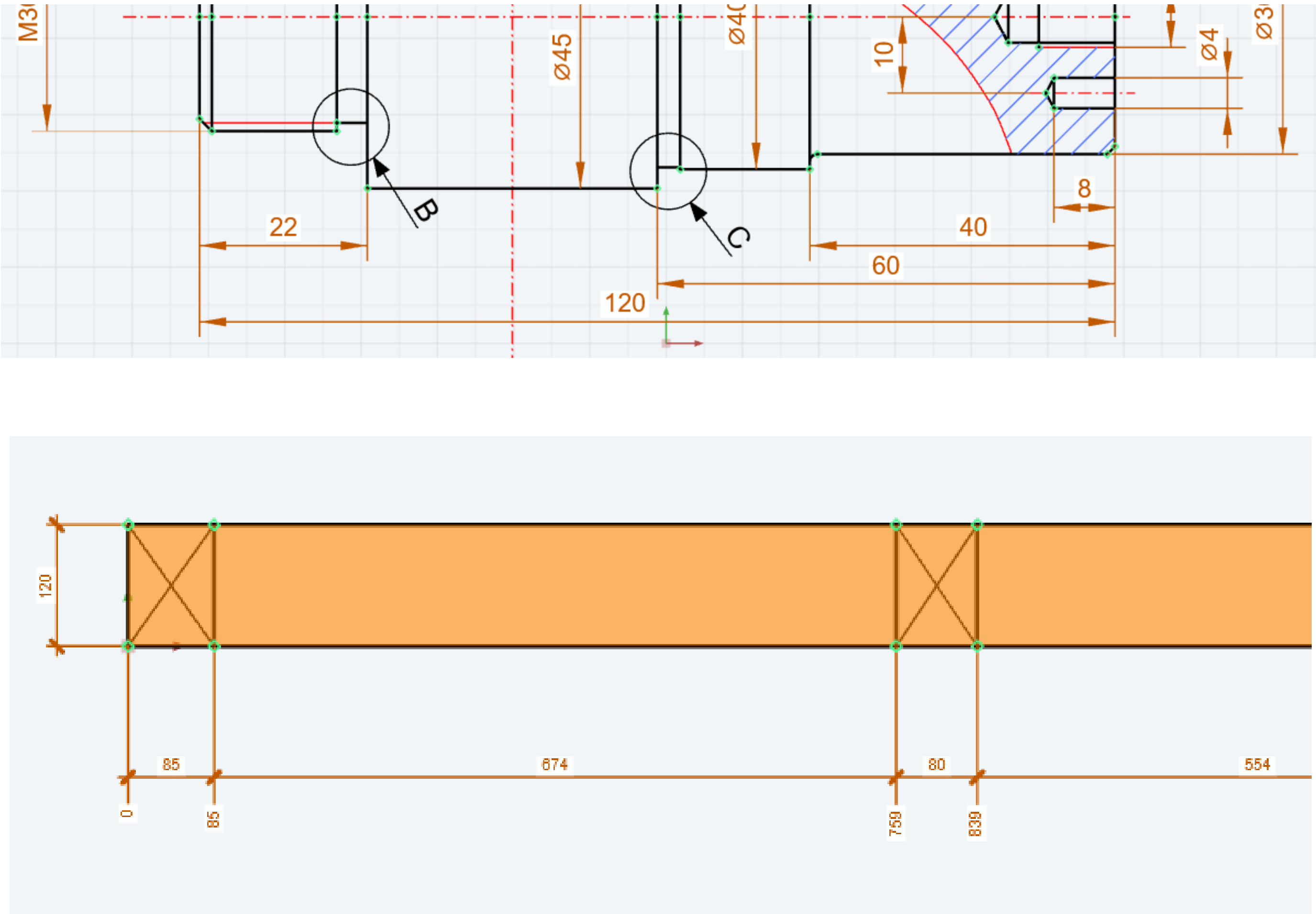
The **decimal places that have been set are retained as an upper limit** -> only zeros that are actually present are removed, the precision is not reduced. The option affects **all dimension values on screen and in print** and is saved.

15.6 Unit

The Unit selection field determines the unit of measurement in which the values are given – mm (millimeters) in the example. Make sure that the chosen unit suits your drawing and its further use, so that no misunderstandings arise later in production or when exchanging the data.

15.7 Dimension styles and settings per dimension

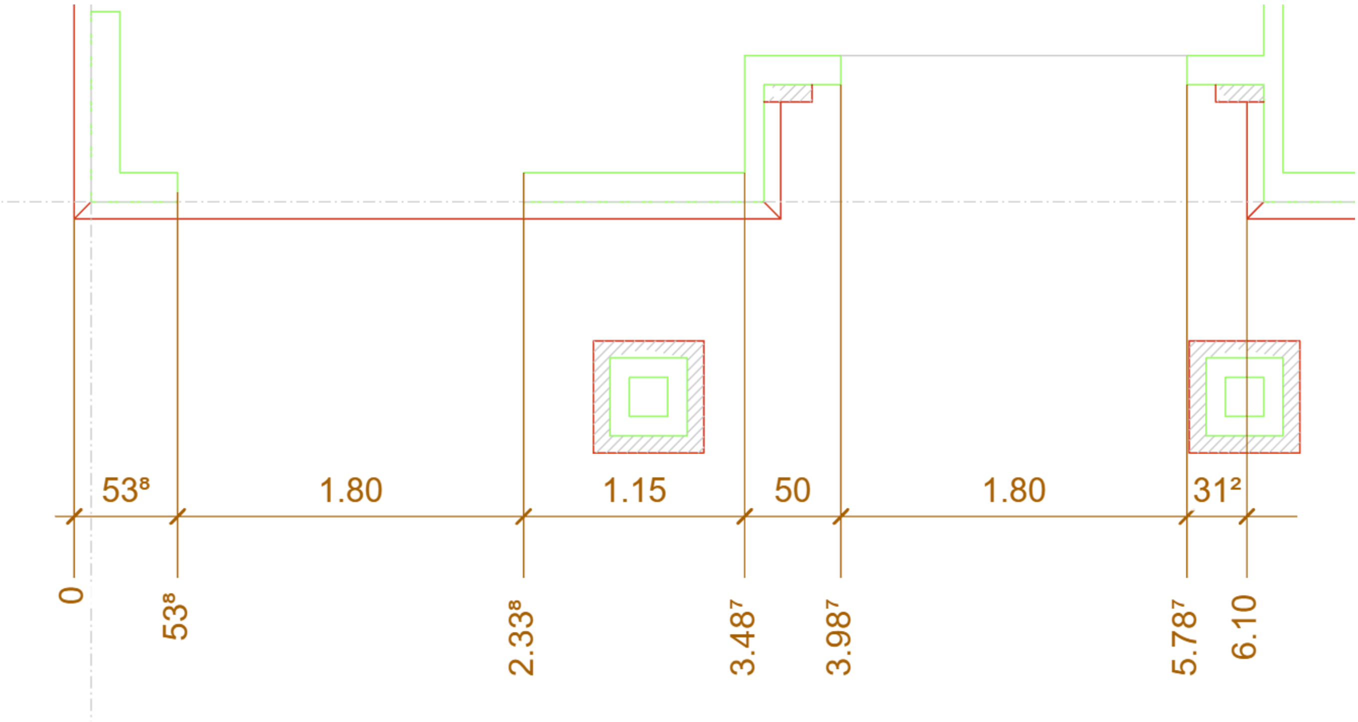
At the top of the dimension dialog you choose a dimension style. Alongside «Mechanical engineering (mm)», «Timber construction» and «Construction dimensioning» are available. Instead of arrowheads these two use the slash, as is customary in the building trade.



[FIGURE: a dimension once with arrows and once with slashes]

A dimension style defines every setting of the dialog, not just the deviations. When you switch over you therefore always get a complete, predictable result.

With construction dimensioning the values are shown in meters. If «Hide zeros» is active, whole meters appear without decimal places: 2.00 becomes a plain 2, while 2.40 stays unchanged. Everything below one meter is shown in cm. The millimeters are shown superscript as a small number:



15.7.1What OK changes — and what it does not

An important principle: settings that you adopt in the dialog with OK apply from that moment on to NEW dimensions. Existing dimensions are only changed by the «Apply» button.

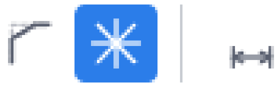
This lets you use dimensions with different appearances in the same drawing. Every dimension carries its own settings — on screen, in print and in the DXF alike.

You reach the settings of an individual dimension via the context menu, via Ctrl+double-click on the dimension, or via the button in the dimension dialog.

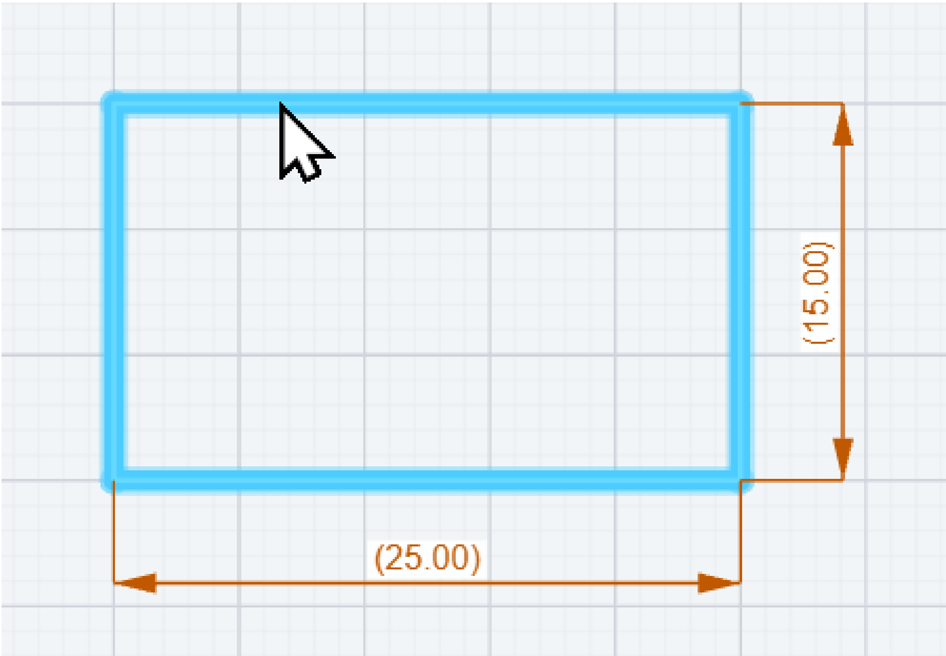
16 How do you recognize a block?

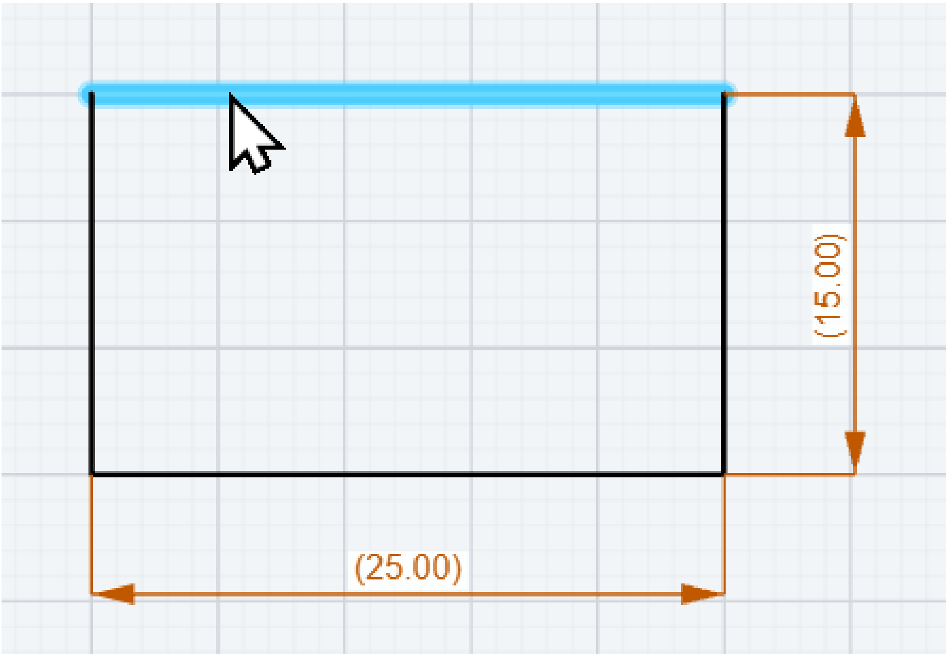
Explode block

By selecting only one element with the mouse. As soon as further elements change color as well, you know that it is a block.



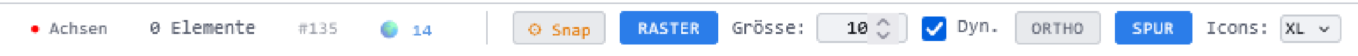
In that case the block can be broken up with the **Explode** function and split into its individual parts.





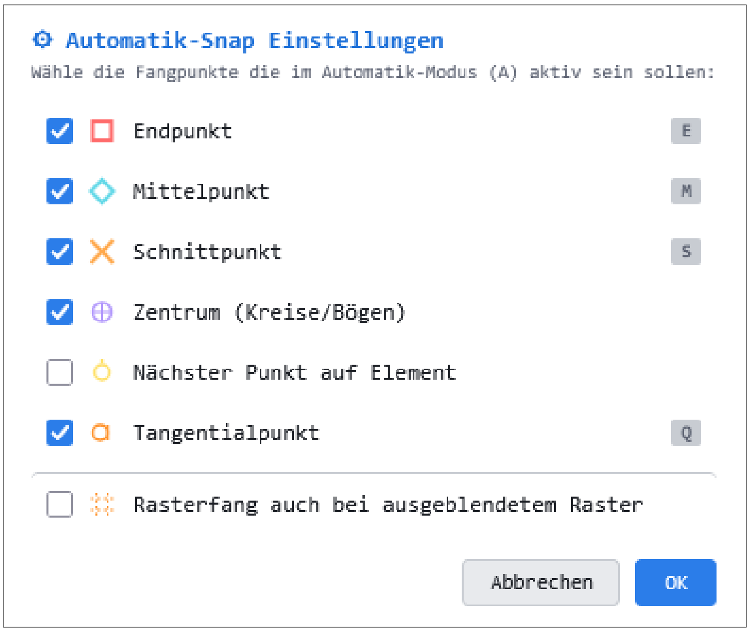
To check, a line of the rectangle can be selected afterwards. If only one line changes color, it has worked.

17 Status bar



The **status bar** is the narrow line at the edge (bottom right) of the drawing window. It summarizes the most important information about the current state of your drawing and at the same time provides a number of frequently needed switches. On one side you see at a glance details such as the active layer, the number of elements drawn and further status values. On the other side you find quick switches for central drawing aids – for example for snapping to snap and grid points, for orthogonal guidance and for setting the symbol size. This bar lets you control important auxiliary functions without leaving the drawing area or opening a menu.

17.1 Auto snap settings



What does "snap" mean?
When drawing on screen it is almost impossible to hit a particular point exactly with the mouse – the end of a line or the center of a circle, for example. This is precisely where the **snap function** helps. As soon as you move the mouse pointer close to a distinctive point, the cursor automatically "jumps" exactly onto that point and latches on there. This lets you draw to the millimeter without having to zoom and aim laboriously. The snap function is therefore one of the most important drawing aids of all: it ensures that your geometry hangs together cleanly – lines really do meet in one point, connections sit exactly, and designs stay precise.

LiberoDraft knows various kinds of such snap points. In this dialog you define **which** of these snap points are to be active in **automatic mode**. You call up automatic mode with the A key; it then checks all the snap point types ticked here at the same time and latches onto the nearest suitable point in each case. So you do not have to decide on a single snap type, but let the program automatically offer the most sensible point.

Hotkeys: behind most entries you see a letter in a gray box (for example **E, M, S**). That is the respective **keyboard shortcut assignment**: with it you can activate just one single snap type, in case you want to snap exclusively to that one type of point in a particular situation.
With **OK** you adopt your selection; with **Cancel** you discard the changes.

The individual snap points

- **Endpoint (E):** snaps the endpoints of elements, that is the start or the end of a line or an arc. This is the most frequently needed snap point, since it lets you connect new elements cleanly to existing ones.

- **Midpoint (M):** snaps the middle of an element, for example the midpoint of a line segment. That is helpful when you want to start something exactly in the middle or build something symmetrically.

- **Intersection (S):** snaps the point at which two elements cross. This lets you start exactly where two lines intersect, for example, even if there is no actual endpoint there at all.

- **Center (circles/arcs):** snaps the center of a circle or arc. This reliably hits the geometric center of a rounding – to start a center line or a further construction from there, for example.

- **Nearest point on an element:** snaps the nearest point on an element, that is any position along a line or curve, without it having to be a special point such as an end or a middle. This option is not activated in the figure. It is useful when you want to connect to an element but do not want to be tied to a particular distinctive point. Because this snap point reacts very "generously", it is often deliberately left deactivated in automatic mode so that it does not override the more precise snap points.

- **Tangent point (Q):** snaps the point at which a line meets a circle or arc tangentially (that is, exactly touching it without cutting through). This snap point is particularly important for design tasks in which straight lines are to join roundings smoothly.

17.2 Grid snap even when the grid is hidden

This option does not concern the geometry snap points mentioned above, but the **grid** – the regular pattern of points or lines in the background onto which you can likewise latch. If the check box is ticked, LiberoDraft also snaps to the grid points when the grid is not currently visible. This lets you benefit from the grid aid without the grid visually overlaying your drawing. In the figure this option is switched off.

17.3 ORTHO (orthogonal mode)

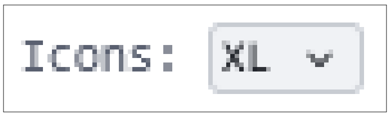
The **ORTHO** button switches orthogonal mode on and off. If this mode is active, LiberoDraft only allows horizontal and vertical lines while drawing – your movements are therefore automatically "forced" into the exactly horizontal or vertical direction. That is a great help when you want to create clean right-angled designs, such as floor plans, frames or rectangular outlines. You then no longer have to guide the mouse with painful accuracy; the program makes sure that the line really does run straight.

If you want to draw a slanted line at an arbitrary angle in between, simply deactivate **ORTHO** again. Orthogonal mode complements the snap and grid functions well: while snap makes sure that you hit exactly the right points, ORTHO keeps the drawing direction cleanly at right angles.

17.4 TRACK (object snap tracking)

The **TRACK** button switches object snap tracking on and off. While drawing, this function automatically displays invisible guide lines (so-called tracking lines) that start from distinctive points of your geometry – from endpoints or midpoints, for example. As soon as you move the mouse pointer into the extension of, or at right angles to, such a point, a dashed orientation line appears onto which you can latch. New elements can therefore be aligned conveniently with existing points without your having to draw guides of your own.
Object snap tracking is particularly useful when you want to place something exactly above, below or beside an existing point – for example to align two components cleanly with each other. It works closely together with the snap function: the snap points supply the reference points from which the tracking lines are spanned. If you do not currently need object snap tracking, or find the displayed guide lines distracting, simply deactivate **TRACK** again.

17.5 Icons (symbol size)



The **Icons** selection field sets how large the symbols (icons) of the tools and buttons are displayed in the interface – in the example the size XL is selected. Several steps from small to very large symbols are normally available.

This setting serves purely your operating comfort and has no influence whatsoever on your drawing itself. Larger symbols are easier to hit and easier to read – that is pleasant above all on high-resolution or large screens, as well as when operating via a touchscreen. Smaller symbols, on the other hand, need less space and thus leave more room for the actual drawing area. Choose the size that is most comfortable for your eyes and your screen; you can adjust it again at any time.

18 LiberoDraft -> Keyboard shortcuts (hotkeys)

Tastaturkürzel — Schliessen (Esc)

WERKZEUGE

V	Auswählen
I	Messen
N	Linie
R	Rechteck
C	Kreis
Y	Polylinie
D	Bemassung
X	Text
T	Trimmen
B	Brechen
U	Runden / Fillet
O	Offset
Ctrl+M	Verschieben

OBJEKTFANG

G	Raster
F	Frei
E	Endpunkt
M	Mittelpunkt
S	Schnittpunkt
Z	Zentrum
L	Element / Kante
P	Punkt
Q	Automatik
K	Koordinateneingabe (Startpunkt)

AUTOTRACK

Shift+T	Temporärer Spurpunkt
Shift+M	Mitte zwischen 2 Punkten

DATEI

Ctrl+N	Neue Zeichnung
Ctrl+O	Öffnen
Ctrl+S	Speichern
Ctrl+Shift+S	Speichern unter
Ctrl+P	Drucken

BEARBEITEN

Ctrl+Z	Rückgängig
Ctrl+Y	Wiederholen
Ctrl+X	Ausschneiden
Ctrl+C	Kopieren
Ctrl+V	Einfügen
Ctrl+A	Alles auswählen
Entf / ⌫	Löschen
Esc	Abbrechen

ANSICHT & ANZEIGE

W	Fenster-Zoom
A	Zoom einpassen / Alles zeigen
H	Herauszoomen
F5	Neu zeichnen
Ctrl+F	Funktion suchen
F1	Tastaturkürzel ein / aus
Ctrl+Shift+G	Gizmo ein / aus

STAPEL (Z-REIHENFOLGE)

Bild ↑	Eine Ebene nach oben
Bild ↓	Eine Ebene nach unten
Ctrl+↑	Ganz nach oben
Ctrl+↓	Ganz nach unten

BEIM ZEICHNEN / EINGABEFELD

Enter	Bestätigen
Tab	Feld wechseln
Leertaste	Polar / kartesisch umschalten
K	Startpunkt per Koordinate

NORMTEIL PLATZIEREN

R	Drehen +90°
Shift+R	Drehen -90°
H	Horizontal spiegeln
V	Vertikal spiegeln
+ / -	Vergrössern / Verkleinern
B	Basispunkt setzen

DRUCKVORSCHAU

H	Hochformat
Q	Querformat
Esc	Schliessen

Einzeltasten wirken ohne aktives Eingabefeld; Objektfang-Tasten (F/E/M/S/Z/Q/L/P/A/G) auch im fliegenden Eingabefeld. Die Gruppen «Normteil platzieren» und «Druckvorschau» gelten nur im jeweiligen Modus. **Maus:** Rad = Zoom · mittlere Taste oder Alt+Ziehen = Verschieben · Rechtsklick = Kontextmenü.

[**F1**] produces the current hotkey overview.

19 LiberoDraft -> Keyboard shortcuts

Manual chapter · as of version 0.9.068 · source: the program's internal shortcut table (identical to the F1 dialog)

LiberoDraft can largely be operated from the keyboard. **The complete shortcut overview can be called up in the program at any time with [F1]** (pressing [F1] again closes it). Single keys (tools, object snap) only work when the focus is **not** in an input field; combinations with *Ctrl* / *Shift* work everywhere. Two groups are tied to a mode: "Place standard part" (while a part is hanging at the cursor) and "Print preview" (when the preview is open).

Note: the F1 list is always up to date. The overview below may not be, since the LiberoDraft CAD program is always a step ahead and the book follows on :-)

19.1 Tools

Key	Function
V	Select
I	Measure
N	Line
R	Rectangle
C	Circle
Y	Polyline
D	Dimension
H	Hatch
X	Text
T	Trim
B	Break
U	Fillet
O	Offset
Ctrl+M	Move

19.2 Object snap

Key	Snap mode
Q	Automatic

Key	Snap mode
E	Endpoint
M	Midpoint
S	Intersection
Z	Center
Q	Tangent
L	Element / edge
P	Point
G	Grid
F	Free
K	Coordinate entry (start point)

19.3 AutoTrack (TRACK)

Key	Function
Shift+T	Temporary track point
Shift+M	Midpoint between 2 points

19.4 File

Key	Function
Ctrl+N	New drawing
Ctrl+O	Open
Ctrl+S	Save
Ctrl+Shift+S	Save as
Ctrl+P	Print

19.5 Edit

Key	Function
Ctrl+Z	Undo
Ctrl+Y	Redo
Ctrl+C	Copy (with base point prompt)
Ctrl+V	Paste -> the copied elements hang at the

Key	Function
	mouse; if the clipboard contains a **new image** (a screenshot, for example), the image is pasted instead and hangs at the mouse
Ctrl+A	Select all
Del / ⌫	Delete
Esc	Cancel (running action, dialogs, clipboard image hanging at the mouse)

19.6 View & display

Key	Function
A	Zoom to fit
W	Zoom window
H	Zoom half (half a zoom step back)
F5	Redraw
Ctrl+F	Search functions (command palette)
F1	Keyboard shortcut overview on/off
Ctrl+Shift+G	Gizmo on / off *

19.7 Z-order (stacking)

Key	Function
PgUp	One level up
PgDn	One level down
Ctrl+↑	Bring to front
Ctrl+↓	Send to back

* More information about the gizmo: chapter 32 Topic: Gizmo Page (126)

19.8 While drawing / in the input field

Key	Function
Enter	Confirm

Key	Function
Tab	Switch field
Spacebar	Toggle polar / cartesian
K	Start point by coordinate

19.9 Placing a standard part / block

Key	Function
R	Rotate +90°
Shift+R	Rotate −90°
H	Mirror horizontally
V	Mirror vertically
+ / –	Enlarge / shrink
B	Set base point
Click	Place (several times possible)
Esc / right-click	Finish

19.10 Print preview

Key	Function
H	Portrait
Q	Landscape
Esc	Close

19.11 Context-dependent keys (special situations)

Extend/shorten (blue triangle at the end of a line or arc):

Key	Function
Digits (±)	Type an amount -> the display "Extension: …" appears in the floating window
Enter	Apply the amount
L	Arcs only: switch between length and angle
Esc	Cancel

Note: a click in the gizmo panel* ends live mode -> the gizmo fields take precedence over the floating input.

Clipboard image (after Ctrl+V with an image):

Key	Function
Mouse movement	The image follows the cursor (with snap)
Left-click	Place the image
Esc	Cancel pasting, remove the image

Dimension mode:

Situation	Behavior
Hovering over an element that cannot be dimensioned (a polyline, for example)	🚫 cursor + status note; remedy: right-click → ✱ Explode
Point snap active (endpoint, midpoint, …)	A point-to-point dimension is always possible
Double-click on a dimension	Opens the dimension attributes

19.12 Mouse

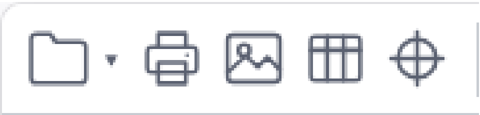
Input	Function
Left-click	Set a point / select / tool action
Right-click	Context menu (or end placement/mode)
Mouse wheel	Zoom
Middle button (hold pressed)	Pan the view
Double-click on an image	Image properties

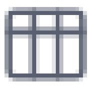
* More about the gizmo in chapter 32 Topic: Gizmo Page (126)

20 LiberoDraft -> The toolbar

Description of all buttons of the upper toolbar, in order from left to right. Tools with a small triangle (▼) or a dot have a **drop-down menu** with related variants.

20.1 File

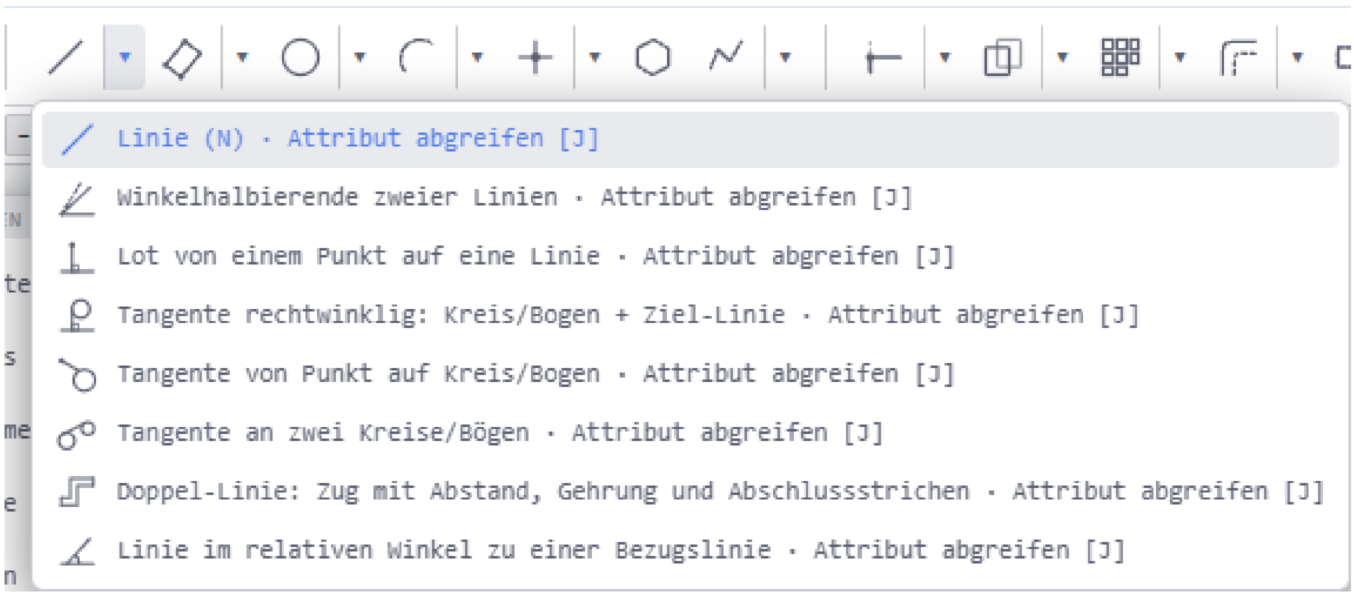


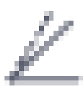
Symbol	Function	Description
Folder ▼ 	**File menu**	Expands all file commands: New drawing (Ctrl+N), Open (Ctrl+O), Save (Ctrl+S), Save as (Ctrl+Shift+S) as well as DXF and SVG import/export.
Printer 	**Print** (Ctrl+P)	Opens the print dialog with scale, paper format, orientation, black-and-white/images option and line width control.
Image 	**Insert image**	Loads a raster image file into the drawing as a template that can be scaled and rotated (for tracing over, for example).
	**Coordinate table**	Creates a numbered coordinate table from inserted points.
	**Axis cross**	On request, automatically inserts the coordinate cross into circles and/or arcs.

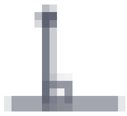
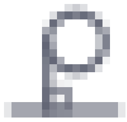
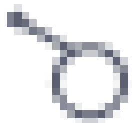
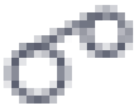
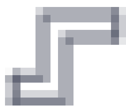
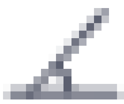
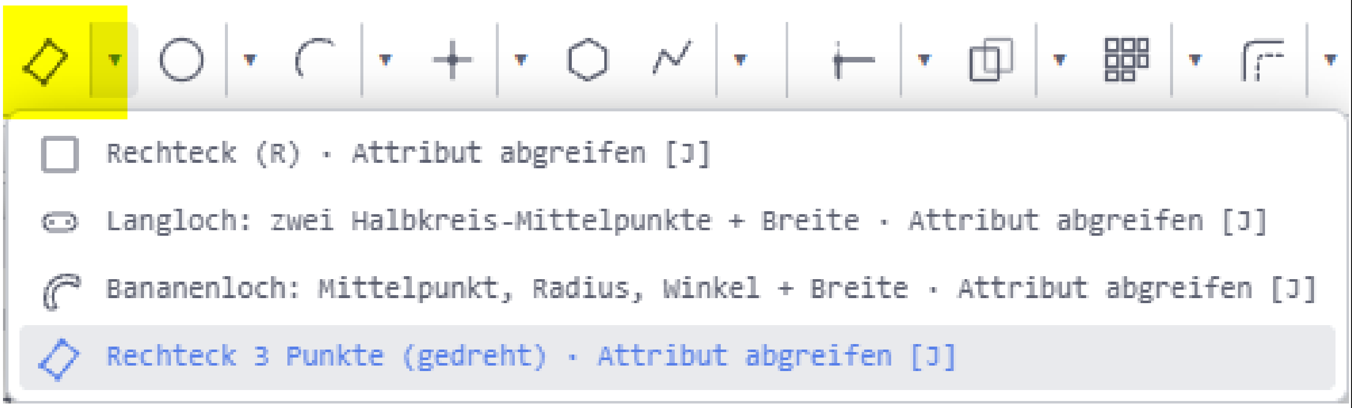
20.2 Navigation

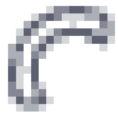
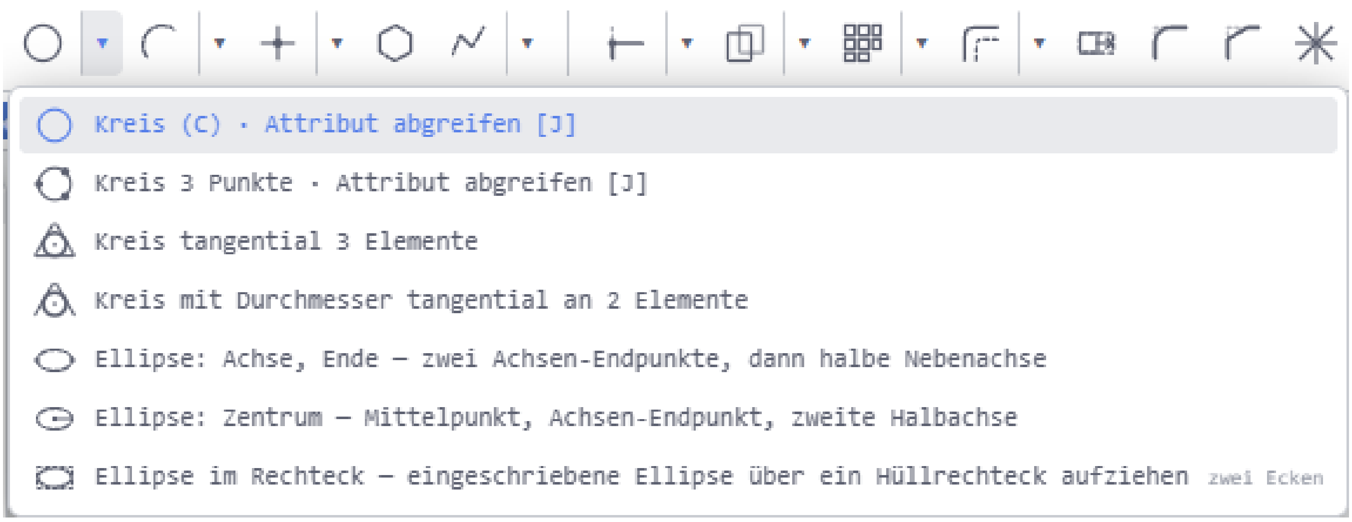
Symbol	Function	Description
Arrow 	**Select** (V)	Standard mode for marking, moving and editing elements via grips.
Ruler 	**Measuring functions** (I)	Extended measuring functions with additional details up to area, and an insertable centroid.

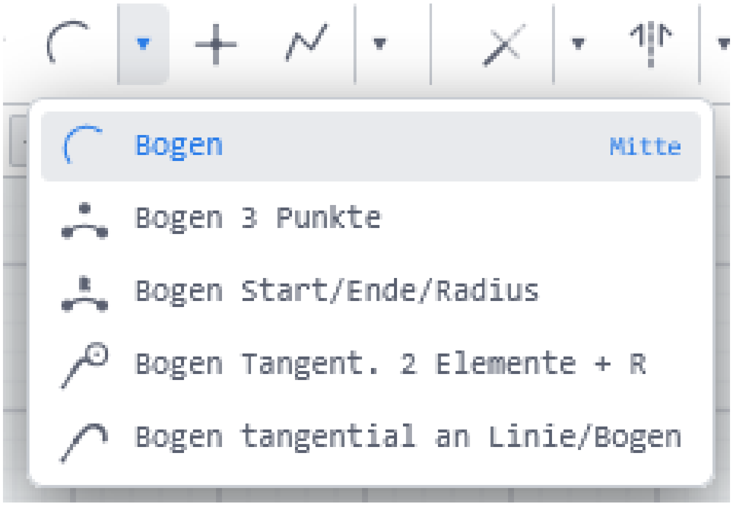
20.3 Draw

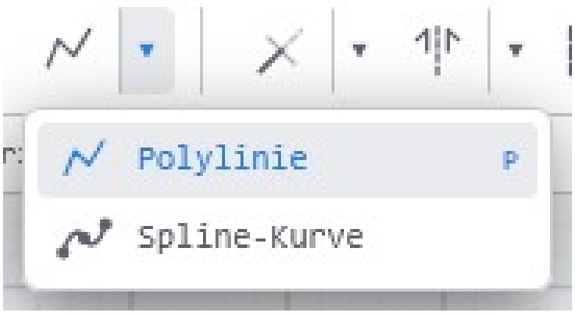


Symbol	Function	Description
Line 	**Line** (N)	Draws a straight segment between two points.
	**Angle bisector**	Draws the bisector between two non-parallel lines, including the length

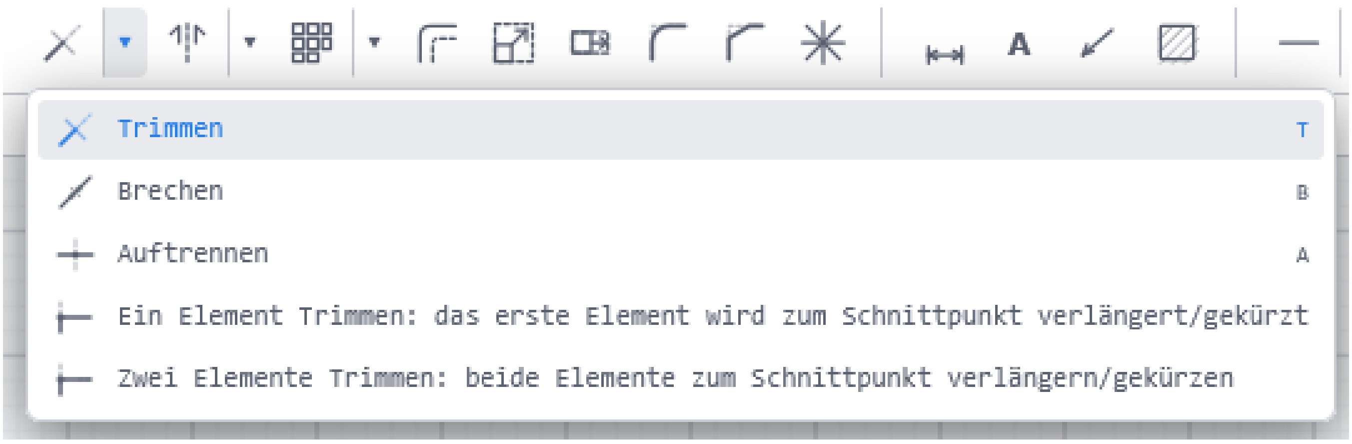
	**Perpendicular from a point onto a line**	Draws the perpendicular line at a right angle from a point onto a line.
	**Perpendicular tangent**	Draws a tangent line running at a right angle from a line onto a circle or arc.
	**Tangent from a point to a circle**	Draws a tangent from a point onto a circle or arc.
	**Tangent to two circles**	Draws a tangent to two circles or arcs.
	**Double line**	Draws a double line with mitred corners and closed ends.
	**Line at a relative angle**	Draws a line at a relative angle to another line.
		
Rectangle ▼ 	**Rectangle** (R)	Ordinary axis-parallel rectangle via two corner points.
└ Variant	**Rectangle 3 points**	Arbitrarily rotated rectangle via three points (base edge +

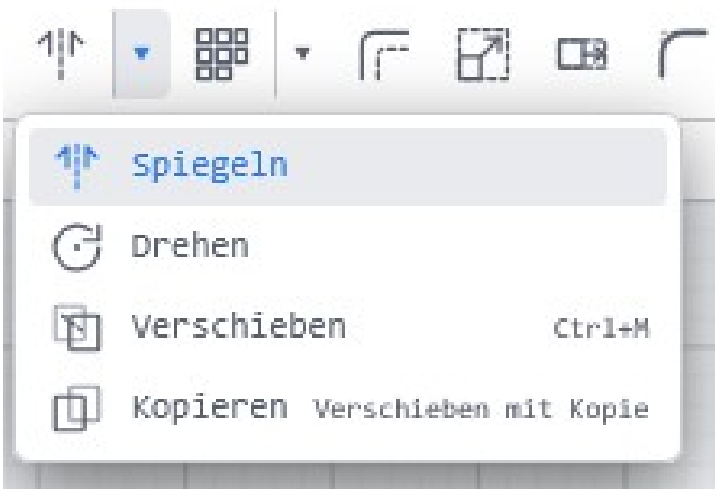
		height).
	**Slot**	Self-explanatory :-)
	**Curved slot**	Self-explanatory :-)
		
Circle ▼ 	**Circle** (C)	Circle via center and radius.
└ Variant 	**Circle 3 points**	Circle through three arbitrary points.
└ Variant 	**Circle tangent to 3 elements**	Inscribed circle touching three existing lines/circles.
└ Variant	**Circle with a given diameter, tangent to 2 elements**	Circle with a specified diameter that touches two

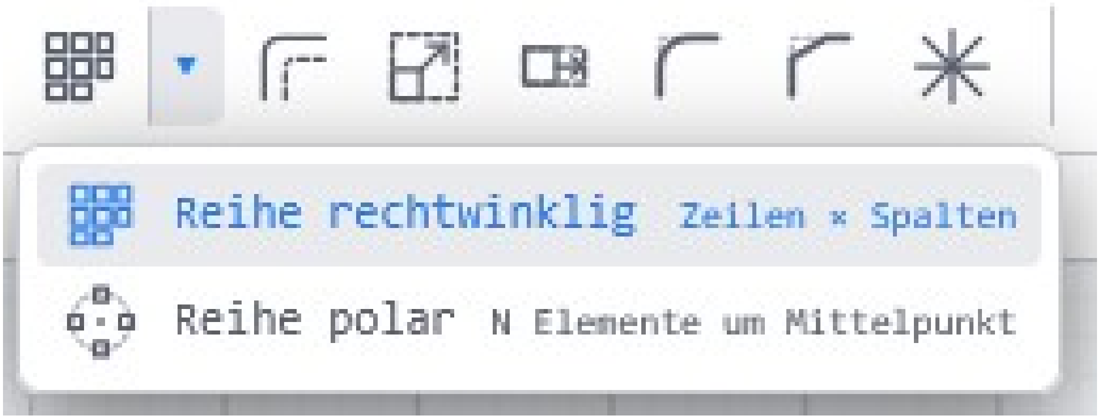
		elements.
		
Arc ▼ 	**Arc (center)**	Circular arc via center, start angle and end angle.
└ Variant 	**Arc 3 points**	Arc through a start point, an end point and an intermediate point. The radius can be entered at any time!
└ Variant 	**Arc start/end/radius**	Arc via two endpoints and a radius.
└ Variant 	**Arc tangent to 2 elements + R**	Fillet arc with radius R that joins two elements tangentially.
└ Variant 	**Arc tangent to a line/arc**	Arc that joins an existing element tangentially. **Perfect for tracing over images!**
Point 	**Set point**	Places an independent point element (as a snap or

		reference point, for example).
		
Polyline 	**Polyline** (P)	Connected run of lines made up of several segments.
Curve 	**Spline curve**	Smooth curve through several control points; the C key closes it.

20.4 Modify

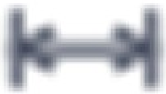
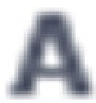
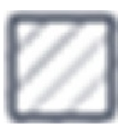
Symbol	Function	Description
		
┤ ▼ 	**Trim** (T)	Cuts an element off at a cutting edge.

Symbol	Function	Description
└ Variant 	**Trim one element**	Extends/shortens only the first element up to the intersection with the second.
└ Variant 	**Trim two elements**	Extends/shortens both elements up to their common intersection.
┤ 	**Break** (B)	Separates an element at a point into two parts (a gap is possible).
┤ 	**Split** (A)	Divides an element at a point without a gap.
		
Copy ▼ 	**Mirror**	Mirrors elements about an axis.
└ Variant 	**Rotate**	Rotates elements about a base point.

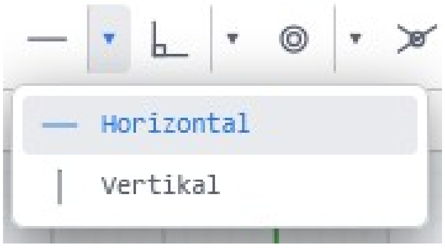
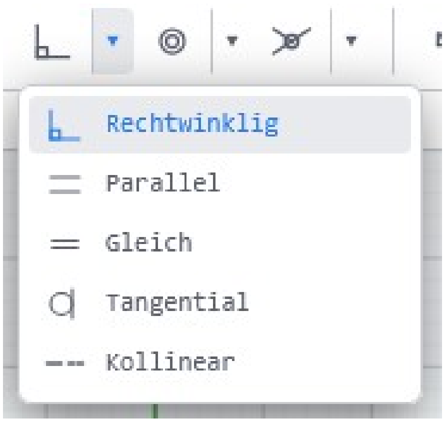
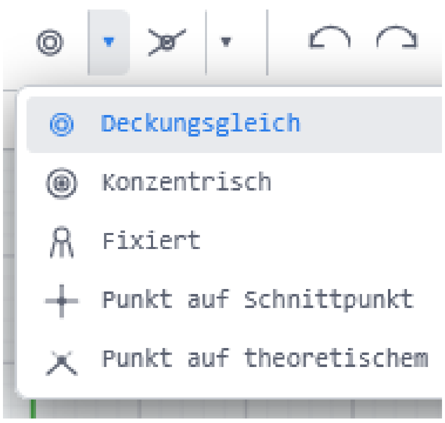
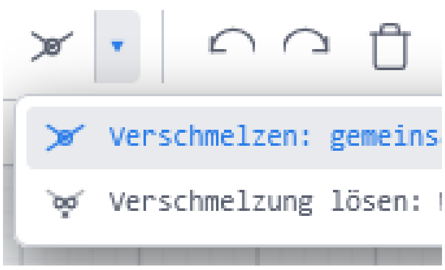
Symbol	Function	Description
└ Variant 	**Move** (Ctrl+M)	Moves elements by a vector.
└ Variant 	**Copy**	Moves with a copy – the original is retained.
		
Array ▼ 	**Array rectangular**	Duplicates elements in a grid of rows × columns.
└ Variant 	**Array polar**	Duplicates N elements in a circle around a center point.
Offset 	**Offset** (O)	Creates a parallel contour at the distance entered.
Scale 	**Scale**	Enlarges/reduces elements about a base point by a factor.
Stretch 	**Stretch**	Moves only the points inside a window (for stretching contours, for example).

Symbol	**Function**	**Description**
Corner ▼ 	**Fillet** (F)	Replaces a corner with a fillet arc of a given radius.
└ Variant 	**Chamfer**	Replaces a corner with a bevel (distances D1/D2 or angle/distance).
Explode 	**Explode**	Breaks a rectangle/polyline/spline down into individual line elements.

20.5 Annotate

Symbol	**Function**	**Description**
Dimension 	**Dimension** (D)	Creates dimensions (linear, aligned, radius, diameter, angular, point-to-point …).
A 	**Text** (X)	Inserts single-line text with a font, size and alignment.
Arrow 	**Leader**	Annotation arrow (leader) with an arrowhead and free text.
Hatch 	**Hatch** (H)	Fills areas with a pattern or a solid fill – with automatic or manual area selection.

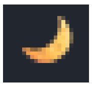
20.6 Constraints (geometric conditions) * experimental *

Symbol	Function	Description
	**Horizontal**	Forces an element/segment into the horizontal.
	Vertical	Forces an element into the vertical.
	**Perpendicular**	Keeps two elements at an angle of 90°.
	Parallel	Keeps two elements parallel.
	Equal	Forces two elements to have the same length/radius.
	Tangent	Keeps an element tangential to a circle/arc.
	Collinear	Places two lines on the same straight line.
	**Coincident**	Places two points on top of each other.
	Concentric	Common center of two circles/arcs.
	Fixed	Anchors a point/element in place.
	Point on intersection	Binds a point to the intersection of two elements.
	Point on theoretical intersection	As above, but on the imaginary intersection of the extended lines.
	**Weld**	Permanently joins the common endpoints of the selected elements.
	Unweld	Releases the weld of the selected elements again.

20.7 Action

Symbol	Function	Description
↶	**Undo** (Ctrl+Z)	Undoes the last step.
↷	**Redo** (Ctrl+Y)	Restores a step that has been undone.
Wastebasket	**Delete** (Del)	Removes the current selection.
	**Zoom to fit** (Ctrl+F)	Zooms so that the entire drawing becomes visible.
↻	**Redraw** (F5)	Rebuilds the screen display.

20.8 On the right in the toolbar

Symbol	Function	Description
⚙ Dim.	**Dimension settings**	Font size, arrows, decimal places, style of the dimensions.
⚙ Hatch	**Hatch settings**	Floating window: pattern, angle, spacing, color, opacity, area selection.
	**Save/load settings**	Save, load, export/import the program settings.
 	**Day/night mode**	Switches between the light and the dark design.
i	**Info / imprint**	Version details and program information.

20.9 Addition: control bar at the bottom right (status bar)

Functionally, these switches belong to the tool set:

Element	Description
⚙ Snap	Object snap settings (which snap points are active).

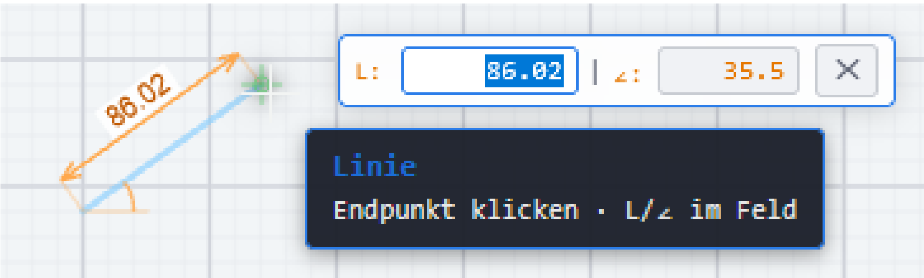
Element	Description
GRID	Show/hide the grid lines (the switch lights up when active).
Size	Grid size in mm.
Dyn.	Dynamic grid – the grid spacing adapts as you zoom.
ORTHO	Ortho mode – restricts drawing to horizontal/vertical.
Icons	Symbol size of the toolbar (S/M/L/XL).

21 LiberoDraft -> Functions in detail

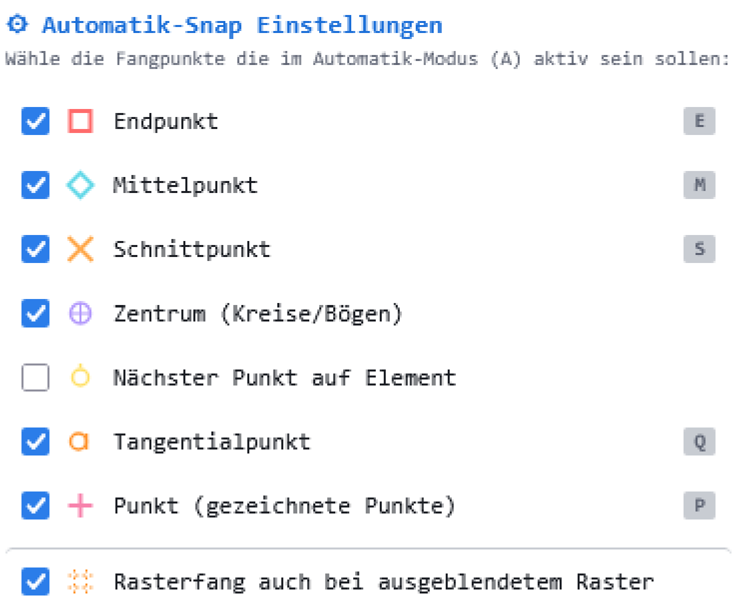
Detailed description of every toolbar function with step-by-step operation. The order follows the toolbar (left → right).
(As of v86.87.)

General conventions:

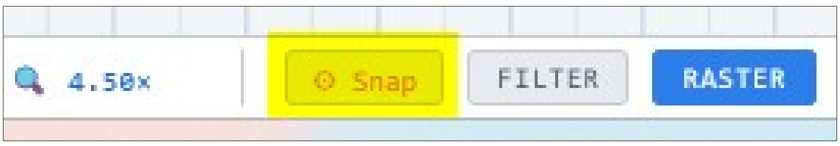
- **Floating input field**: with many tools a small field appears next to the crosshair (length, angle, dX/dY, radius, for example). Type in a value, change the field with the **Tab key** and confirm with **Enter** – or use the mouse and click.



- **Object snap automatic [Q]**: snaps these snap points via the status bar (Snap) or by key (see the hotkey list [F1]).



In the status bar the **Snap** button is used to set the desired snap modes for automatic snap / auto snapping / snap automatic:



- **Esc** cancels any running action.

22 File

22.1 File menu (folder symbol ▼)

Expands all file commands.

1. Click the folder symbol – the menu opens.
2. Choose a command: **New drawing** (Ctrl+N), **Open** (Ctrl+O), **Save** (Ctrl+S), **Save as** (Ctrl+Shift+S), **Import DXF/SVG** or **Export DXF/SVG**.

Print (Ctrl+P)

Produces a printout to scale.

1. Call up the tool – the print dialog opens.
2. Choose the paper format, the orientation (portrait/landscape) and the scale (1:1, for example).
3. Set the options: print **B/W**, **frame**, **images** as well; adjust the line widths per element group if required.
4. Check the print preview, then print.

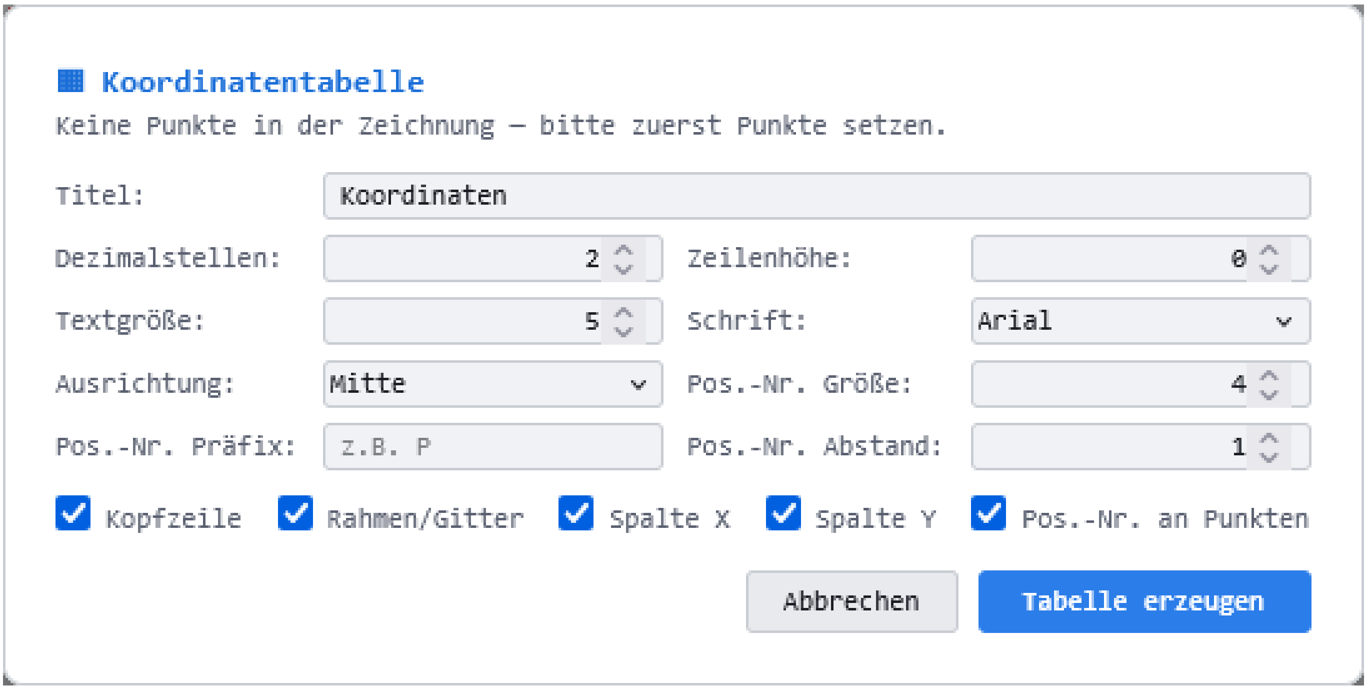
Insert image

Loads a raster image as a template (for tracing over a sketch, for example).

1. Click the tool and choose the image file.
2. The image is placed; size, position, rotation and transparency can be changed by double-clicking (or right-click → Image properties).

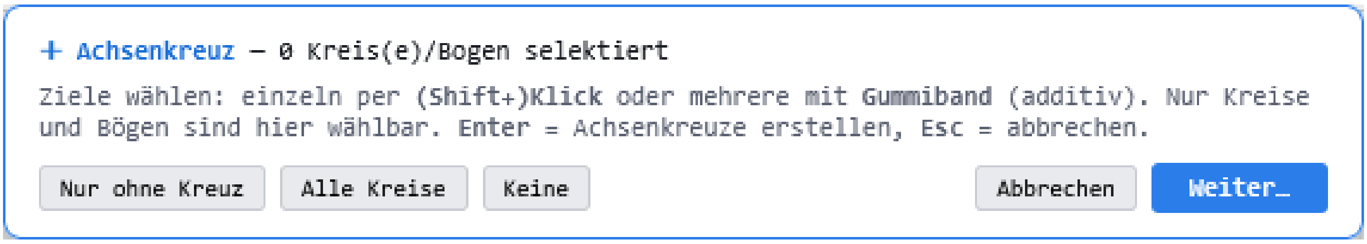
Coordinate table

Creates a clean coordinate table from the points present in the drawing.
Every point is automatically referenced with a number in the drawing.
The settings available are shown below.

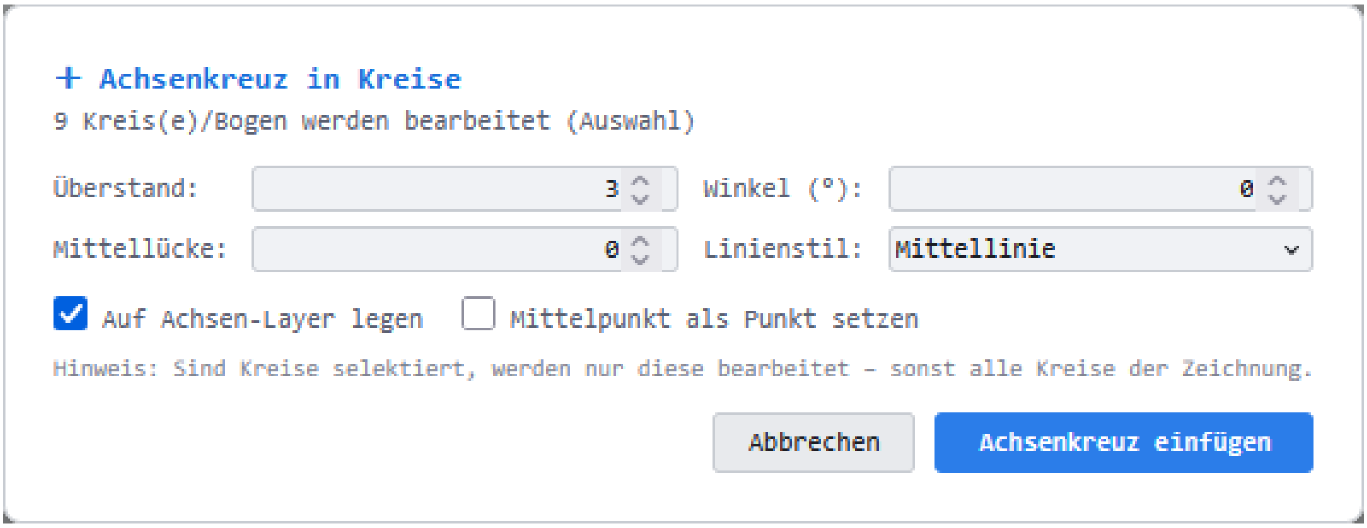


Inserting axis crosses automatically

Definable axis crosses are drawn into all selected...



circles or arcs...



23 Navigation

23.1 Select (V)

The basic mode for marking and editing.

1. **Single selection**: click an element.
2. **Multiple selection**: click further elements with the **Shift** key held down.
3. **Rubber band**: click in an empty area and drag
   - dragging a window from left to right selects the elements that are completely **enclosed**.
   - dragging a window from right to left also selects all **crossed** elements.
4. Selected elements show **handles (grips)** for moving/deforming them.

23.2 Window zoom

With the window zoom you frame an area and fill the screen with it. The button carries the note «Window zoom: frame an area».

Proceed as follows:

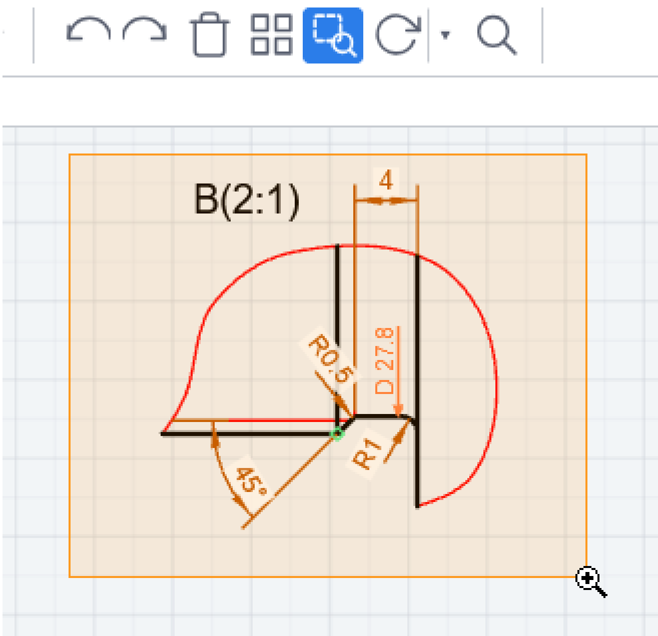
• Click the button or choose the command from the command search.

• Click the first corner of the area you want.

• Click the second corner — or drag with the mouse button held down and release.

Important: the window zoom stays active afterwards. You can therefore work your way towards a spot with several frames one after another without having to press the button again every time. It is ended with the Esc key, with the right mouse button or by pressing the button again.

A tiny frame drawn by accident is ignored — a mere click therefore does not lead to a useless zoom.

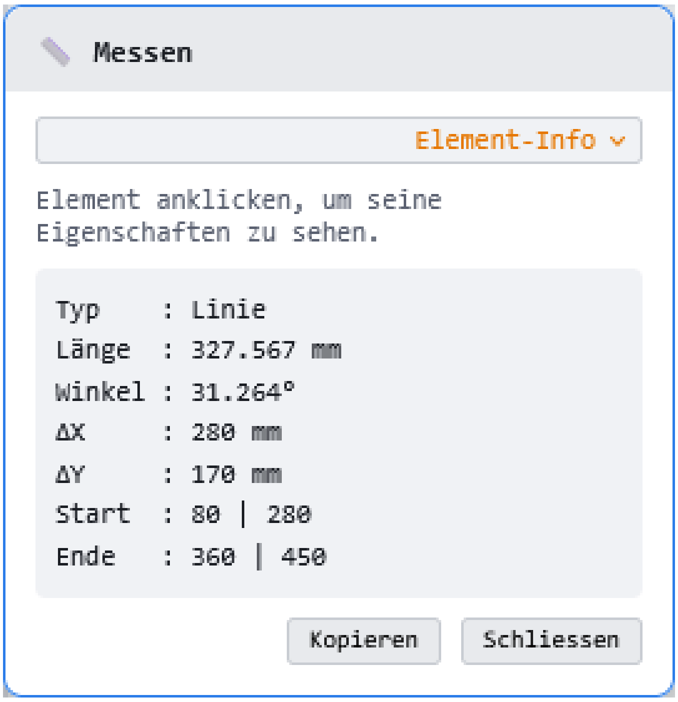


24 Measure (query tool)

The **Measure** tool lets you read geometric details directly from the drawing **without** creating a dimension. It serves purely as a query – the values are listed in the measurement window and additionally shown in color on the drawing.

Calling it up: the ruler symbol in the toolbar (*Navigation* group), the **I** key, or via the right-click menu (the *Measure* entry).

24.1 The measurement window



When you activate it, a window opens at the top of the screen that can be moved freely with the mouse. The selection field at the top is used to choose the **measurement mode** you want. The values determined appear in the lower area and can be copied to the clipboard with **Copy** (for a parts list or documentation, for example). **Close** ends the tool.

24.2 The measurement modes

Element info – click an element; depending on its type, all the important properties are displayed:

- *Line:* length, angle to the horizontal, ΔX, ΔY, start and end coordinates
- *Circle:* radius, diameter, circumference, area, center
- *Arc:* radius, central angle, arc length, chord length, center
- *Rectangle:* width, height, area, circumference
- *Polyline / spline:* number of points, open/closed, total length, and for a closed boundary the area
- *Point:* X/Y coordinates

The element you clicked is highlighted; the most important value is additionally displayed directly at the element (for lines also an angle arc as well as ΔX in red and ΔY in green).

Distance – click two points (with object snap). The distance, the angle to the horizontal and ΔX and ΔY are output. During the second click, the distance is previewed live at the cursor with ΔX (red) and ΔY (green).

Min. distance – click any two elements one after another. The shortest distance between them is determined and the shortest connection is drawn in – also between curved elements, for example between a circle and a straight line.

Angle – click two lines. The enclosed angle and the supplementary angle are output; the angle arc is shown at the intersection.

Area & circumference – determines the area, circumference and centroid of a closed boundary. The boundary is determined in two ways, just as with hatching:

- *Automatic:* click inside a closed area; the boundary (including any islands inside it) is detected automatically and island areas are subtracted.
- *Manual:* click the boundary elements one by one and finish with **Enter** (**Esc** cancels).

Once an area result is available, the **Insert centroid as a point** button additionally appears, with which the calculated centroid can be taken over into the drawing as a real point on the active layer.

24.3 Notes

- Measuring creates **no** dimension – it is purely a query.
- All measured values are shown on the drawing in a uniform color (ΔX in red, ΔY in green as the axis convention).
- **Esc** discards a multi-point measurement that has been started, or the manual area selection.

25 Draw

25.1 Line (N)

Straight segment between two points.

1. Click = start point.
2. Move the mouse – the floating field shows **length** and **angle**.
3. Enter the values + Enter, or 2nd click = endpoint.

25.2 Angle bisector

Select two lines and enter the length.

25.3 Perpendicular onto a line

Perpendicular line from a point onto the target line.

25.4 Line at a relative angle

Line at a relative angle to an arbitrary line, with the length given. If only the angle field is filled in and confirmed with the TAB key, the line keeps its orientation and can be drawn freely or with a given length.

Rectangle (R)

Axis-parallel rectangle.

1. Click = first corner.
2. Click = opposite corner – or enter the **width/height** in the field + Enter.

Rectangle 3 points (rotated)

Arbitrarily rotated rectangle.

1. Click = start of the base edge, 2nd click = end of the base edge (defines the direction).
2. Click = height/width perpendicular to the base edge.

Circle (C)

Circle via center and radius.

1. Click = center.
2. Enter the **radius** + Enter, or 2nd click on the circumference.

Circle 3 points

1. Click P1, P2.
1. Click a point or enter **R** + Enter.

Circle tangent to 3 elements

Inscribed circle touching three elements.

1. Click three elements one after another (lines/circles).
2. Move the mouse – the matching circle follows; a click places it.

Circle with a given diameter tangent to 2 elements

1. Click the first and the second element.
2. Enter the **diameter** ⍰ + Enter (or click); the solution follows the cursor.

Arc (center)

Circular arc via the center.

1. Click = center.
2. Click = start point (radius + start angle).
3. Click = end angle (the arc runs counter-clockwise).

Arc 3 points

1. Click the start and end point.
2. Define the bulge with the mouse or enter **R** + Enter.

Arc start/end/radius

1. Click the start and end point.
2. Enter the **radius**.

Arc tangent to 2 elements + R

Fillet arc with a fixed radius between two elements.

1. Click the first and the second element.
2. Enter **R** + Enter (or click).

Arc tangent to a line/arc

Arc that joins an existing element tangentially.

1. Click = line/arc at the connection point.
2. Click = endpoint of the new arc – the radius follows automatically.

Tip: excellently suited to vectorizing images precisely.

Set point

Places an independent point element.

1. Click at the position you want (with snap).

Polyline (P)

Connected run of lines.

1. Click the points one after another; every floating field allows a length/angle to be entered.
2. **Double-click** or **Esc** ends the run; **C** closes it into an area.

Spline curve

Smooth curve through control points.

1. Set the points – the curve runs through all of them.
2. **Double-click/Esc** ends it, **C** closes the curve.

25.5 Relative angle

With this function you draw a line at a particular angle to a line that already exists. The button is located with the line functions and carries the note «Line at a relative angle to a reference line».

The procedure has three steps; the status bar and the hint at the mouse pointer guide you through them:

• Step 1: click the reference line — the line the angle is measured from.

• Step 2: set the start point of the new line. The object snap switches to «Automatic» on its own while doing so.

• Step 3: click the endpoint — or type the angle and the length into the floating fields and draw with the Enter key.

[FIGURE: reference line, new line and the displayed fields for angle and length]

The angle is measured from the direction of the reference line, that is from its start point to its endpoint, and is positive counter-clockwise. It always lies between −180° and +180°. That distinguishes it from the angle field of the ordinary line, which refers to the horizontal axis.

Tip: if you type the angle in and confirm it with the Tab key, the direction is locked. The mouse then only determines the length — this gives you an exact angle and still a freely

dragged length.

26 Modify

26.1 Trim (T)

Cuts an element off at a crossing.

1. Click **on the side that is to be kept** – the protruding part up to the next cutting edge is removed.

Trim one element

1. Click = the element to be changed (on the keep side).
2. Click = the reference element (stays unchanged). The first element is extended or shortened up to the intersection.

Trim two elements

1. Click each of the two elements on its keep side – both are extended/shortened up to their common intersection (ideal for clean corners).

Break (B)

Separates an element with a gap.

1. Click **on the section that is to be removed**.

Split (A)

Divides an element without a gap.

1. Click on the split position (line/arc).
2. With a circle: 1st point and 2nd point for the two split positions.

26.2 Mirror

1. Select the elements.
    1. Click = start of the axis, 2nd click = end of the axis – the selection is mirrored.

Rotate

1. Select the elements.
2. Click the base point, then enter the **angle** + Enter, or rotate with the mouse.

Move (Ctrl+M)

1. Select the elements.
    1. Click = base point (the floating **dX/dY** field appears).
    2. Click = target point, or enter dX/dY + Enter.

Copy

Like Move, but the original is retained; several copies one after another are possible.

26.3 Array rectangular

Duplicates in a grid.

1. Select the elements, call up the tool.
2. Enter **rows × columns** and the spacings – the array is created.

Array polar

Duplicates in a circle.

1. Select the elements, call up the tool.
2. Choose the **center**, give the **number** of copies and the total angle.

Offset (O)

Parallel contour at a fixed distance.

1. Choose an element (line/circle/arc/rectangle/polyline/spline).
2. Move the mouse – the preview shows the side and the distance.
3. A click adopts the current distance, or enter **D** + Enter.

Scale

1. Select the elements (or click an element).
2. Click the base point (with snap).
3. Enter the **factor** + Enter, or drag with the mouse and click.

Stretch

Moves only the points inside a window (stretching contours).

1. Drag the **stretch window** with two clicks – points inside the window move, points outside stay fixed.
2. Click the base point, then the target point – or enter **dX/dY** + Enter.

Fillet (F)

Replaces a corner with a fillet arc.

1. Click the first element (line/circle/arc), then the second element.
2. Move the mouse – the radius and sector follow; a click places it. Alternatively enter **R** + Enter.

Chamfer

Replaces a corner with a bevel.

1. Click the 1st and the 2nd line, each on the side that is to be kept.
2. In the floating field, enter either **D1/D2** (two distances) or ∠**/D** (angle + distance). ⇄ swaps D1/D2, → changes the mode.
3. A click or Enter adopts the values.

Explode

Breaks compound elements down.

1. Either make a selection (all rectangles/polylines/splines in it are broken down) or click a single such element – it is dissolved into individual lines.

26.4 Cutting with Ctrl+X

Ctrl+X cuts out the selected elements: they disappear from the drawing and at the same time lie in the clipboard. With Ctrl+V you place them again at the mouse position. Together this is the most convenient way to move a selection.

- Select the elements, press Ctrl+X.
- Press Ctrl+V — the part hangs at the mouse pointer.
- A click places it. Further clicks place further copies; Esc finishes.

Dimensions and hatches belonging to the elements that were cut travel with them and come back again when they are pasted.

Unlike copying with Ctrl+C, cutting does not ask for a base point: the original has already been deleted, after all. Instead the part hangs centered at the mouse pointer.

A single Ctrl+Z brings everything that was cut back to its old place.

Note: if only dimension values or hatches are selected without the elements they belong to, nothing is cut — a message explains why. Dimensions and hatches always travel with their elements.

27 Annotate

27.1 Dimension (D)

Creates dimensions.

1. Click an element (line, rectangle, circle/arc …) – a suitable dimension (length, radius, diameter …) is proposed.
2. Position the dimension line and place it with a click.
3. For an **angular dimension**, click a second line and choose the sector.

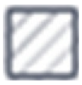
Text (X)

1. Click at the insertion position and enter the text; font, size and alignment can be adjusted in the properties panel.

Leader

1. Click = arrow tip (reference point).
2. Further clicks set the bend points; enter the text.
3. The arrow side, arrow size and fill can be set in the properties.

Hatch (H)



When it is called up, the floating hatch window opens.

1. **Automatic** (default): click inside a closed area – the boundary and the islands are detected automatically.
2. **Manual** (radio button): click the boundary elements one by one, then **Enter** or "Hatch" – useful when the automatic detection fails.
3. In the window, set the pattern, angle, spacing, color and opacity; all changes take effect immediately.

27.2 Item balloon, datum feature and feature control frame

Alongside text, remarks and leaders there are three further kinds of annotation.

27.2.1Item balloon

A circle with the item number from the parts list that is attached to a block instance. If the item number changes in the parts list, the balloon follows.

[FIGURE: drawing with item balloons on several blocks]

27.2.2Datum feature and feature control frame (form and position)

For form and position tolerances to ISO 1101 there are two tools: the datum feature (A, B, C …) and the feature control frame. The frame is assembled from a symbol, a tolerance value and datums, and is attached to an edge.

[FIGURE: feature control frame with a datum feature on a component]

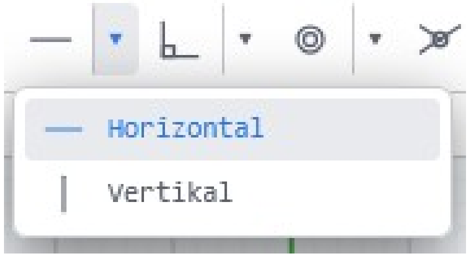
27.2.3Properties of the leader

When a leader is selected, the text, arrow size, text size, text offset, fill of the arrowhead and the arrow side can be set in the panel on the right. Every change is visible immediately and can be undone with Ctrl+Z.

28 Constraints (geometric conditions)

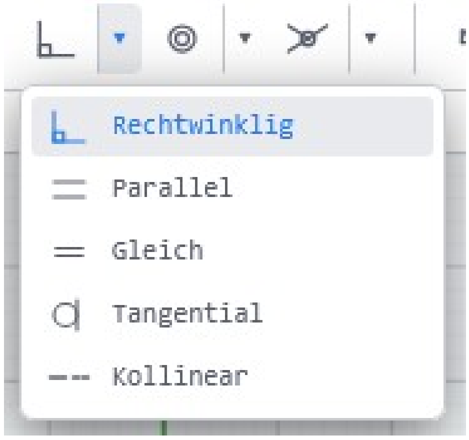
* Experimental * Constraints define relationships between elements that are preserved while editing. Right-clicking a constraint symbol deletes it.

28.1 Horizontal / vertical



1. Click a line – it is forced into the horizontal or the vertical.

Perpendicular · parallel · equal · collinear · concentric

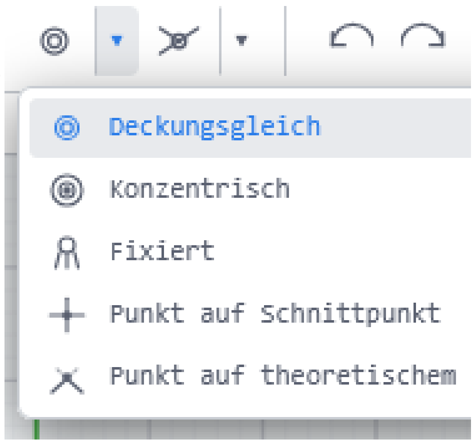


1. Click = reference element.
2. Click = second element – it is made perpendicular/parallel/equal in size/placed on the same straight line, or given the same center.

Tangent

1. Click a circle/arc, then a line or a further circle – both are kept tangential.

Coincident



1. Click = element near the endpoint that is to be moved.
2. Click = target: an endpoint (point on point) **or** a position on another element (the point then slides along it).

Fixed

1. Click an element – it is anchored completely (it can neither be moved nor changed in its dimensions). Release it by right-clicking the ⚓ symbol.

Point on intersection

1. Choose a point, then element A and element B – the point moves onto their intersection.

Point on theoretical intersection

1. Choose a point, then line A and line B – the point sits on the **imaginary** intersection of the extended lines.

Weld / unweld

1. Select the elements.
2. **Weld** joins their common endpoints permanently; **unweld** releases that again.

29 Action

Function	Description
Undo (Ctrl+Z)	Take back the last step.
Redo (Ctrl+Y)	Restore a step that was taken back.
Delete (Del)	Remove the current selection.
Zoom to fit (Ctrl+F)	Adjust the zoom so that everything is visible.
Redraw (F5)	Rebuild the screen display.

30 Settings

Dimension	Font size, arrows, decimal places and style of the dimensioning.
Hatch	Floating window: pattern, angle, spacing, color, opacity, area selection.
Setup	Save/load/export/import the settings; among other things the cursor distance of the floating field.
Theme	Light/dark display mode.
Info	Version and program information.

31 Edit dimension

The **Edit dimension** dialog is used to design the text of a dimension and – with driving dimensions – to control the geometry via the numerical value. Here you can add a prefix and a suffix (Ø or an assembly note, for example), apply tolerances in various notations and change the dimension value.

Calling it up: double-click an existing dimension. The input focus is then immediately on the **Value** field (its content selected), so that a new value can be typed straight away. **Enter** confirms the dialog, **Esc** cancels.

31.1 Preview

At the very top the dialog shows a **live preview** of the finished dimension text, exactly as it will later appear in the drawing – including prefix, suffix and stacked tolerance. Every change in the fields below is adopted immediately.

31.2 Prefix

Text **before** the dimension value. The two quick buttons on the right let you set the prefix with a single click:

- **Ø** – diameter symbol (for holes and shafts)
- **R** – radius marking (for radii)

Any text of your own can also be entered.

31.3 Value driving or driven

The actual dimension value. To the right of it there is a button that switches between two operating modes:

- **driven** – the dimension is a *reference*: it measures the existing geometry and shows its actual value. If the geometry changes, the dimension automatically changes with it.
- **driving** – the dimension *determines* the geometry: the value entered defines the length, the distance or the angle, and the associated element is adjusted accordingly.

If the value in the field is changed, the dimension switches to **driving** on its own – the geometry then follows the value entered. If the value stays unchanged (because only the prefix or the tolerance is adjusted, for example), the previous operating mode is retained. With dimensions on fixed elements the value field is locked; there the dimension serves exclusively as a reference.

31.4 Suffix

Text **after** the dimension value, for example a machining note such as "Drill during assembly" or a unit.

31.5 Tolerance

The selection field defines the type of tolerance. Depending on the choice, the matching input fields are displayed:

- **none** – no tolerance is shown.
- **symmetrical (±)** – a single value that stands as ± after the dimension (50 ±0.1, for example).
- **upper/lower deviation** – the upper and lower deviations are entered separately and stacked in small type one above the other after the dimension.
- **tolerance class** – only the short ISO designation of the fit is shown (H7 or g6, for example), without numerical values.
- **class + deviations** – the class is shown and in addition the upper and lower deviation are set in small type in brackets below it (H7 with the deviations +0.035 / 0, for example).
- **limit sizes** – instead of the deviations, the calculated largest and smallest size are given one above the other.

The input fields are arranged below: on the left **Class** (with the **determine** button), on the right **upper** and **lower** for the upper and lower deviation.

31.5.1Determining the deviations automatically

With the tolerance types *class + deviations* and *limit sizes*, the **determine** button calculates the deviations to **ISO 286** automatically from the tolerance class entered and the nominal size (the value):

- A class with **capital letters** (H7, G7, for example) is evaluated as a **hole**.
- A class with **lower-case letters** (g6, h6, for example) is evaluated as a **shaft**.

The table values come from the file **toleranzen.json**, which is loaded together with the program. It can be extended with further classes or nominal size ranges if required; a combination not stored there is reported by the program in the status bar.

31.6 Confirming

- **OK** (or **Enter**) adopts all entries.
- **Cancel** (or **Esc**) discards the changes.

Example

In the case illustrated, the prefix **Ø**, the value **88.00**, the suffix **"Drill during assembly"** and the tolerance type *class + deviations* with class **H7** (determined deviations **+0.035 / 0**) produce the dimension text:

Ø88.00 Drill during assembly H7 ($^{+0,035}$ / $_{0}$)

31.7 Note

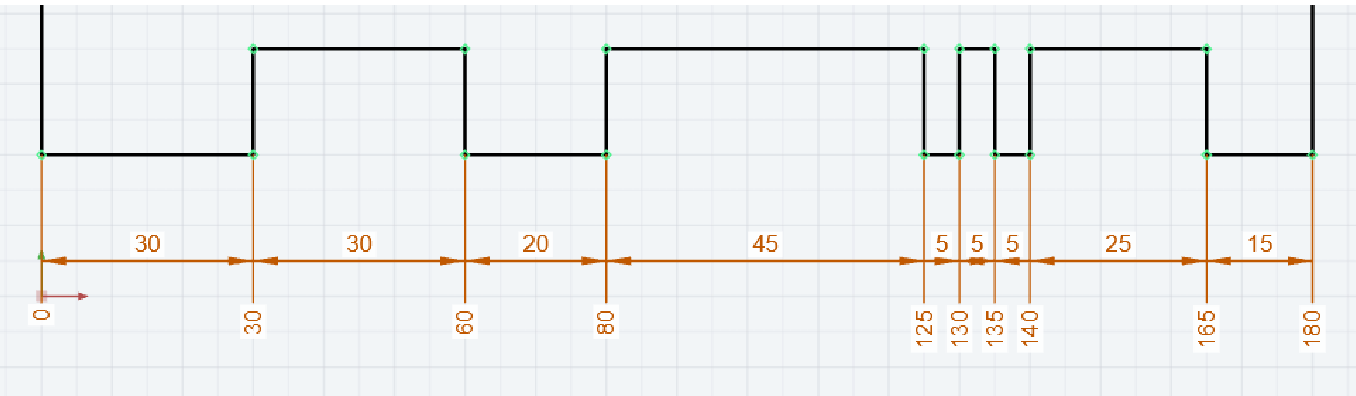
The general appearance of the dimension value – the number of decimal places and the hiding of trailing zeros – is not defined here but in the **dimension settings (Dim.)**, and applies to all dimensions.

31.8 Further dimension types: chain, baseline, arc

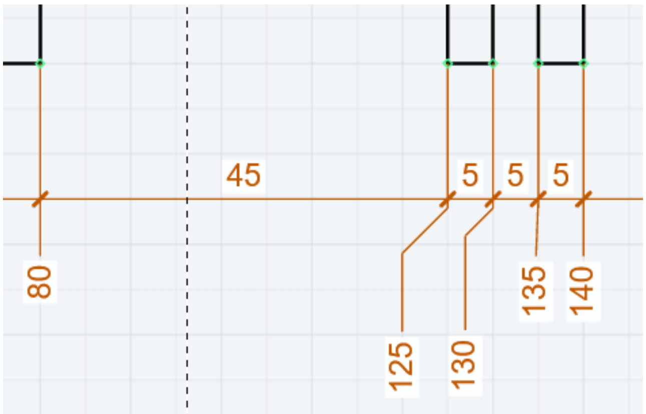
Alongside the familiar dimension types, three further ones are available. You find them in the toolbar with the dimensions.

31.8.1Chain dimension

With the chain dimension, the individual dimensions line up one after another on a common dimension line. You click the points one after another; every new dimension joins onto the previous one.



[FIGURE: chain dimensioning across several points]



A dimension chain is one unit. If you move one link, the whole chain follows — all links therefore always lie at the same height. If you reposition an extension line point, the common point of the neighboring link moves with it so that the chain does not break apart.

In the dimension settings, a total dimension from the origin can additionally be displayed («Chain dimension: total dimension from the origin»).

31.8.2Baseline dimension

With the baseline dimension, all dimensions start from a common reference point and are staggered one above the other.

[FIGURE: baseline dimensioning with staggered dimension lines]

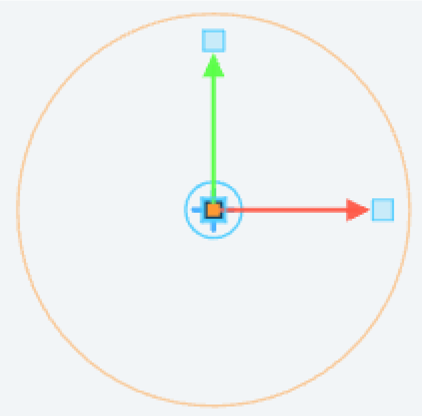
31.8.3Arc length dimension

The arc length dimension dimensions the length of an arc, not the chord. The dimension line follows the arc, as do the arrowheads; the dimension value sits on the arc.

[FIGURE: arc length dimension on an arc]

32 Gizmo

32.1 1. What the gizmo is



The **gizmo** is the tool with which you **move, rotate, scale and mirror** a selection — either with the mouse or by entering exact numerical values. It consists of two parts that always belong together: the **graphic on the drawing area** (a pair of axes with arrows, a rotation ring, a move square and a reference point marked in red) and the **input window** beside it:

The red reference point is called the **pivot**. It is the point about which rotating and scaling take place, and whose absolute coordinates the window displays. Everything the gizmo does refers to it.

The figure above shows the window in a typical position: it is aligned to a line (display U/V), and because exactly one line is marked, the third row carries a length field instead of a scale field — see section 6 on that.

32.2 2. When the gizmo appears

The gizmo appears as soon as at least one element is marked in the select tool. It stays hidden as long as you are drawing, dragging a grip, using another tool, or while the context menu is open.

With **Ctrl+Shift+G** you switch the gizmo off and on again completely. The × button at the top right does the same; the status bar then reminds you of the shortcut. With ▼ the window can also merely be **collapsed** so that the graphic remains available — this state is retained across sessions.

32.3 3. The graphic on the drawing area

Part	Color	Meaning
Target mark at the origin	red	the pivot — ring, crosshair and center point, always the same size on screen
Axis with an arrowhead	red	first axis (X or U)
Axis with an arrowhead	green	second axis (Y or V)
Square at the origin	cyan	free movement in both directions
Ring around the origin	orange	rotating; when hovered over, a scale appears every 15 degrees
Small boxes at the ends of the axes	cyan	scaling
Highlighted line	orange, pale	the line the gizmo is currently aligned to
Ring with a crosshair	violet	point mode (see section 8)

All grips have **fixed screen sizes**: they are equally easy to hit at any zoom factor. The grip under the mouse pointer is highlighted before you click.

32.4 4. Working with the mouse

Free movement — grab the cyan square at the origin and drag. The selection follows in both directions.

Moving along an axis — grab the red or green axis shaft. The movement is forced onto that axis. At the same time the text cursor automatically jumps into the matching numerical field of the second row, so that you **can also type the distance** instead of dragging: the value you type overrides the mouse position immediately, and Enter completes the move.

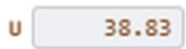
Rotating — grab the orange ring. The selection rotates about the pivot; dimensions, hatches and constraints are carried along.

Rotating only the axes — drag the ring with the Shift key held down. Then only the pair of axes rotates, and the geometry stays unchanged. That is the way to go if you subsequently want to move at an angle to the world.

Scaling — drag one of the cyan boxes at the ends of the axes. The mapping is one to one: the distance the box travels corresponds exactly to the growth of the selection in that direction.

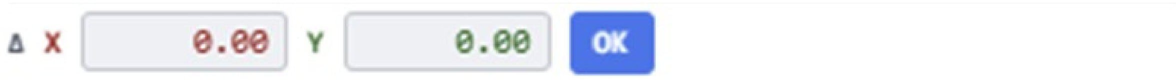
32.5 5. The input window in detail

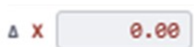
32.6 Row 1 — position of the pivot

Detail	Meaning
	Grip: move the window with the mouse button held down; a double-click makes it follow the element again
	Mode display: WORLD (horizontal axes), U/V with an angle (aligned or rotated), or POINT with the element type
	absolute X or U coordinate of the pivot — type a value and press Enter to place the pivot and the selection exactly there
	the same for Y or V
	pin the window position (see section 10)
	expand and collapse the window
	hide the gizmo (Ctrl+Shift+G shows it again)

The two coordinate fields are the fastest way to place an element at a known coordinate: type it in, press Enter — done.

32.7 Row 2 — displacement



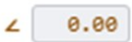
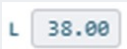
Detail	Meaning
	displacement along the first axis (X or U)
	displacement along the second axis (Y or V)

Detail	Meaning
	transfer the typed values to the selection; Enter in the field does the same

The delta sign in front indicates that this is a **displacement** and not a position. While dragging, both fields run along live; afterwards the values stay put and can be reworked.

What matters is the bookkeeping behind it: the gizmo remembers how much has already been applied and calculates the **difference** for every entry. So if you type 10 and then 12 one after another, the selection moves by 12 in total — not by 22. If you want to specify the absolute position instead, use the fields in row 1; they then take precedence.

32.8 Row 3 — rotation, size, axes

Detail	Meaning
	rotation about the pivot in degrees; with a single line, its absolute angle to the horizontal instead
	scale factor (1 = no change); with a single line the length in millimeters (the case in the figure above), with a circle or arc the radius
	absolute direction of the pair of axes in degrees — changes only the axes, not the geometry
	align to a line: the next click on a line aligns the axes to it (Ctrl+click on a line does the same directly)
	restore the world axes — back to horizontal and vertical
	set the pivot anew: the next click in the drawing defines the reference point, with object snap
	point mode: the next click on an endpoint places the gizmo there; pressing it again ends the mode
	mirror the first axis — reverses the direction of X or U
	mirror the second axis — reverses the direction of Y or V

A selection that has been started (aligning, pivot, point) can be cancelled at any time with

Escape. The buttons for mirroring and point mode stay highlighted in color as long as they are in effect.

32.9 Row 4 — layer

The list shows the **common layer of the selection** — "SICHTBAR" in the figure; with mixed layers three dots appear. If you change it, all marked elements move onto the chosen layer and adopt its color and line width, exactly as when changing the layer in the properties panel.

32.10 6. Special behavior depending on the selection

The gizmo adapts when exactly **one** element is marked.

A single line. The axes align themselves to it automatically — which is why the screenshot above shows U/V instead of X/Y. The angle field then no longer contains a rotation but the **absolute angle of the line to the horizontal**; a value entered rotates the line to exactly that angle. The scale field becomes the **length field** and carries the label L:

— it shows the current length in millimeters, and a new value lengthens or shortens the line **on one side**. Which end stays put is clearly defined:

- If exactly one end is welded to another element, that end stays fixed and the free one moves.
- Otherwise the end on the back side of the arrow direction stays put; swaps the sides.

If **both ends are welded**, there is no end that could give way. Length and angle are then refused, and a short message appears directly above the field concerned — right where you are looking anyway while typing. Moving remains permitted, because there the whole line travels.

A single circle or arc. The same field becomes the **radius field** and carries the label R: it shows the current radius, and a new value changes it while the center stays put.

A single block instance. The pivot sits on the **insertion point** of the block by itself — that is, the point at which the block hangs on the mouse while being placed.

Several elements. The pivot sits in the middle of the enclosing rectangle as long as you have not set it yourself.

32.11 7. Setting the pivot

With you define the reference point freely. The click that follows **always works with object snap**, regardless of the snap mode that has been set: endpoints, midpoints and intersections are preferred; if the click hits no snap point but does hit an element, the point is cleanly projected onto that element. Only if neither applies does the bare click point count.

A pivot you have set yourself stays in place until you replace it. It is at the same time the basis for the insertion point when creating a block.

32.12 8. Point mode

In point mode the gizmo is placed on an **endpoint** of an element, and the axes move **only that one point** — the rest of the element stays put. This lets you pull the end of a line by an exact amount, for example, without hunting for grips.

Two things are important here:

1. **Points lying on top of each other move along.** If several elements touch at that point, they stay connected — exactly as when dragging the grip.
2. **Only the pair of axes is rotated.** A single point cannot be rotated; in point mode the ring therefore always rotates only the gizmo itself and turns violet so that the difference is visible.

In that case the display on the left of the window reports POINT together with the element type, and the button stays highlighted as long as the mode is running.

32.13 9. What all travels along when moving

The gizmo does not move only the marked lines and arcs. The following are carried along as well:

3. **dimensions** that belong to the selection or travel freely with it,
4. **hatches** whose boundary moves along,
5. **constraints**, which are resolved again after the move.

That applies equally when dragging with the mouse and when applying typed values. Elements on locked layers stay untouched.

32.14 10. Keeping the window under control

The input window normally follows the marked element. If you push it aside by its grip  , the new position initially applies only to the current selection — with the next selection it jumps back to the element. A **double-click on the grip** restores the following behavior immediately.

If you want to keep the window permanently in a particular place, switch **pinning** on with  . It then stays there even when you mark something else. The setting is saved and still applies at the next start; the tooltip of the button names the current state and what a click will do.

32.15 11. Keyboard and operation at a glance

Input	Effect
Ctrl+Shift+G	switch the gizmo off and on again
Enter in a numerical field	apply the value; during an axis drag, Enter completes the move at the typed value
Escape in a numerical field	discard the entry, the original value appears again
Escape with a selection started	cancel aligning, setting the pivot or point mode
Tab	jump from field to field (fixed order through all rows)
Shift and dragging the ring	rotate only the axes, the geometry stays put
Ctrl and clicking a line	align the axes to that line
Double-click on the window grip	the window follows the marked element again

All numerical fields calculate: you may type in an expression, and comma and point count equally as the decimal separator.

32.16 12. Practical tips

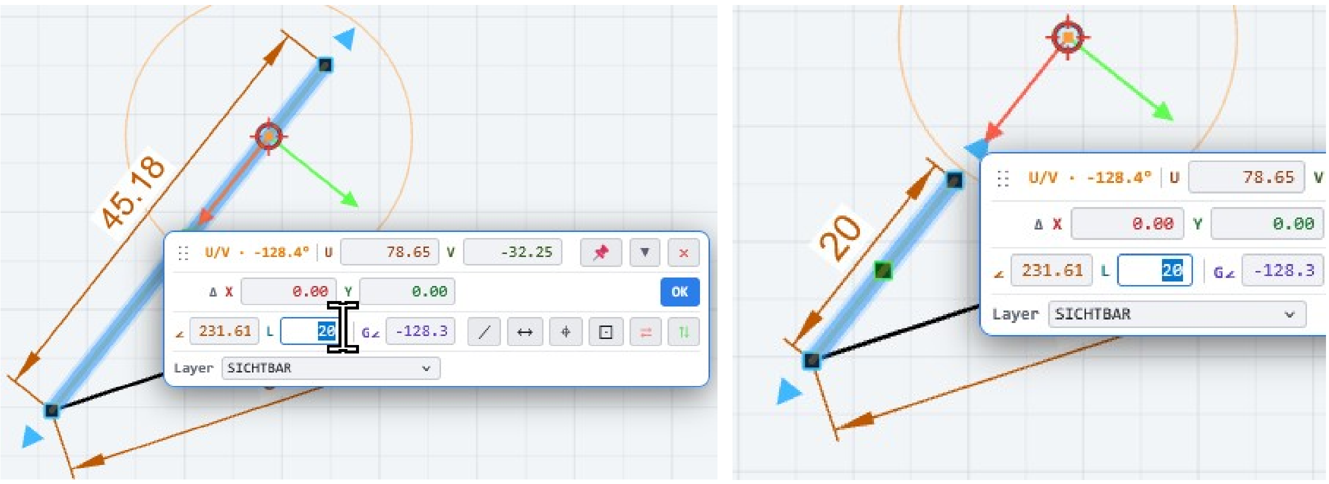
6. Set the pivot deliberately before rotating — on the center of a hole or on a corner. That saves moving things afterwards.
7. To move at an angle to the world: first Ctrl+click a line (or rotate the pair of axes with

Shift on the ring), then drag along the axis or type the distance.

8. For exact positions, going via the absolute coordinates in row 1 is the most reliable route — they always refer to the world, regardless of what was applied before.
9. With a single line you handle length and angle entirely via the two fields of the third row; you do not need to drag for that.
10. If a weld at both ends blocks the change, first release one of the welds — the message above the field tells you so in plain words.

32.17 The gizmo with welded elements

If you use the gizmo's input fields to change the length or the angle of a line one of whose ends is welded, the following applies: the welded end stays fixed and the free end moves or rotates about it. The connection to the neighboring element is thereby retained, regardless of how the gizmo is aligned.

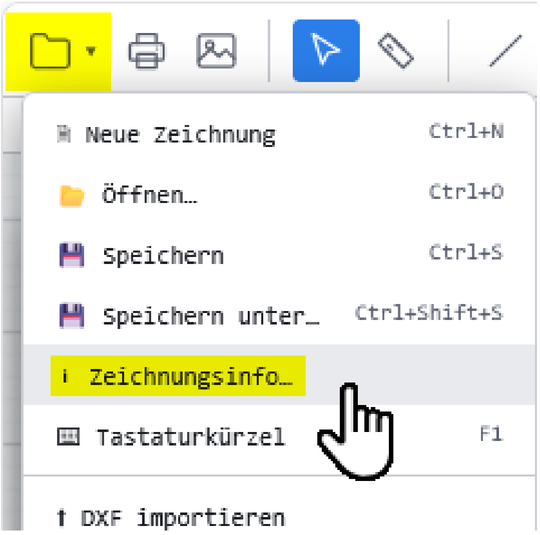


[FIGURE: line with a welded corner before and after a change of length]

With a line without a weld, the previous behavior applies: the direction of the gizmo axis decides which end stays put.

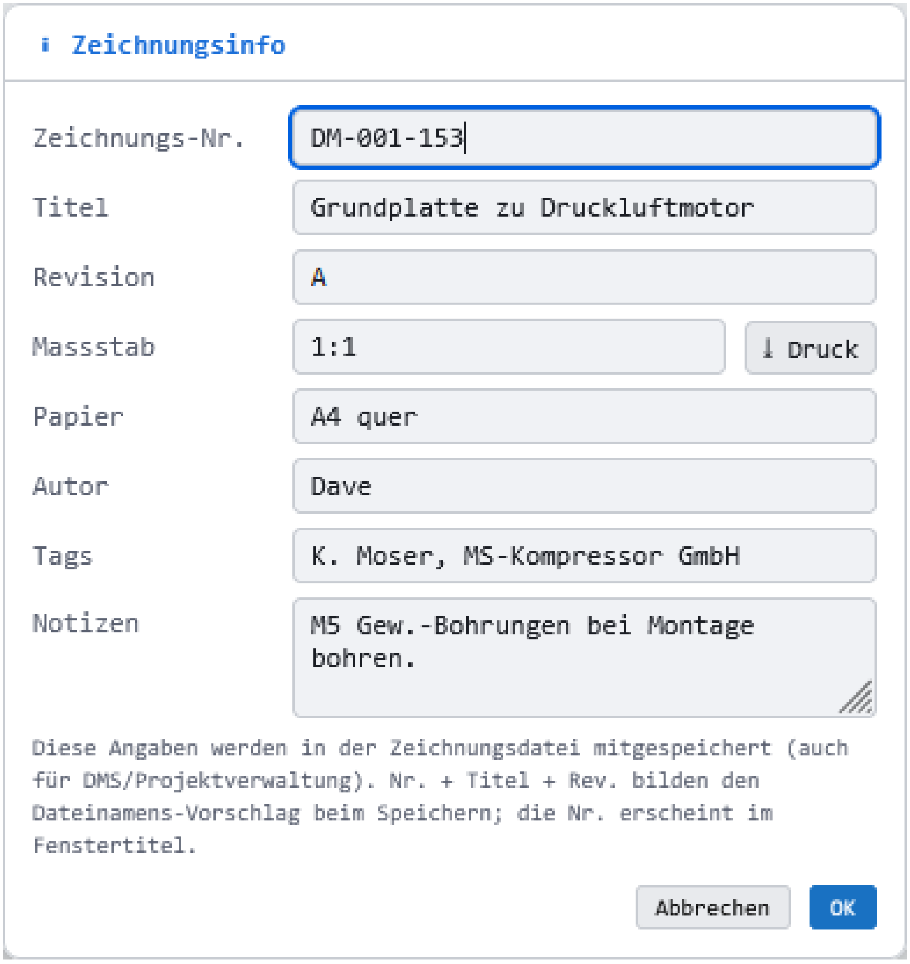
If BOTH ends are welded, there is no free end that could give way. Length and angle can then not be changed; a message appears briefly above the field concerned with the reason and the way out — release one end. Moving via the fields for ΔU and ΔV remains possible, because there the whole line travels and no connection is torn apart.

33 Procedure for creating a drawing



Before you start a design drawing, it is advisable to pass the known **drawing information** to the file right away.

Tip: in ***LiberoDraft DMS*** *you can search by all fields and find your drawings again in a fraction of a second. Guaranteed!*



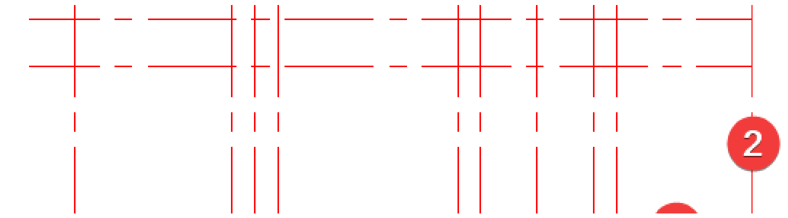
Drawing information is read out and managed directly by our DMS/project management **LiberoDraft DMS**.

Info: if you would like to manage the LiberoDraft drawings in your own system, it should be mentioned that **LiberoDraft** stores this information as **plain text** in the drawing file.

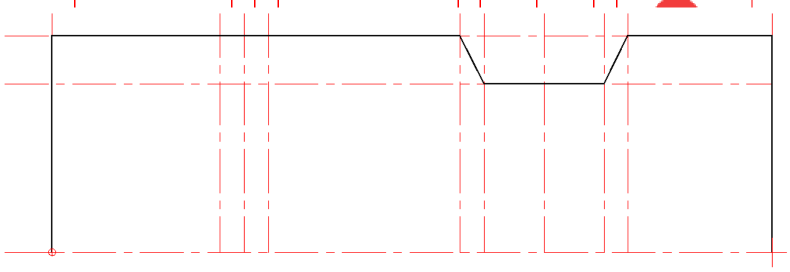
The following example is intended to show the working steps involved in creating technical drawings.

A shift claw is drawn at a scale of 1:1 in front view with a half section and dimensioning:

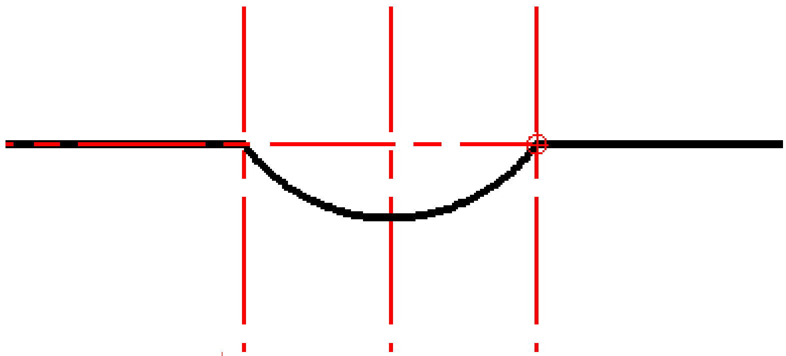
Create all edges and the center line.
Start with line **(1)** and **(2)**



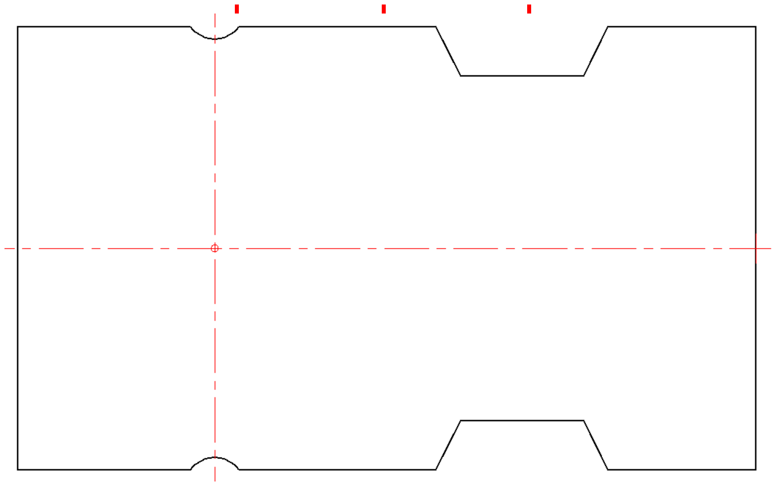
Create the contour including the chamfers



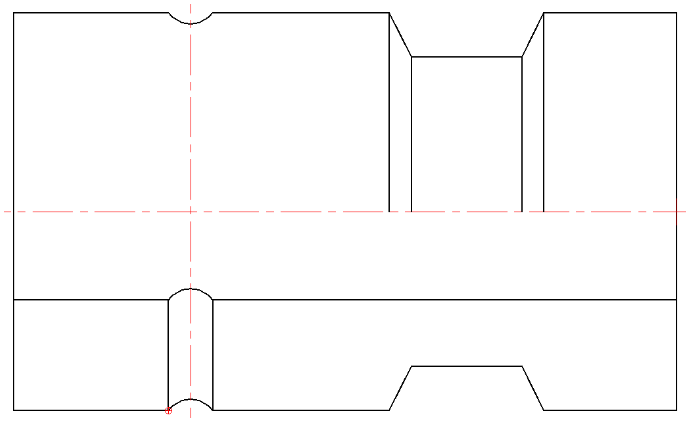
Construct the hole with a circular arc



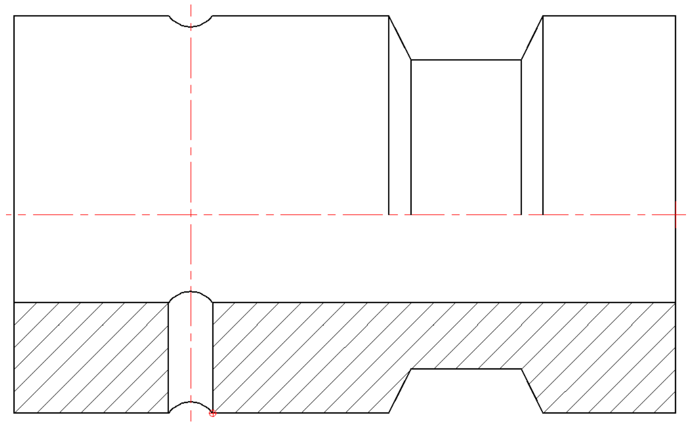
Mirror the contour



Construct the circumferential edges and the inner hole and create the hole edges



Prepare and activate the hatch areas, generate the hatch



Oops!!! The forgotten inner chamfer has to be added. To do so the hatch must be deleted, the chamfer drawn, hatched again and dimensioned...

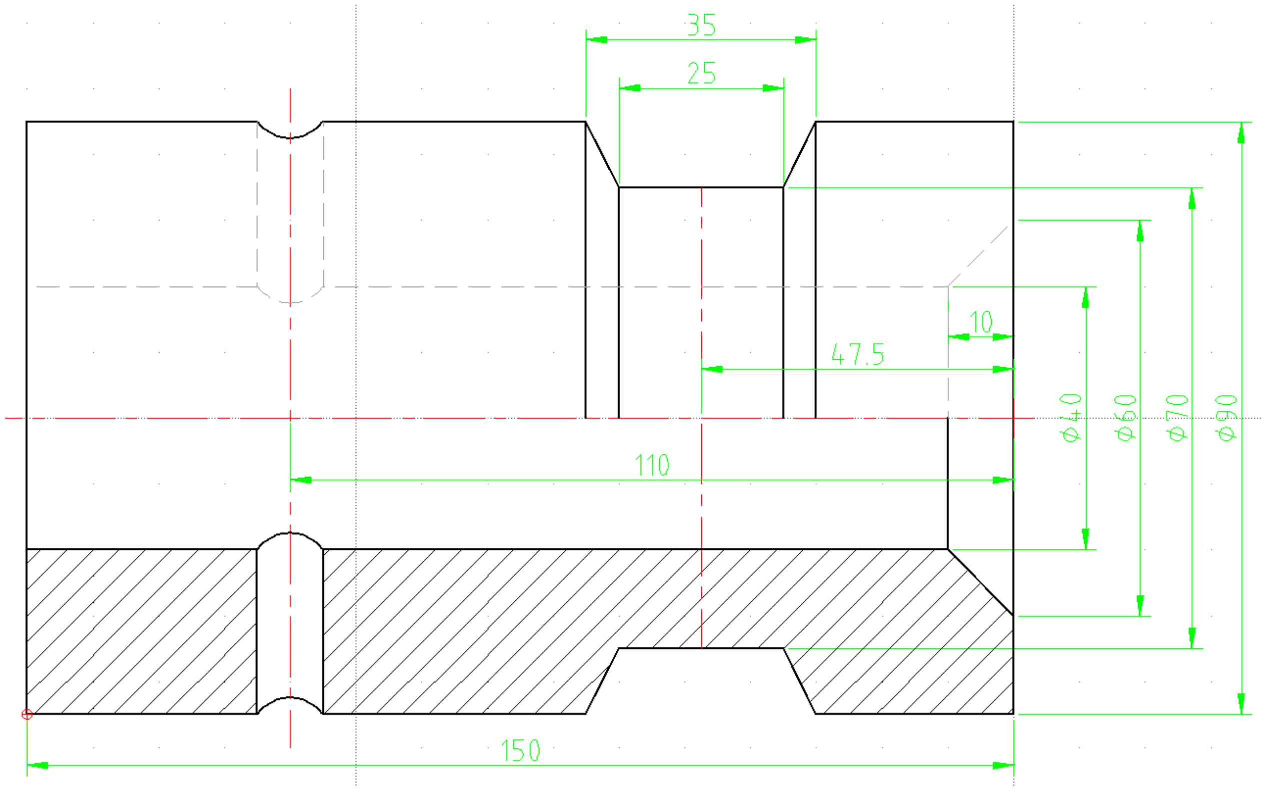
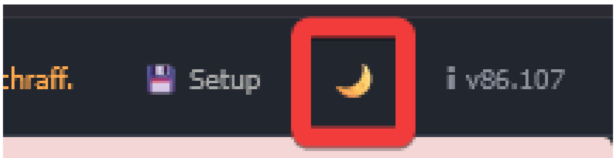


Diagram 1: source: Zeichnen für technische Berufe, HT 3241

34 Your first drawing on your own

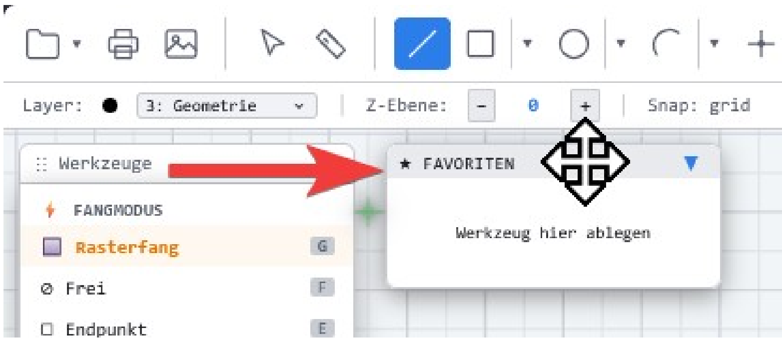
Create a new drawing. LiberoDraft shows an empty window. The origin lies in the middle of the drawing area.

In this book we work in day mode. You can switch over to night mode at the top right:



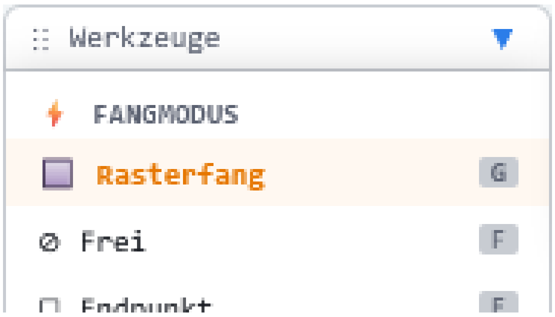
34.1 Favorites window

In the favorites window you can place your favorite functions from the menu bar by drag and drop. It is best to move the window right next to the tool window straight away.



34.2 First of all, something important about the grid snap

Please use the grid snap [G] with care while drawing. It is mostly used only for the **first** object, or when you deliberately want to draw on the grid. The grid snap is also frequently used when positioning dimensions and texts.



Reason:
When designing in a technical context, where you want to snap **not the grid** but mostly **endpoints and intersections**, the grid snap can – to put it bluntly – have «traumatic effects».

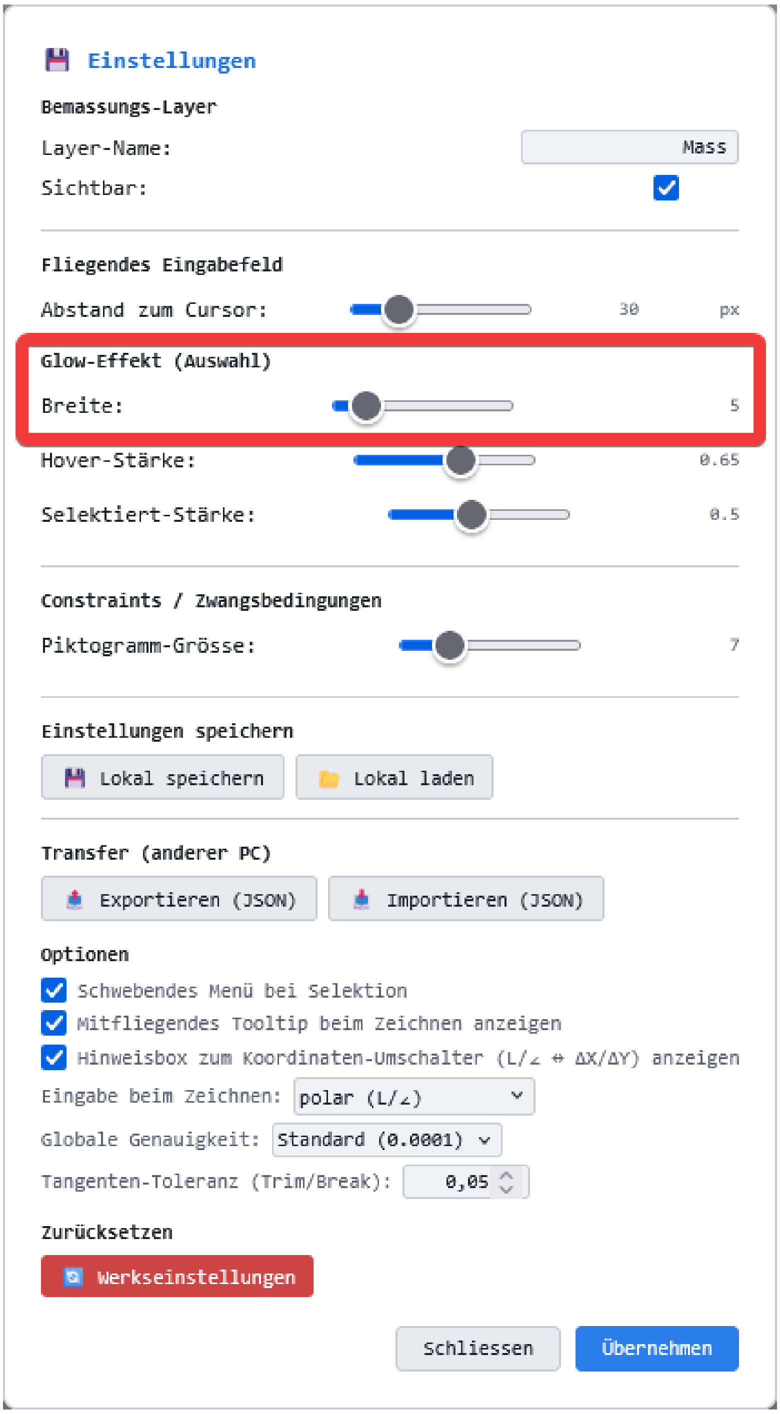
If a grid point lies very close to an important intersection or endpoint, it can happen that the wrong point is snapped, which has negative consequences.

A point snapped wrongly will make itself felt negatively in the design history as the work goes on and has to be **taken back completely**.

That means you have to correct everything in such a drawing until it is right, and with extensive drawings that can lead to considerable extra work. This manual therefore recommends activating **the grid snap with care.**

34.3 First settings

We briefly recommend making the following basic setting in Setup:



Depending on your screen resolution the glow effect may be too large, so it is best to start with the value 7.

All other settings are OK.

Save the settings locally.

Transfer the settings to another PC

.

Various settings...

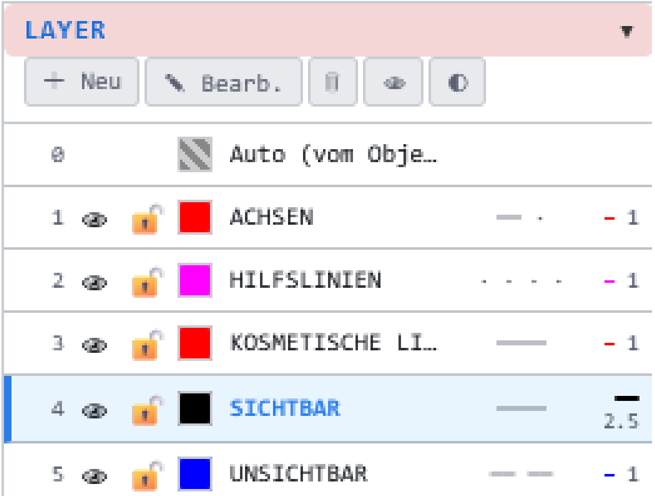
If you have "mis-configured" the program, the red button resets the whole program cleanly to the factory state.

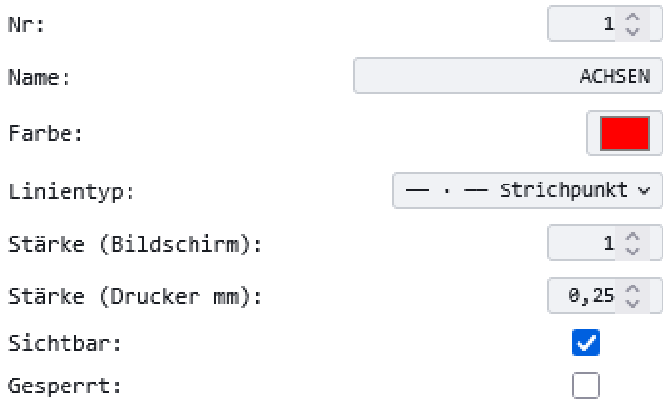
With Apply we are ready for the first drawing :-)

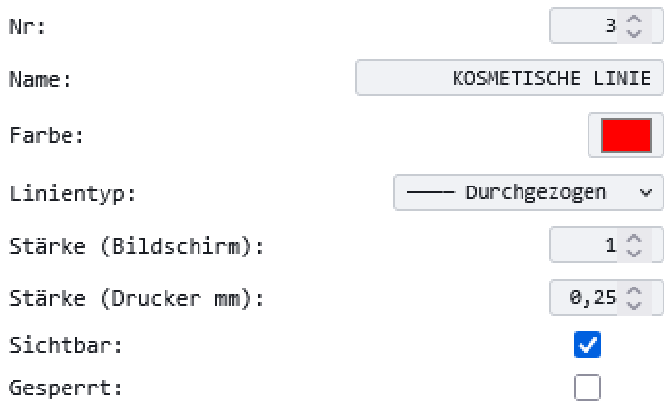
34.4 Layer settings

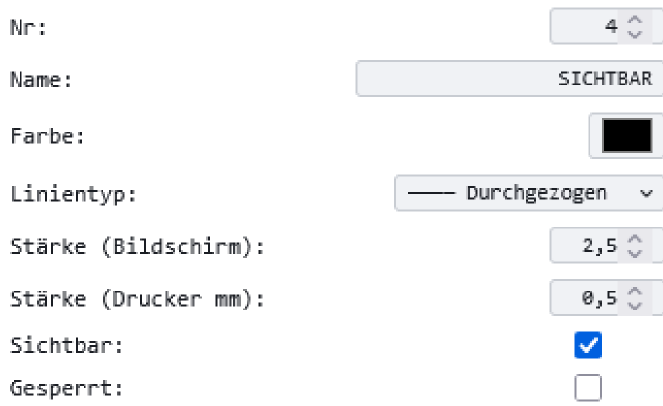
In the layer panel on the right you can manage the settings yourself.
For the further course of the book we set the layers up as follows:

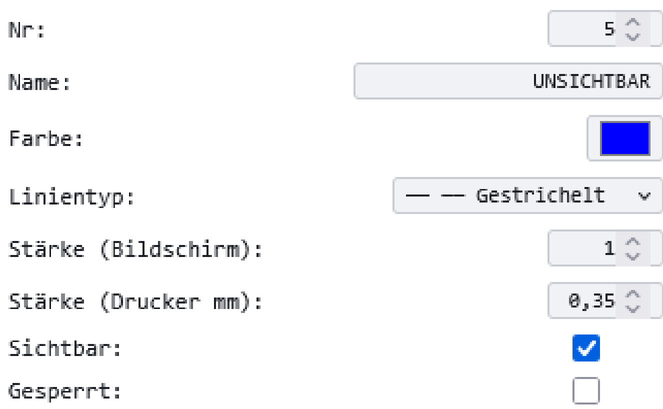
Click the button "+ NEW" or "Edit"







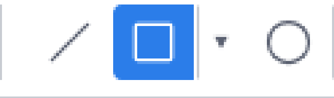




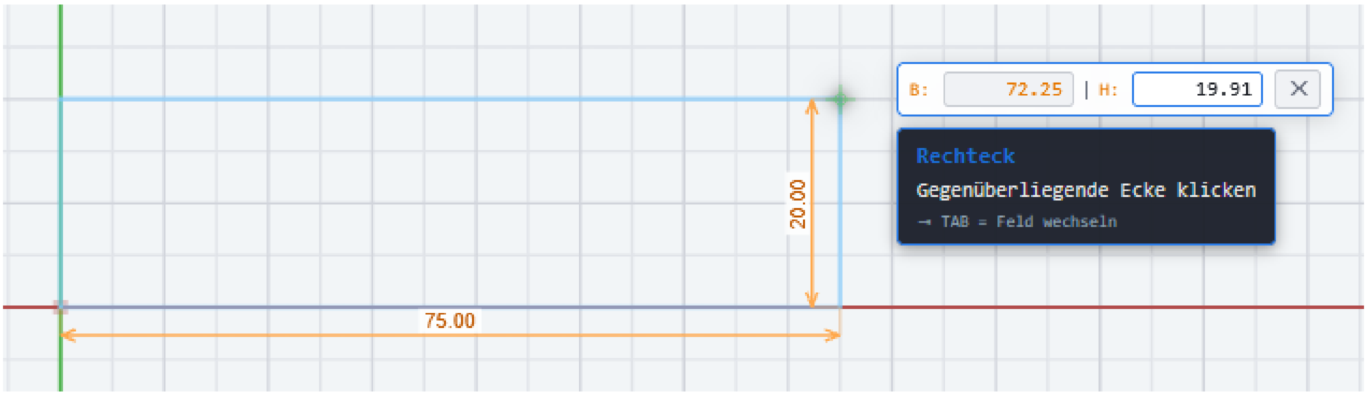
These details are only meant as a free suggestion that you can adapt yourself at any time.

Tip: if you would like a template of your own, save the empty sheet as **_Vorlage-001** or **_Template-001** so that you can load it at any time with those settings as a default.

34.5 Drawing a first rectangle...



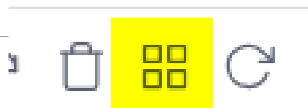
The rectangle starts at the **origin** with a **width** of **72.25** mm and a **height** of **19.91** mm. In this case the first point can be set at the origin with the **grid snap [G]** activated.



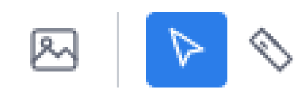
After the first point the rectangle can be dragged out and the floating input fields filled in directly. The input fields can be jumped to with the **Tab key [TAB]**.
In the W: field enter 72.25 → Tab key, and for H: enter 19.91
The Enter key draws the precise rectangle.

The rectangle is created, including its dimensions, as follows:

If the rectangle is shown only very small or not at all, click the icon:
"**Zoom to fit**"



or press the key combination **[Ctrl] + [F].** This zooms all existing drawing elements into the visible area.

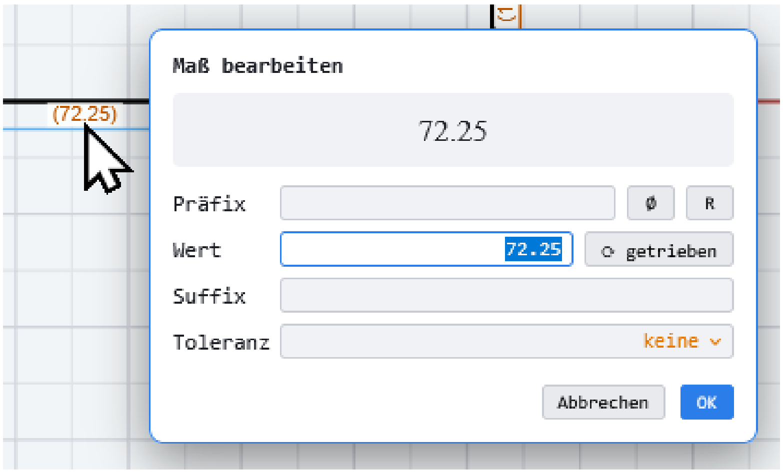


If you want to move a dimension or change it, that is only possible in **select mode**:

Tip: *pressing the [ESC] key several times likewise activates select mode.*

Now the dimension can be grabbed with the **LMB** and moved.

If you want to **change the dimension value**, you likewise have to be in select mode. With a double-click on the dimension value...

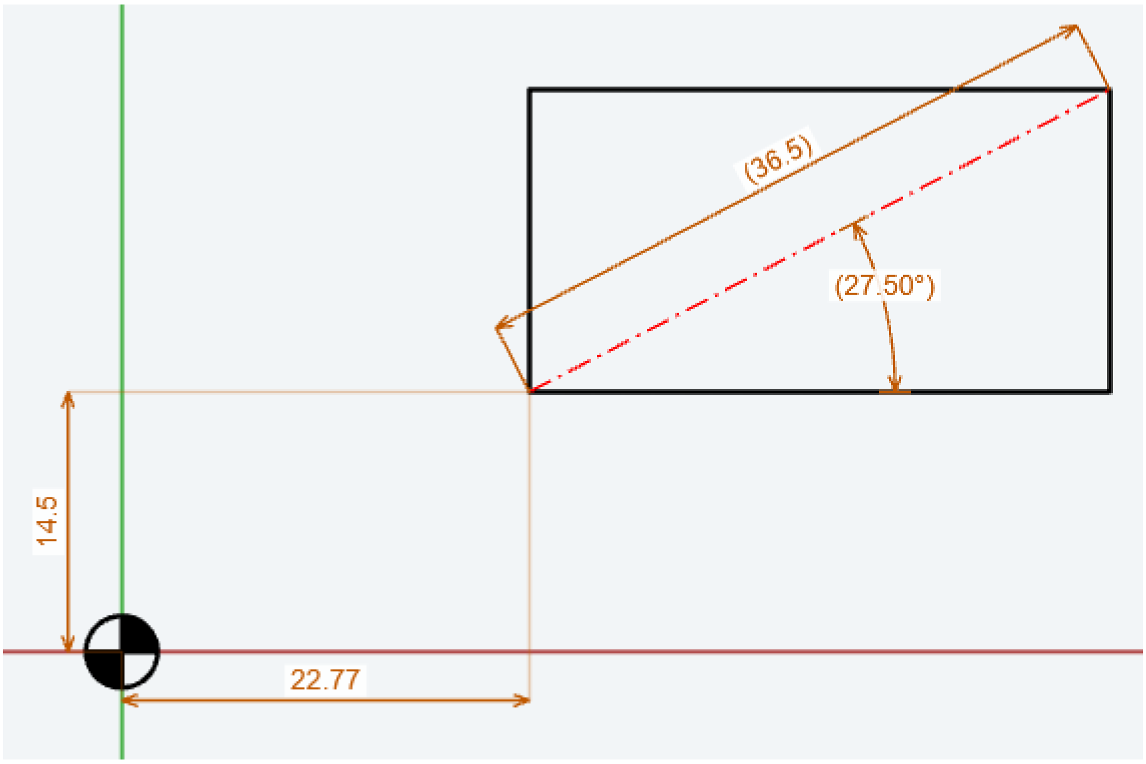


... the dimension can simply be entered anew and the geometry follows the dimension.

Whether a dimension is driving or driven is switched with the toggle button behind the value field.

34.6 Another rectangle with cartesian and polar coordinates...

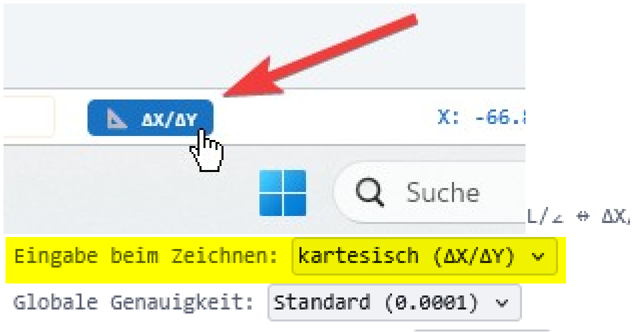
Now we draw a new rectangle, precisely at these coordinates:



LiberoDraft already has many functions that offer several ways of reaching the result.

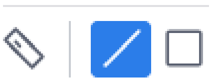
Make a mental note of switching between polar and cartesian coordinates. **Polar coordinates** are entered via a **distance and an angle**. **Cartesian coordinates** quite classically via an **X and a Y value**.

At the bottom in the status bar, this **toggle button** switches quickly between the two coordinate systems:



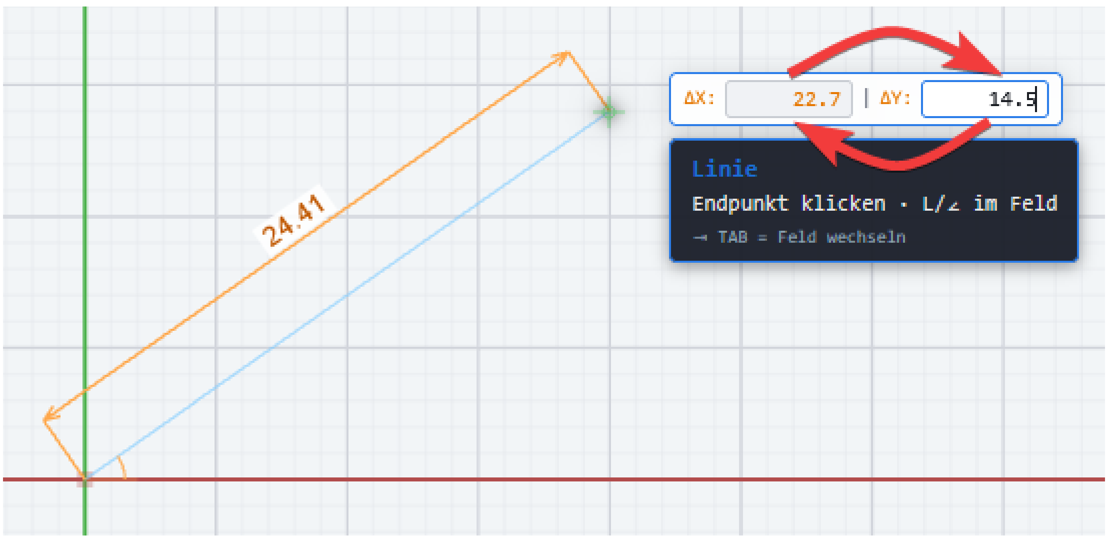
***Tip:** a default choice can be made in the settings:*

So let us simply take the following route:



Draw a line starting from the origin. The easiest way is to snap the origin with the **grid snap [G]** and drag the line out.

While dragging, a floating input dialog is displayed in which the values can be entered:



With the Tab key [TAB] you can jump back and forth between the fields.

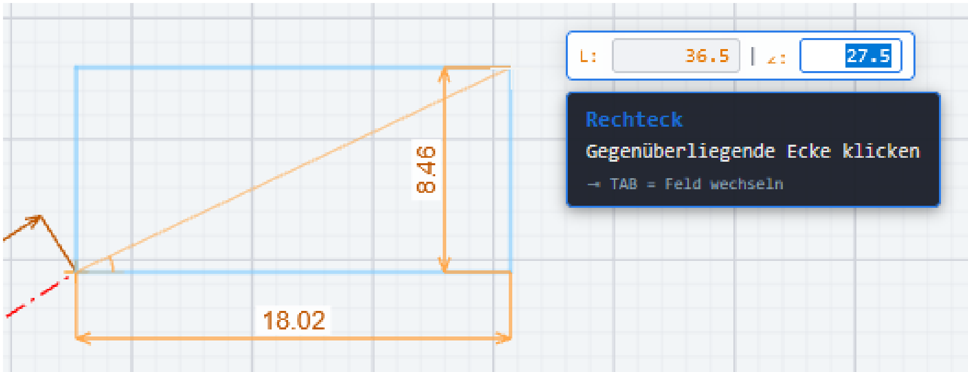
Every function can be cancelled with the **Escape key** [ESC] or with the **RMB**.

OK, now the **rectangle** with **polar coordinates** (switch the toggle button in the status bar)

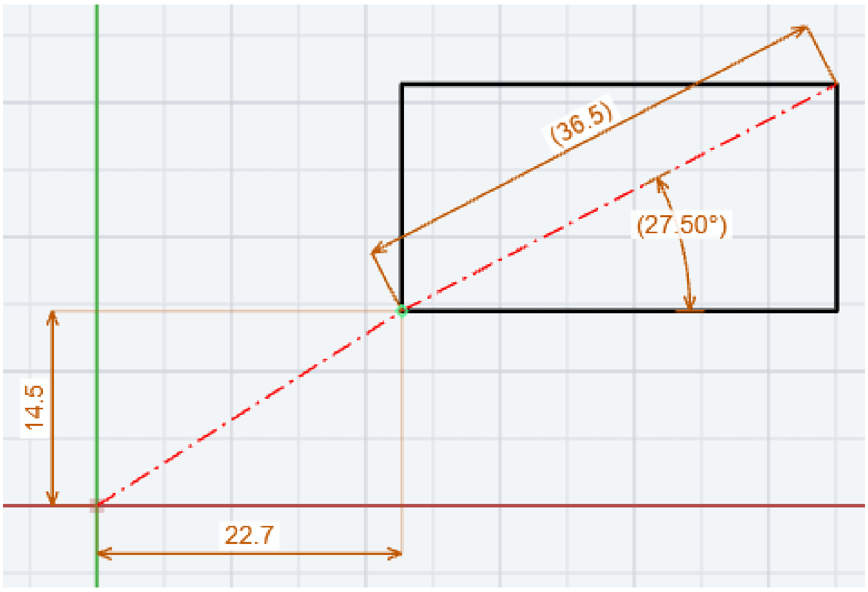

Choose the Rectangle function.

Snap the first corner of the rectangle cleanly with snap mode **Endpoint [E]**.

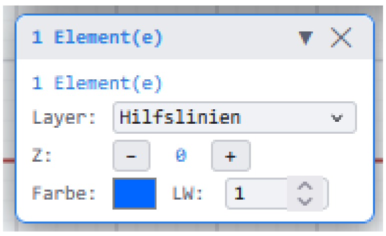
Enter the second corner point via the coordinates accordingly:



Done :-)



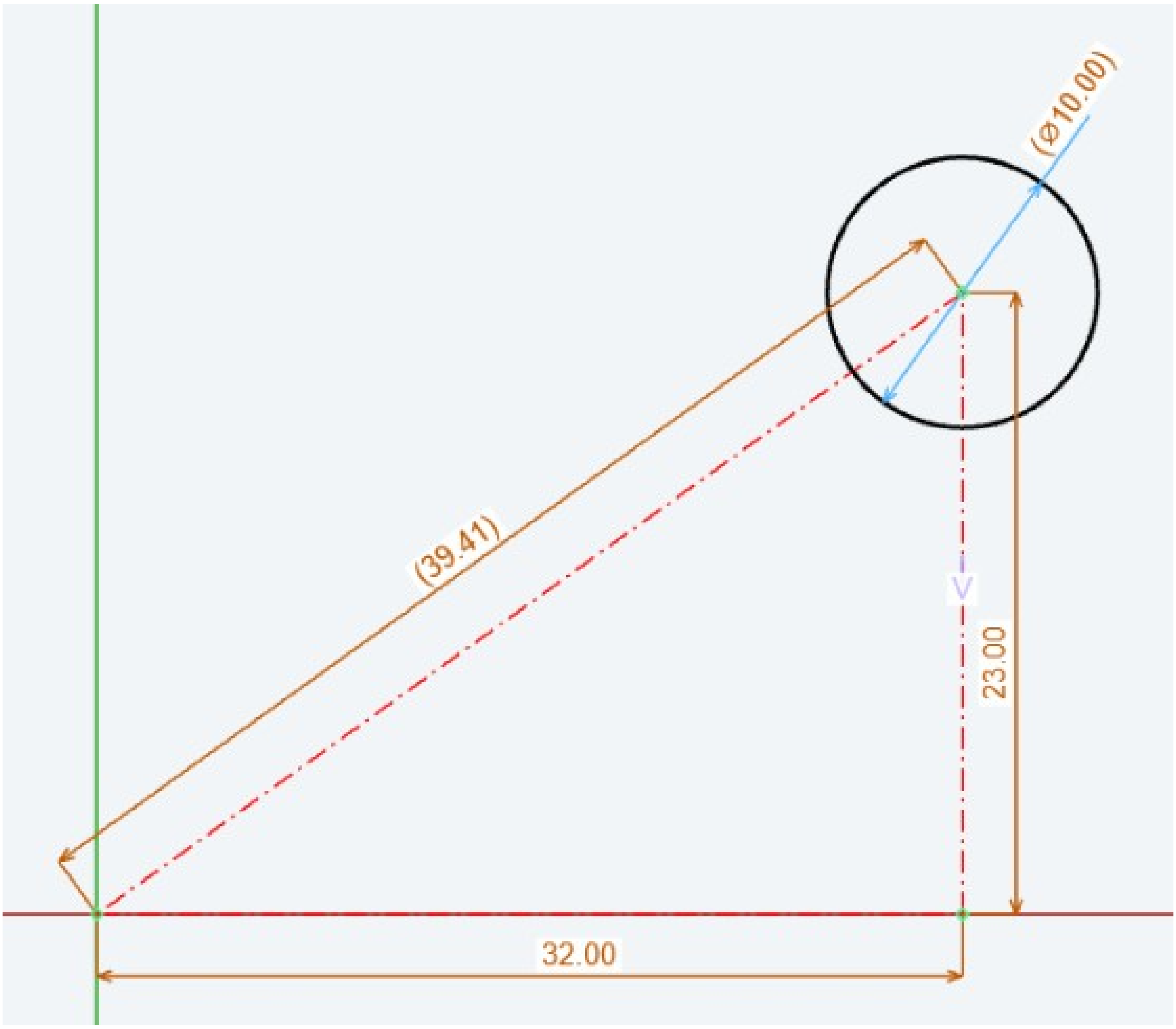
Set the **line types** by clicking the elements you have drawn and adjusting them in the dialog that follows, or select the layer beforehand and then draw.



The dimensioning can be applied with the **dimension function**:

Info: since the rectangle is a block (one part), it has to be broken down into its individual parts with the **Explode** function. Only then can the angle be dimensioned cleanly.

Draw a hole of D10 mm at the following position:



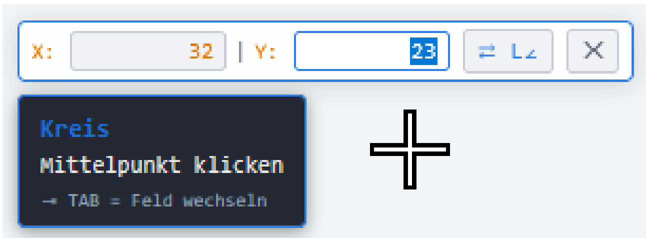
There are essentially two ways of achieving this:

1) The circle function and positioning of the circle center directly by entering **X and Y coordinates** to set the position
2) or with lines, horizontal and vertical with a dimension entry

With variant 1 the procedure is as follows:

Call up the **Circle** function.
→ Activate layer: **SICHTBAR**!

Press the [**K**] key for **coordinate entry**, or click "Coordinate entry" in the **Tools** panel on the left.



The floating coordinate menu is displayed. With the [**TAB**] key you can jump back and forth between the fields.
[**Enter**] or a **left-click** carries the function out.

Info: if the menu shows polar coordinates instead of cartesian ones,

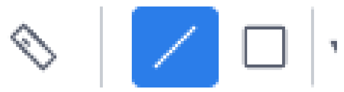
the type of coordinates can be switched over with the toggle button.

Toggle button for switching between cartesian and polar coordinate entry.

Tip: the mode can likewise be switched with the **spacebar**:

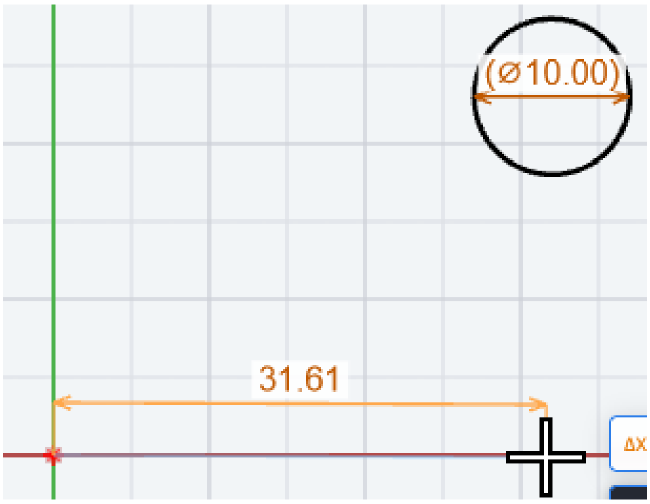
Enter the radius, and **Enter** or a **left-click** positions the circle at the coordinates with a radius of 5 mm.

If you like, you can first place a **point** at the origin. Place the origin with snap mode grid snap [**G**], for example, or X0 → Y0 → [**Enter**]



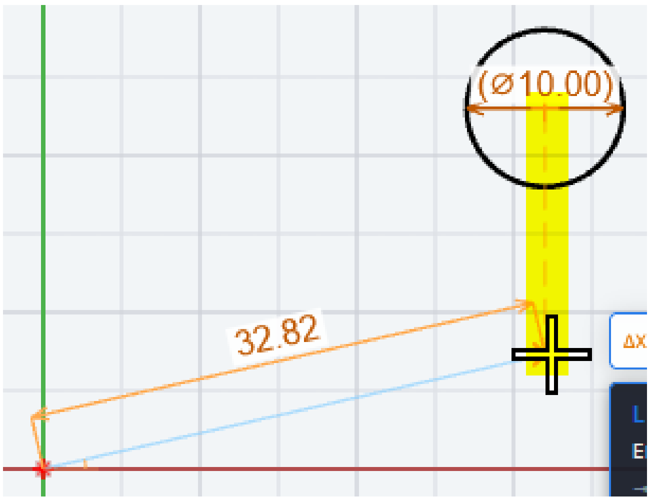
Connect all three axis lines from the origin to the center of the circle. Choose the **Line** function.
→ Activate layer: **ACHSEN**.

Check that the **TRACK** function is activated at the bottom right. **The TRACK function is worth its weight in gold while drawing.**



With snap mode Automatic [**Q**] start the first line at the origin. The automatic snap mode catches the point set earlier by latching onto it magnetically.

Drag the line...
We could enter the coordinate. But if we did not know the position of the circle, the TRACK function helps enormously...



Move the mouse onto the center of the circle, wait 0.4 seconds and then move away downwards to see the magnetic tracking line.

The TRACK lines run off at 90° in all four directions.

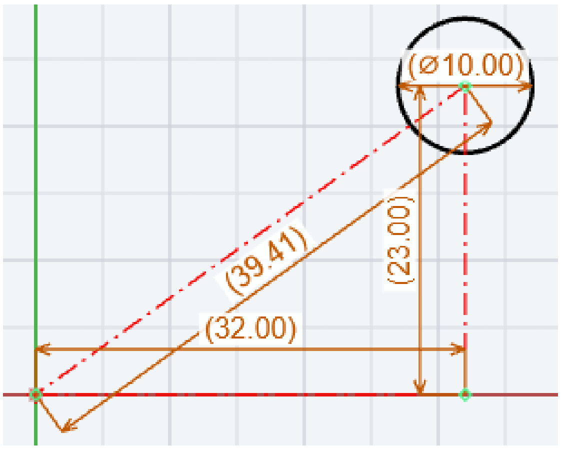
Move the mouse once around the center of the circle and watch how the TRACK lines are shown and hidden.

To catch the horizontal precisely, move the mouse onto the origin and again wait 0.4 seconds **WITHOUT clicking the mouse.**

Then move away from the origin along the horizontal axis until the vertical **TRACK** line is displayed and the mouse cursor is perfectly positioned and snaps on magnetically.

Now place it with a click.

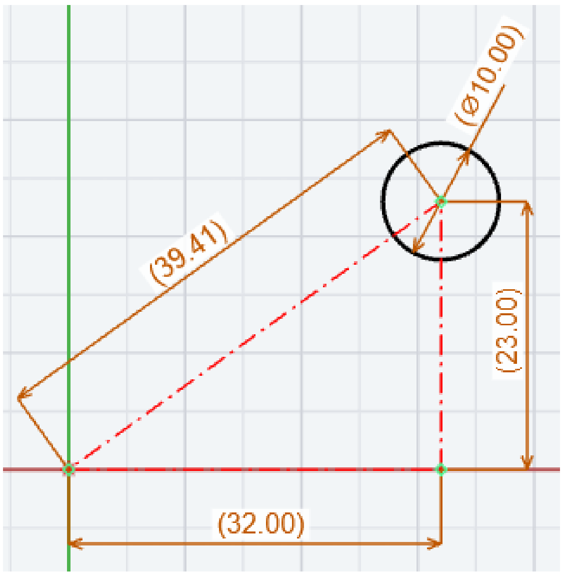
Make a mental note of the workflow of the **TRACK** function; it will always serve you well.

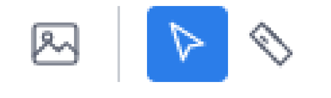


The other two lines can be snapped quite normally from now on.

It is best to draw everything else with the automatic snap [**Q**].

You will find the snap functions and the hotkeys on the [**F1**] key





To make the drawing look good, grab the **dimension lines** with the select function and move them with the mouse button held down...

Done! Bravo!

IMPORTANT: from here on only compact instructions

In the further course of the book not everything will be shown in such detail, but in compact form as follows:

Circle function → [K] → X: 32 [TAB] Y: 23 → [Enter] Radius: **5**

Point function → [G] or X0, Y0 → [Enter]

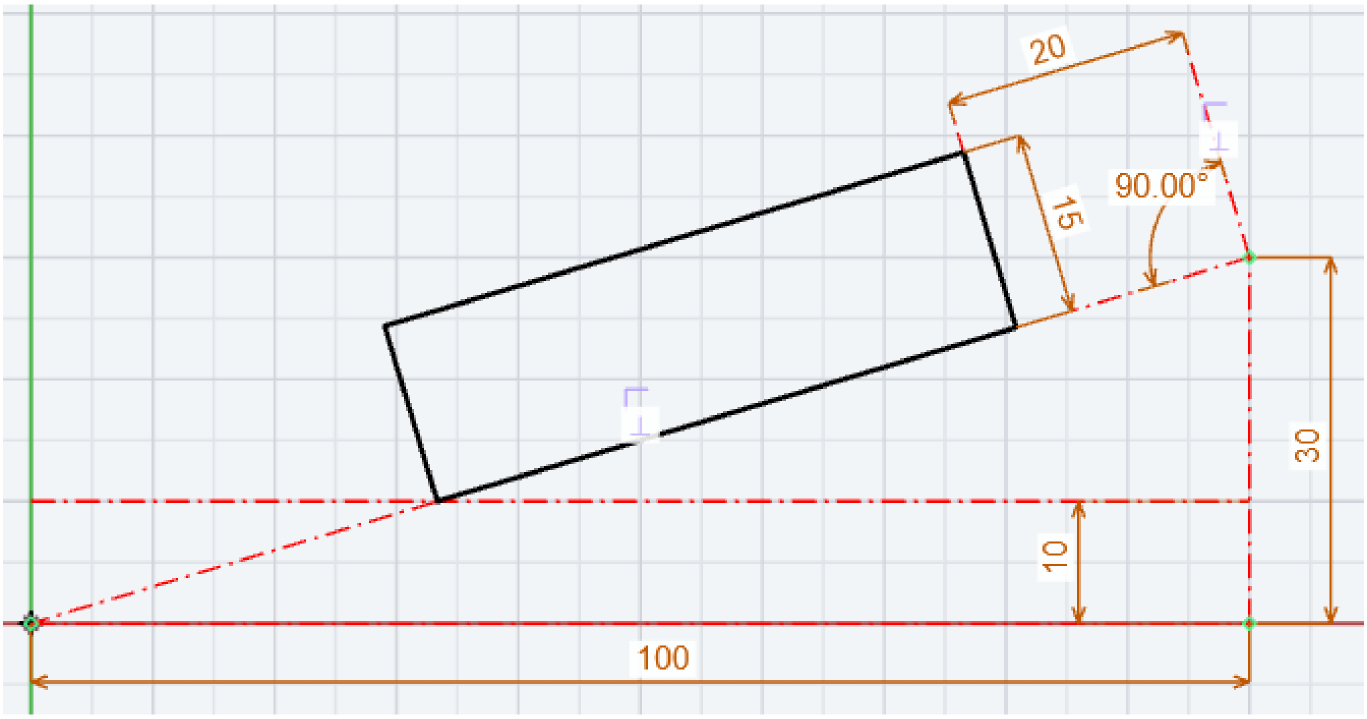
Line function → [Q] → draw all lines... (watch the [TRACK] lines)

Drawing 1: this is what the previous example could look like in compact notation...

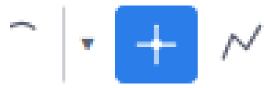
Notes:

35 Constructing a slanted rectangle...

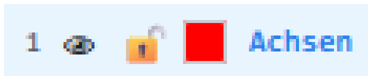
Now we construct the following rectangle, which is created at a particular angle with given distances:



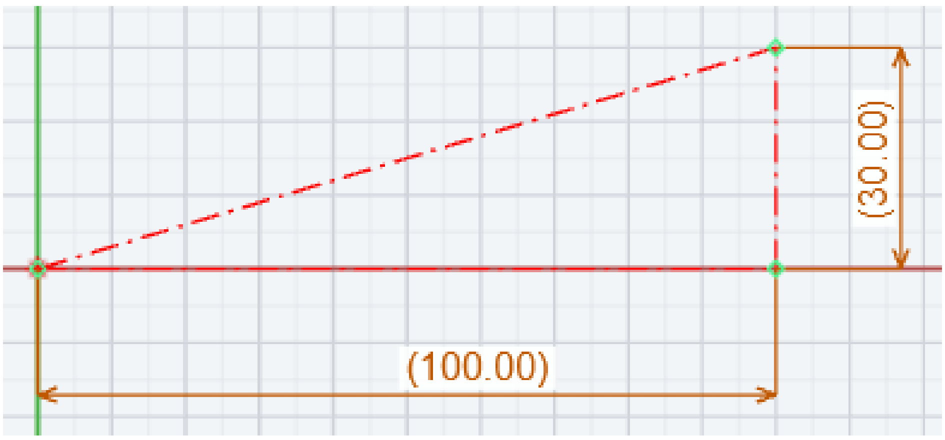
1. Place a **point** at the origin:
   Point function → [G] or X0, Y0 [Enter]



2. Select layer **ACHSEN**.



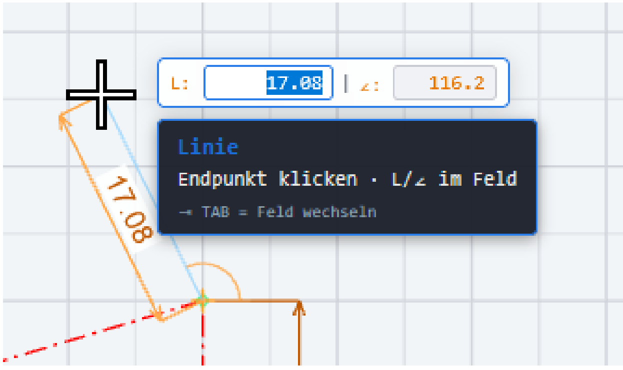
3. With the **Line → [A]** function, first draw only the triangle 100, 30...
   *(Remember the short pause on the intersections for the **TRACK** line.)*

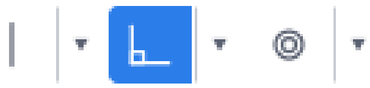


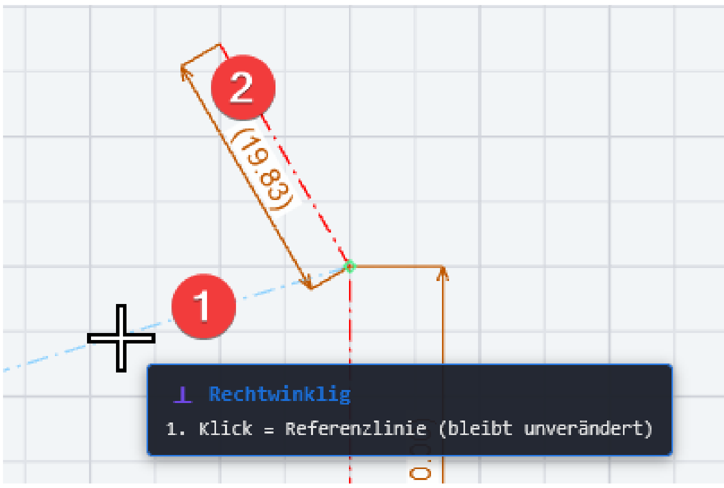
4. Now create the parallel line at a distance of 10 with the **Offset (O)** function. The function is self-explanatory.

5. Then, in order to set off the dimension 20, proceed as follows:

   Draw a line from the endpoint **roughly** at a right angle:

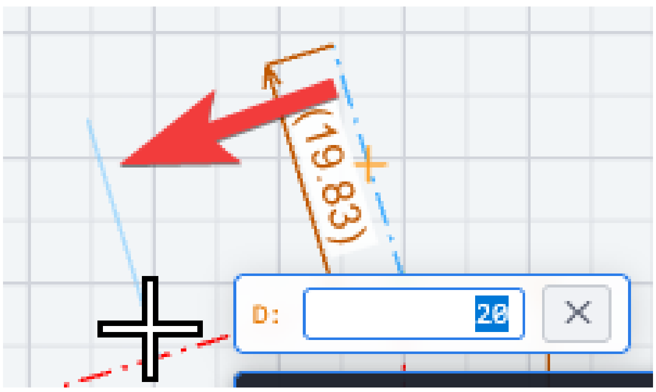


6. Make it **perpendicular** with this function: 



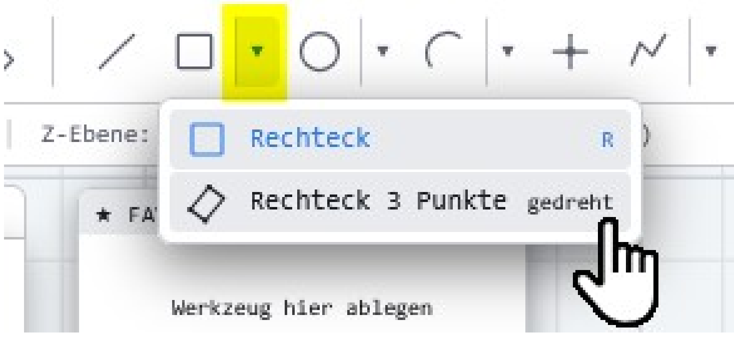
   Read the sequence of steps in the floating tooltip and follow it, then all will go well :-). The red numbers show the order of the clicks.

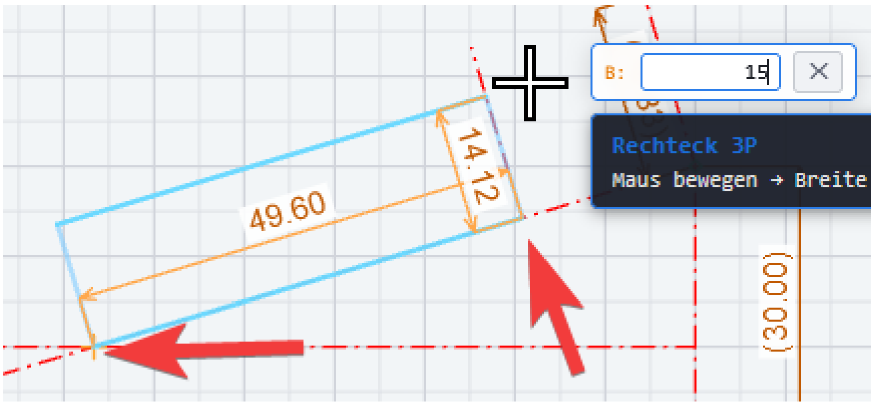
7. Next, an offset line at a distance of 20 



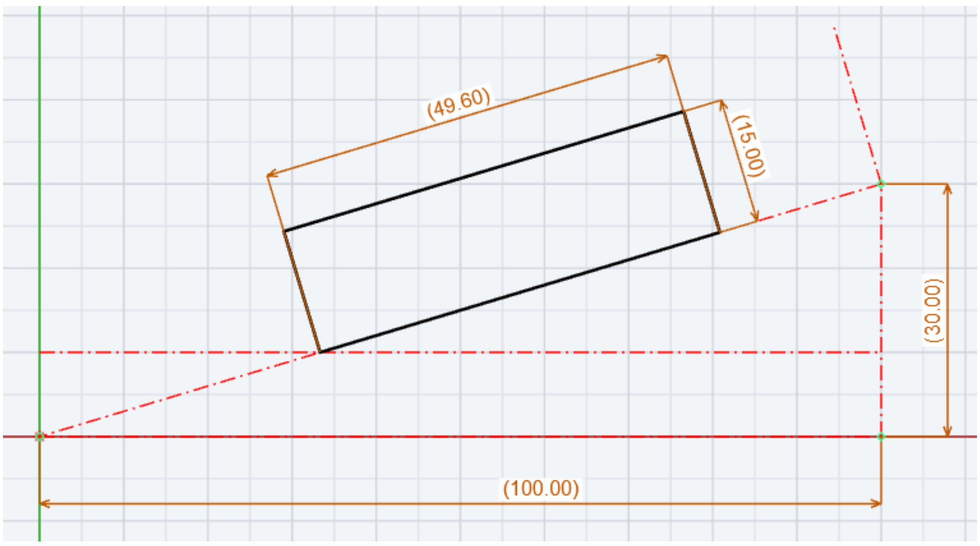
8. Activate the layer **SICHTBAR**:

9. The rectangle is easy with the function **Rectangle 3 points**





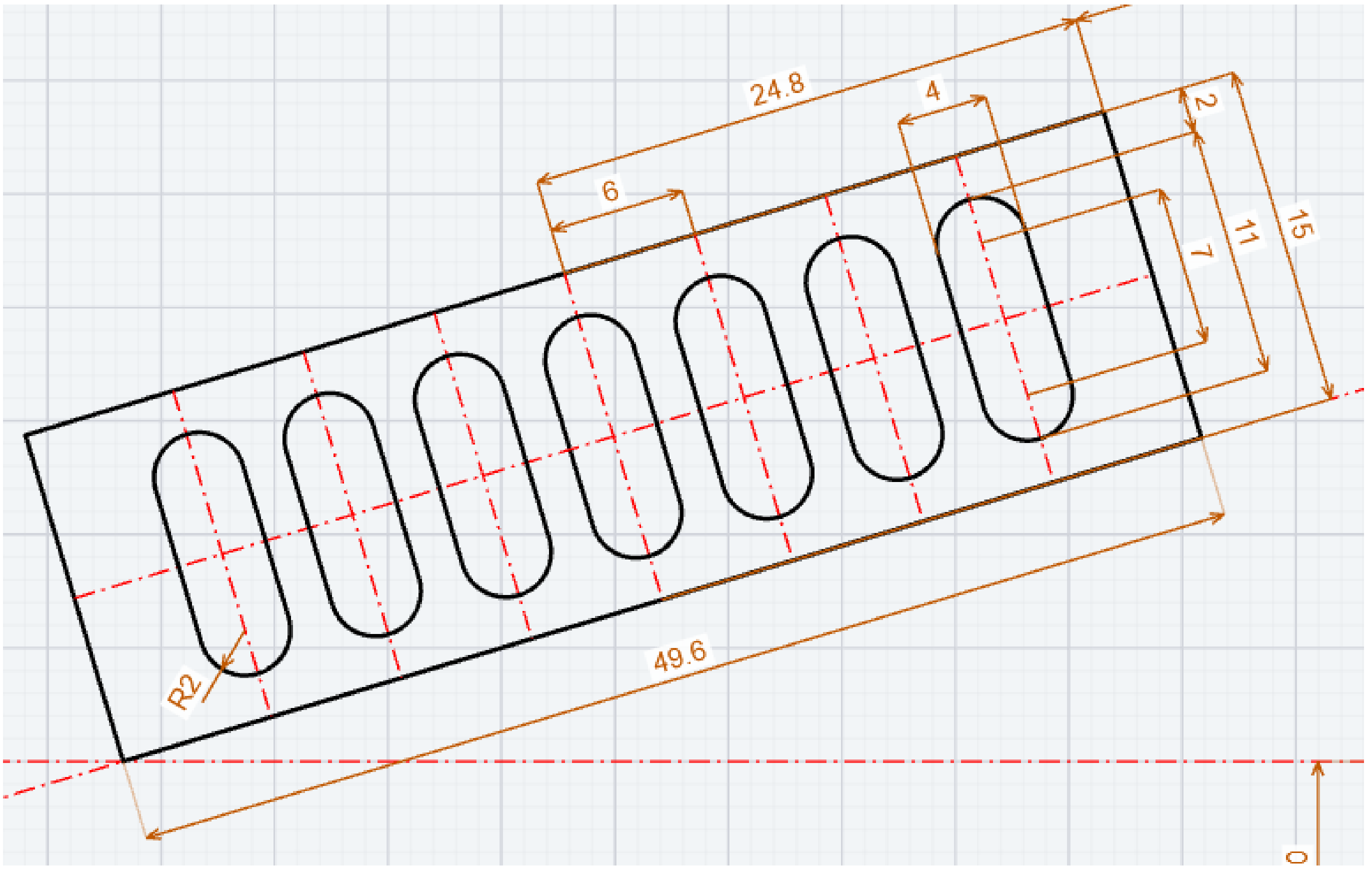
Watch the snap mode [**A**] and draw the rectangle on these **intersections**, with the dimension W: **15**



Done :-)

35.1 Adding ventilation slots

Next, ventilation slots are added...



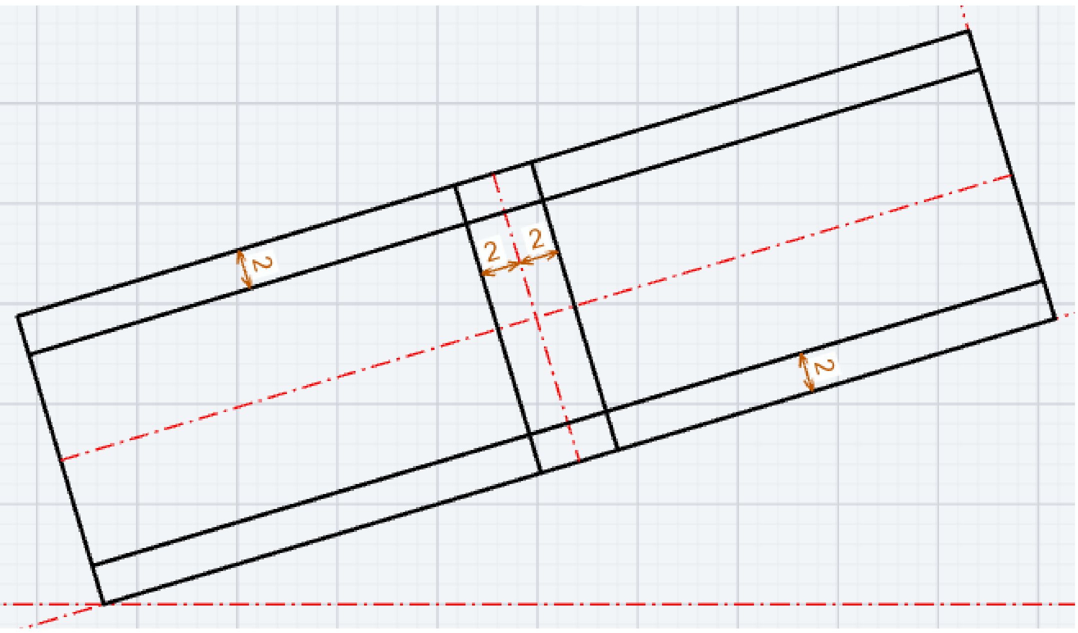
Activate layer: **ACHSEN**.

Function: **Line**   Draw both axes centered into the rectangle.

Activate layer: **SICHTBAR**.

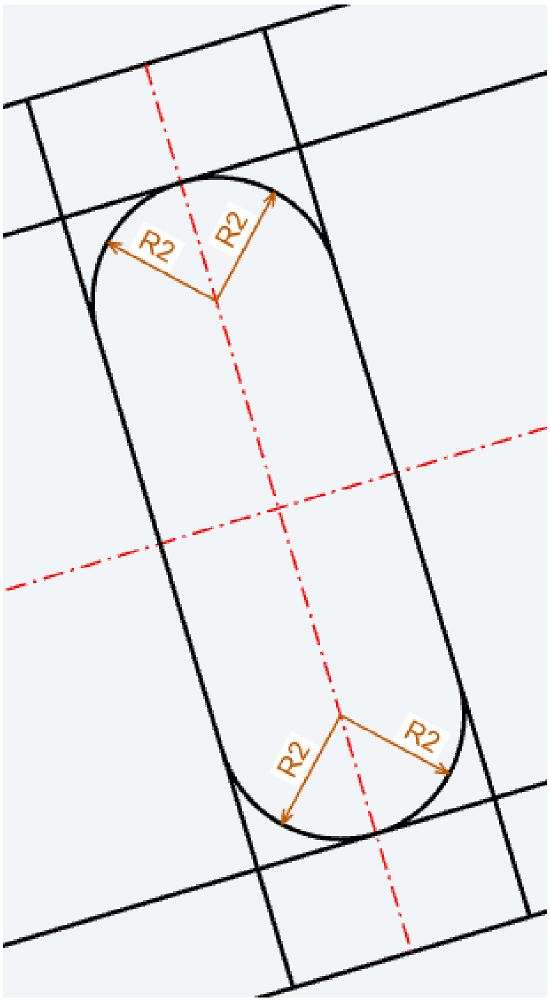
Function: **Offset**, distance: **2**

Create the parallels as follows:



Next, the semicircles are created with the **Fillet** function as follows:

Function: **Fillet →**  snap mode: **Free [F]**

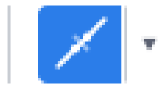


Always select two lines to fillet and move the mouse into the **sector** where the fillet is to be placed.

Then enter the radius value **2** and carry it out with **Enter**

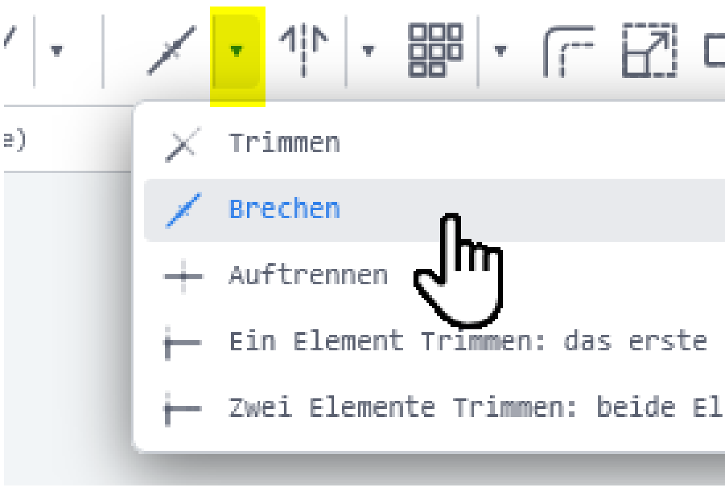
Repeat this procedure at all four corners.

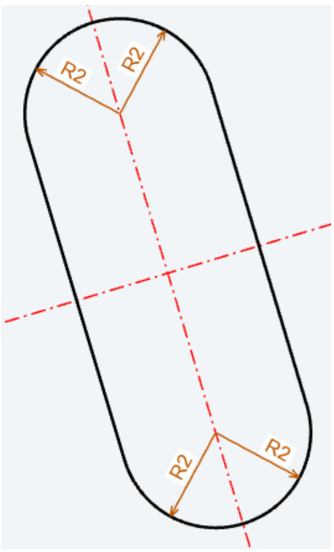
Info: the semicircles now consist of two quarter circles each, which result automatically from the use of the trim function and that is fine in this case.

 Function: **Break**

Now all superfluous **line** pieces are removed (broken away) with Break...

The Break function automatically breaks away everything between intersections and endpoints...





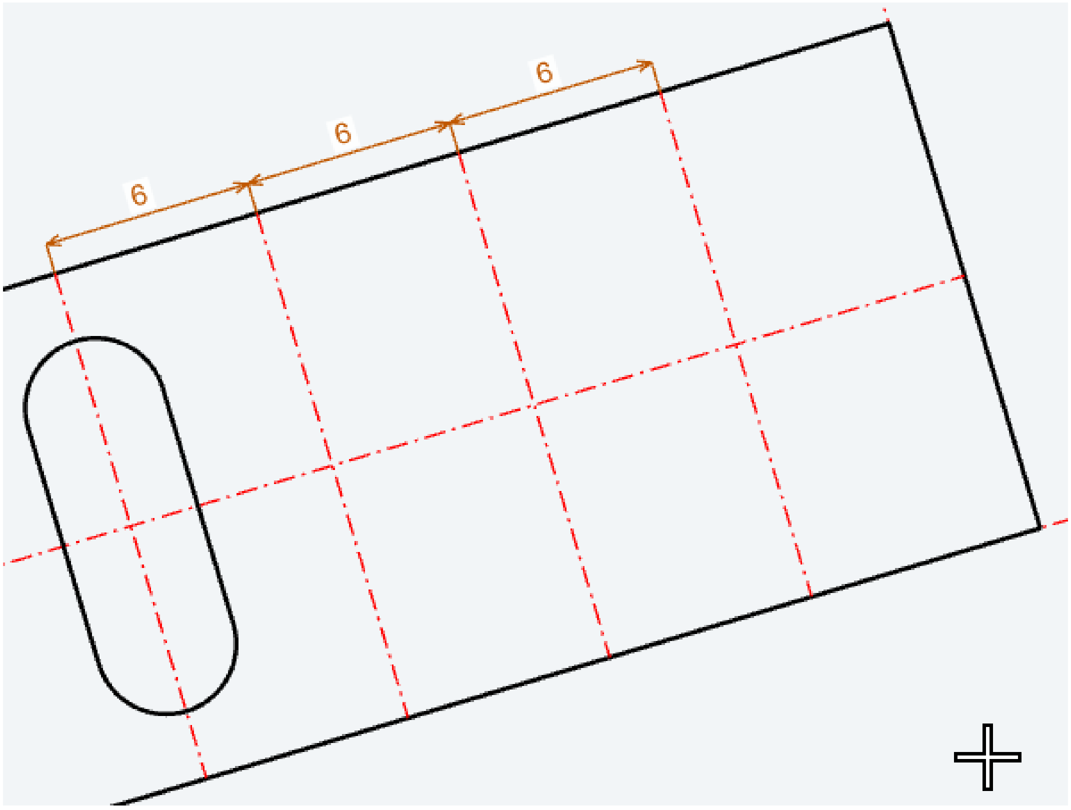
Perfect!

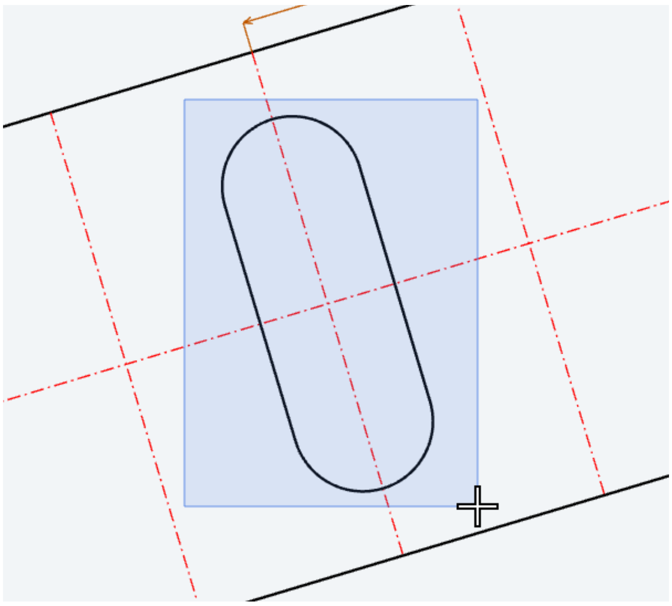
(Internal remark: programming that was a hard piece of work!)

Next we draw the further axes that define the slot spacing...

Activate layer: **ACHSEN**

Function: **Offset [O]**, distance → D: **6** **Create all three** axes to the right ...



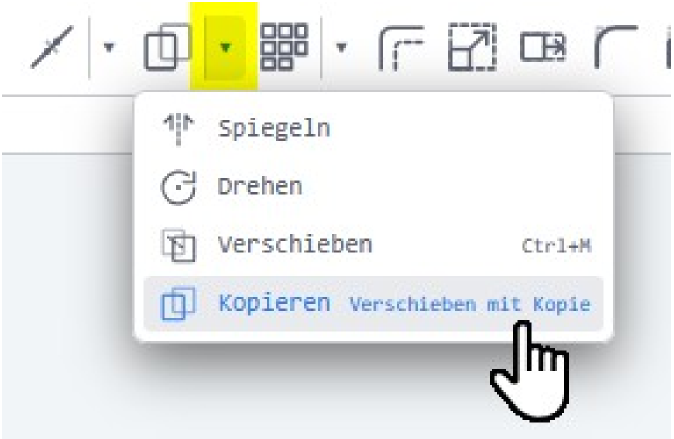


Next the ventilation slot is selected, either with a selection frame ...

or individually with the [**Shift**] key held down... (the [**Shift**] key) select all elements one by one.

Select all the elements to be copied... *(4 arcs and 2 lines)*

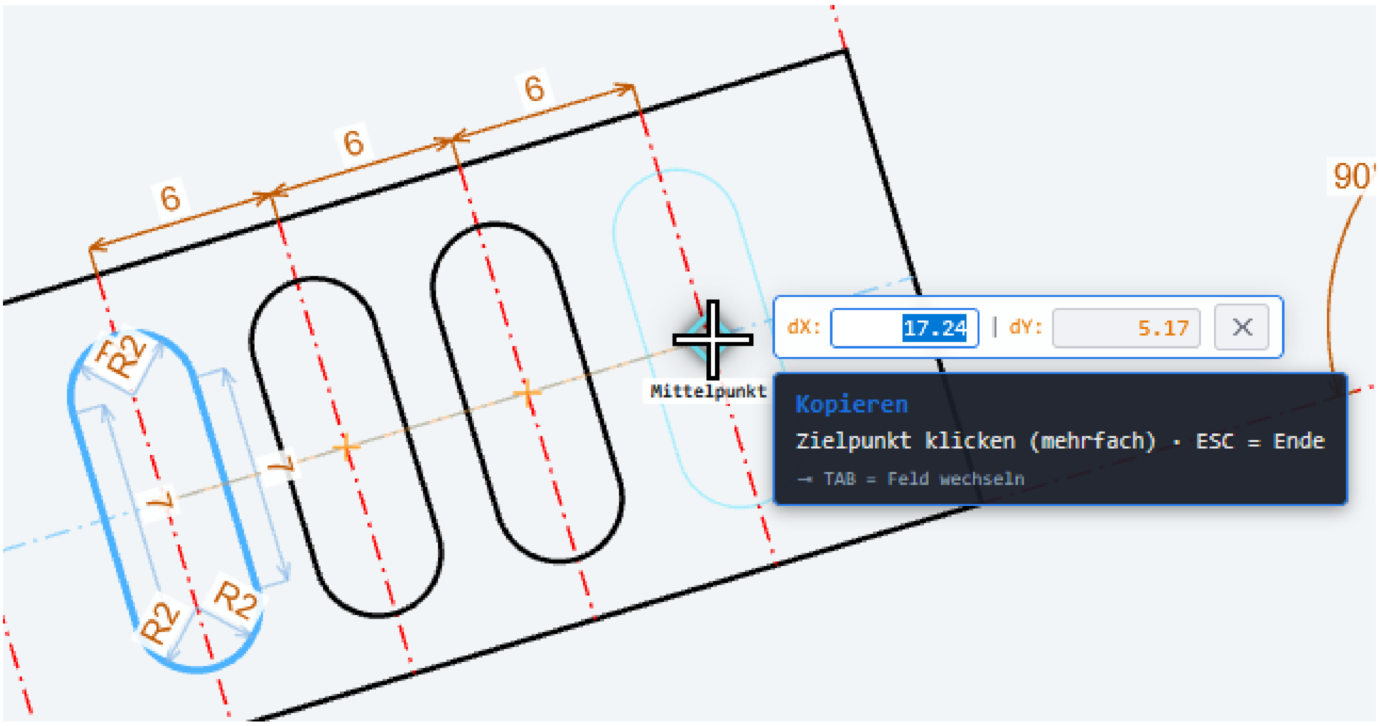
Function: **Copy**



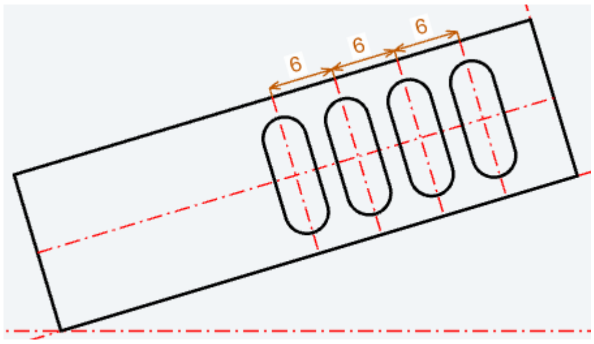
Set the snap mode to Automatic [A] or [S] for intersection.

Set the base point in the center of the slot.

Set the target point at all the further intersections:



The slots are positioned perfectly:

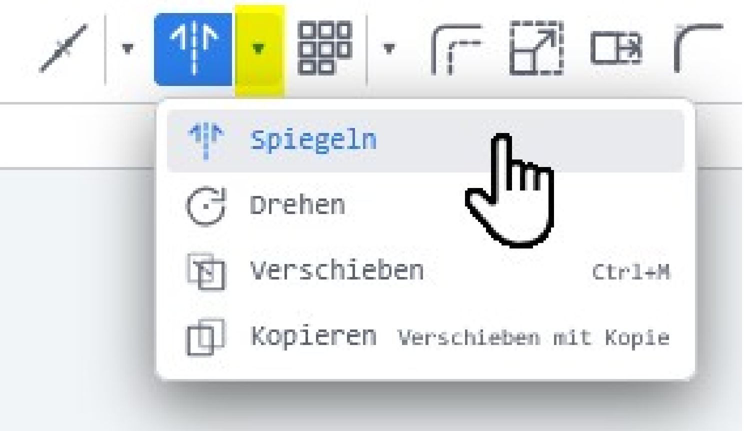


You could now do the same again for the left side, which would be completely fine.

But in order to get to know the «**Mirror**» function, we will use that one.

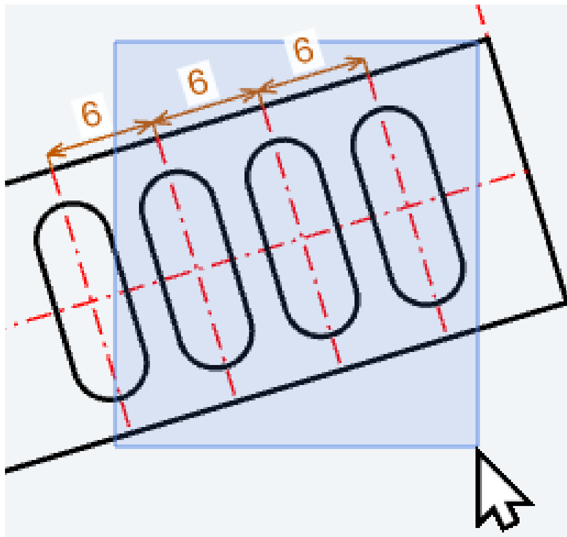
First select the elements to be mirrored.

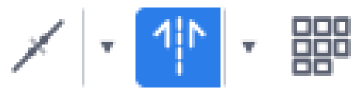
Call up the Mirror function as follows:



*Select all elements of the **three** right-hand slots by dragging a rectangle from top left to bottom right over all three slots. A rectangular area dragged from **left to right** means that all elements **must lie completely inside** this area in order to be selected.*

All elements to be mirrored selected.

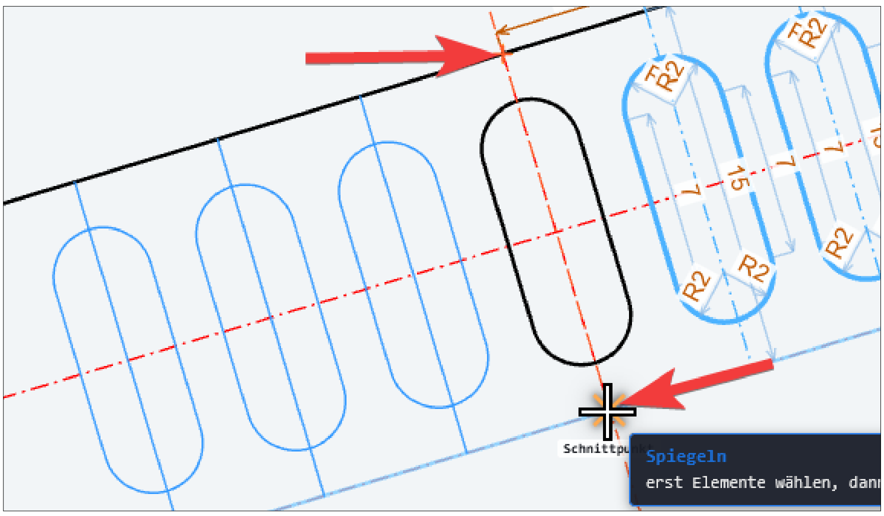




Function: **Mirror → snap mode [A]**

Specify the **first point** of the mirror axis (1)

Specify the **second point** of the mirror axis (2)



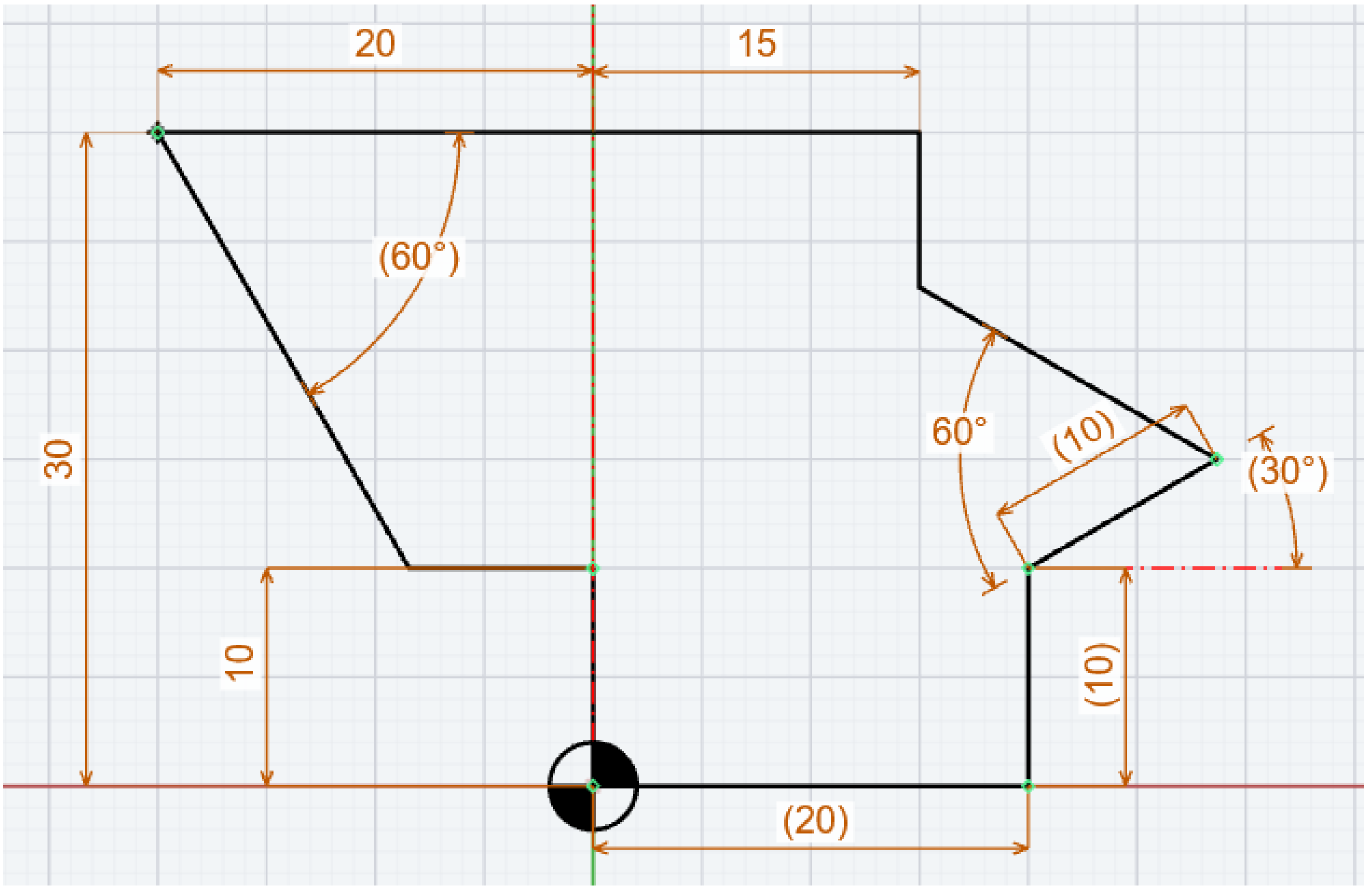
The mirror axis must lie exactly on the two intersection points.

Snap mode [**A**] or [**S**] catches them perfectly.

Tip: if you do not want to keep the original, simply press the **Delete** key.

36 Coordinates exercise 001

With the previous exercises, you should be in a position to master the following task:



Are all dimensions present at first glance?

Can any missing dimensions be determined with the LiberoDraft functions?

How would you proceed?

- Will you do all of it via the **coordinate entry**?
- Or do you prefer constructing **guides**?
- Or differently? There are several ways. Thanks to LiberoDraft you get to the goal comfortably.

Both ways are absolutely fine, the main thing is that they lead to the goal.

I would do it with a mixture of both variants. So first have a look at which main lines can be created by means of **parallel lines**.

In addition, I occasionally use **guides**, some of which I only need temporarily in order to get to the construction points that matter to me.

The journey is the reward.

How you decide is also a matter of routine. The more you work with a CAD program, the more efficient you become :-)

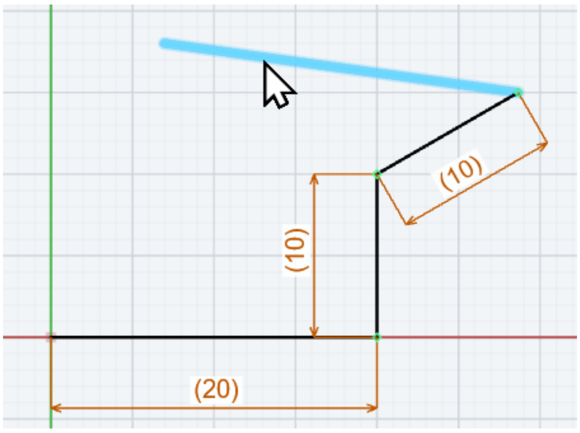
A possible way of solving the task

Since LiberoDraft has many good functions, there can also be different ways of solving it. Only one of them is shown below. Perhaps you will find another one that is easier for you, which is absolutely fine :-) All that matters is the precise, identical result of the design drawing.

Start on layer **SICHTBAR** from the **origin** towards the right with ...

Function: **Line → on the grid**

The first **two** lines are easy. With the **third**, angled line you may have to switch the coordinate system to **polar** with the **spacebar**.

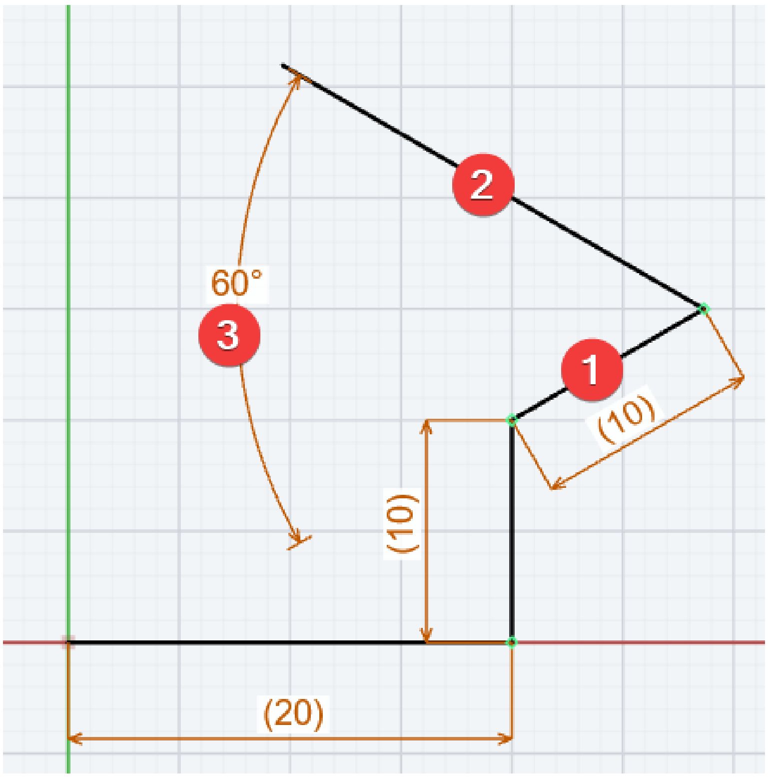


Simply draw the fourth line **freely** (shown in blue in the figure); the angle and length do not matter at all.

Now dimension the angle to 60° with the dimension function.

When dimensioning angles and driving the lines to the 60° angle, note the following:

1. Always select the **fixed** line **first**.
2. Use snap mode **F** for **Free**.



(1) First select the **fixed** line.

(2) Select the driven line.

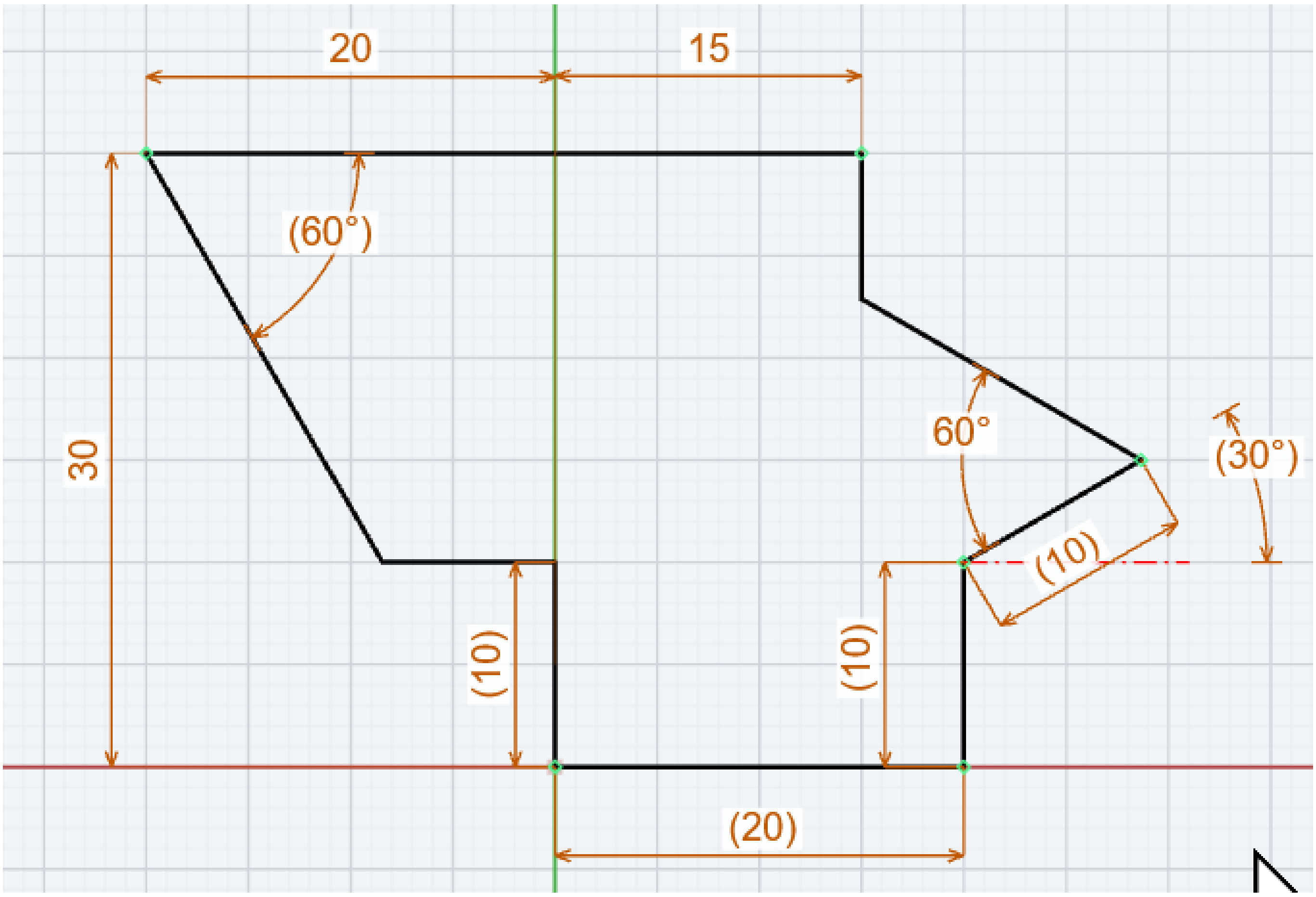
(3) Place the dimension, enter the value = **60** and confirm with **Enter** or **OK**.

Excellent!

From here on you have to draw the remaining lines yourself.

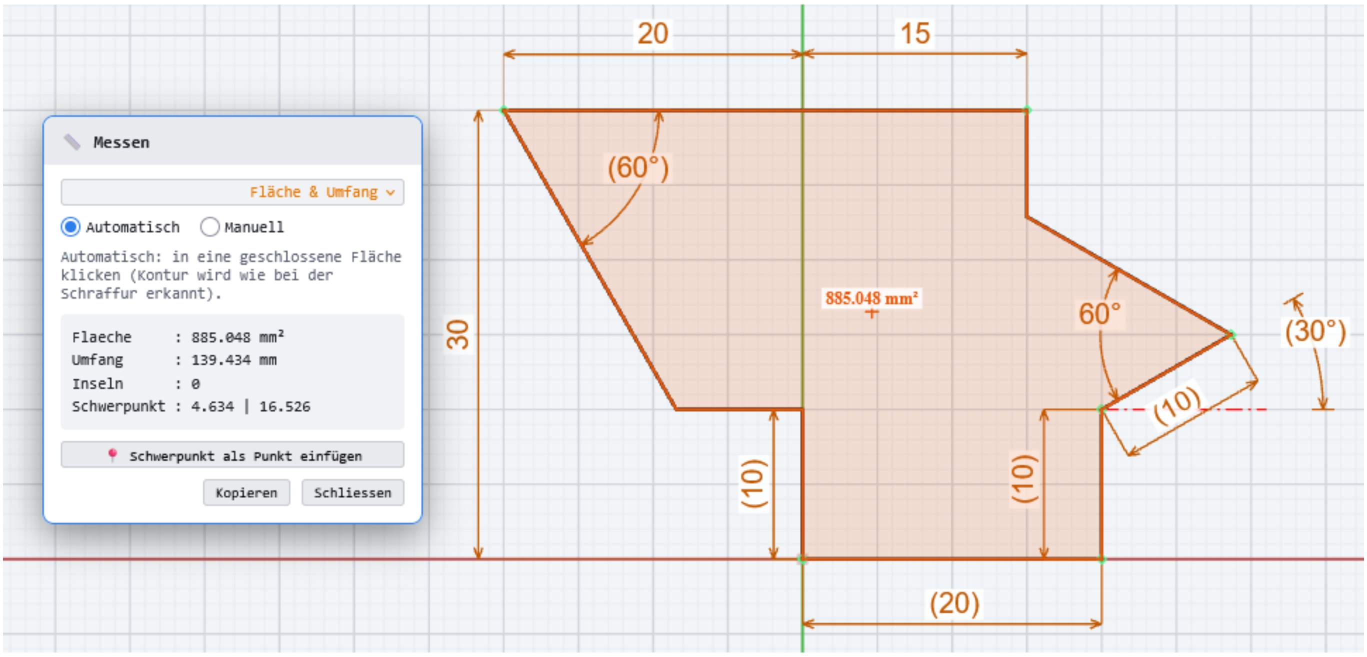
Use whichever line function suits you to reach the goal.

You can see that in LiberoDraft, thanks to the many functions, you can reach your goal in different ways. Simply try out everything possible... :-)



Checking whether your values match ours…

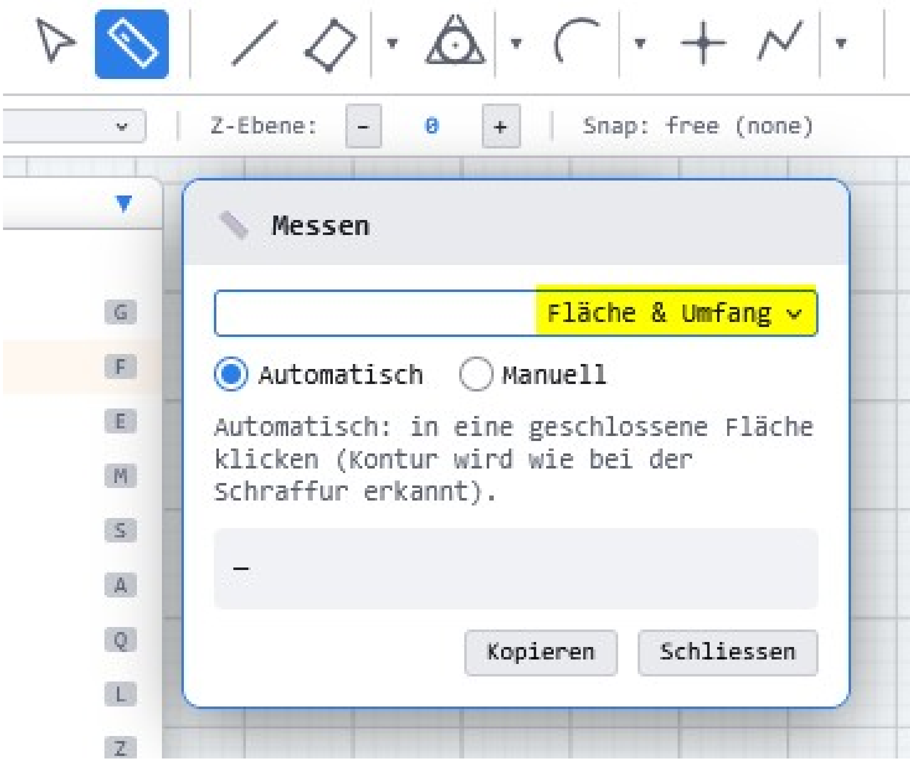
The easiest way is to do it approximately by calculating the area and the circumference with the Measure function: **Area & circumference**



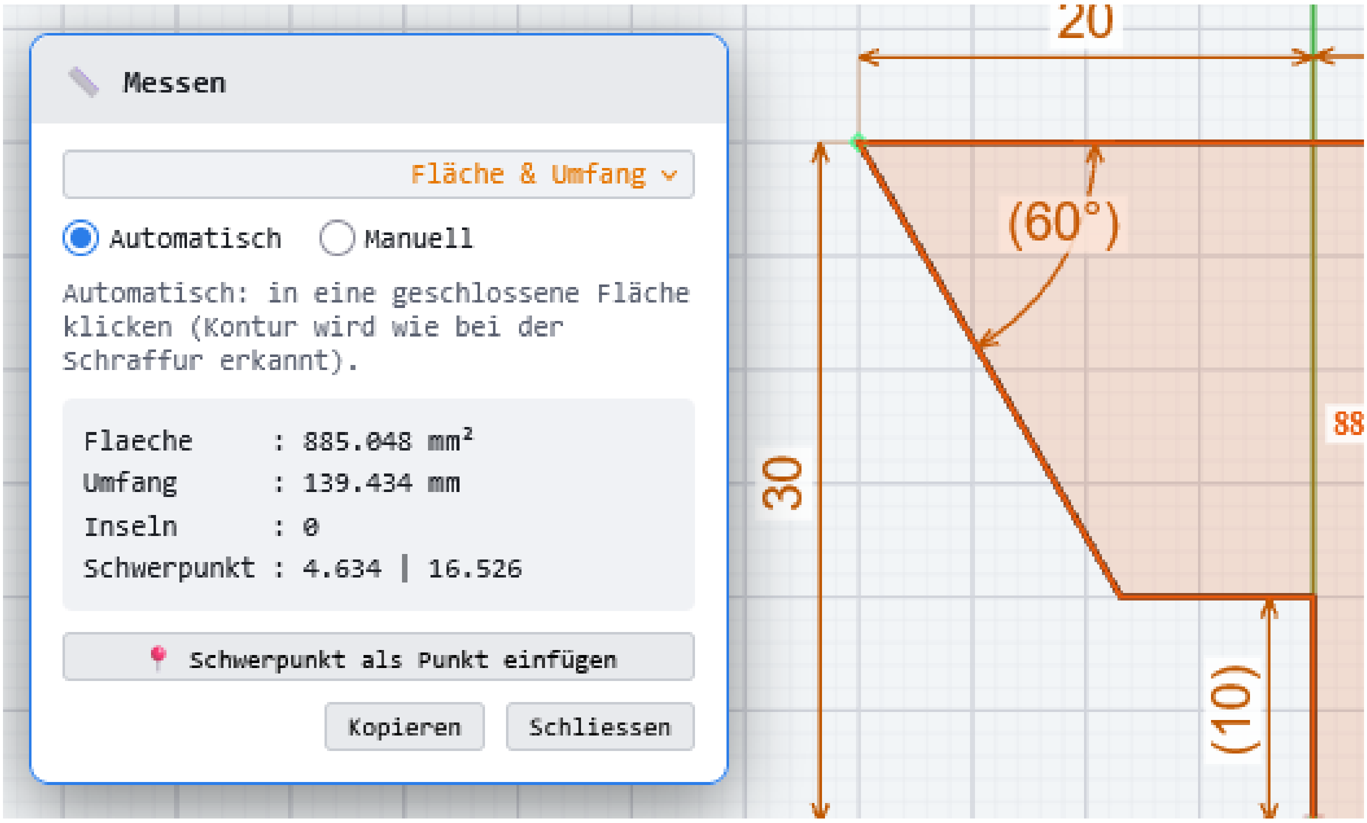
Area : 885.048 mm²

Circumference : 139.434 mm

The check is carried out with the Measure function:

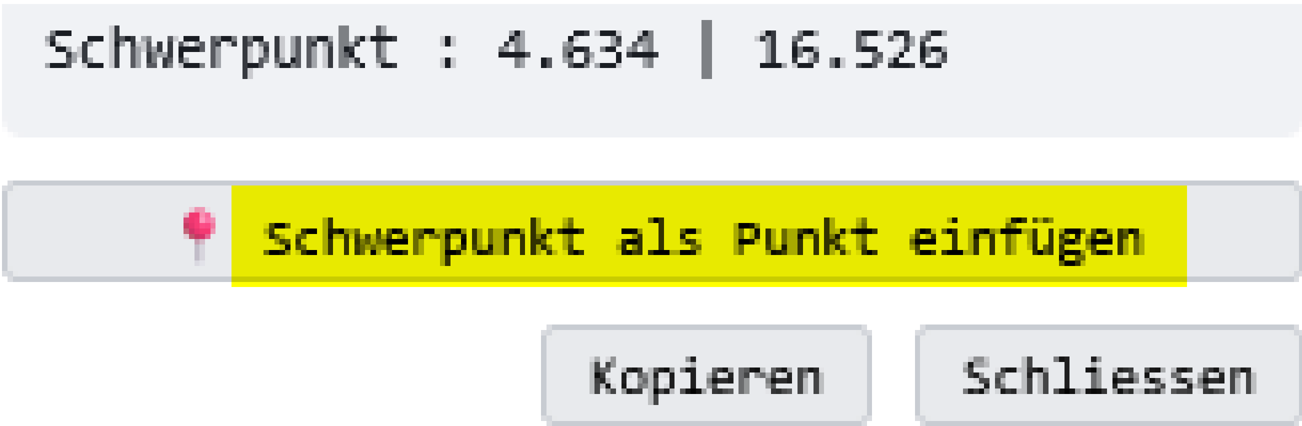


Choose Area & circumference and click inside the area:

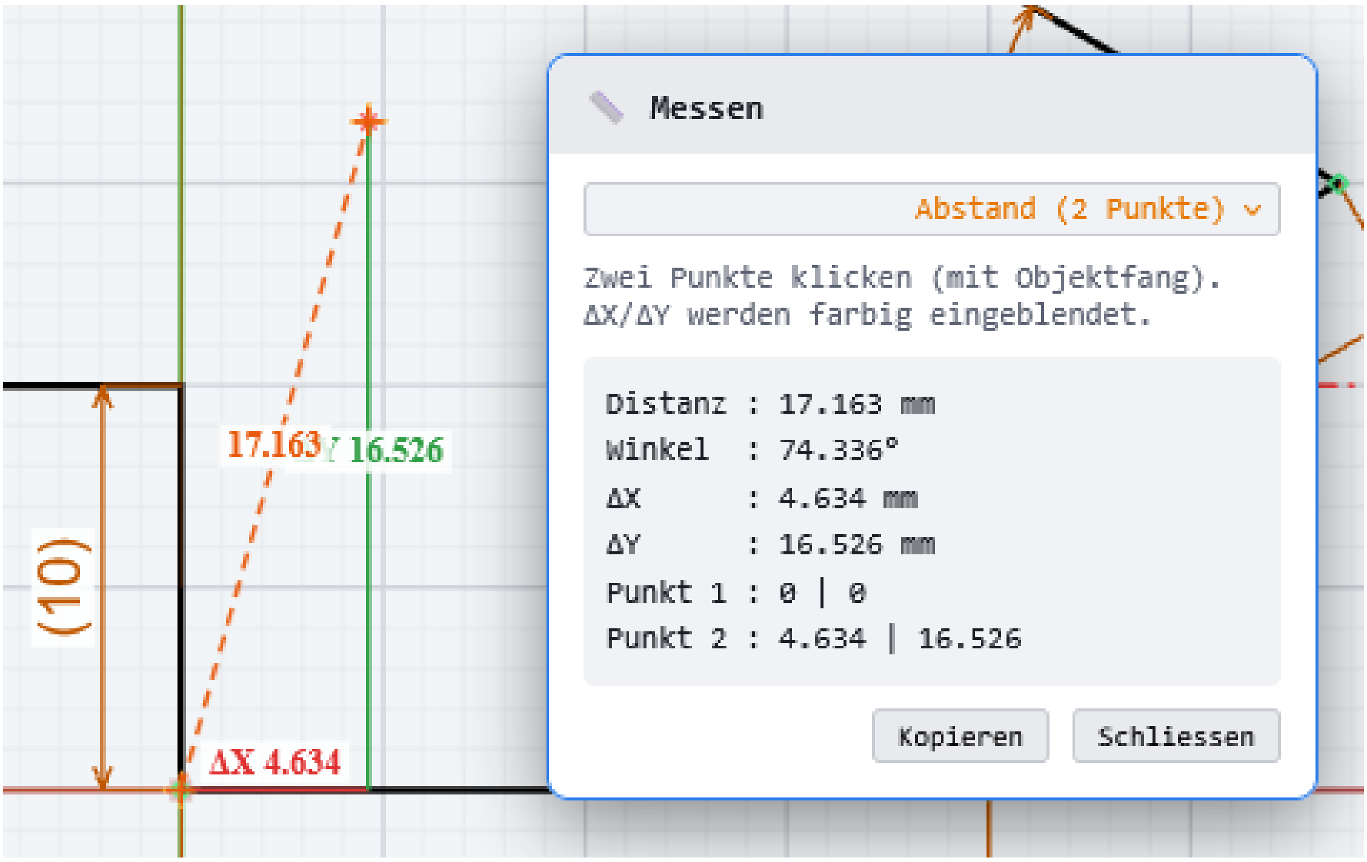


Perfect, that is the first one!

Checking the **centroid** of an area:



Insert centroid as a point creates a real 2D point in the drawing that you can measure…



With the **Measure function** you can carry out the most important measurements directly:

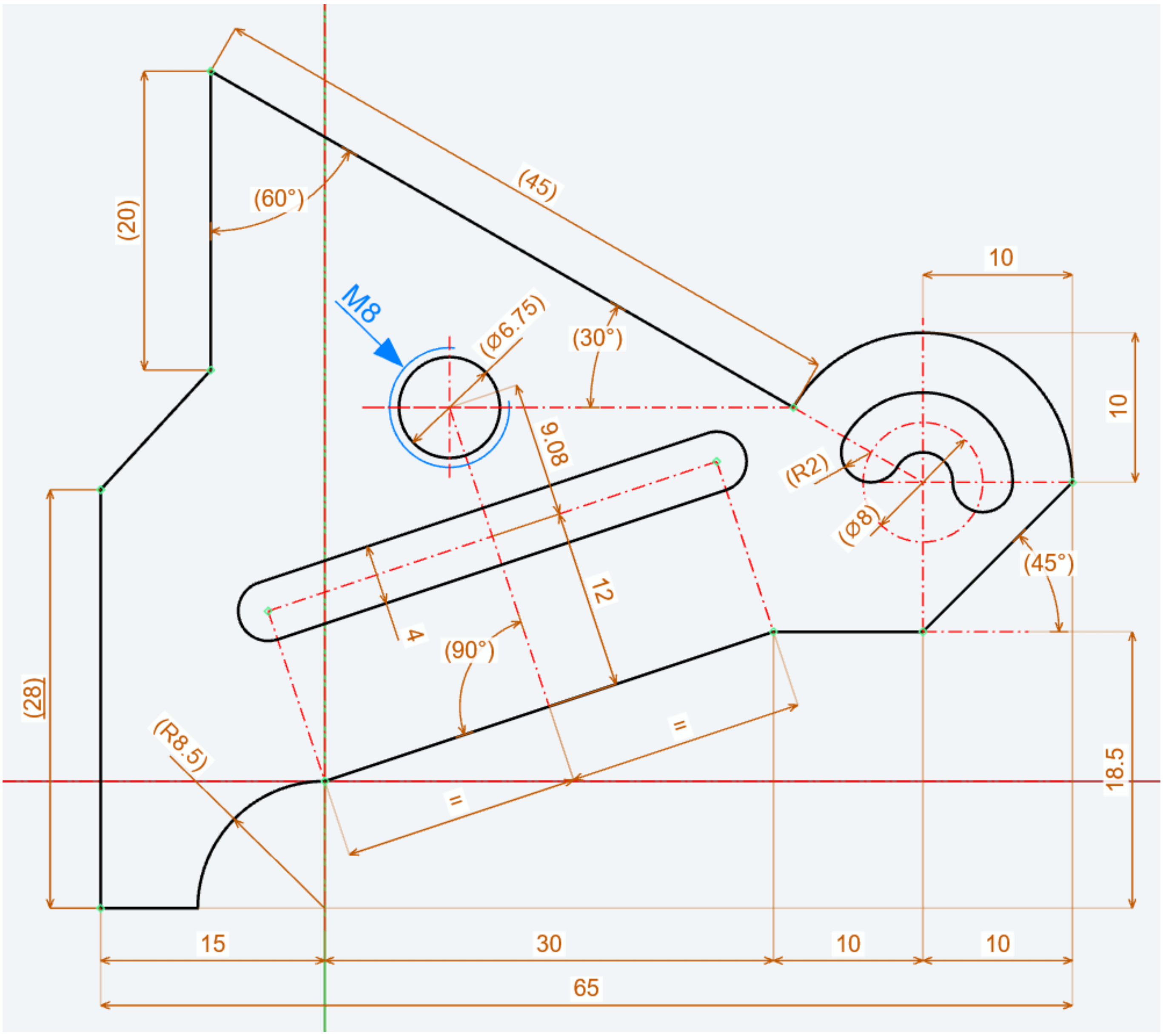
For example, Distance (2 points) shows all the relevant information…

It is hidden again automatically as soon as you end the Measure function.

If your centroid lies at X 4.634 and Y 16.526, everything is perfect — that is the second one!

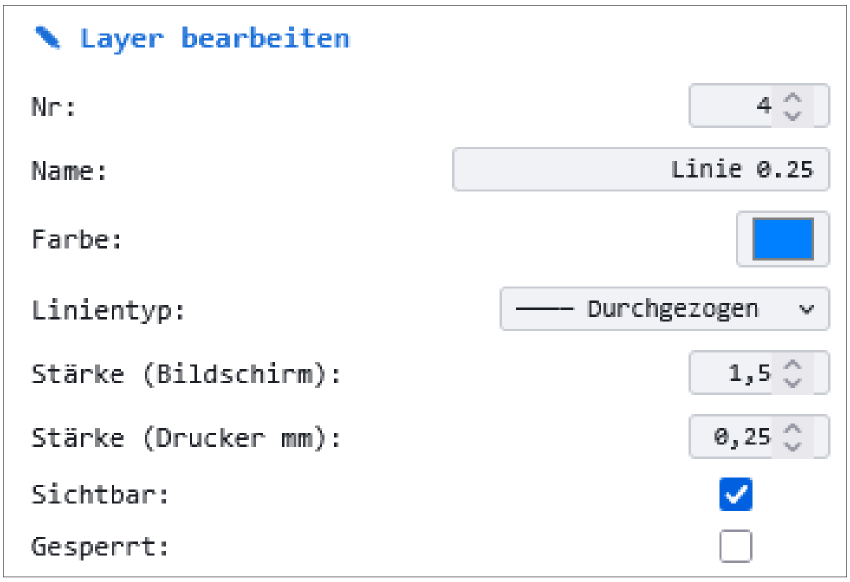
37 Coordinates exercise 002

Try to complete the exercise without help:



Not all dimensions are drawn in completely. They can, however, easily be constructed with the LiberoDraft functions.

For the thread line of the M8 thread we create an additional layer with the name: **Linie 0.25** →



Furthermore, this drawing requires cartesian, polar, absolute and relative coordinate entries as well as various drawing, trim, parallel and circle functions in order to reach the goal.

Draw guides and axes wherever they are needed. In the layer panel

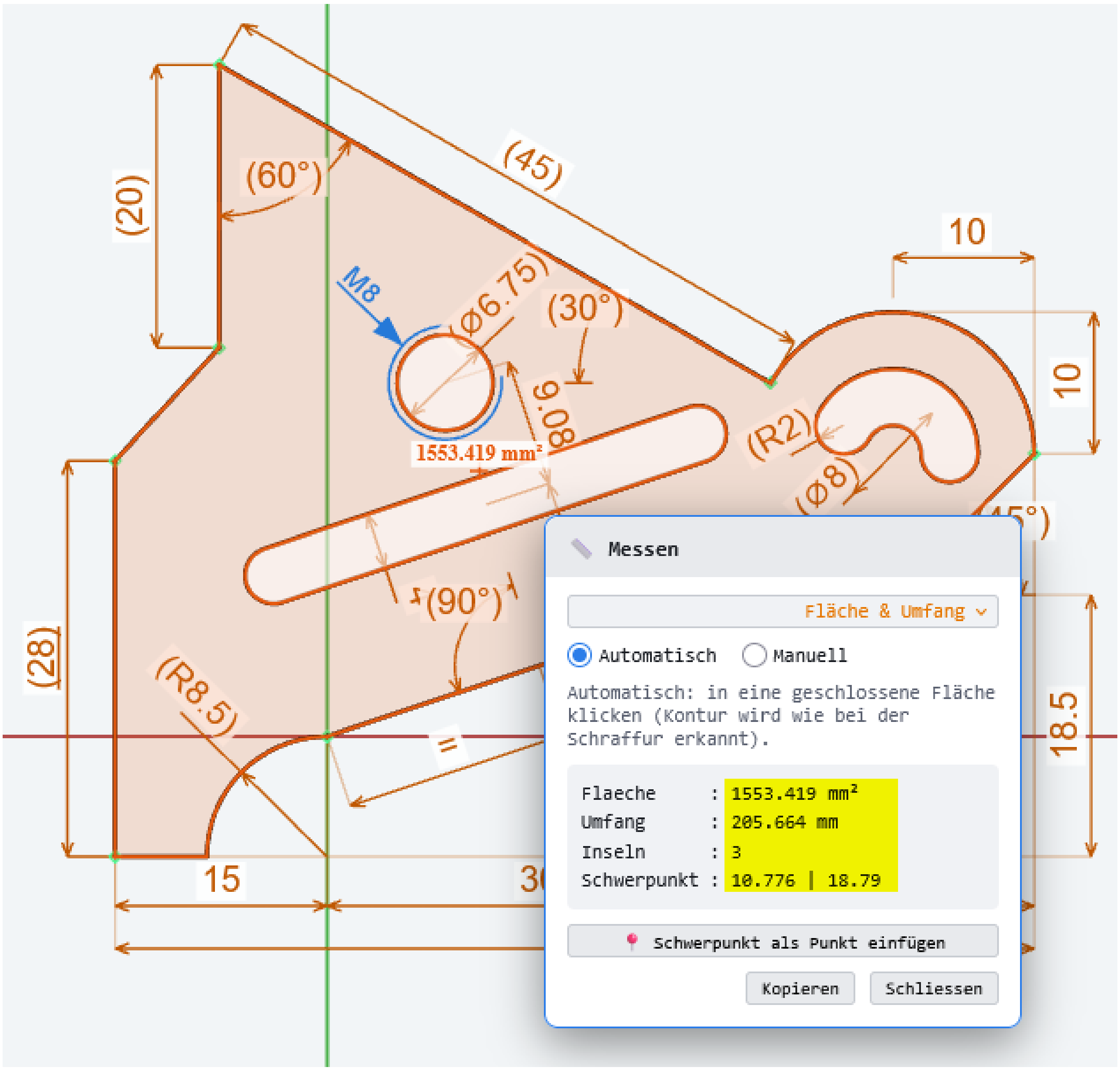
Extend or shorten lines with the Trim function.

Total length of all visible lines including the tapping drill hole: **336.4511 mm**

Tapped hole M8 → see chapter 50.1 **Creating your own standard part (M8)**

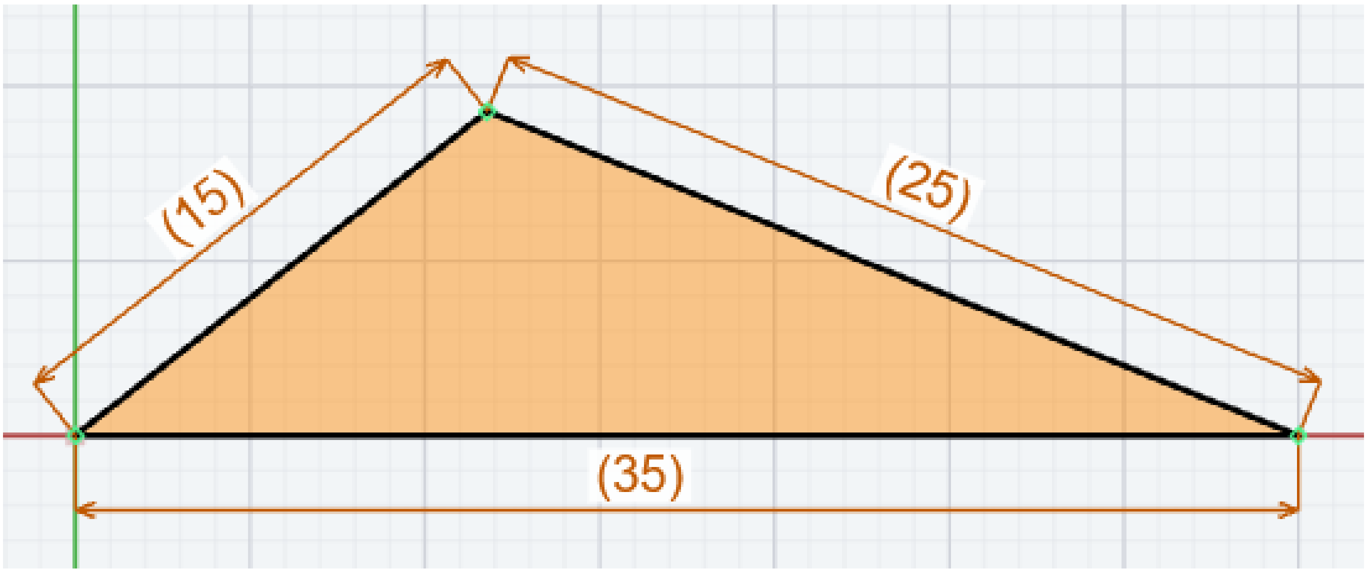
But for practice we will draw the tapped hole ourselves.

So first a circle, then an arc of about 3/4 of a circle :-)



Use the Measure function to check whether you get the same values :-)

38 Constructing a triangle



Task:

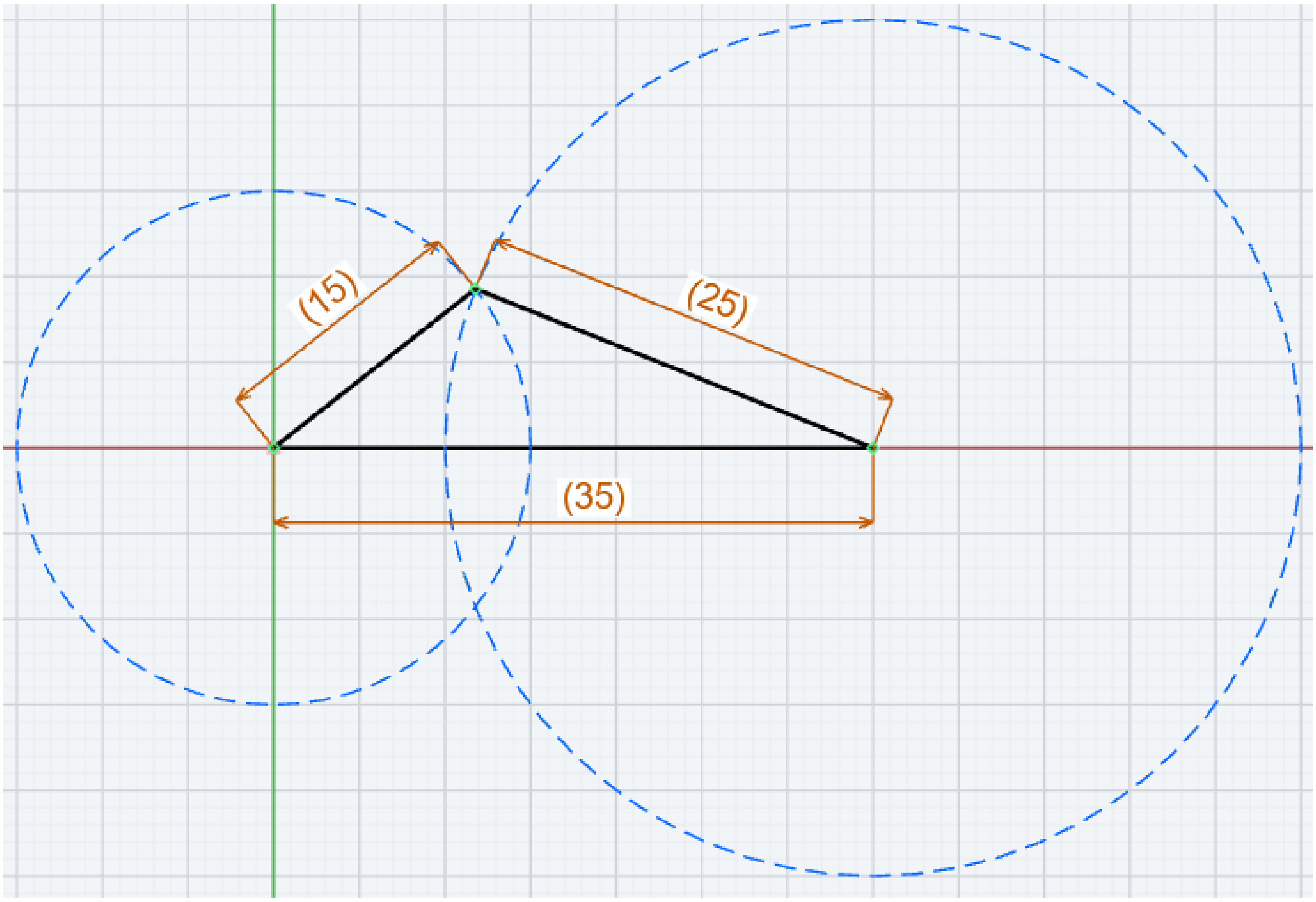
The triangle above is to be constructed.

All we have are the lengths of the three sides: **35, 25 and 15 mm**

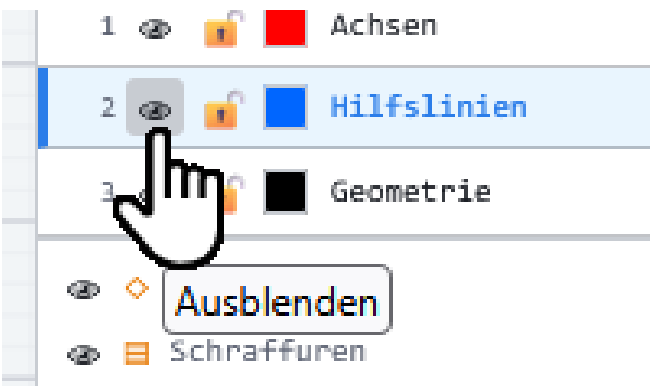
Form a triangle from them in which the corner points coincide exactly and whose resulting area is **162.38 mm²** . Furthermore, all three angles are to be dimensioned.

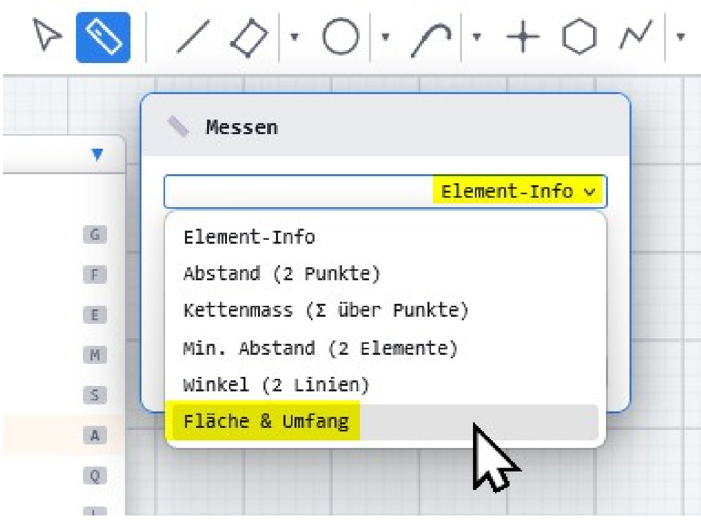
Steps:

- Activate layer **SICHTBAR**
- Create a **horizontal line: 35 mm**
- Activate layer **Hilfslinie**
- Draw two circles with the function: **Circle**
  Activate snap option **Automatic [A]** .
  Place one circle with **R25** and one with **R15**
  on the **endpoints** of the line.
- Activate layer SICHTBAR
- Draw two **lines** with snap mode **Automatic** [**A**]
- The triangle is finished :-)



To query the area, hide the layer **Hilfslinien**...





Query the area...

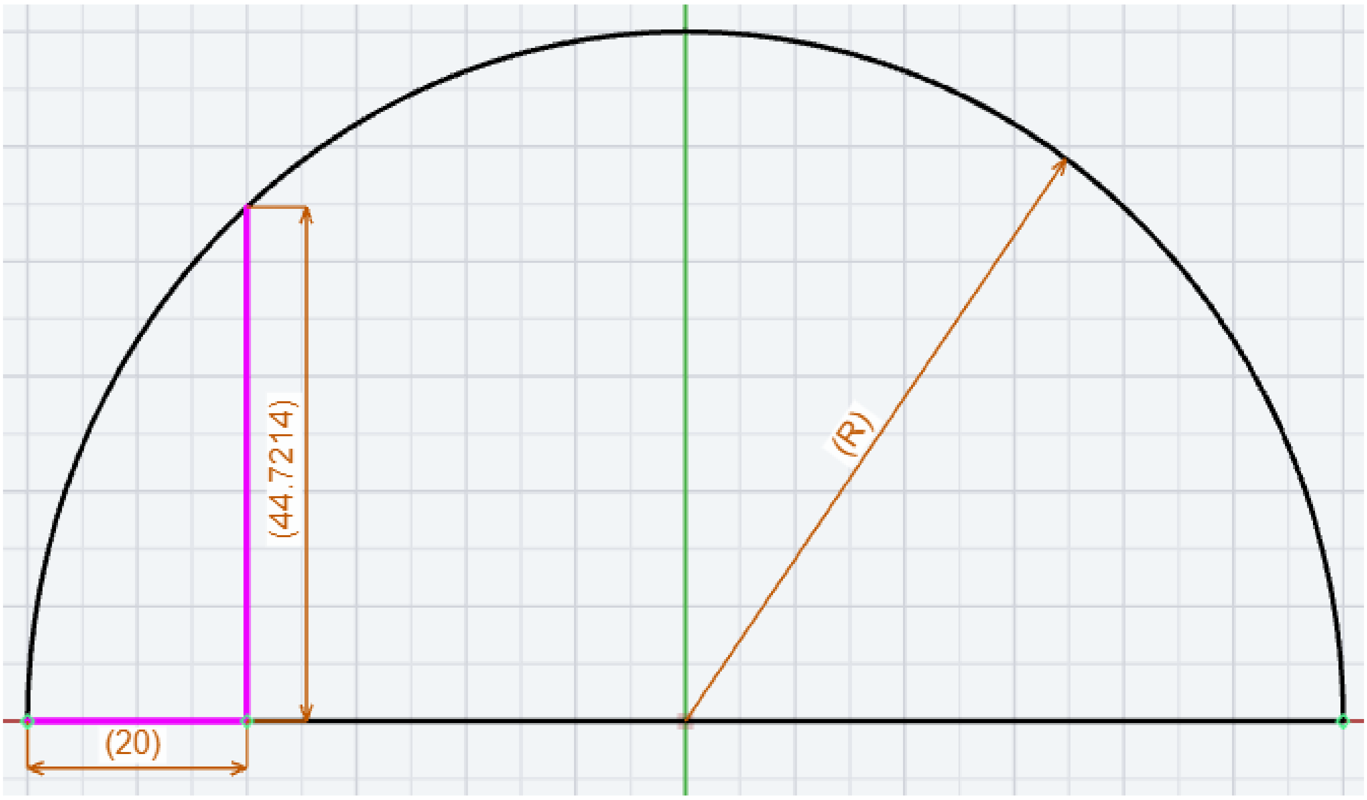
Area = 162.38 mm² → perfect!

39 A semicircle and two lines — how large is the radius?

There is a semicircle and two lines. One line lies horizontally on the axis of the semicircle and has a length of 20 mm.

The second line is vertical and has a length of 44.7214 mm.

How large is the circle radius **R**?



Try to solve this task on your own first.

There is the route via calculation with trigonometric functions, and there is the construction with guides in LiberoDraft.

Please show both routes.

1. The constructive route via the drawing functions of LiberoDraft.

2. And the route via pure calculation.

If you do not find the solution, a clever Greek could help you along (no, it is not Pythagoras). :-)
First think about it yourself and then do some research.

The solution follows on the next pages...

39.1 Thales's theorem

Thales's theorem is a fundamental theorem in geometry, named after the Greek mathematician and philosopher **Thales of Miletus**. Thales lived from about 624 to 546 BC and was one of the Seven Sages of ancient Greece.

Thales's theorem states that every triangle in a semicircle is a right-angled triangle. More precisely, if A, B and C are distinct points on a circle and the segment AB forms the diameter of the circle, then the angle at C is a right angle (90 degrees).

Conversely it also holds that every right-angled triangle can be inscribed in a semicircle. That means that if a triangle has a right angle, you can always find a circle on whose diameter the hypotenuse of the triangle lies and whose corners lie on the circle.

Thales's theorem is an important theorem in geometry and is frequently used to solve problems in plane geometry. It can also be applied to other geometric figures such as trapezoids and parallelograms.

There are various ways of proving Thales's theorem. One possible proof uses the fact that the sum of the interior angles of a triangle is always 180 degrees.

So if you can show that the two angles at A and B are each 90 degrees, the angle at C must also be 90°. This can be shown with the congruence theorems for triangles, for example.

There are also other formulations of Thales's theorem that are equivalent to the one given above.

For example, the theorem can also be formulated like this: if a triangle is inscribed in a circle and one side of the triangle forms the diameter of the circle, then the triangle is a right-angled triangle.

Core statement:

If a point C lies on a circular arc whose diameter is the segment AB, then the triangle ABC is always right-angled at point C.

Reformulation:

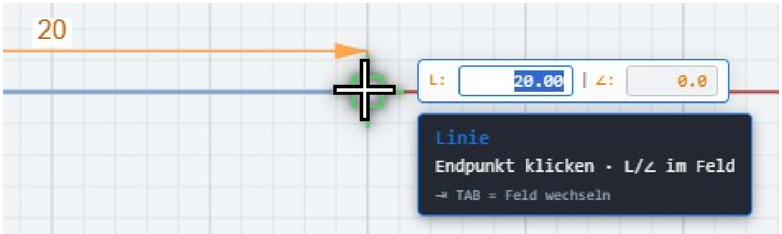
All angles subtended by a semicircular arc are right angles. With this information you should be able to find the solution.

39.2 Constructive solution

First draw the two magenta-colored lines at right angles to each other:

The horizontal line with...

Function: **Line** length: **20,** angle: **0**

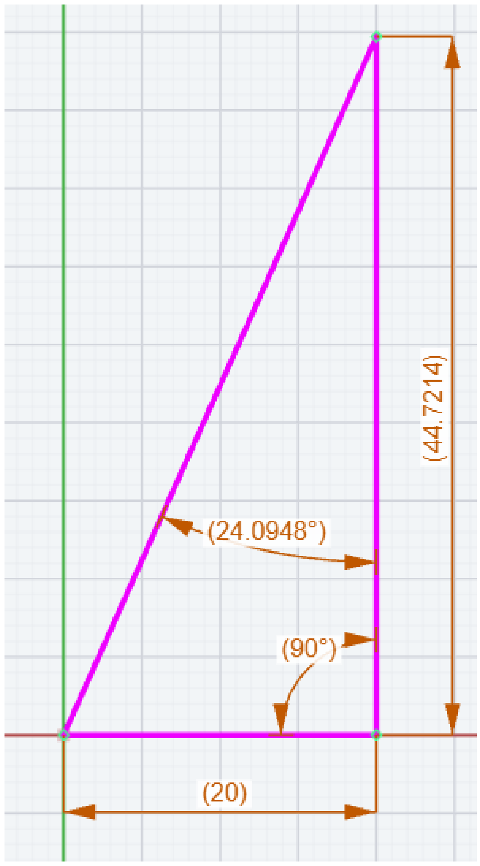


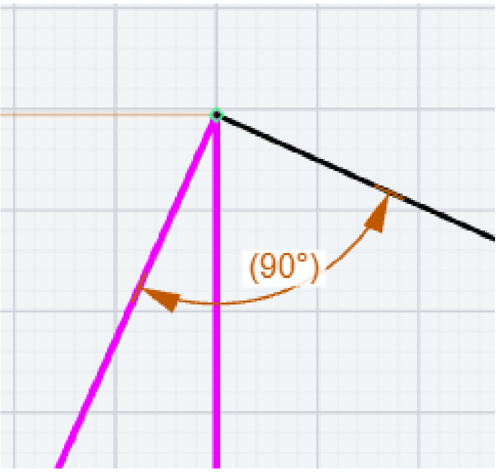
The vertical line with angle: **90**, length: **44.7214**

Now connect the two endpoints with...

Function: **Line** with snap mode **Automatic [A]**

This gives the angle alpha1 → 24.0948°





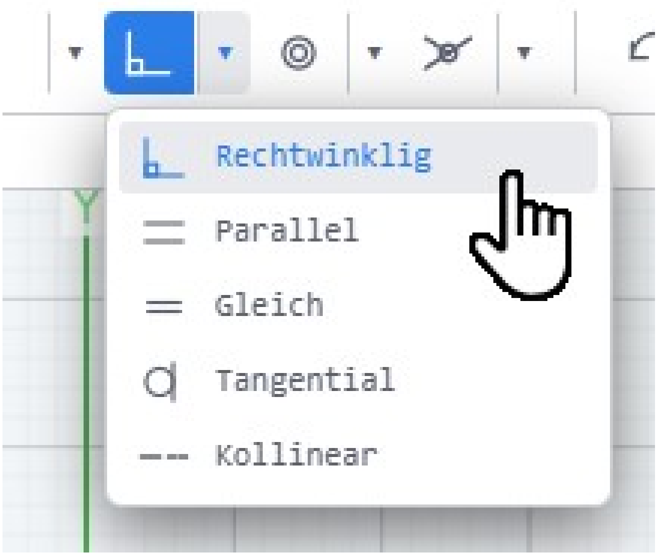
Now we need a perpendicular line here (the black line in the figure)

Function: **Line → snap [A]** and draw roughly from the intersection towards the bottom right.

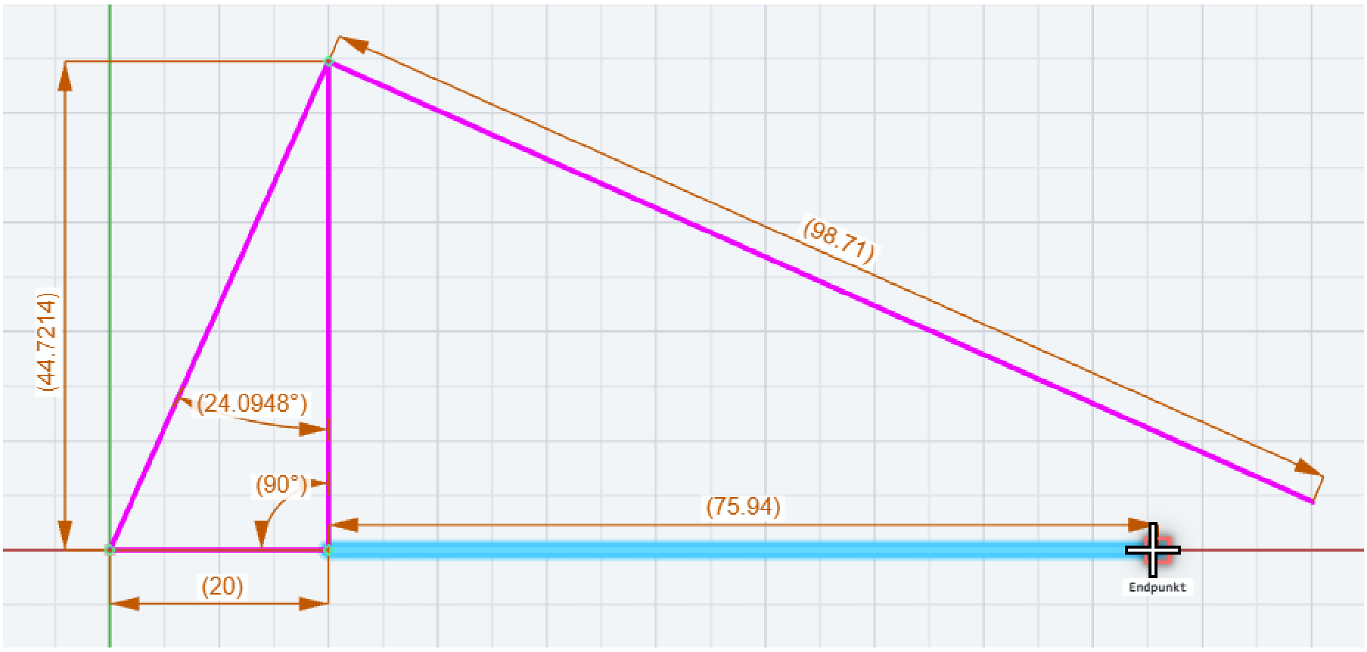
This line is aligned cleanly to 90° with the function **Perpendicular**.

INFO:
Always select the fixed line first, only then click the second line.

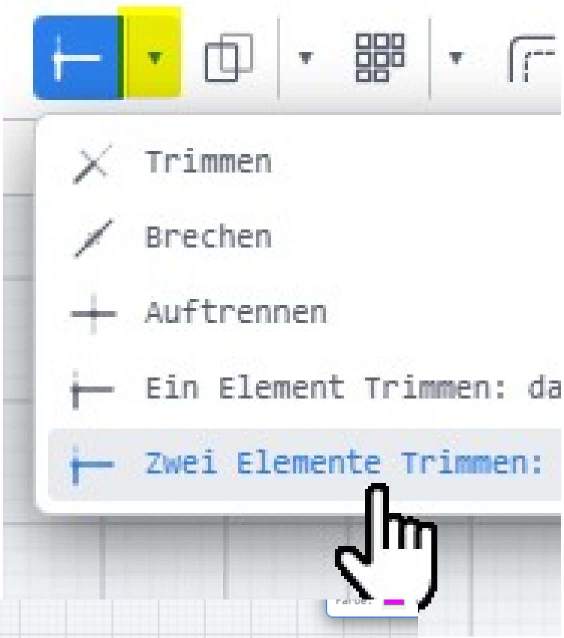
The latter then aligns itself automatically at a right angle to the first line.



Now draw the blue **line** **<u>horizontally</u>** (0°) → the length does not matter:

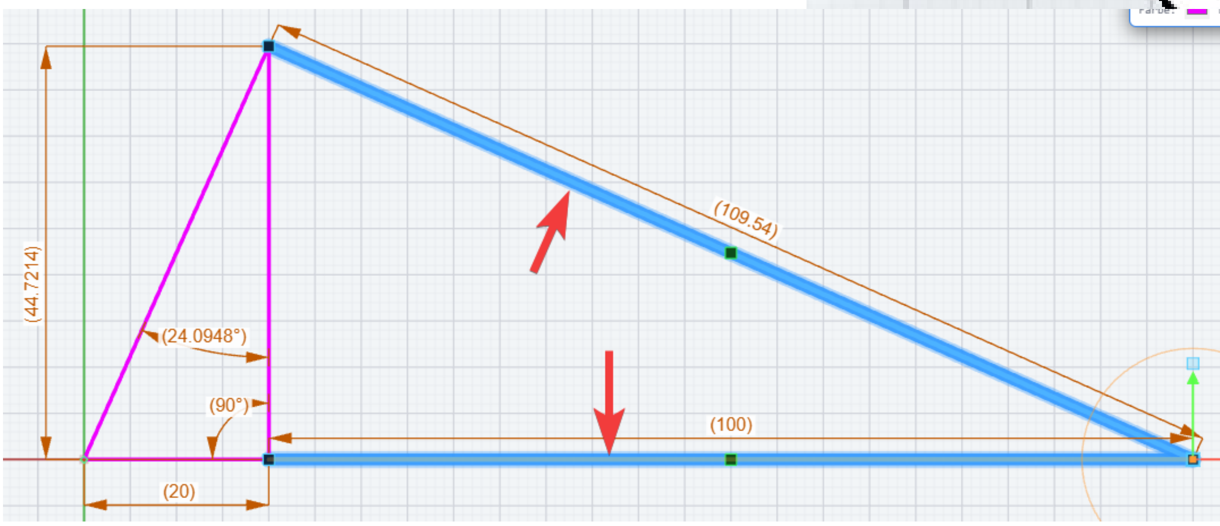


Now two lines are trimmed together:



Function: **Trim two elements**

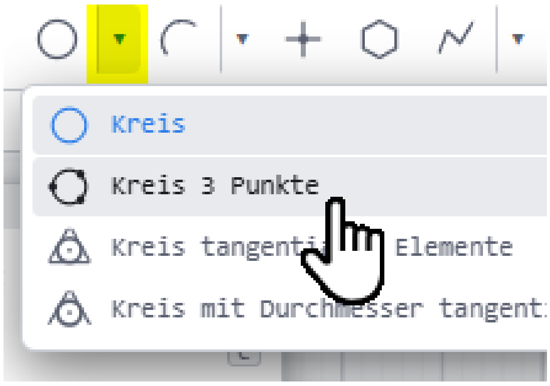
Click these lines:



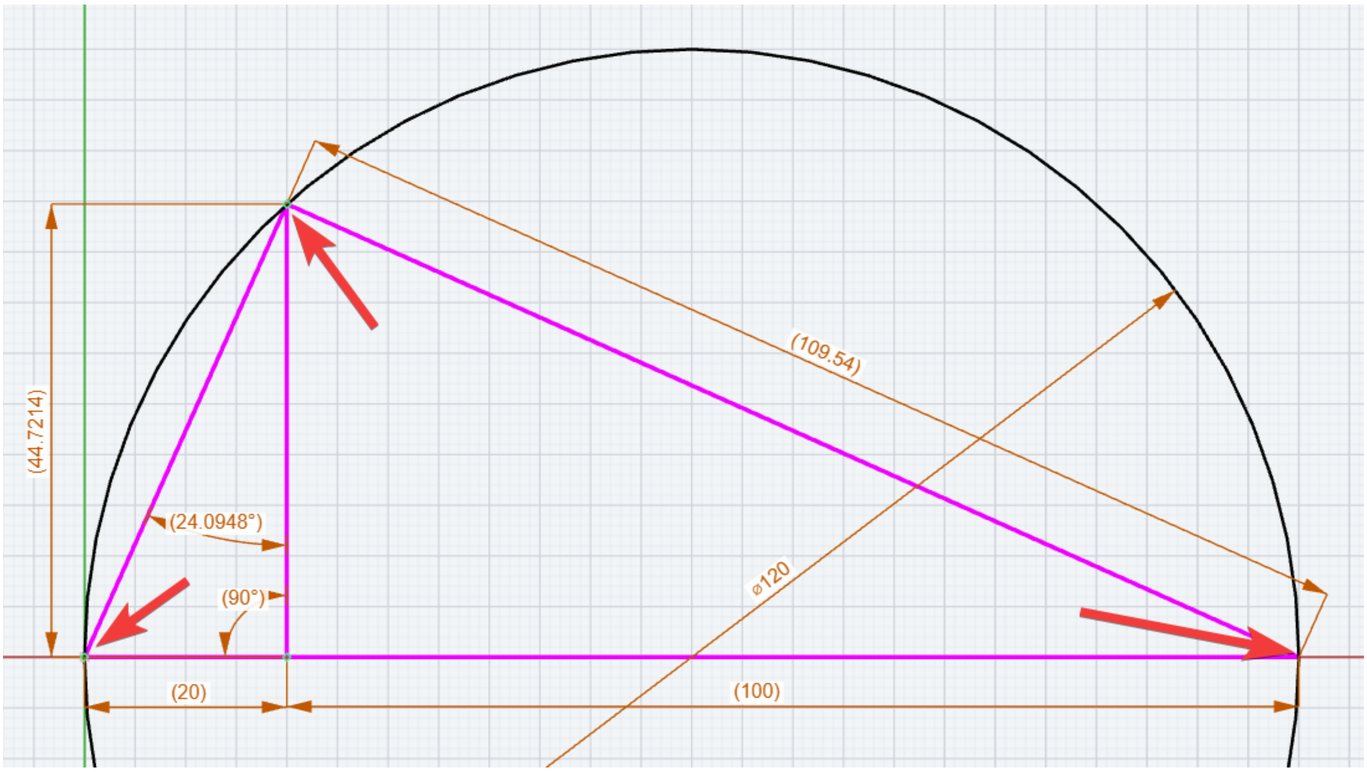
Perfect! With this we have created the basis for drawing a circle through three points.

39.2.1Draw a circle through these three points

Function: **Circle 3 points** with snap mode **[A]** →



Click all three corner points, et voilà, the circle is there :-)

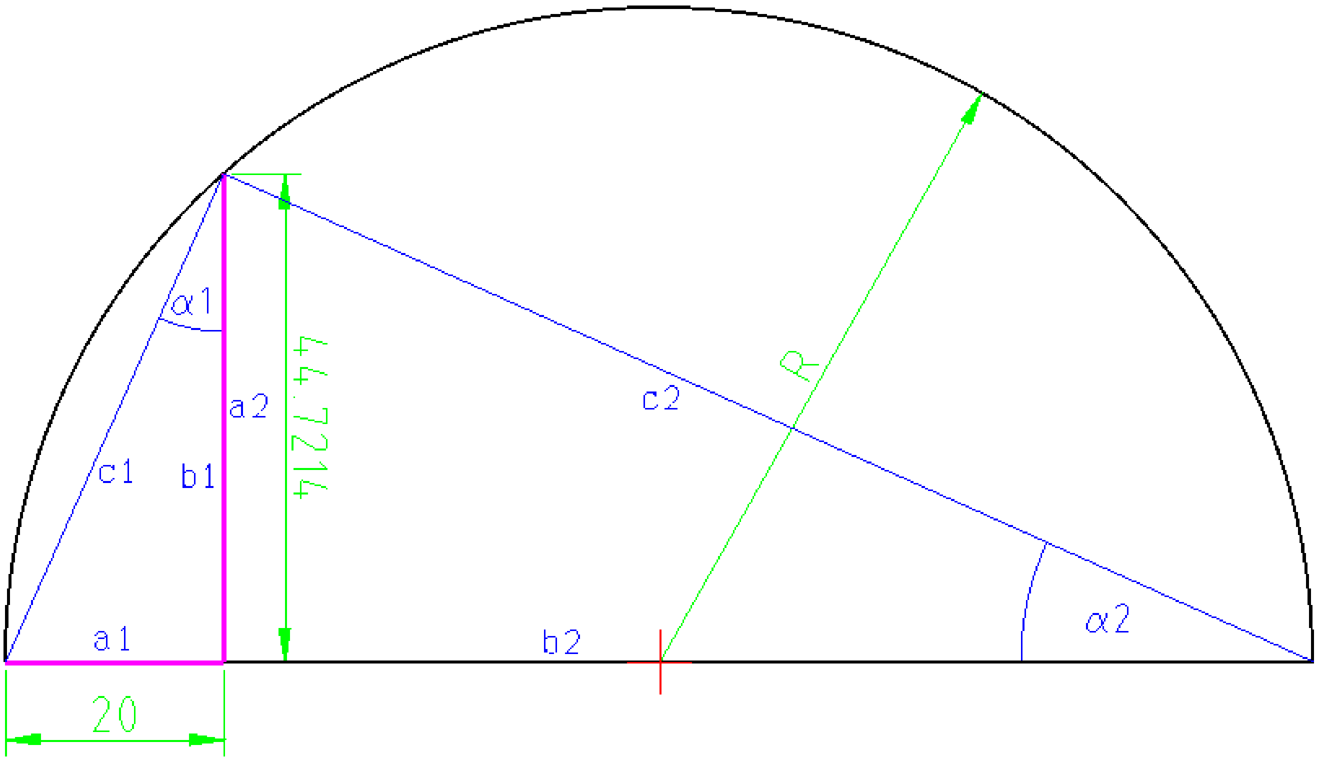


Perfect; now dimension the circle diameter and you have the result: 120 mm.

The radius is accordingly R60.

Below is the other method of calculation according to **Thales's theorem**...

Solution to the task: 39 A semicircle and two lines — how large is the radius?



$$\mathrm{Tan}(\alpha 1) = a1/b1 \rightarrow 20/44.7214 = 24.0948°$$
$$\arctan(20/44.7214)$$

$$\mathrm{Tan}(\alpha 2) = \mathrm{Tan}(\alpha 1)$$

$$b2 = a2/\mathrm{Tan}(\alpha 2) \rightarrow 44.7214/\mathrm{Tan}(24.0948) = 100$$

$$R = (100+20)/2 = 60$$

Application:

Thales's theorem is applied in various fields, among them:

- **Navigation**
  determining the compass direction with the help of the sun and the stars
- **Construction**
  calculating angles and lengths in structures
- **Surveying**
  establishing angles and distances in geodesy

Thales's theorem is a fundamental theorem of geometry with a wide range of applications. It makes it possible to identify, construct and analyze right-angled triangles and plays an important role in various fields such as navigation, construction and surveying.

39.3 Constructing a circle tangent

One important application of Thales's theorem is, among others, the construction of the two tangents to a circle **k** through a point **P** lying outside that circle.

To sum up, the individual steps of the construction can be described as follows:

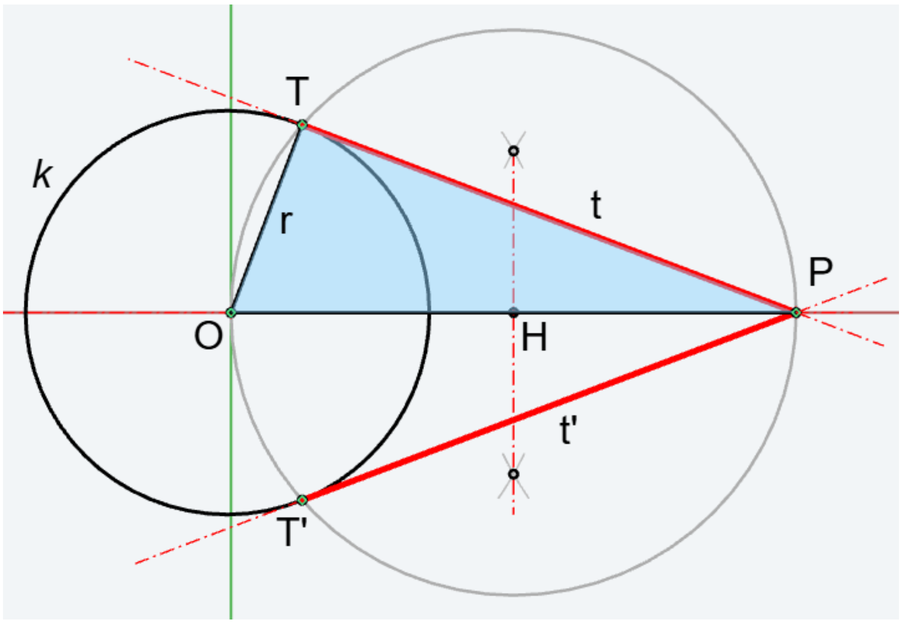


Figure: the graphic above was created with LiberoDraft
LiberoDraft is also excellent for explanatory graphics of any kind :-)

1. Given are the circle k with the center O and the radius r, as well as the point P outside the circle.
2. Draw the segment O P and determine the midpoint H of that segment with the help of the perpendicular bisector.
3. Draw a circle with the radius H O about the center H. This circle is also called the Thales circle.
4. Determine the intersections of the Thales circle with the given circle k. These intersections correspond to the points at which the tangents sought touch the circle k.
5. Join the point P to the points of contact of the tangents and you obtain the tangents t and t' sought.

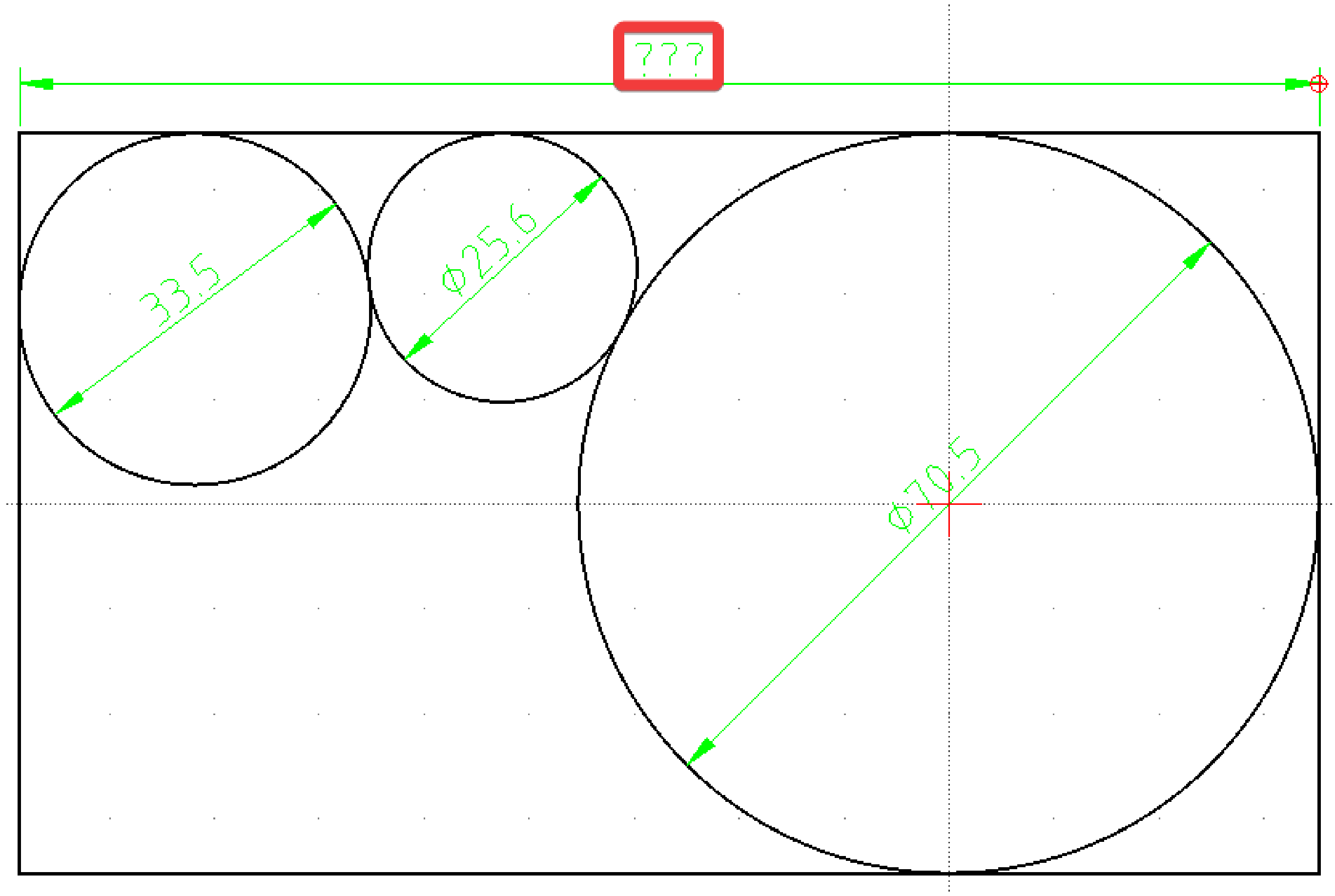
Since Thales's theorem states that every triangle in a semicircle is a right-angled triangle, this also applies to the triangles O P T and O P T' formed by the tangents t and t'. The segments O T and O T' must therefore be perpendicular to the tangents t and t', which satisfies the fundamental property of a circle tangent.

Do not worry! In LiberoDraft you do not have to construct tangents by hand.
The program has very good solutions for that.

40 Tangents exercise

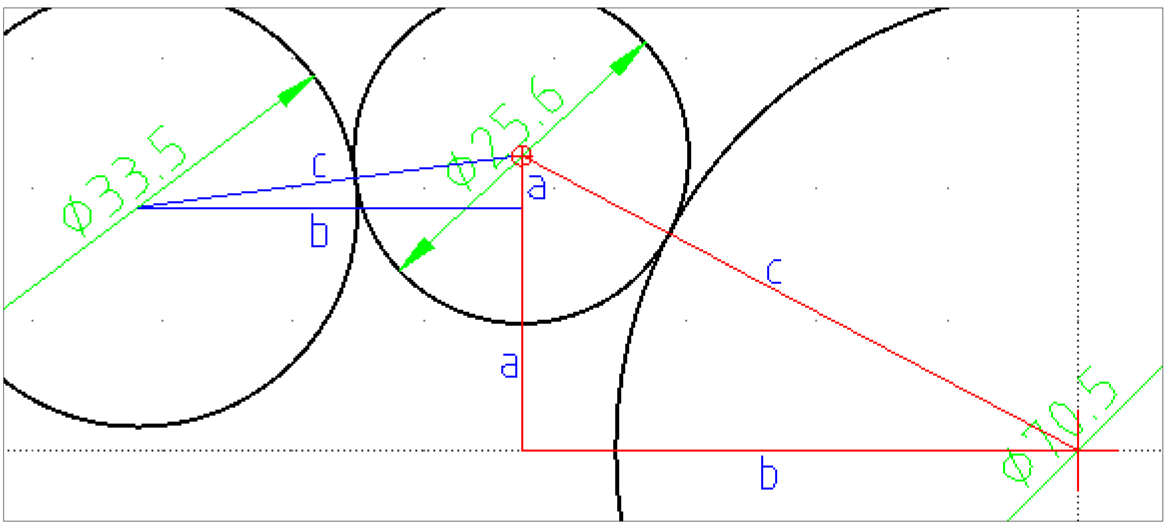
Three tangential circles are drawn in a rectangle.
Only the diameters of the circles are known.

How long is the surrounding rectangle?



First try the traditional calculation by hand.

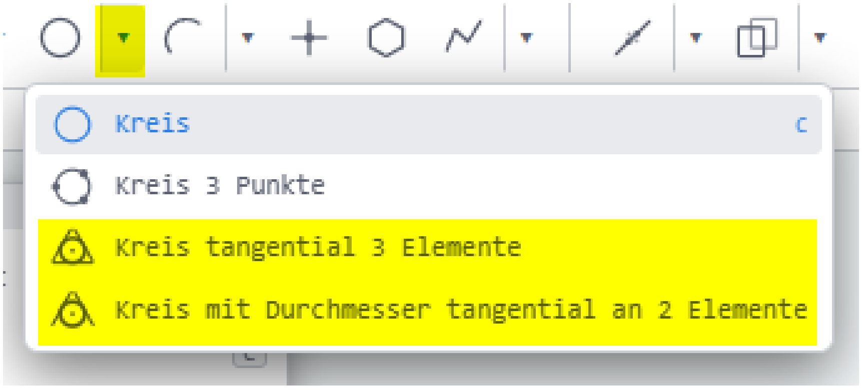
With these two triangles that works without any problem :-)



41 Experimental constraints

41.1 Circle tangent

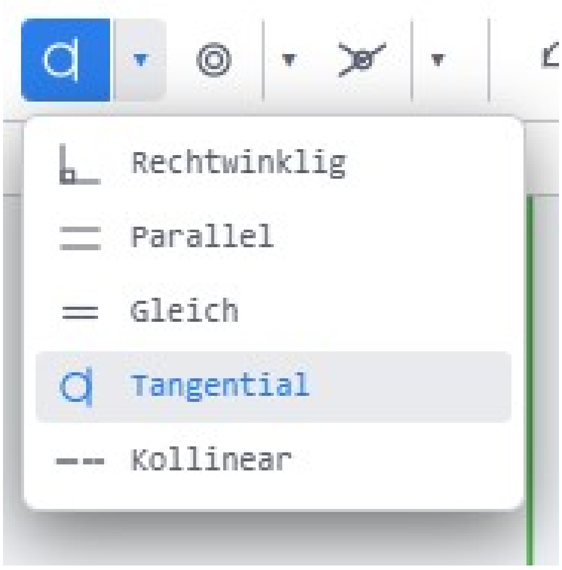
Either you have to draw circles tangent to existing lines, or the other way round.



These functions are self-explanatory: you select the elements and move into the sector you want to keep.

41.2 Line tangent

Either you have to draw lines tangent to existing circles, or the other way round.



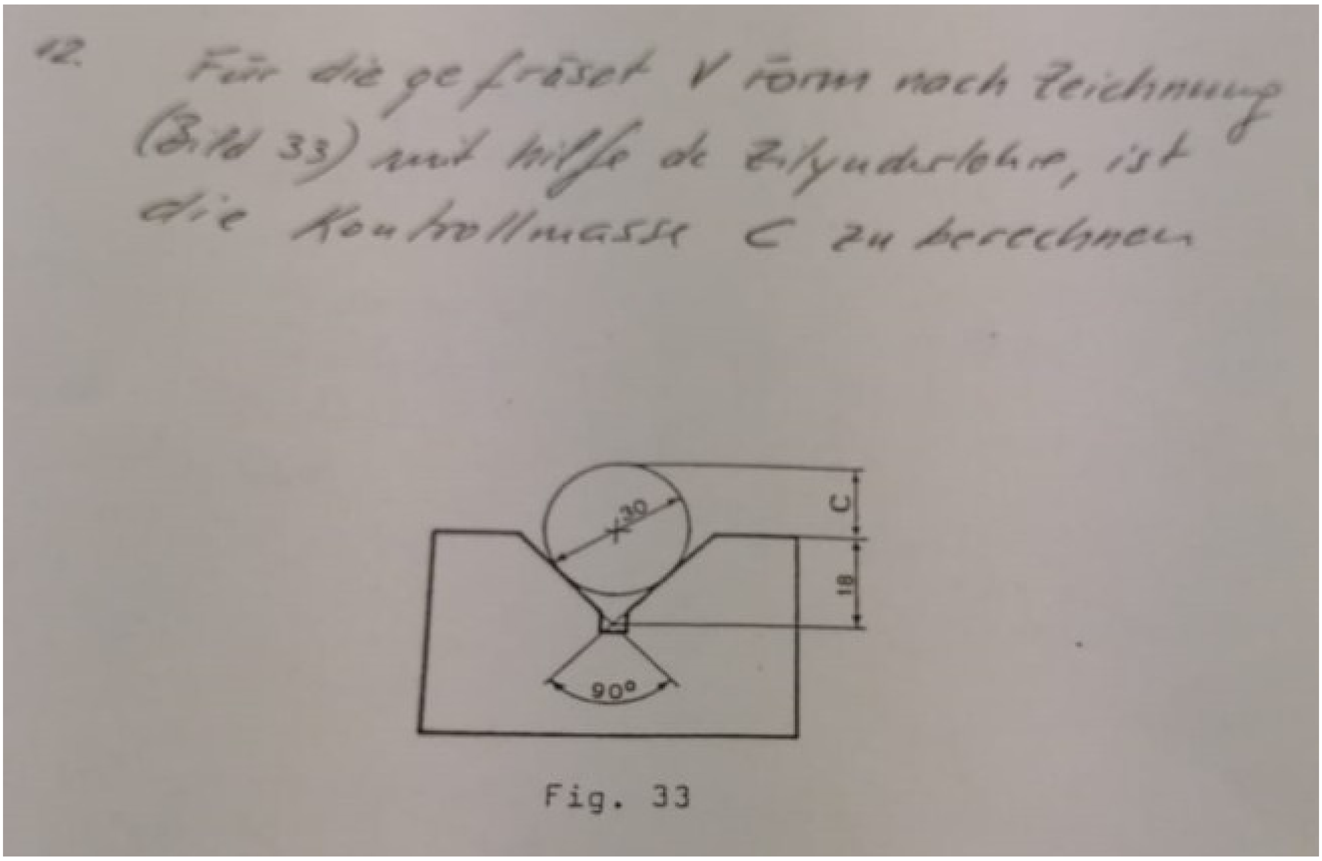
Tangential lines, circles or arcs are created via so-called "geometric relationships". Here too, anyone who reads the notes in the floating menus needs little help :-)
For example, simply draw a circle freely plus a line, circle or arc somewhere and call up the Tangent function.

INFO: tangential and other relationships follow changes in the geometry.

42 Prism check dimension

IMPORTANT: left mouse button = LMB, right mouse button RMB

We have received the following sketch and have to determine the check dimension C.



The check dimension C can be calculated nicely with the theorem of Pythagoras :

C = sqrt(a^2 + b^2) → sqrt(15^2 + 15^2) = 21.213

21.213 + 15 = 36.213 ...continuing... 36.213 – 18 = **18.213**

Calculator

In the command line **type cal → [Enter]**, and the input line turns into a calculator!

Example: **Sqrt(15^2+15^2) → [Enter]** = 21.2132034356

Typing **cal → Enter** again ends the calculator.

Whether that is correct we will now re-construct and check with LiberoDraft by redrawing the sketch.

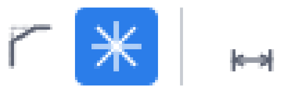
To do so, start LiberoDraft and use **File → New drawing** to load the template created earlier in this book: **___Vorlage-001.cad2d**

The template is loaded. So that it is not overwritten by accident, save the file under a new name right away: **Prisma** would be a good choice :-)

For the prism we simply draw a rectangle of 60 x 40 mm

Select layer: **SICHTBAR**

Menu: **Rectangle → W: 60, H: 40 → Enter**



INFO: the rectangle is a block (that is, a single element consisting of four lines). The **function after next**, **Offset**, with a distance of 18 mm would create the offset from all four lines, which is fine in itself but is not needed in our case. The rectangle therefore has to be dissolved into its individual parts beforehand with the "**Explode**" function.

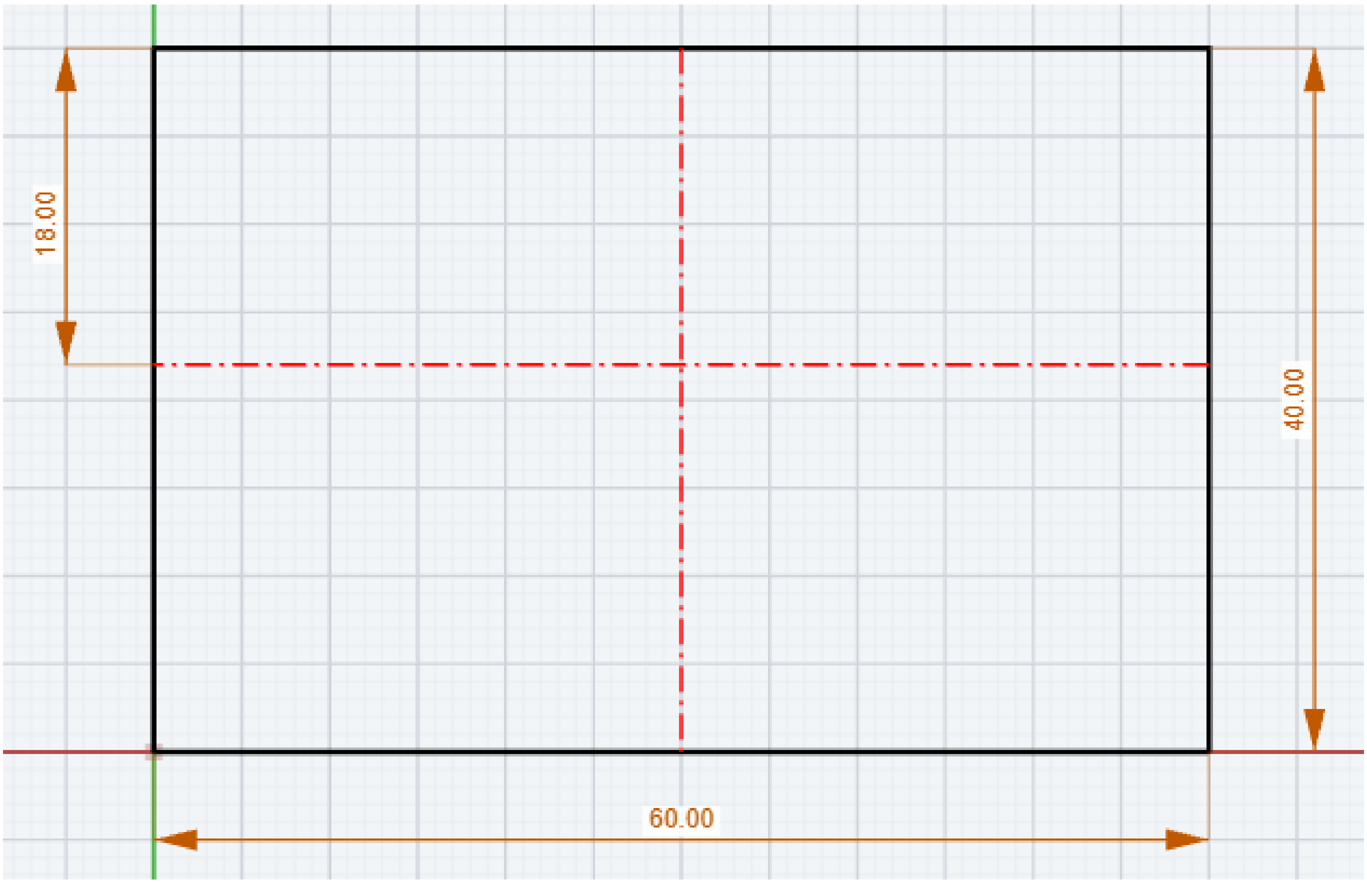
First the **vertical** axis of symmetry at the center...

Select layer: **Achsen 0.25**

Menu: **Line → automatic snap [A]** and draw the vertical axis...

Menu: **Offset → D: 18** draw the parallel line...

The rectangle with the two axes should now look like this:

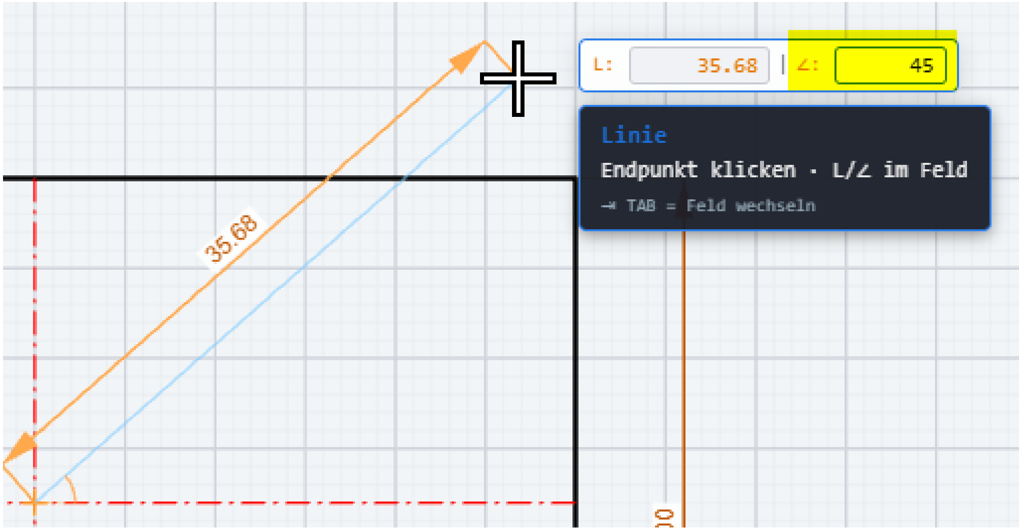


The dimensioning is there only for checking purposes.

The two lines for the prism are now drawn from the intersection:

Layer: **SICHTBAR**

Menu: **Line** → **automatic snap [A]** draw from the intersection



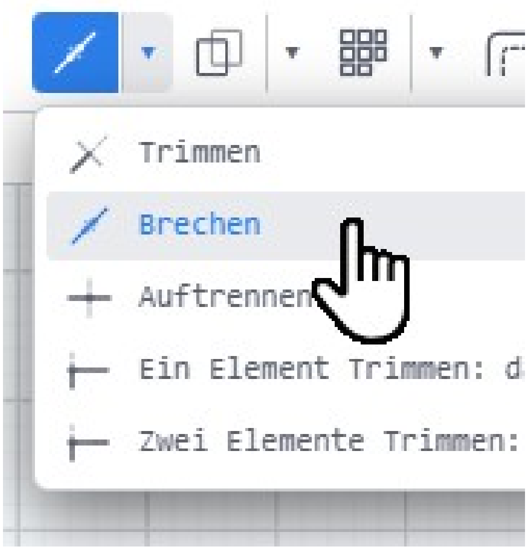
Length: does not matter, simply beyond the line of the rectangle

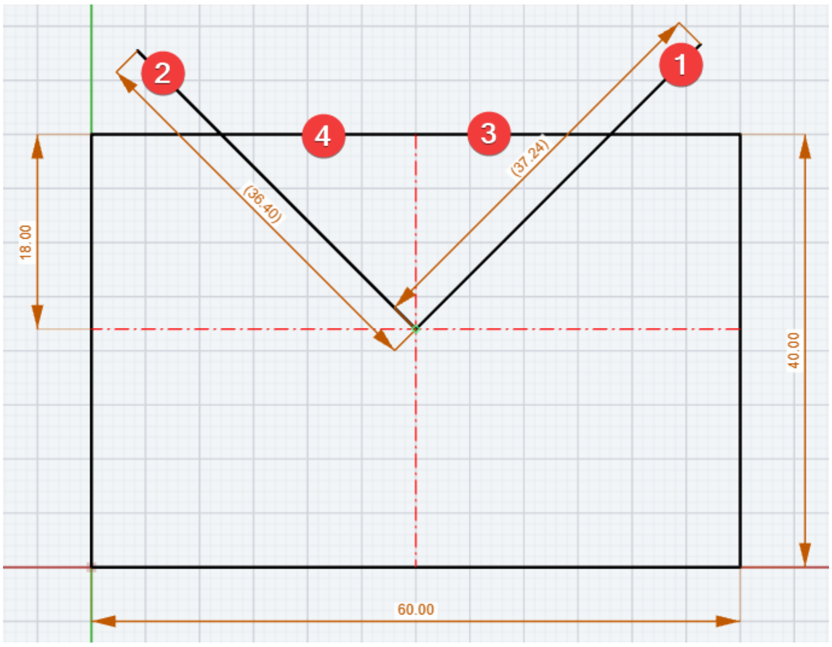
Angle: 45° → Enter → Esc

The second line on the **other** side likewise with **angle: 135°**.

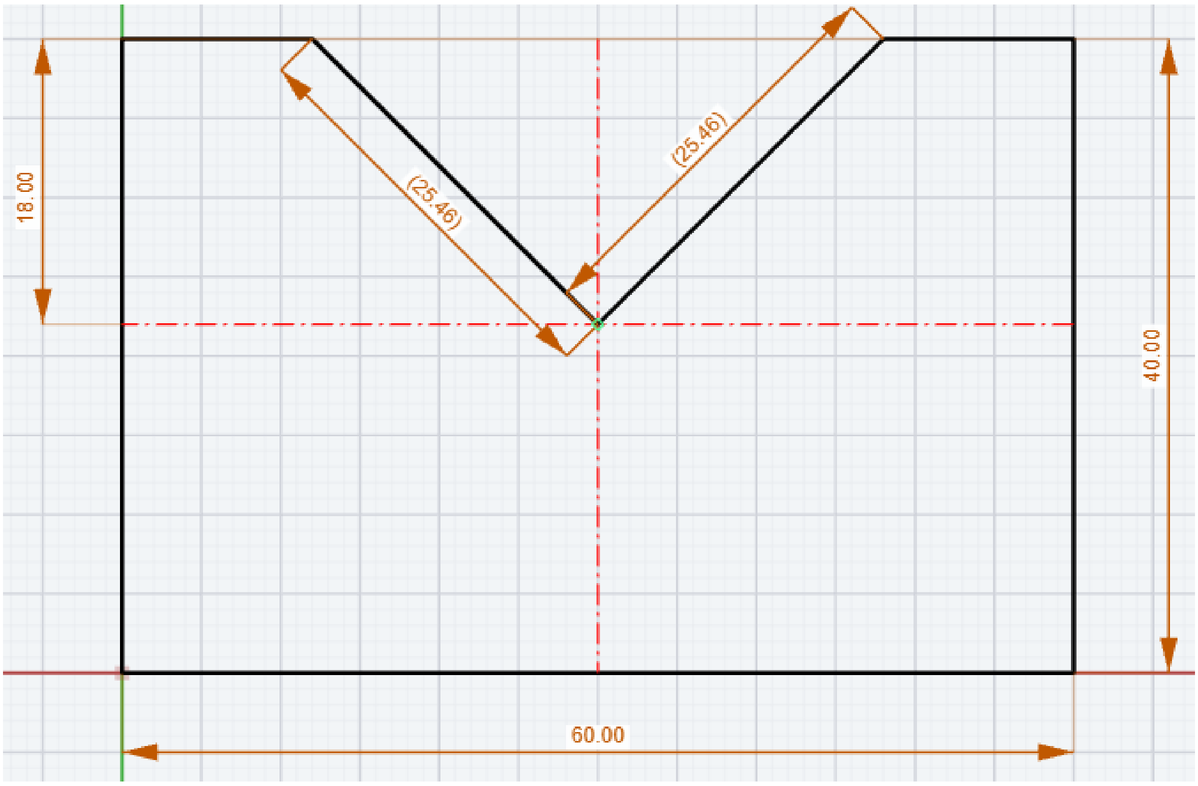
The Break function is used to remove all superfluous line segments.

Menu: **Break →** if you like, **snap mode Free [F]**
Break away lines (1), (2) as well as above the "V" of the prism, (3) and (4)...





So far so good :-)



Next, the circle is placed tangent to both flanks of the prism…

Layer: **SICHTBAR**

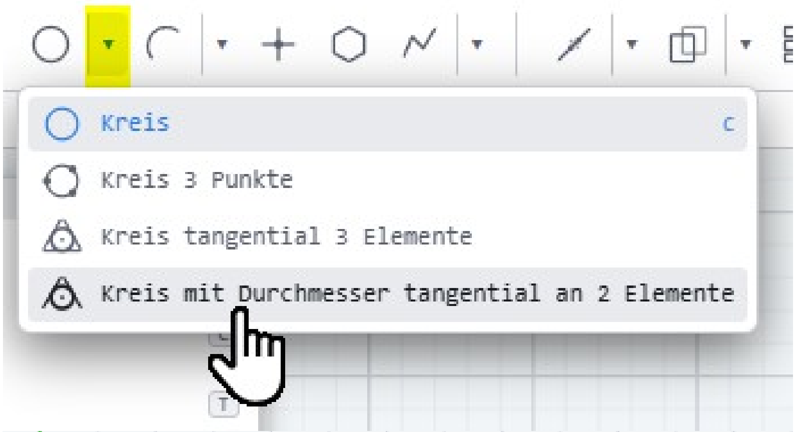
Menu: **Circle → Circle with a given diameter tangent to 2 elements** → **snap mode [F]**

Select both flanks.

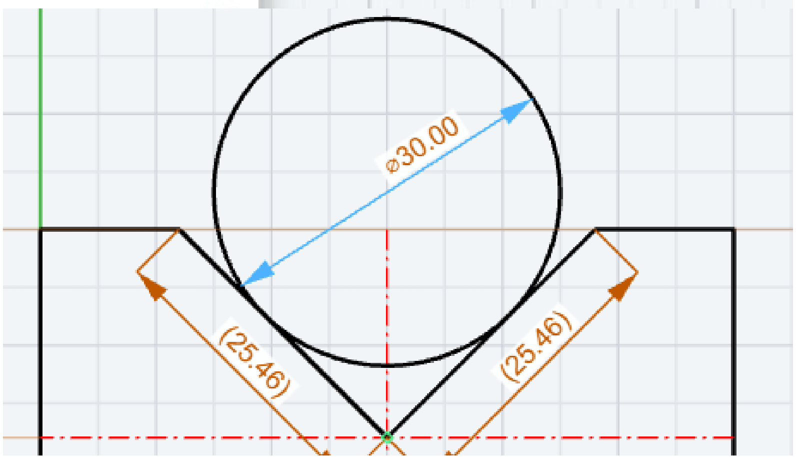
Enter the diameter **30**

Enter key

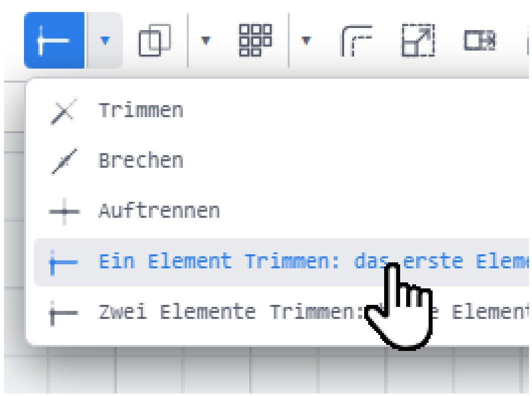
Choose the sector with the **mouse** and **click**.



Perfect! The circle sits cleanly on both flanks.



42 Prism check dimension

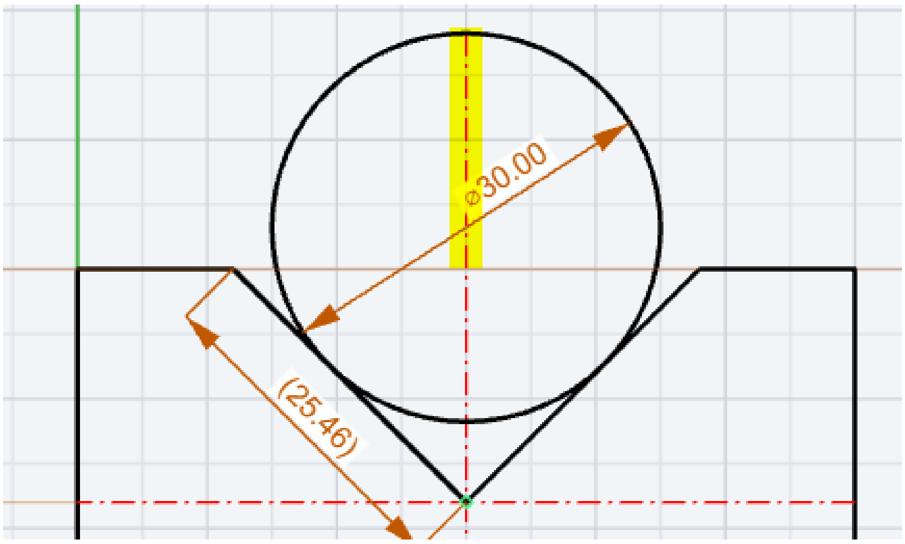


So that the check dimension C can be determined, it would make sense to extend the vertical axis upwards to the circumference of the circle.

Function: **Trim one element**

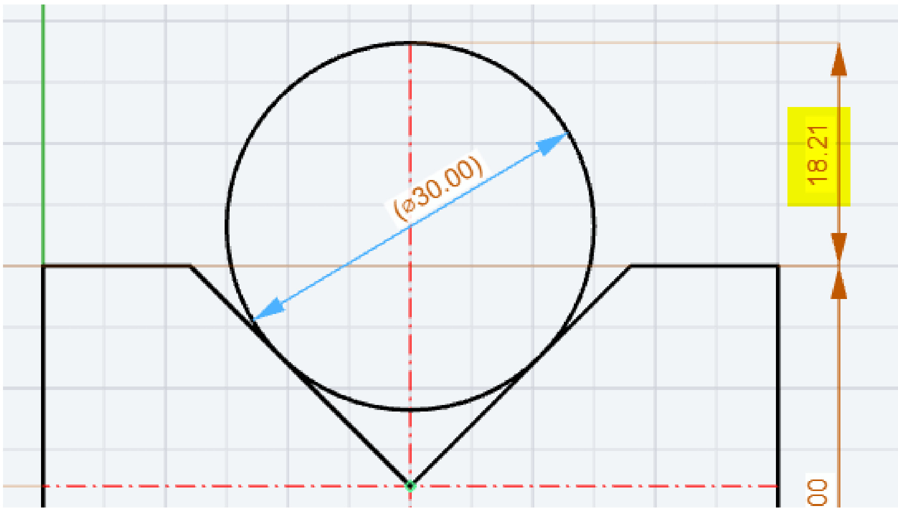
Select the axis, select the circle at the top.

The axis is extended cleanly:



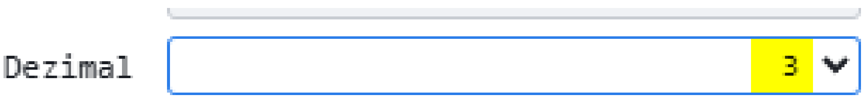
42.1 Placing the check dimension C

Dimension → automatic snap [A]



We see that the check dimension **C = 18.213**, which we calculated correctly with the theorem of Pythagoras earlier :-)

Stop! we only read 18.21 → Solution: **double-click the dimension value** and **change the decimal places** to **3**.



Now it fits :-

43 Inserting an image and tracing over it

With the **Insert image** function you place a pixel image into the drawing as a background. The image serves as a template over which you lay your clean vector geometry with the normal drawing tools. The most frequent application is **vectorizing**: from a sketch, a photo, a scanned plan or a screenshot, a true-to-scale CAD drawing of real lines, arcs, circles and splines is created.

43.1 What the function is good for

- **Vectorizing templates ->** a hand sketch, an old paper data sheet or a logo is traced into a CAD drawing that can be dimensioned and printed.
- **Taking measurements to scale ->** a photo or scan with a known dimension (a ruler in the picture, for example) is calibrated; afterwards lengths can be taken off directly and correctly.
- **Reference and comparison ->** an existing image serves as a background in order to insert new geometry with a precise fit or to check proportions.
- **Annotating and taking off quantities ->** dimensions, notes and symbols are placed directly over the image.

43.2 Supported image formats

The common pixel formats **JPG/JPEG, PNG, GIF** and **WebP** are read in. A PDF is not loaded directly -> export or "photograph" the page you want as a PNG or JPG beforehand (by taking a screenshot at a good resolution, for example).

43.3 Inserting an image

1. In the toolbar, click the **Insert image** button (the symbol with a picture frame and a mountain motif).
2. In the file dialog, choose the image file you want.
3. The image appears centered in the current view, about half as large as the drawing area and with 50 % transparency.

> **Note:** The image is automatically placed **behind** all the other geometry. Your drawn lines therefore always lie visibly on top -> the image never covers them.

43.4 Opening and adjusting the image properties

Open the **Image properties** dialog in one of two ways:

- **Double-click** the image, or
- **right-click** the image and choose **Image properties**.

The following settings are available in the dialog:

Field	Meaning

Transparency	Opacity of the image from 0 to 100 %. 50 % ex works, so that the geometry drawn over it stays clearly visible. Takes effect immediately as the slider is moved.
Angle (°)	Rotates the image about its center. Useful for straightening a template that was scanned crookedly.
Width / height	Size of the image in drawing units (mm). The aspect ratio is retained: if you change one value, the other is scaled automatically with it.
X / Y	Position of the lower left corner of the image in drawing coordinates. Allows exact positioning without dragging with the mouse.
Locked	Fixes the image. A locked image can no longer be moved by accident and shows a padlock symbol. Recommended as soon as the template is correctly placed.
Calibration	Scales the image to the true scale (see the section "Calibrating to scale").
Delete	Removes the image from the drawing.

Close the dialog with **OK**.

43.5 Calibrating to scale (for vectorizing)

So that you can take real dimensions off the template, the image first has to be brought to the true scale. For this you need **a known distance** in the image -> a dimensioned edge, a ruler shown in the picture or a scale bar, for example.

1. Insert the image and place it roughly.
2. Use the **Measure** tool (key `I`) to take off the known distance in the image -> that is, click the two endpoints of the reference edge -> and **read off the measured value**.
3. Open the image properties. In the **Calibration** section, enter the value you read off under **Measured** and the true dimension you want under **Effective**.
4. Click **Scale**. The image is enlarged or reduced so that the measured distance corresponds exactly to the effective value (scale factor = effective ÷ measured). Width and height are adjusted uniformly.

> **Example:** An edge measures `47.0` in the inserted image, while in reality it is `100.0` mm long. "Measured" = 47, "Effective" = 100, "Scale" -> the image is enlarged by the factor `2.128` and is then correct to scale.
>
> **Tip:** The longer the reference distance in the image, the more accurate the calibration. Take off the longest reliably known dimension, not the shortest.

43.6 Drawing over the template (tracing)

As soon as the image is right, draw your geometry over it:

1. Recommended: fix the image with **Locked** so that it is not moved by accident while drawing.
2. It is best to create **a separate layer** for the traced geometry -> the image can then easily be hidden separately later on.
3. Trace the contours of the template with **Line, Arc, Circle, Polyline** and **Spline**.
4. Adjust the **transparency** of the image if required: higher, to see details better; lower, to bring out your own geometry.

> **Important:** The image is **not a snap object**. The object snap only latches onto your own geometry (endpoints, midpoints, intersections and so on), not onto image pixels. You therefore trace the template "by eye" and afterwards join your own points exactly via the snap.

43.7 Controlling the image in print

Inserted images can optionally be included in or left out of the printout. In the **print dialog** you control this with the **Images** option. If it is activated, the image is printed with the transparency that has been set (in grayscale in B/W mode). For a clean technical output you will usually switch the template **off** here -> only your vector geometry is then printed.

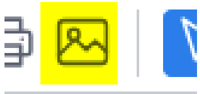
If the image is no longer needed, you can remove it from the drawing completely via **Image properties → Delete**.

43.8 Quick guide at a glance

1. **Insert image** → choose the file.
2. **Calibrate**: measure a known distance with `I`, enter Measured/Effective, **Scale**.
3. Set **Locked** and choose your own layer.
4. **Trace** with the drawing tools.
5. Switch the **Images** option off when printing, or delete the image.

43 Inserting an image and tracing over it

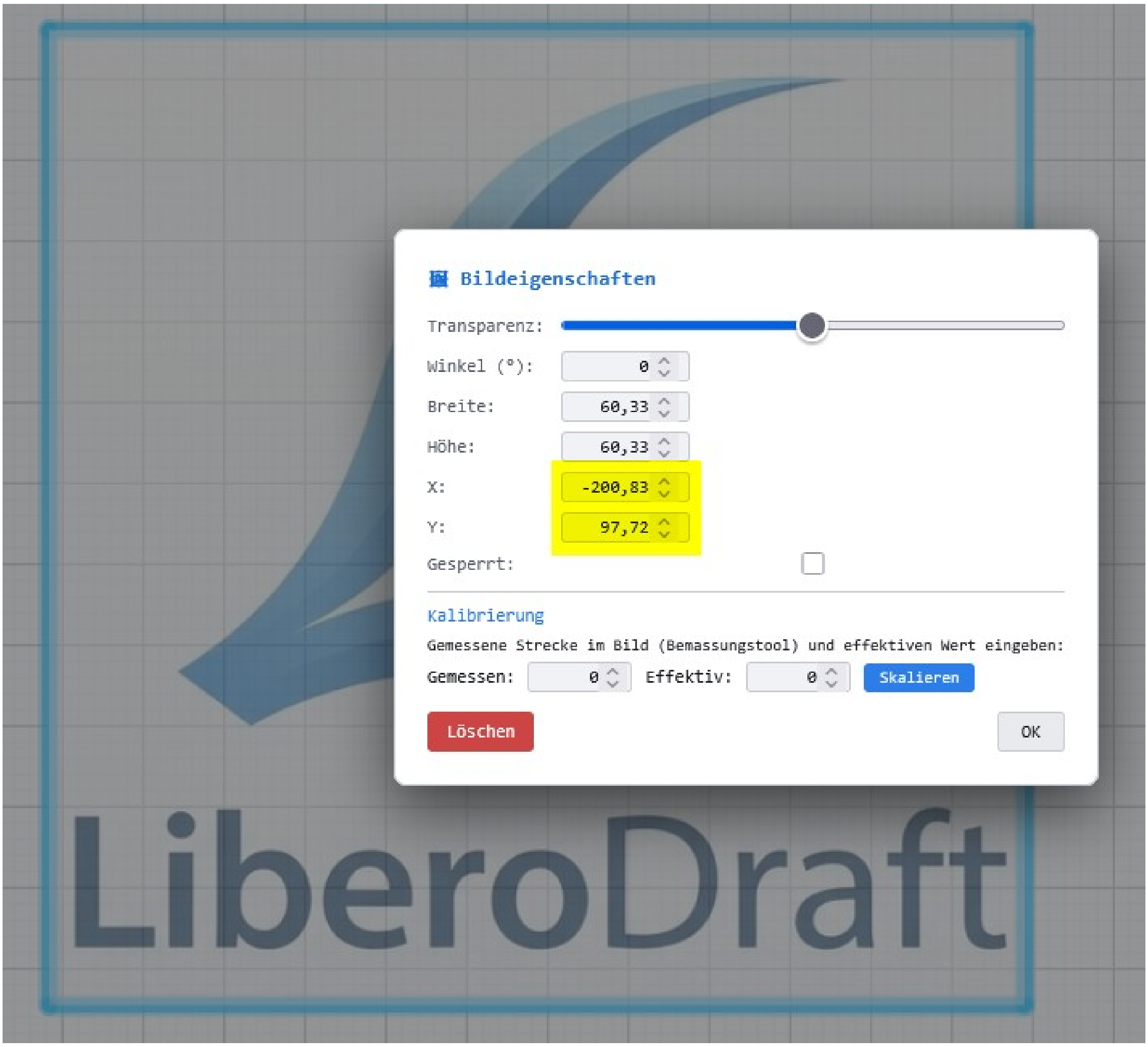
Example

Menu: **Insert image** 

Choose the image and load it. In certain situations the image will load somewhere outside the visible area.

Menu: **Zoom to fit [Ctrl + F]** performs a zoom ALL → the image is displayed.

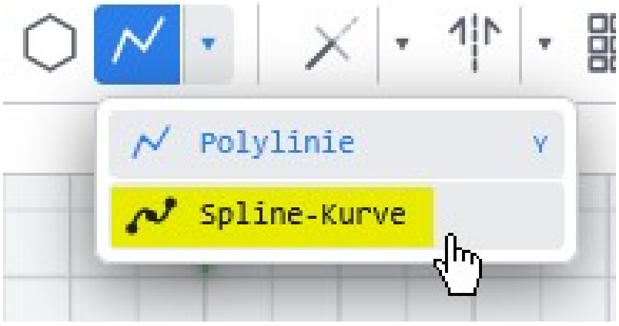
A double-click takes you into the image properties for editing



We enter 0 for both X and Y and confirm with OK.

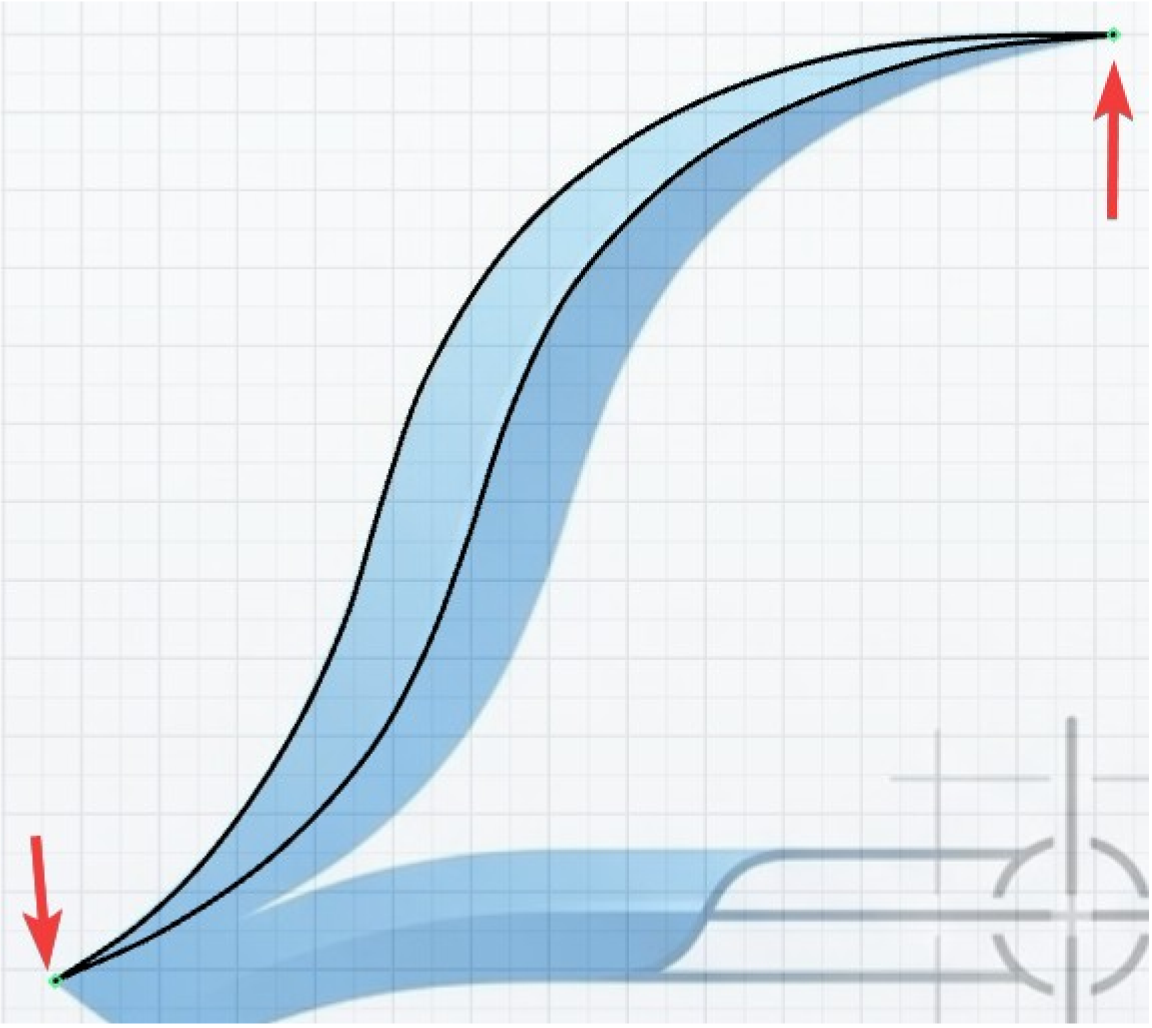
If you want to work on a cadastral plan at a particular scale, for example, the image has to be calibrated as described above.

The spline function is best suited to vectorizing the image:



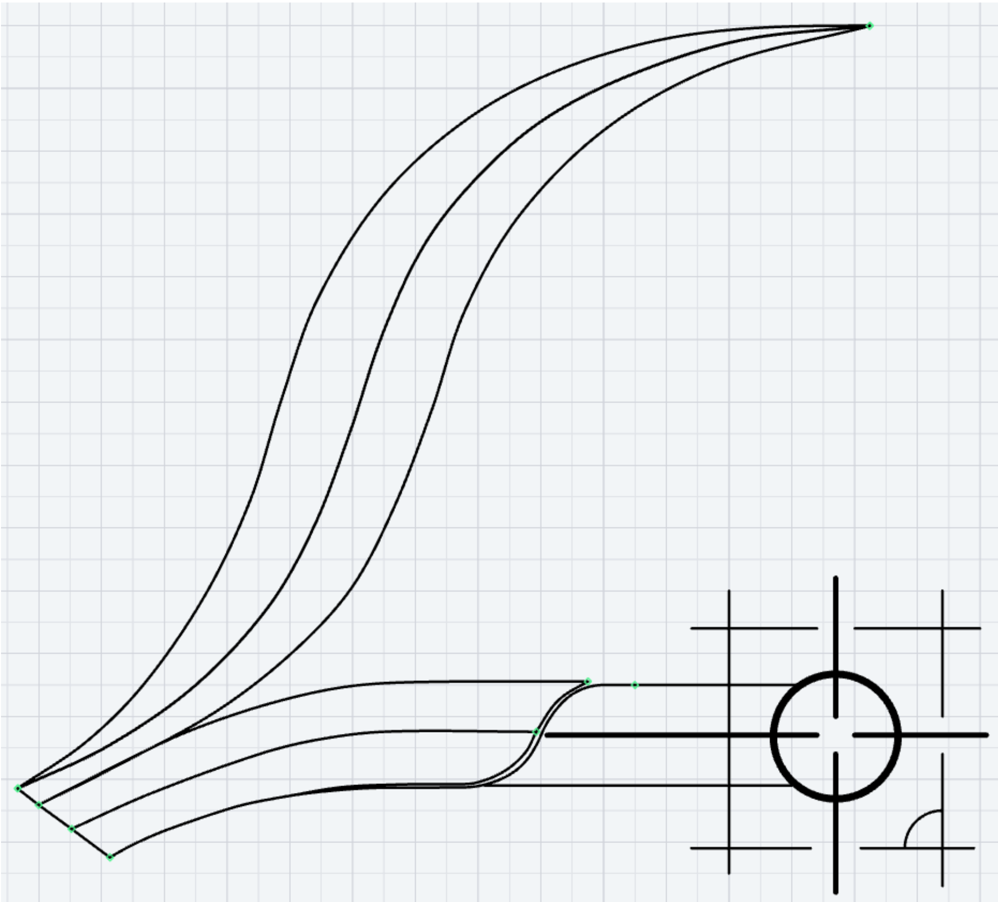
Menu: **Spline curve → snap mode Free [F]**

Take off the points one after another and feel free to leave a little space between the picks…



So that the second spline joins on cleanly, use **Endpoint [E]** snap mode at the start and at the end and [**F**] for **Free** in between...

Make sure everywhere that the areas to be filled are closed.
Afterwards they can be filled with: menu: **Hatch** → pattern: **SOLID**



So that the fill can flow into the small details in vector form, the tolerance may have to be increased:

43.9 Dimensioning over an image

An inserted image — a scanned plan or a cadastral extract, for example — can not only be traced: you can also dimension the geometry drawn over it directly. The image no longer intercepts clicks; dimensions and annotations find the lines above it.

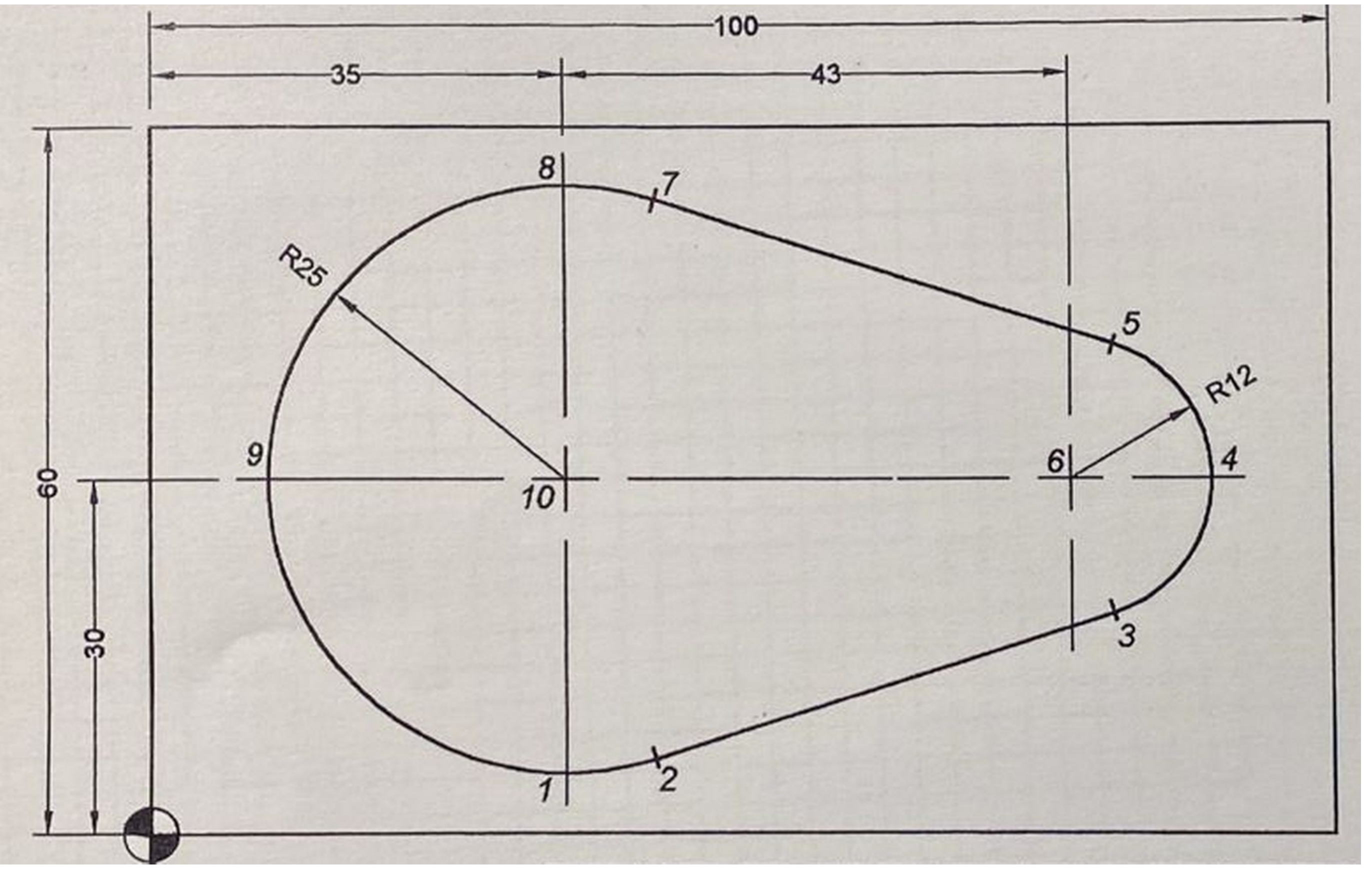
[FIGURE: dimensioning over a scanned plan]

A double-click on a dimension lying over an image likewise opens the dimension editing and not the image settings.

The Z level additionally determines what lies in front. The order applies to images and annotations too, and texts can even be placed behind the drawing so that lines lie over the text.

44 Drawing: belt drive and coordinates

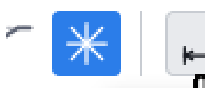
Next, the following belt drive is to be drawn and the coordinates of points 1-10 as well as the angles and the total belt length determined:



Activate layer: **SICHTBAR**.

Menu: **Rectangle**

Tip:
Make sure that **snap mode Grid [G]** is activated:
Drag the rectangle out from the **origin** and either use the mouse on the grid point or enter **100,60** on the keyboard and confirm with **Enter**.


The rectangle is still a block that first has to be dissolved into its individual parts with the **Explode** function.

Activate layer: **ACHSEN**.

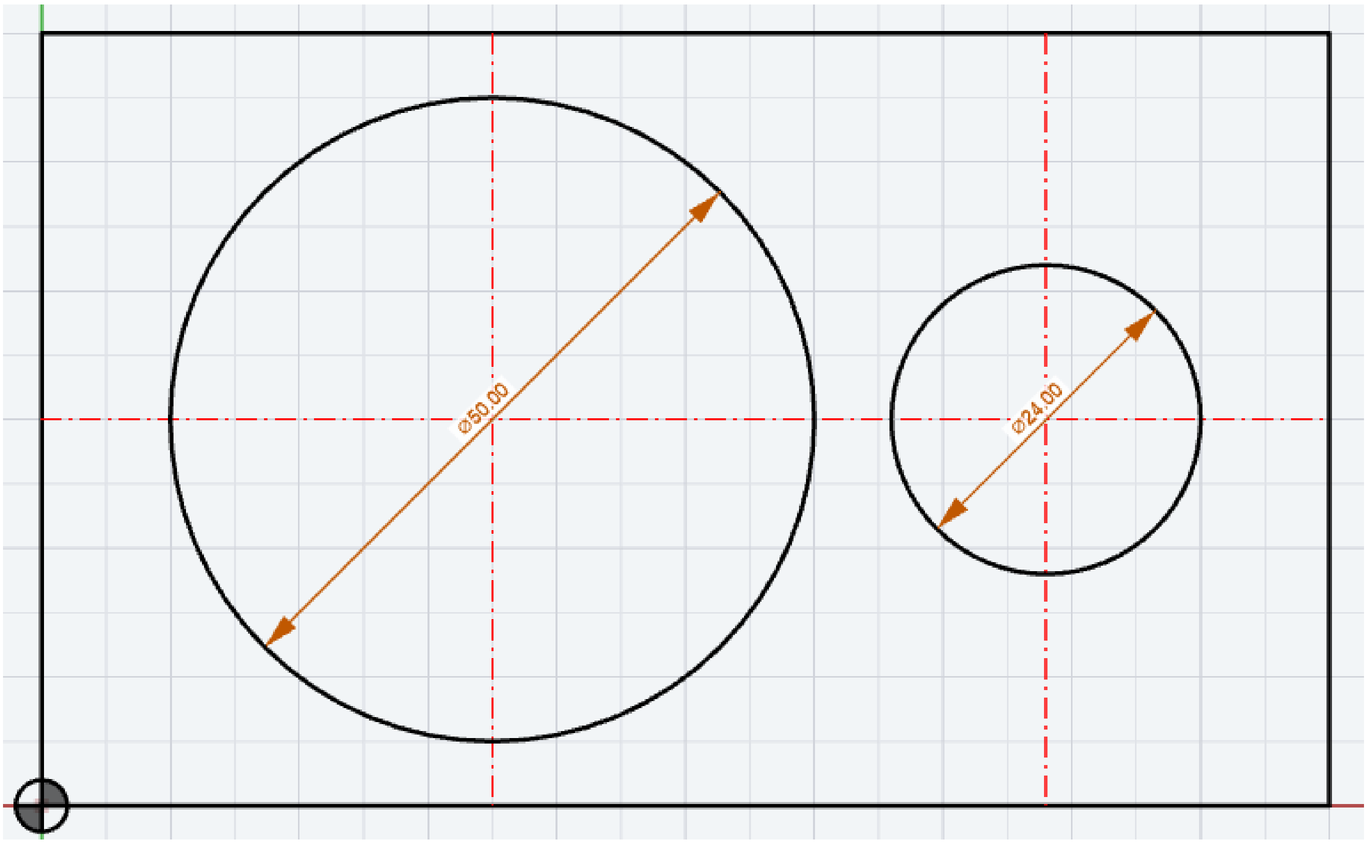
Menu: **Offset** **snap mode** [F], each with the distance **35** and **43**, and horizontally with **30**.

Activate layer: **SICHTBAR**.

Menu: **Circle** **snap mode** [A]and draw with radius **25** and **12**.

Now the circle can be positioned on the intersection of the axes.

But careful: *if the grid snap is still active, you have to make sure that the* ***intersection*** *and not a grid point is really snapped! It is therefore better to draw in* ***automatic*** *mode [****A****].*

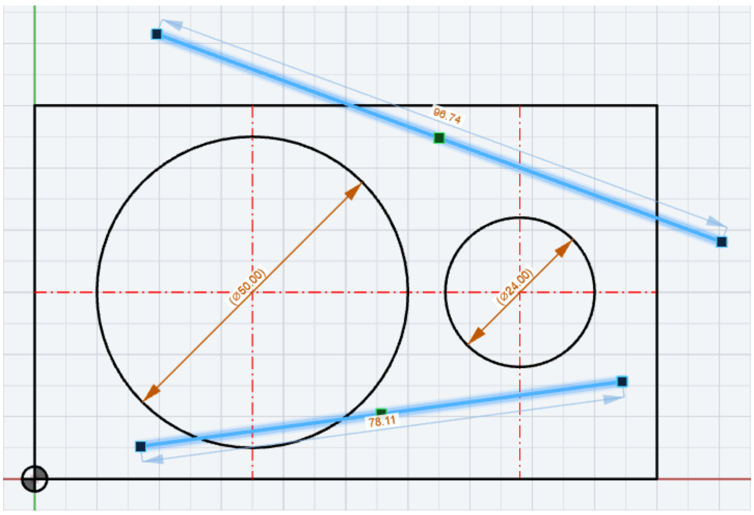


Next come the two tangential lines and the function:

Menu: **Line → snap mode [F]**

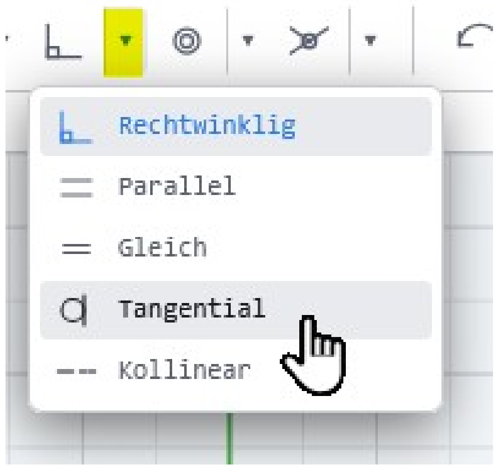
Draw the two lines quite boldly **anywhere** you like...

The lines are now made to hug the circles tangentially by means of a so-called dependency*.

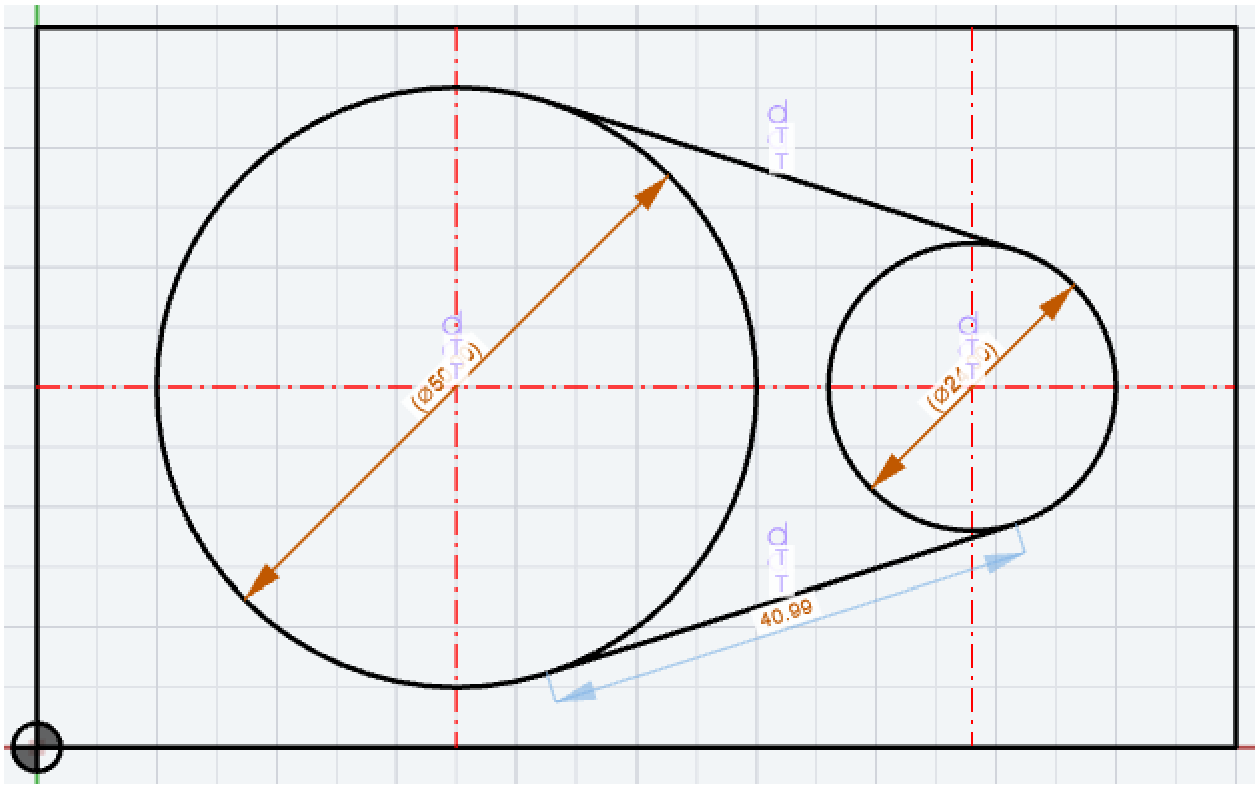


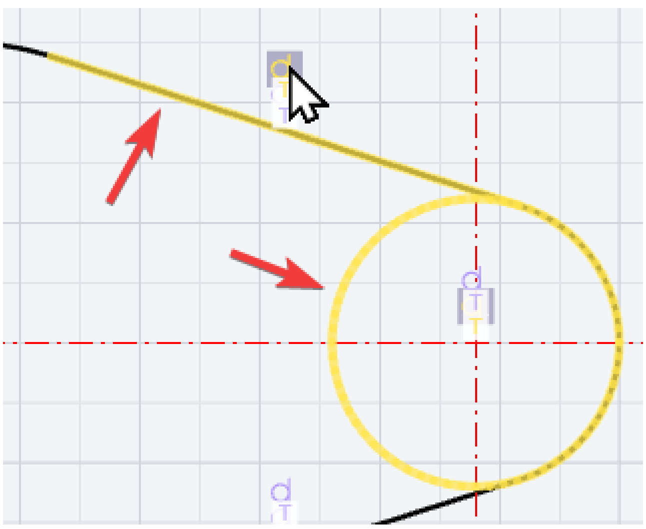
* In other CAD programs this may also be called a geometric condition, a dependency or a constraint. It is all the same thing :-)

Menu: **Tangent snap mode [F] or Automatic [A]**

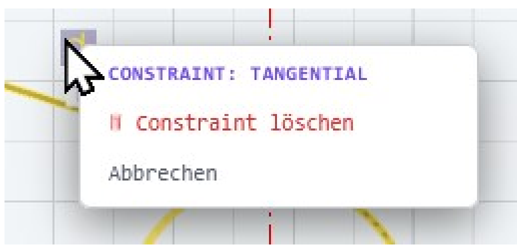


Now tangential dependencies can be assigned in such a way that you always click the circle or arc. Read in the tooltip and in the status bar what the function does or wants.

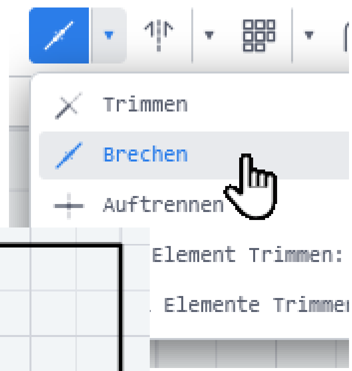




The constraints are shown as small pictograms. In select mode they can be "hovered over" with the mouse and flash accordingly.

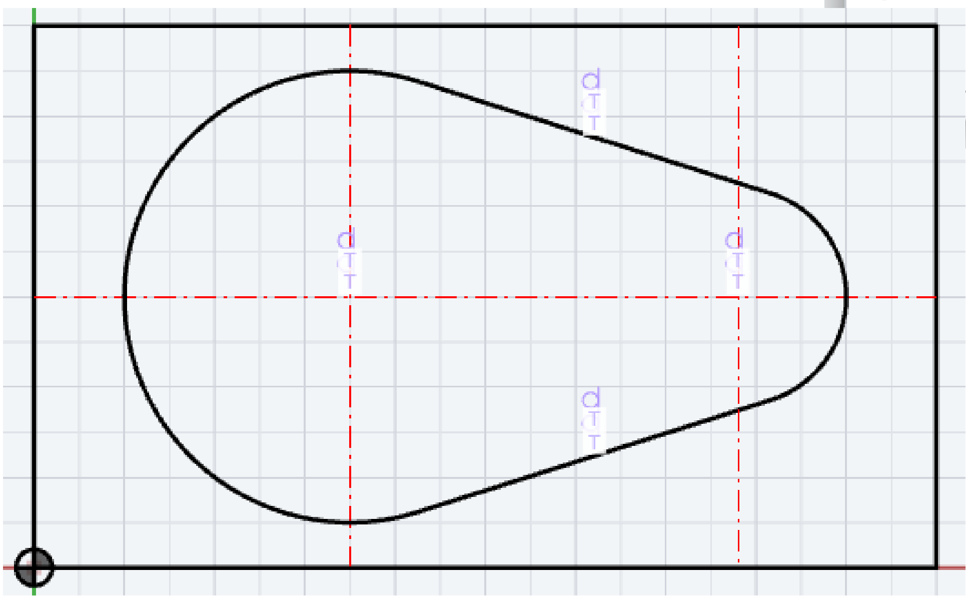


A right-click opens the menu.



Menu: **Break snap mode [A]** or **[F]** and break away all inner arc segments, which requires at least **6** clicks...

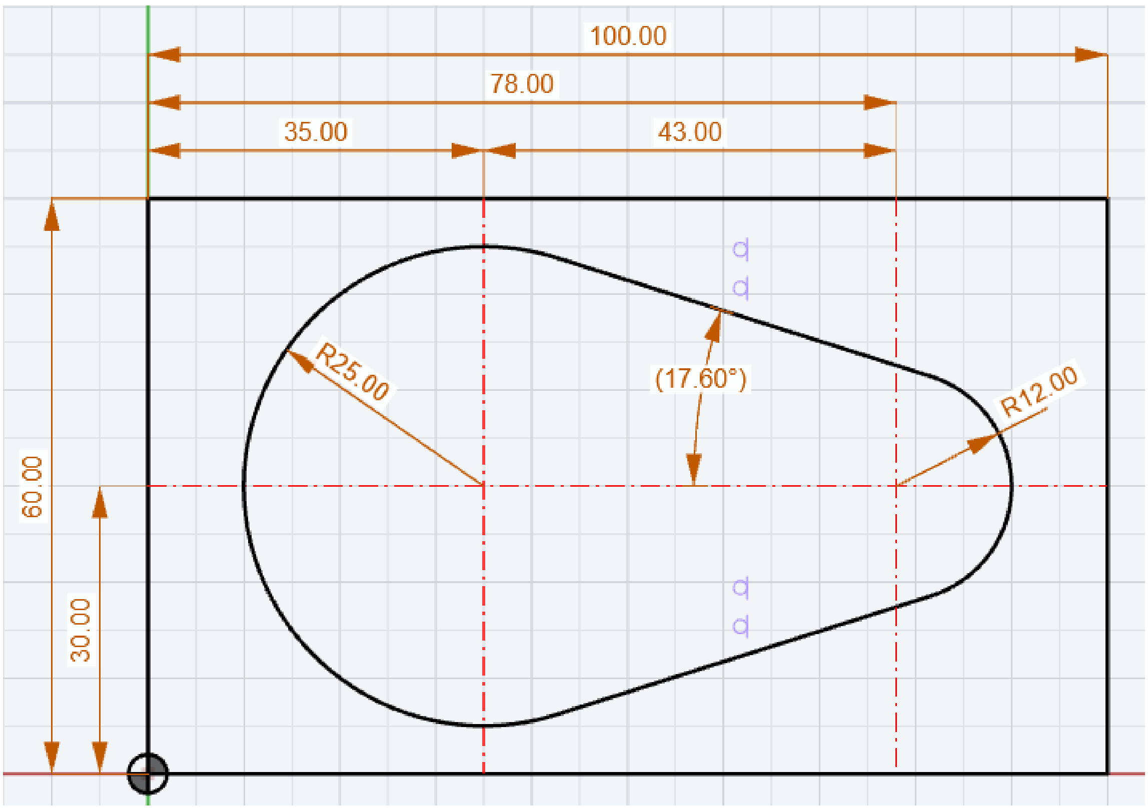
When you have got that far, the drawing looks like this:



Apply all dimensions with the dimension functions.

Snap mode [A]

Menu: **Dimension →** place the dimension with **snap mode [G]** on the paper .

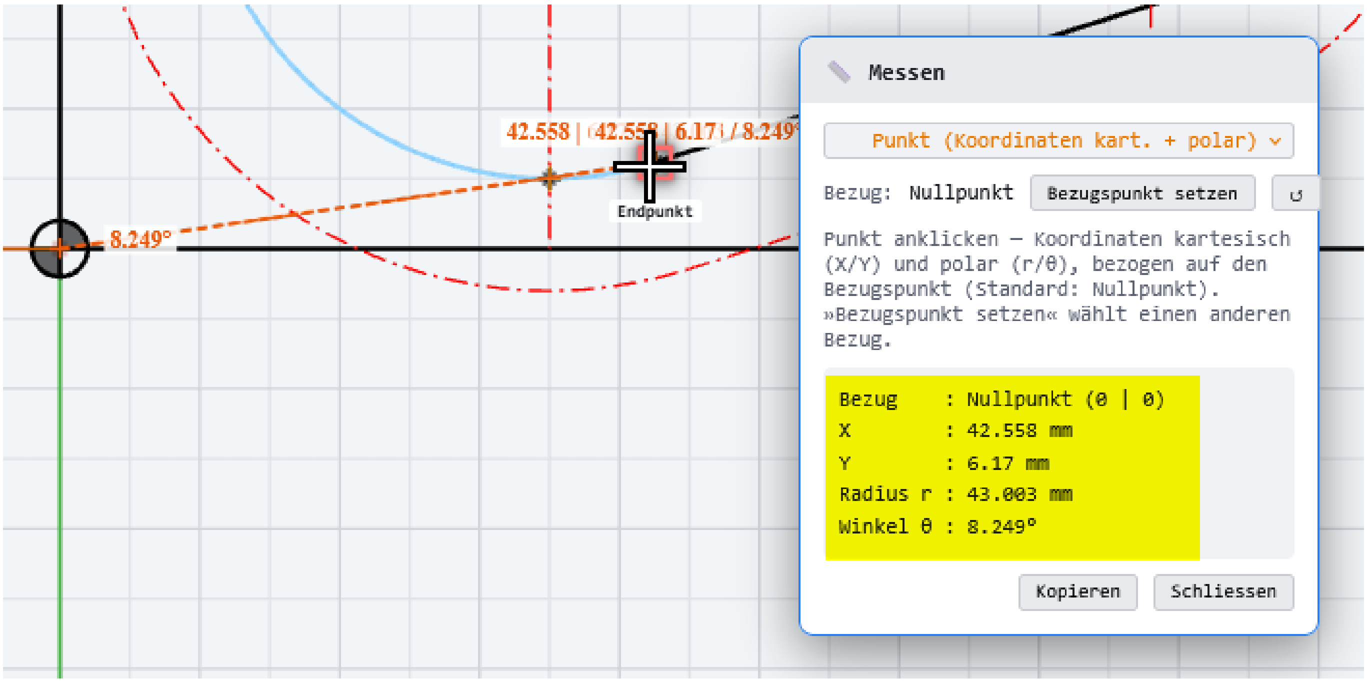


44.1 Workpiece coordinates

Have a look at the beginning of this chapter. Only points 2, 3, 5 and 7 are of interest to us. To query them we use the function:

Menu: **Measure → Point (coordinates cart. + polar)**

Activate snap mode [A] for **Automatic** and click the points:



With this function we can easily query all the further **workpiece coordinates**.

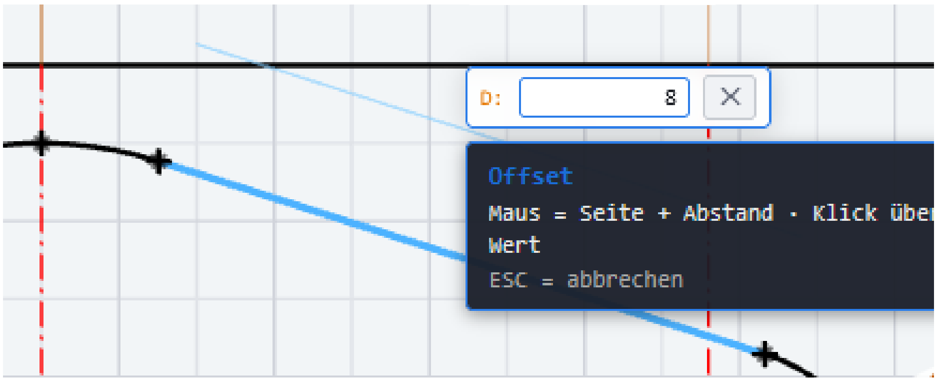
44.2 Milling coordinates

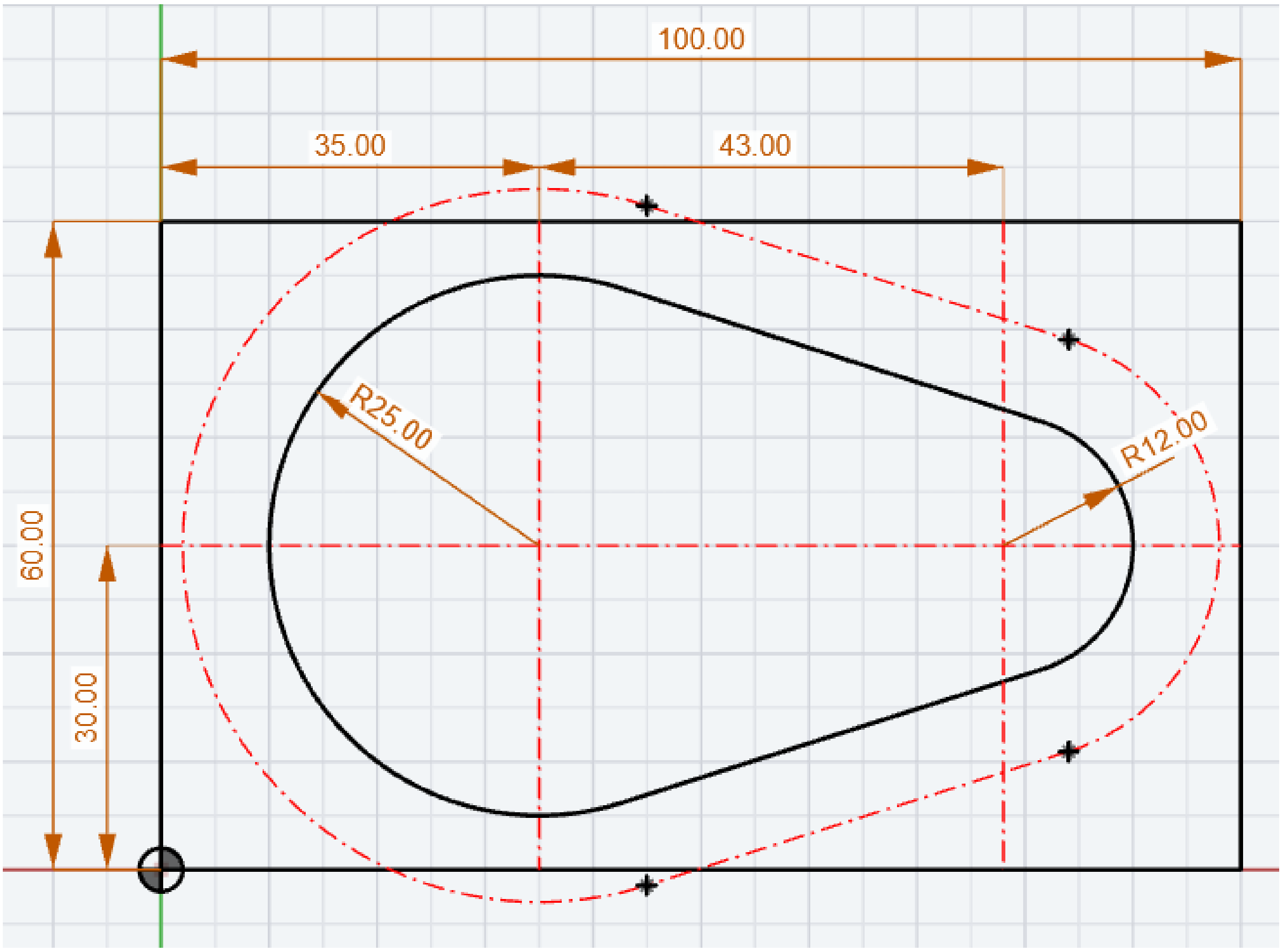
Things get much more interesting now with the actual milling coordinates…
Below we construct the milling path for a **16 mm** cutter.

Layer: **ACHSEN**

Menu: **Offset → snap mode [F] for Free** 

Distance: **8**





Now we are above all interested in the points that lie on the milling path.

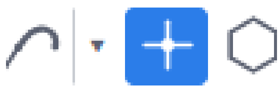
We could query the points on the milling path individually again...

But what is far better is the Coordinate table function, which numbers the **points** including their X and Y coordinates and shows them in a neat table.

So, place points on the milling path:

Layer: **SICHTBAR**

Menu: **Point**



Snap mode: [**A**]

Now set four points at the tangential transitions between lines and arcs.

Do not be sparing with the zoom. The points have to sit precisely.

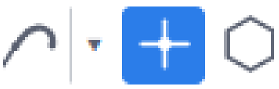
45 Creating a coordinate table from points

With the "Coordinate table" function you automatically collect all the points placed in the drawing into a clear table. For every point you get a consecutive item number as well as its X and Y coordinate. At the same time the corresponding item number is displayed at every point directly in the drawing, so that the table and the drawing match unambiguously.

45.1 What you use this function for

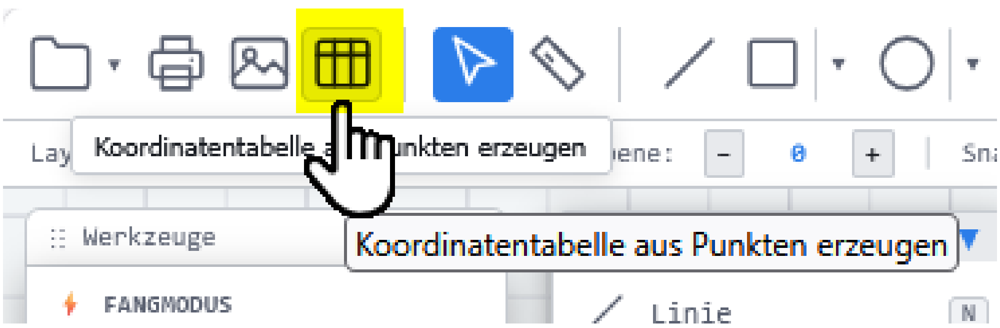
The coordinate table is suitable wherever position points have to be documented – for hole patterns, fixing points, surveying or setting-out points, for example. Instead of reading off and entering every coordinate by hand, LiberoDraft produces the complete table at the press of a button.

45.2 Prerequisite: placing points



The basis of the table is point elements. So first place the points you want in your drawing with the point tool. All visible points are taken into account; the order of the item numbers corresponds to the order in which the points were created.

45.3 Calling up the function



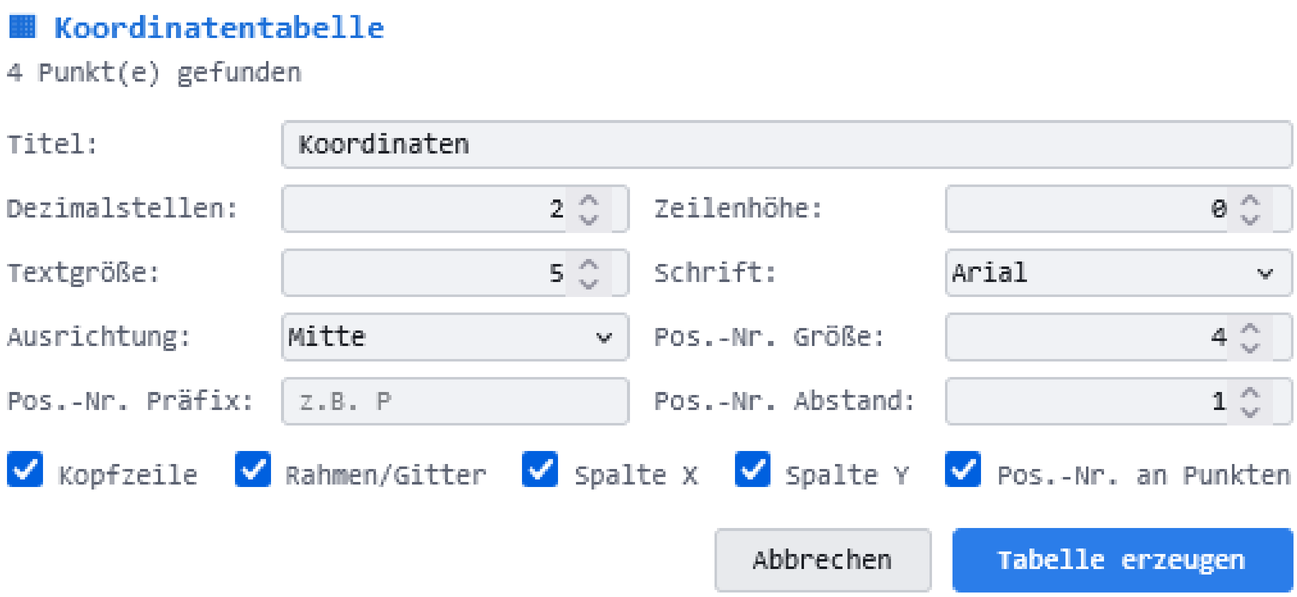
In the toolbar, click the button "Create coordinate table from points" (the symbol with the table grid, directly next to "Insert image"). A dialog opens in which you first define all the table, text and alignment parameters. At the top of the dialog you also see how many points were found.

45.4 The dialog at a glance

Set the following parameters as required and then confirm with "Create table":

Parameter	Meaning	Default
Title	Heading above the table. If you leave the field empty, no title row is created.	Coordinates
Decimal places	Number of decimal places for the X and Y values.	2
Row height	Height of each table row in drawing units. The value 0 calculates the height automatically from the text size.	0 (auto)
Text size	Text height inside the table (in drawing units).	5
Font	Typeface for the table texts.	Arial

Parameter	Meaning	Default
Alignment	Alignment of the text in the columns: left, center or right.	Center
Item no. size	Text height of the item numbers displayed at the points.	4
Item no. prefix	Prefix before every number; "P", for example, gives P1, P2, P3 …	(empty)
Item no. offset	Distance of the item number from its point. The value 0 places the number directly on the point.	1
Header	Shows or hides the column headings (Pos, X, Y).	on
Frame/grid	Draws the frame and the separating lines of the table.	on
Column X / column Y	Defines which coordinate columns appear in the table.	both on
Item no. at points	Shows or hides the item numbers at the points.	on



45.5 Placing the table

After confirming, the finished table "hangs" at the mouse – its upper left corner follows the mouse pointer. The object snap is active, so that you can align the table exactly with existing geometry (at an endpoint, for example) if required.

- Left mouse button: place the table at the position you want.
- ESC or the right mouse button: cancel the operation without creating a table.

In the preview you already see the frame or grid of the table as well as the item numbers at the points.

45.6 The result

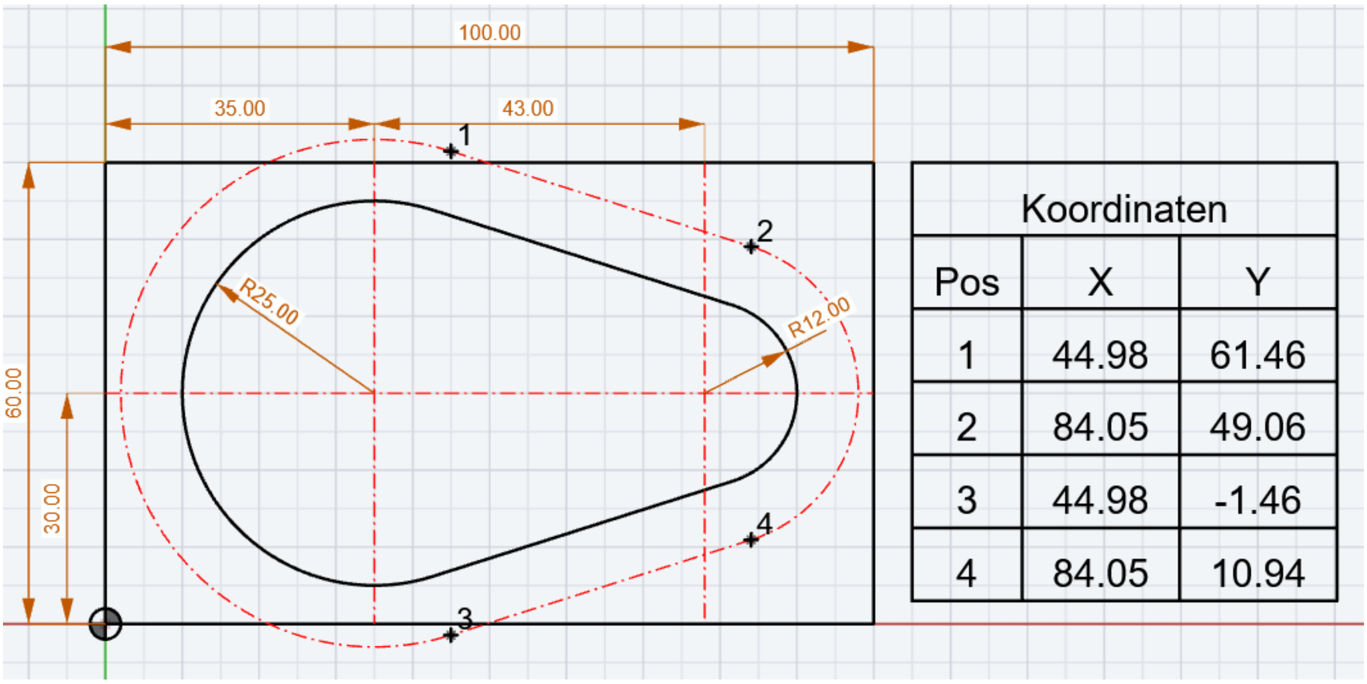
LiberoDraft creates the table as a group of its own with the name "Koordinatentabelle N" (N is a consecutive number). The table consists of ordinary line and text elements and can therefore be edited, moved, scaled and printed like any other geometry. If you want to place the table on a particular layer, make that layer active before creating it.

45.7 Adjusting the item numbers

The item numbers at the points are independent texts and can be moved freely at any time. With a single mouse click you select an individual number (not the whole group) and can drag it to a clearly legible position. The initial distance from the respective marker is already defined in the dialog via "Item no. offset".

45.8 Good to know

- The table is a snapshot: if you move a point later on, the table and the item number are not updated automatically. In that case simply create the table anew.
- If there are no points, the dialog tells you so – place points first in that case.
- With "Column X" and "Column Y" you can restrict the table to the coordinates you need; the "Pos" column is always included.
- With "Item no. at points" the annotation in the drawing can be switched off, in case you only need the table.

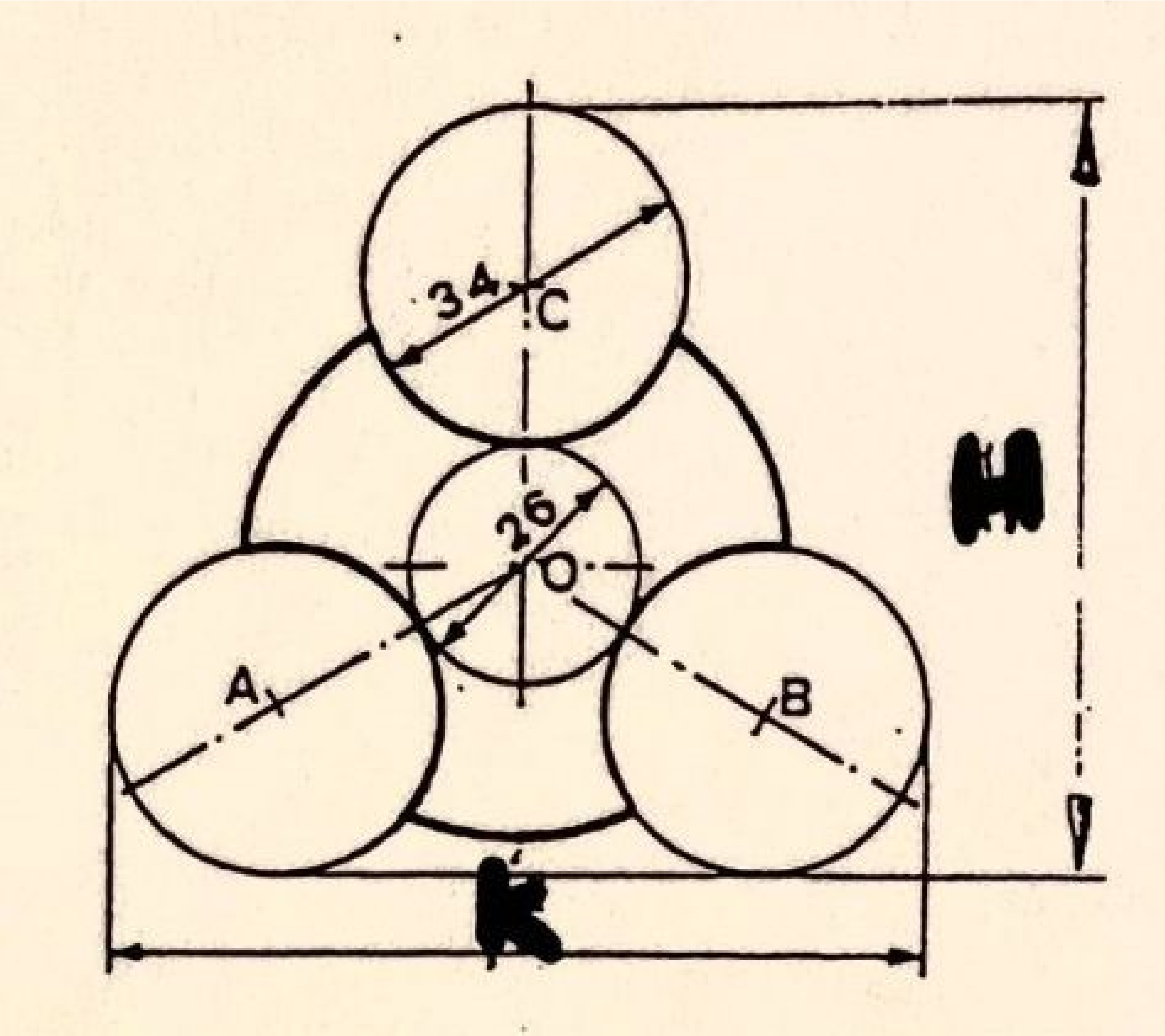


Pos	X	Y
1	44.98	61.46
2	84.05	49.06
3	44.98	-1.46
4	84.05	10.94

The item numbers are set automatically, the table hangs at the mouse and can be placed anywhere you like. Splendid! :-)

46 Drawing: planetary gear

From this manually adapted sketch, the check dimensions **H** and **K** as well as the coordinates **A**, **B** and **C** are to be calculated:



The ***origin*** is at the center.

The planets with D34 are distributed evenly on the pitch circle diameter.

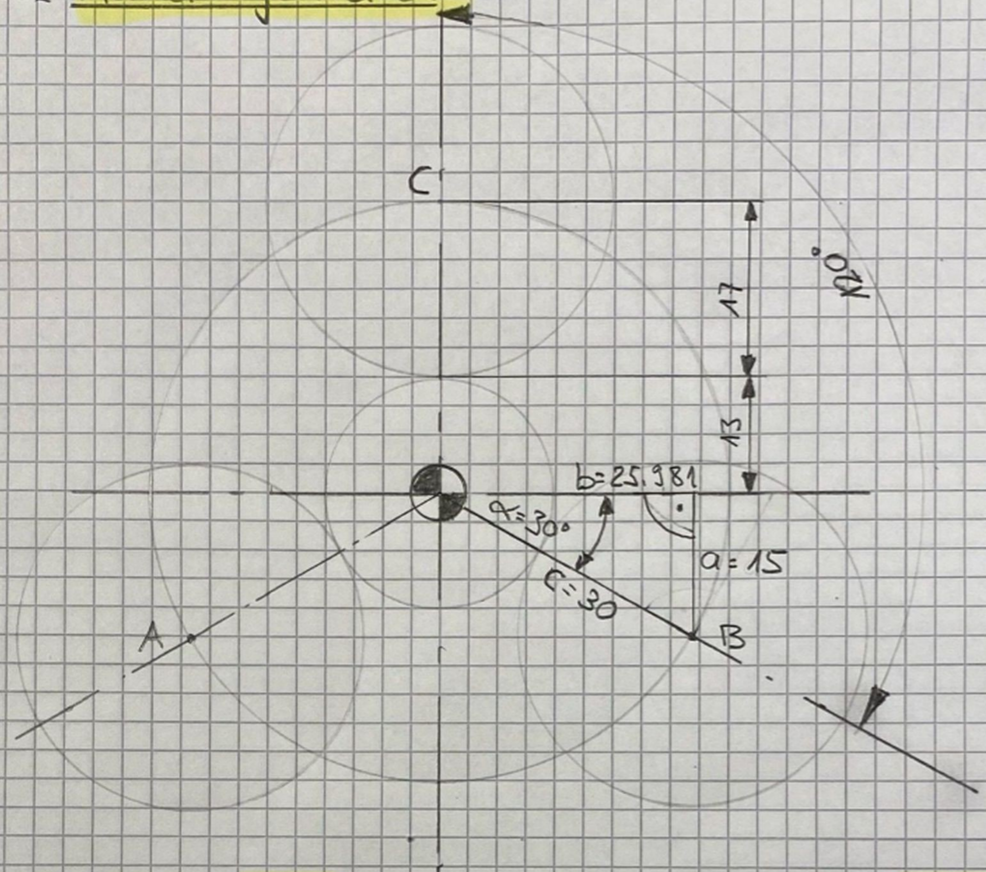
The inner circle D26 lies tangential to the planets.

Several dimensions are missing and have to be derived accordingly.

Here you see a figure showing how it can be done by hand.

Überlegung zur Skizze (16.)

Hierbei handelt es sich um eine schematische Darstellung eines Planetengetriebes.



Dabei sind mit drei Planeten die Winkel bei 360/3 = 120°

$b = c \cdot \cos(\alpha) \rightarrow 30 \cdot \cos(30) = 25{,}981$

$a = c \cdot \sin(\alpha) \rightarrow 30 \cdot \sin(30) = 15$

$H = 17 + 15 + 13 + 34 = 79$ mm

$K = (2 \times 17) + (2 \times 25{,}981) = 85{,}962$ mm

A =	X -25,981	Y -15
B =	X 25,981	Y -15
C =	X 0	Y 30

Above, the calculations done by hand.

Check with the construction in LiberoDraft whether the results are correct.Determine and draw the pitch circle.

Activate layer: **ACHSEN**.

Menu: **Circle** radius: 13 + 17 = **30**

Tip:
Place the circle at the *origin* **with** the grid snap **[G]**

46.1 Drawing the vertical axis

Activate layer: **ACHSEN**.

Menu: **Line**

Draw the axes on the grid through the origin:
one vertical and one horizontal.

46.2 Drawing the first planet

Activate layer: **SICHTBAR**.

Snap mode: [**A**]

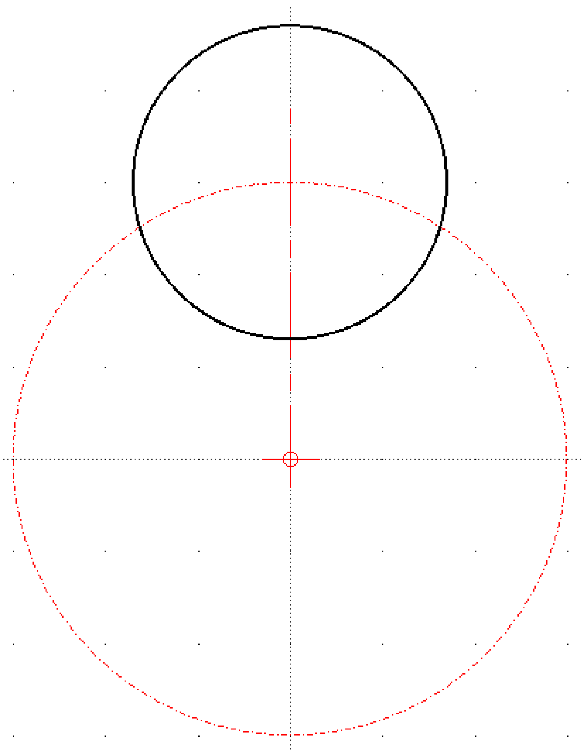
Menu: **Circle** radius = **17**

Place the circle at the **intersection** of the vertical axis and the pitch circle.

The center of the 17 mm planet has to lie absolutely exactly on the intersection of the pitch circle and the vertical axis.

To make sure that works 100 %, it really is advisable to activate the automatic snap with key [**A**].

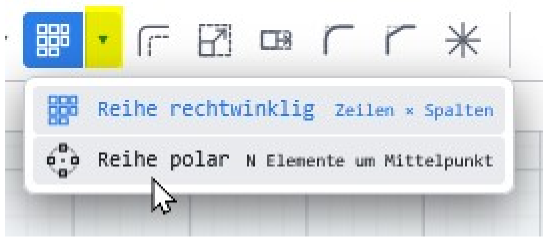
The drawing should now look like this →



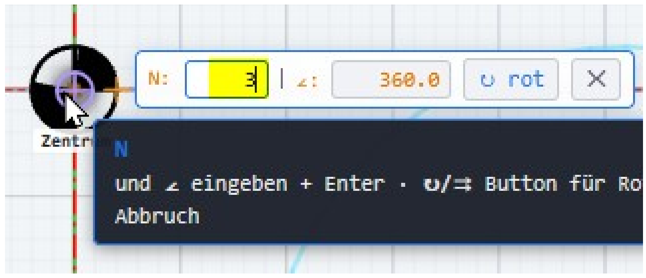
46.3 Patterning the planet 2x in a circle

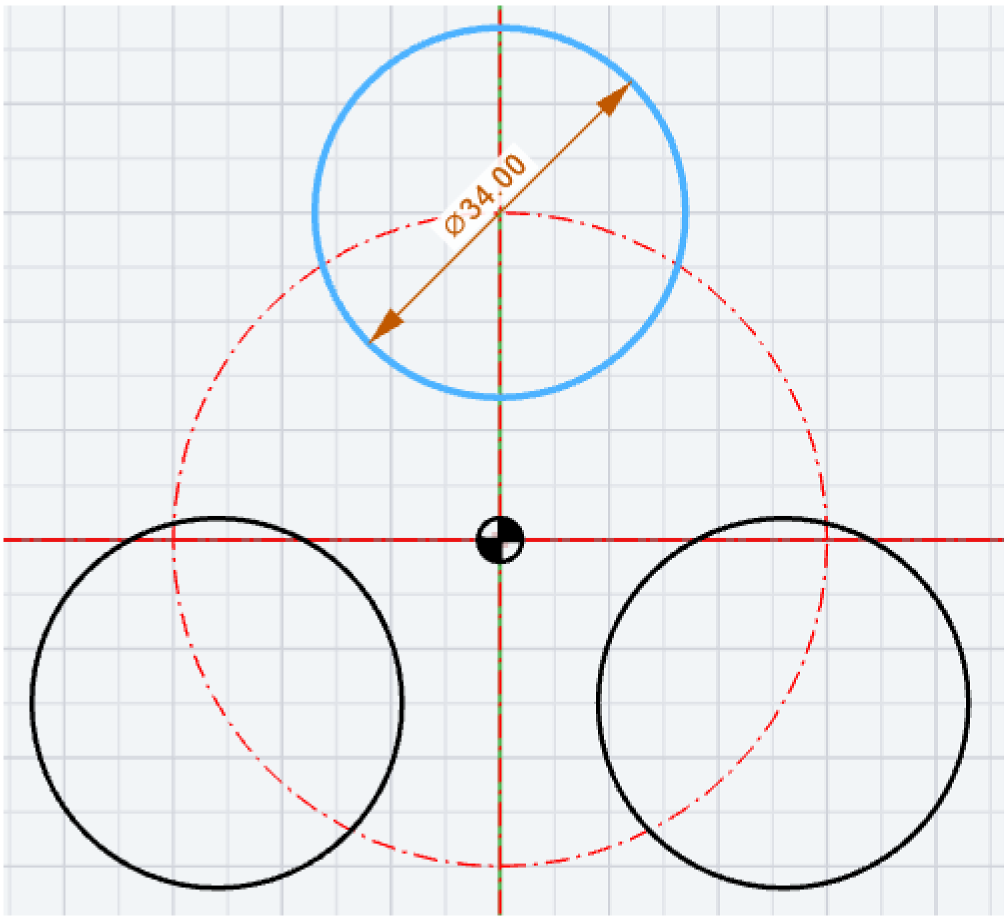
The first planet or circle can now be copied twice at 120°.

1. Select the planet. (Click it with the mouse; the circle **must** glow blue.)
2. Call up the function: **Array polar**



3. Select the **circle center** of the **planet** with the automatic snap [**A**].
4. Specify the **center** of rotation: click the **origin** at the center.
5. In the dialog that follows, enter the number **3** and press **Enter**.





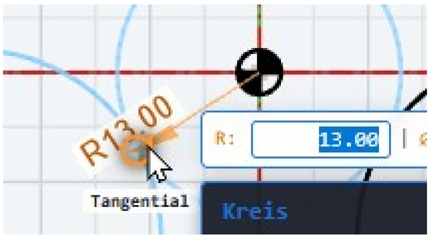
46.4 Circle tangent to three planets

Now only the inner circle with D26 is missing. We could draw it quite normally as a **circle** via the **center** and the **radius**, or **tangent** to the three planets.

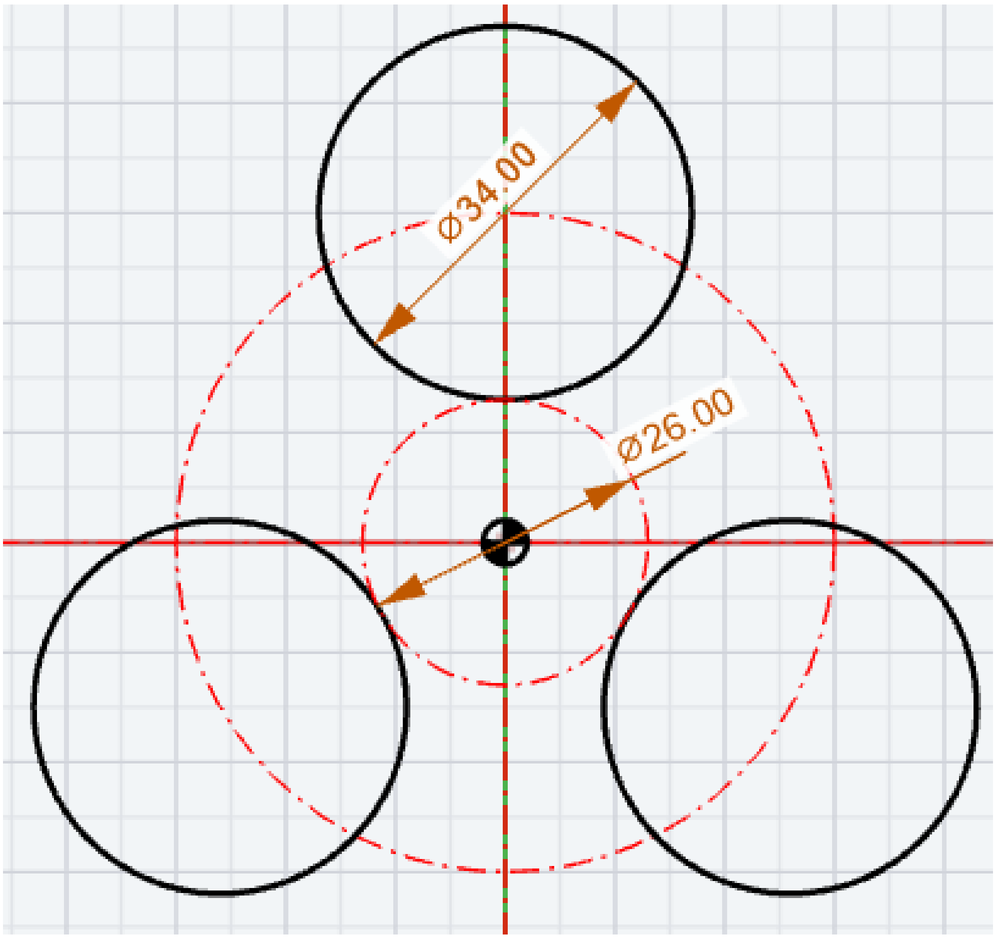
We take the normal circle from the center with the automatic snap [A].

Activate layer: **ACHSEN**.

Function: **Circle →** **click the origin** and move onto a circle until "**Tangent**" is shown at the mouse cursor:



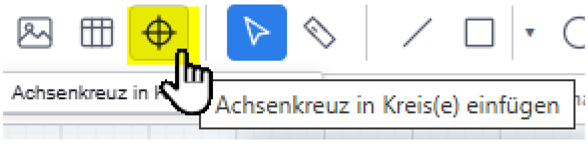
Confirm with Enter or a mouse click.



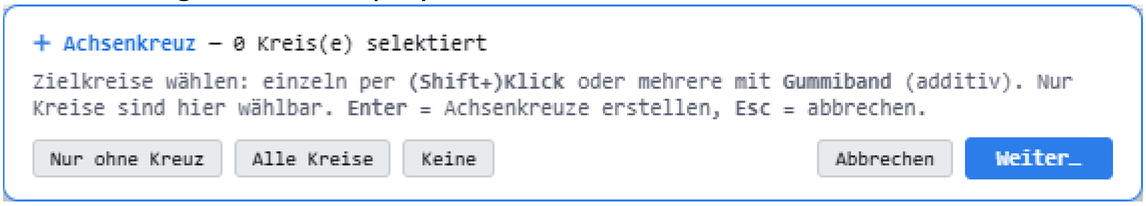
Now the two check dimensions and coordinate points can be dimensioned.

For the check dimensions H and K we may have to optimize two or three things:

1. Adding **axis crosses** to the planets (circles)...

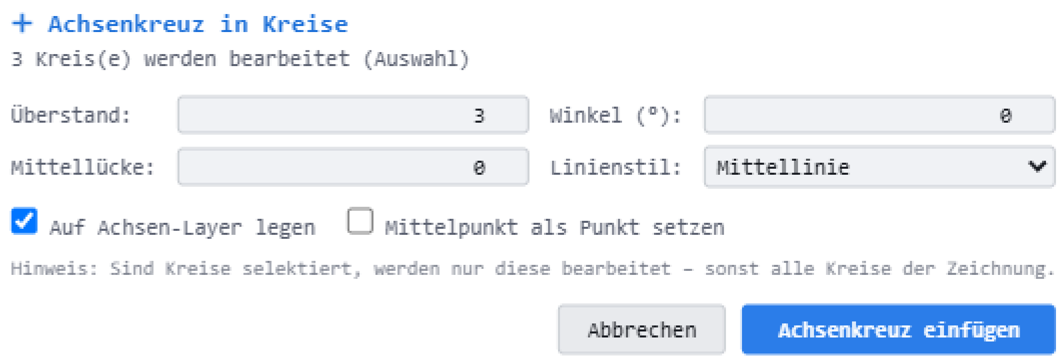


2. The following menu is displayed:

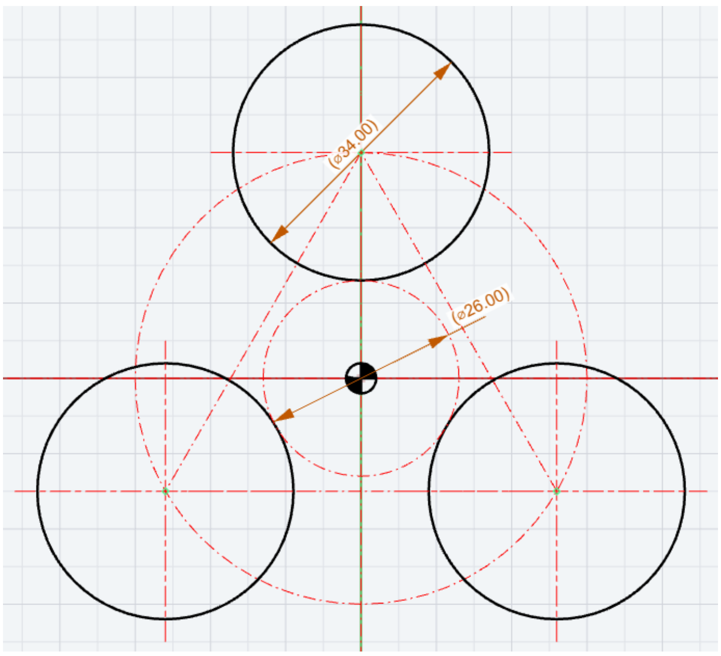


Everything relevant is stated in the dialog...

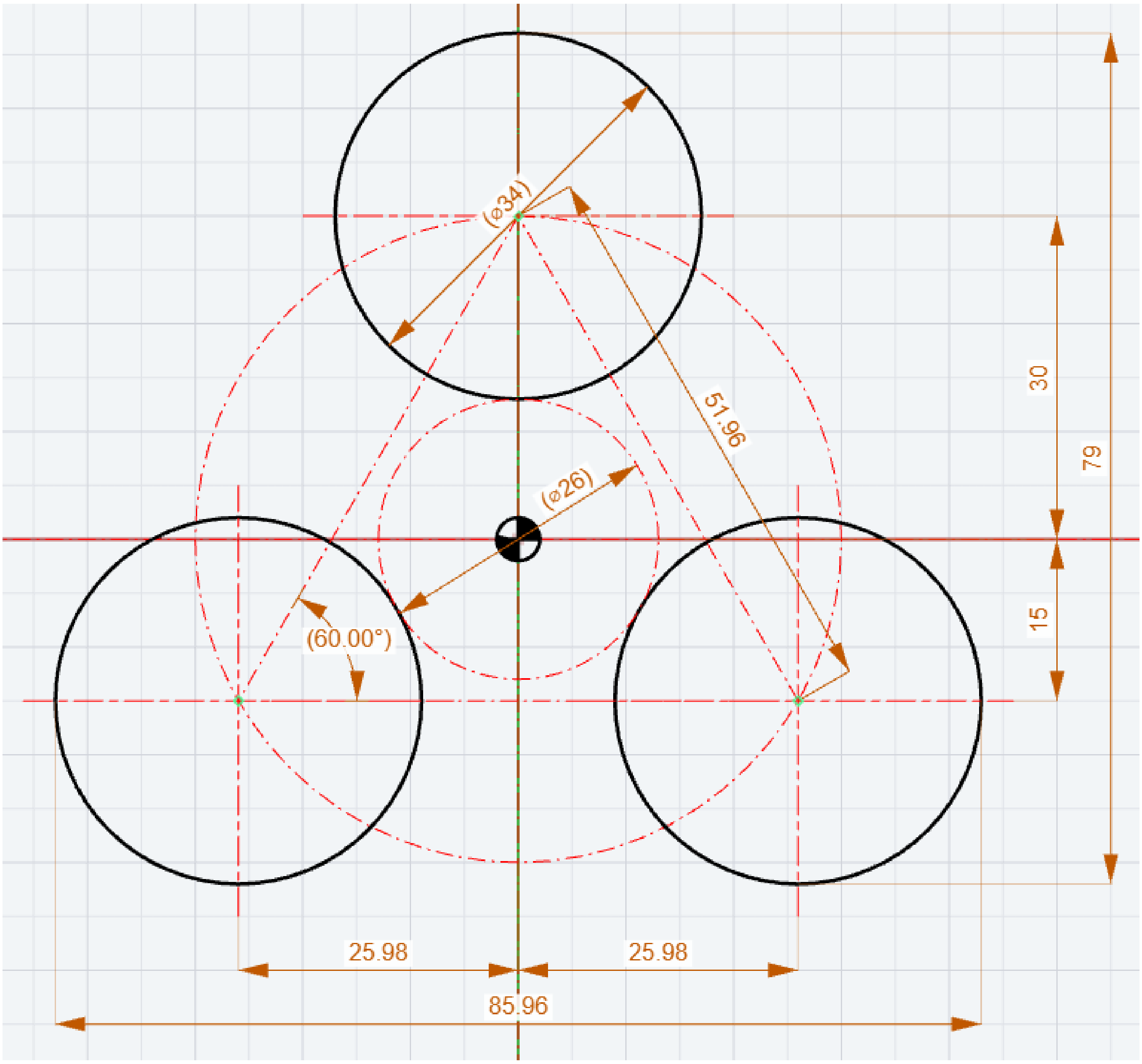
3. So, select the **three planets** and click **Next...**
4. The following dialog for the design of the axis crosses is displayed. As a rule the settings are fine as they are and you can go straight ahead with "**Insert axis cross**":



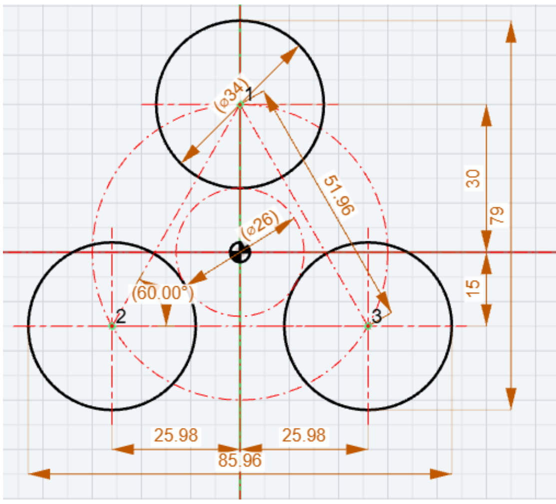
All the axis crosses are then drawn in cleanly:



Now just dimension it as follows and you are done:



Coordinate check: simply place a point on the centers of the three planets and run the **coordinate table** function.

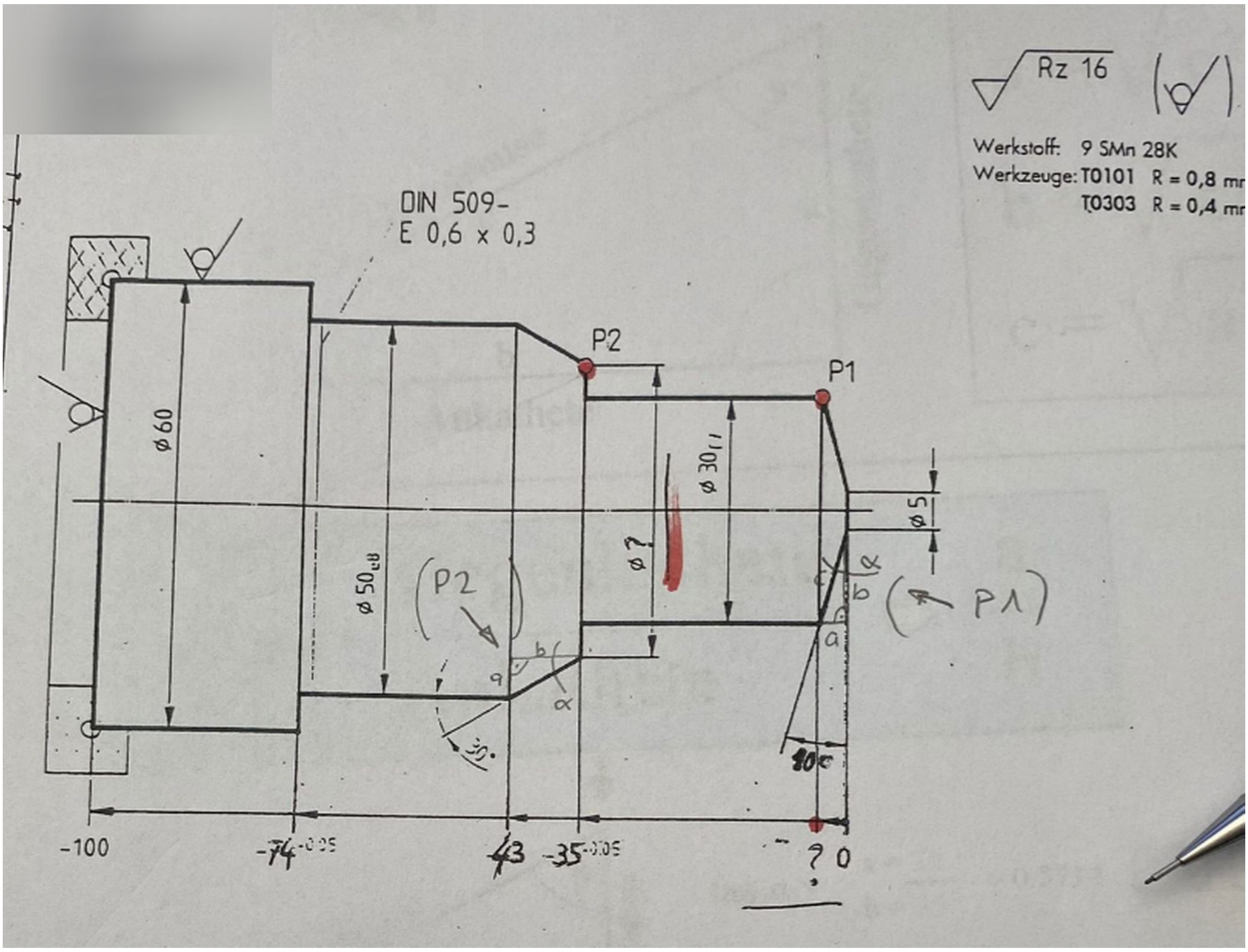


Koordinaten		
Pos	X	Y
1	0.00	30.00
2	-25.98	-15.00
3	25.98	-15.00

Done! :-)

47 Drawing: turned part

We have received this sketch, in which some dimensions are missing:



Open a new drawing.

Activate layer: **ACHSEN** → snap mode **Grid [G]**

First draw the axis to the left as far as about **X -110**

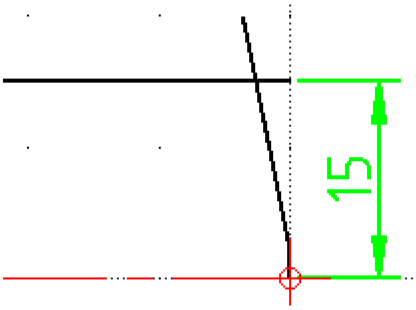
Activate layer: **SICHTBAR**.

With the function **Line**, draw a **vertical line on the grid** from the origin with a length of **2.5** mm (that is half the small diameter at the front).

A further line is attached to it at an angle of 10°:
function: **Line → polar** angle: **100** (from 90°+10°), length: about **15**

Function: **Offset**

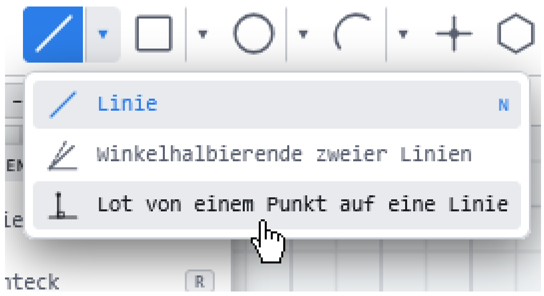
Create a parallel at a distance of **15** from the axis.



Now draw a new line **perpendicularly** from the **intersection** of the slanted and the horizontal 15 mm line down onto the **axis**.

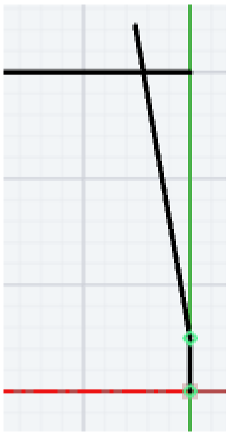
Function: **Perpendicular** **snap mode [A]** for Automatic….

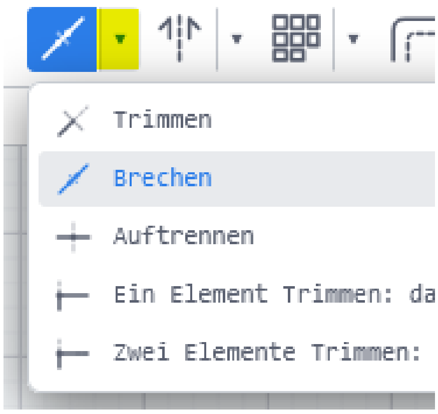
Snap the **intersection** of the slanted and the 15 mm line, move onto the axis and click.



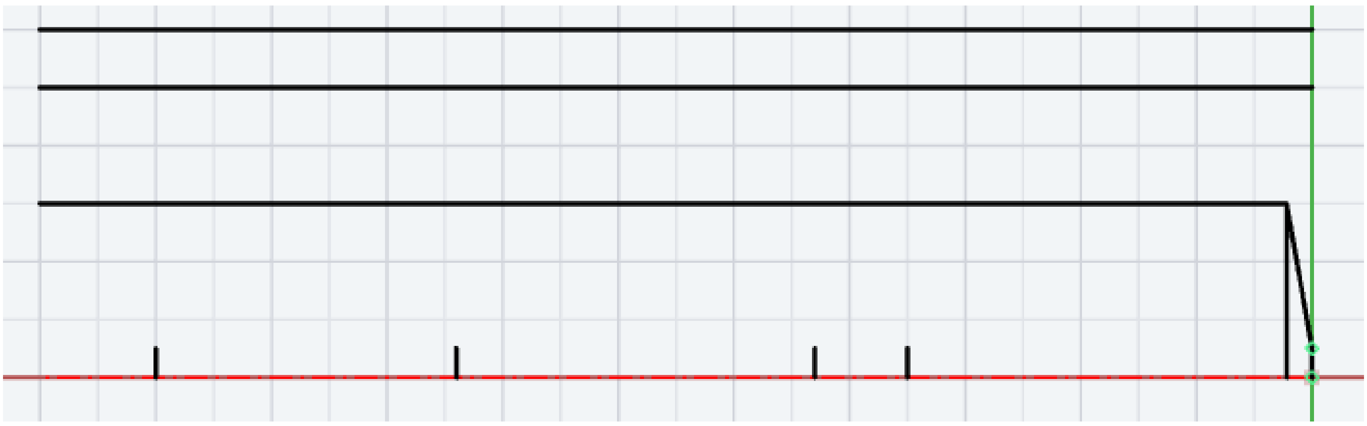
The drawing should now look like this:

The two protruding lines can be broken away or trimmed.





The drawing should now look like this:



47.1 Trim

With the **Trim** function, lines are shortened or extended.

Always read what LiberoDraft wants from you.

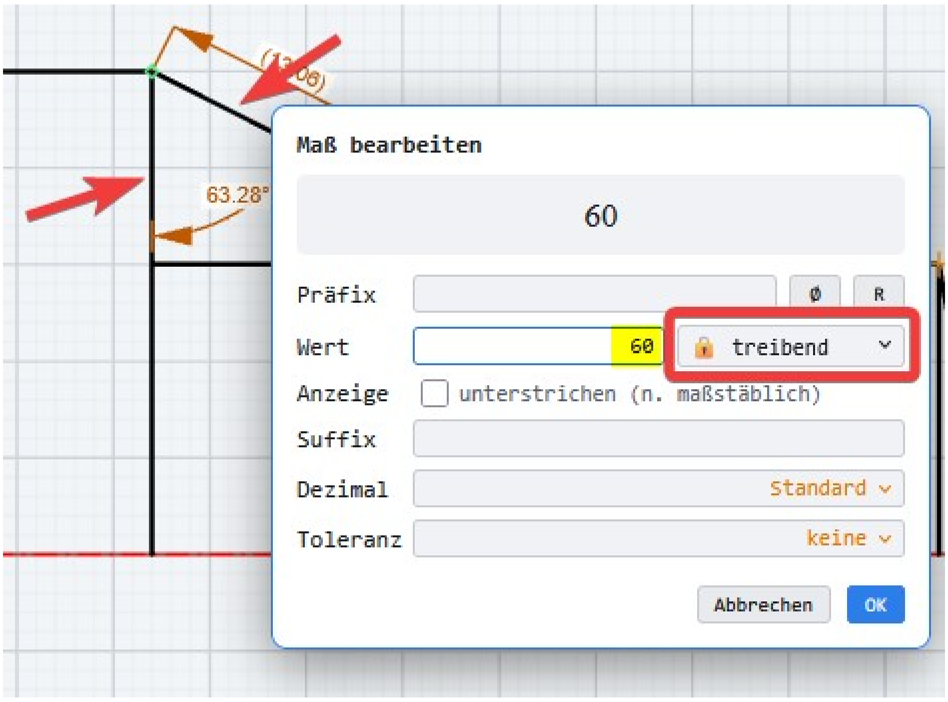
With the function **Trim one element** it will look like this:

Use the **Break** function to remove everything that does not belong there.

The easiest way to do the 30° cone is: **Line -> automatic snap [A]**, drawn roughly.

Then dimension the angle to 60° with **Dimension**.

The dimension has to be **driving** so that the line jumps to 60° after **OK** has been clicked.

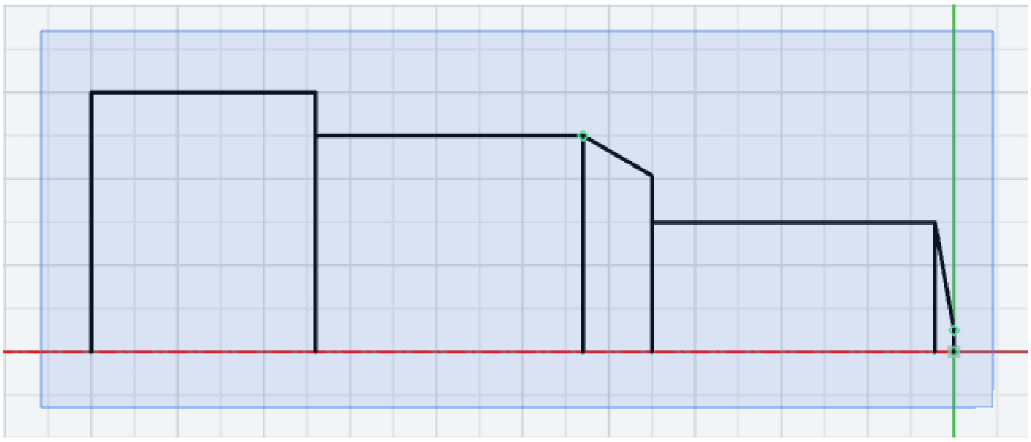


Now just **trim** the two lines to the intersection and break away a line segment with **Break**.

47.2 Mirror

Once everything is trimmed and fits, it can be mirrored.

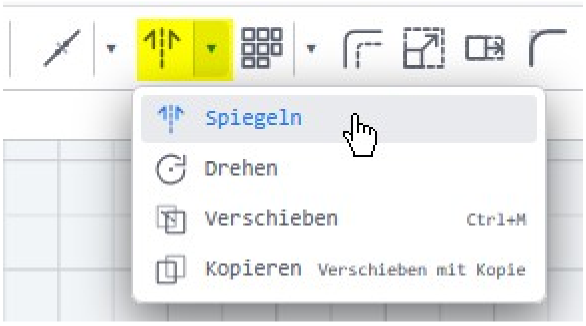
Select all the elements to be mirrored in select mode.



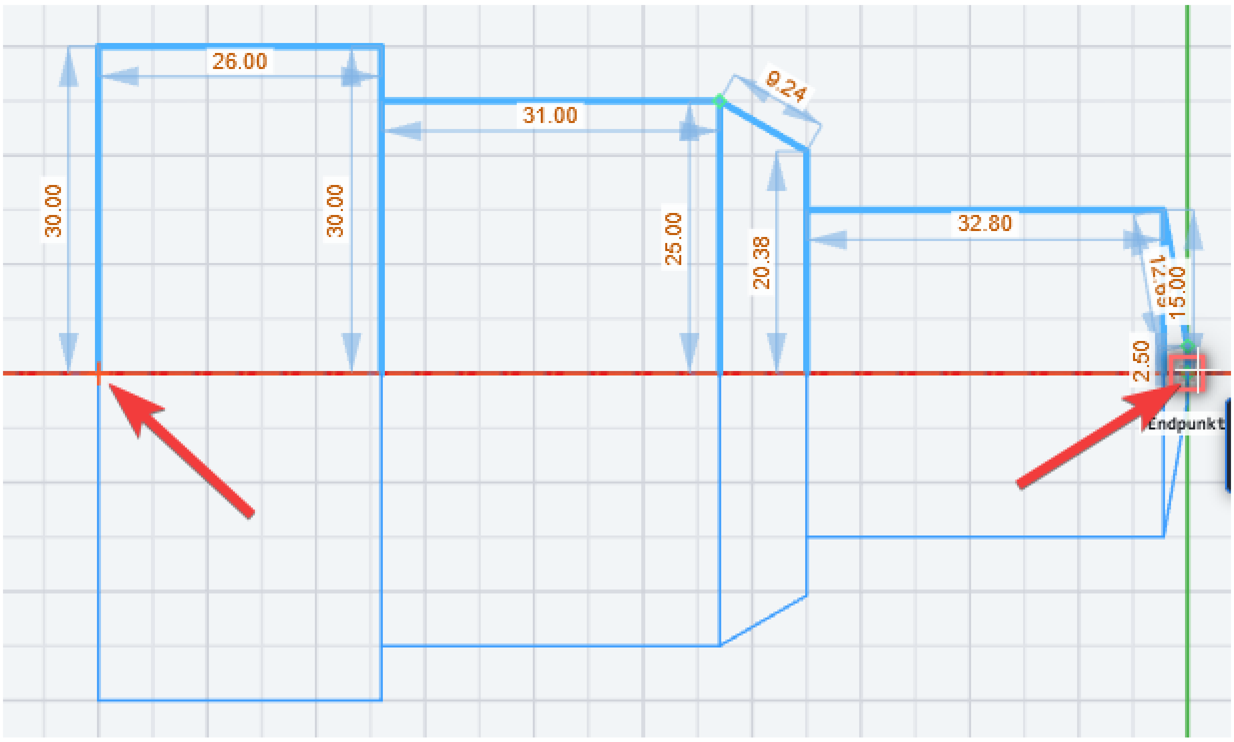
All **selected elements** are now shown **in a different color** and **with dimensions**.

Function: **Mirror**

Mirroring only works if the elements to be mirrored have been selected beforehand with the Select function.



Snap mode: **Automatic [A]** and the two points for the mirror axis selected:

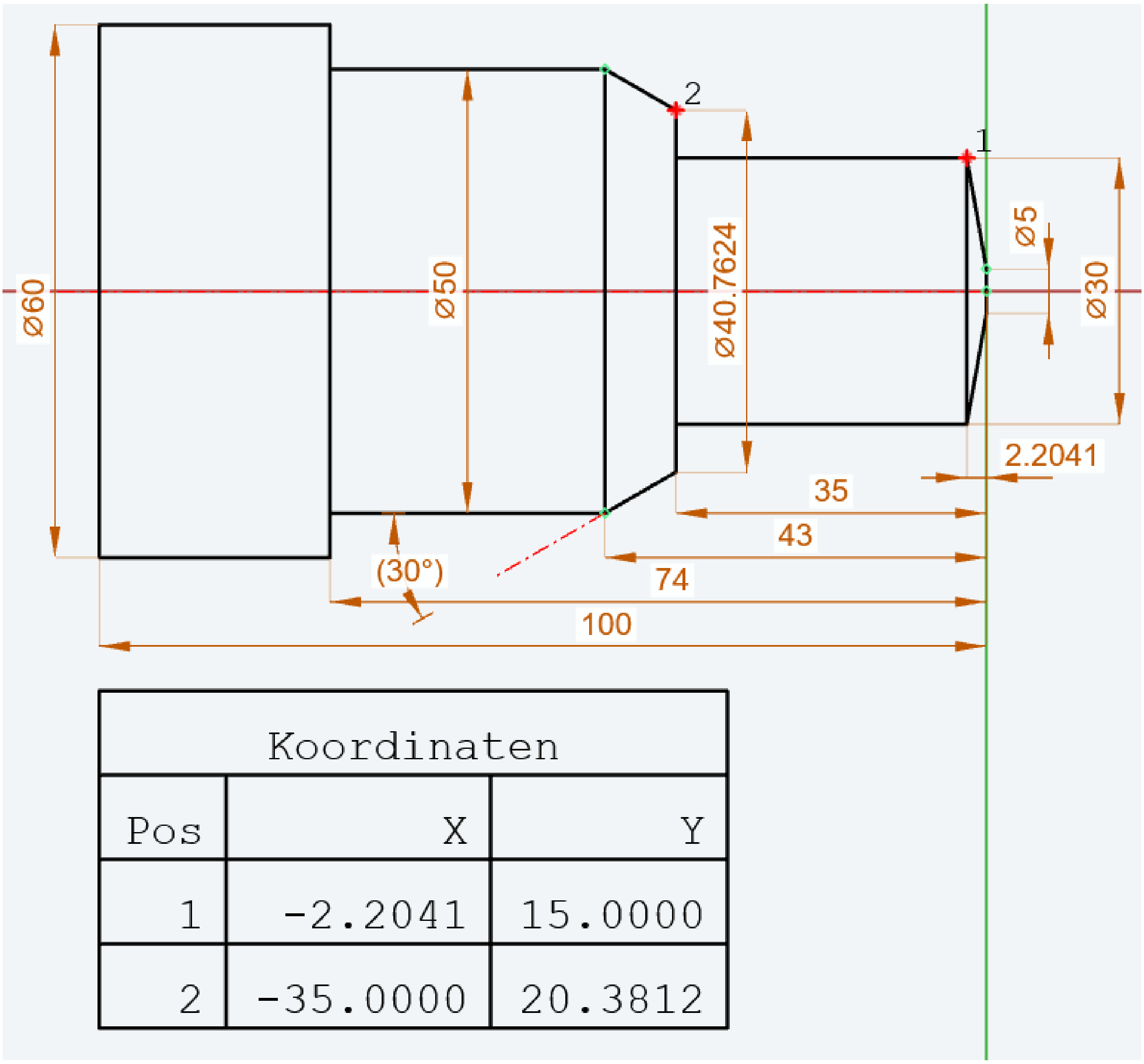


Now we have the drawing far enough along for it to be dimensioned.

Activate the function: **Dimension**.

For the diameter dimensions, activate the diameter button in the upper status bar:

Dimension the part as shown below:



To create the coordinate table, draw two **points** with the **automatic snap [A]** (in the drawing above you can see the two points in red.) Then call up the Coordinate table function:

Done :-)

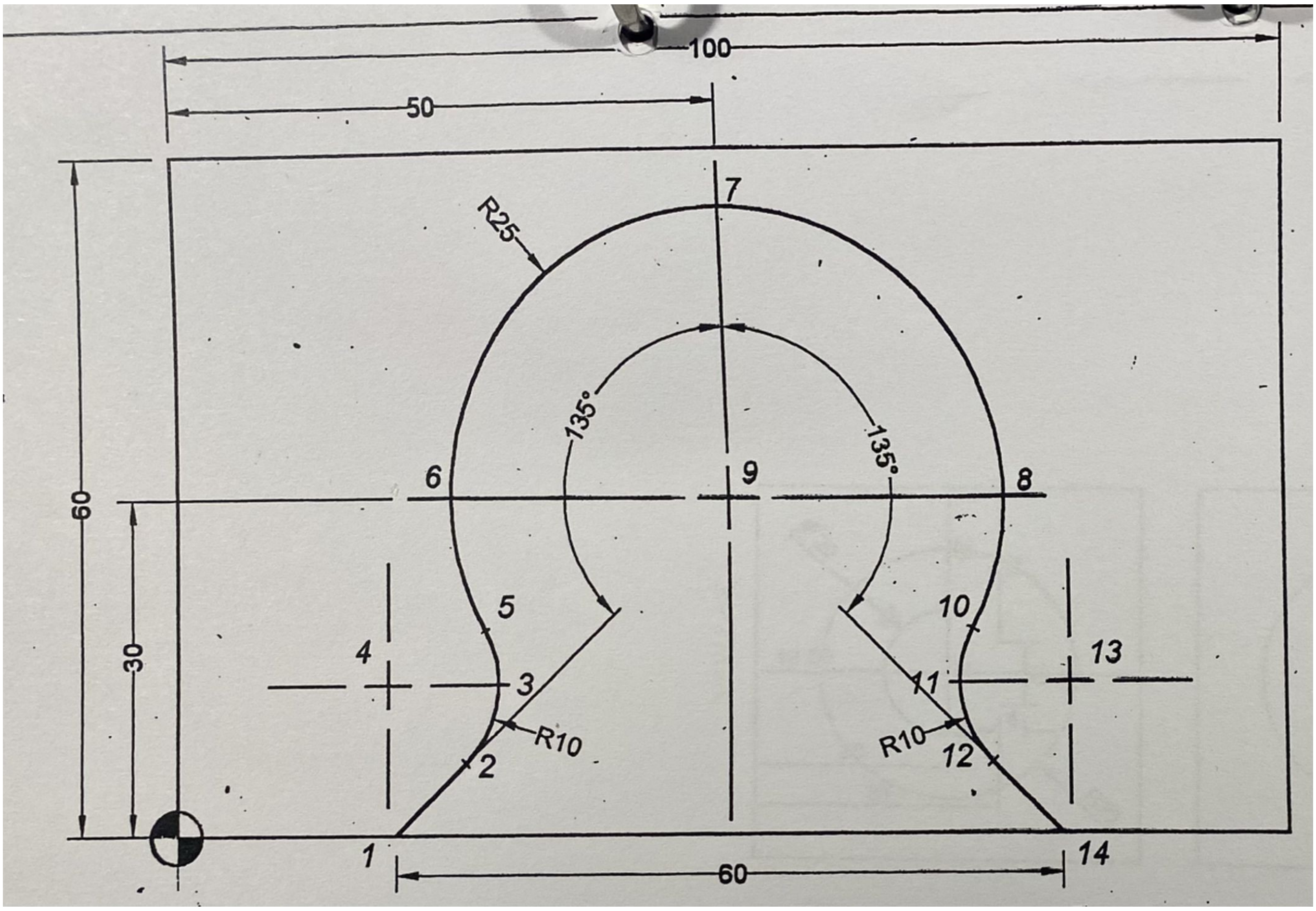
…

48 Drawing the omega shape

As a drawing we have received the following figure, with some dimensions obviously missing.

Draw the omega shape including all the missing dimensions.

In a second step the **milling and drilling coordinates** are determined.



48.1 Draw a rectangle 100 x 60.

Activate layer: **SICHTBAR**

Function: **Rectangle** drawn on the **grid** [**G**]
or via the **coordinate entry** [**K**] → **0,0 [Enter]**   **100,60 [Enter]**

Info: *switch from polar to cartesian coordinates with the* ***spacebar****.*

48.2 Draw the central axes.

Activate layer: **ACHSEN**.

Function: **Line** → draw two through the middle.

Tip:
activate snap option ***[A] for the automatic snap*** *.*

48.3 Creating the circle

Place a circle R25.

Layer: **SICHTBAR**

Function: **Circle → [A]** radius: **25**

Next, place **two parallel axes** from the center at 30 mm distance on both sides.

INFO:
Since the rectangle is a **block**, it first has to be **exploded**

Layer: **ACHSEN**

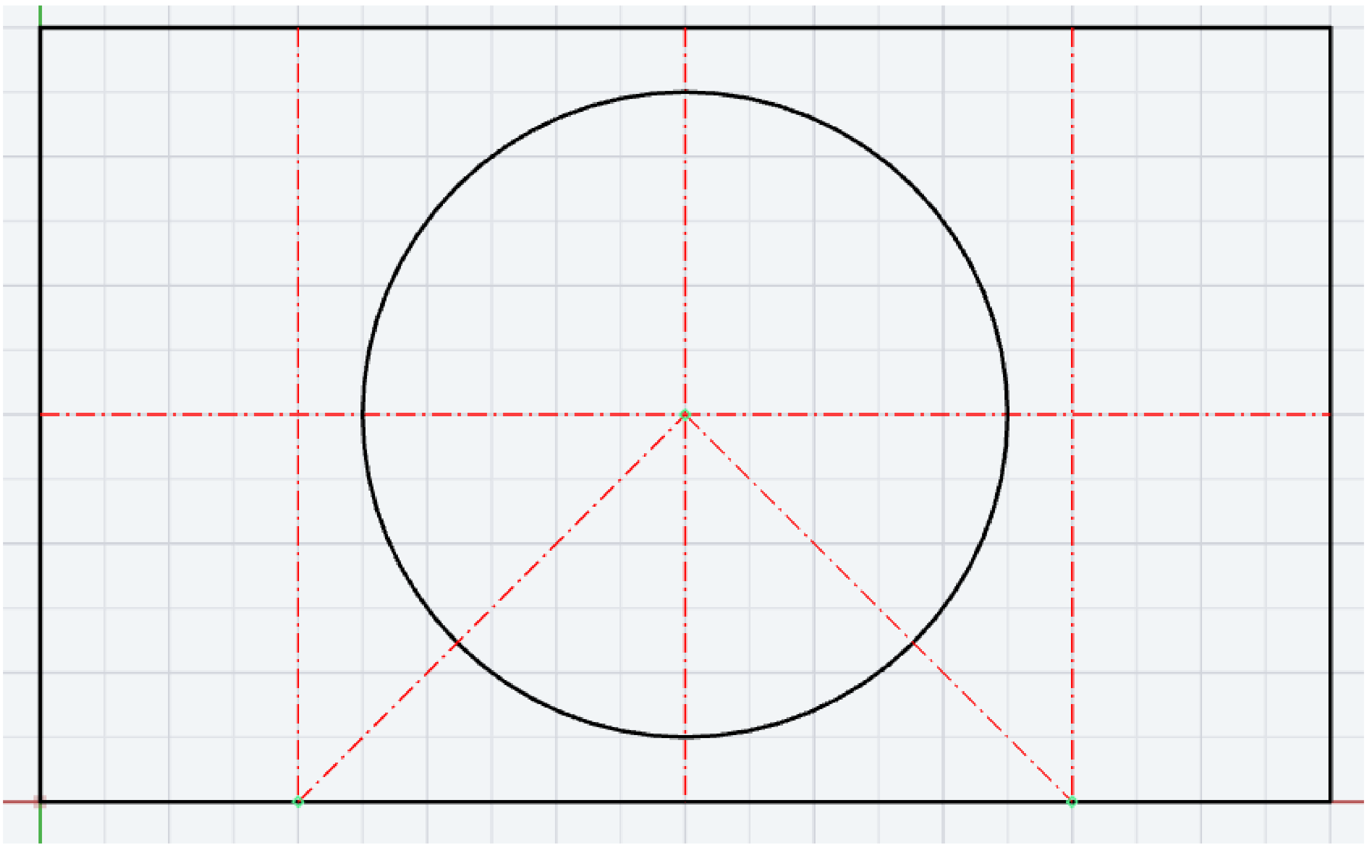
Function: **Offset** distance: **30** place one on the left and one on the right of the vertical axis.

48.4 Draw axes and lines.

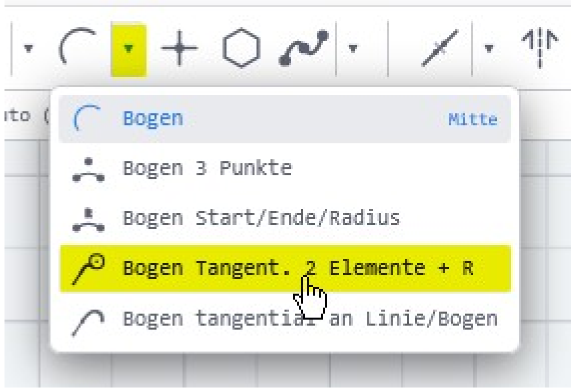
Layer: **ACHSEN**

Function: **Line** angle: **45°** length: **20 mm** (or onto the center) snap point: **start**

Place both lines precisely on the intersections, with the automatic snap [**A**]



48.5 Tangential arcs



The two tangential arcs R10 can be created as follows:

Layer: **SICHTBAR**

function: **Arc tangent to 2 elements + R**

Select the two elements, drag the tangential arc out with the mouse and enter **R 10**. **Enter**.

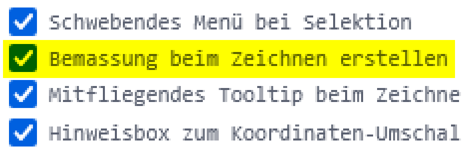
Now draw the two **lines** that join onto the arcs, with [**A**].

Use the Break function [**B**] to **break** away the superfluous arcs.

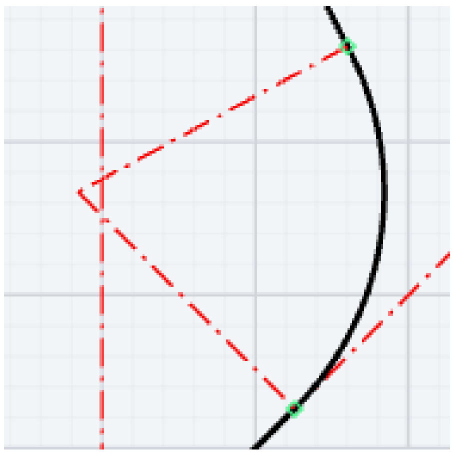
48.6 Center and opening angle

To determine the center of the omega as well as the opening angle at the R10 with full precision, these steps are needed:

Tip: deactivate the automatic dimensioning; in cramped conditions – as here – this can be defined in the Setup at this point →



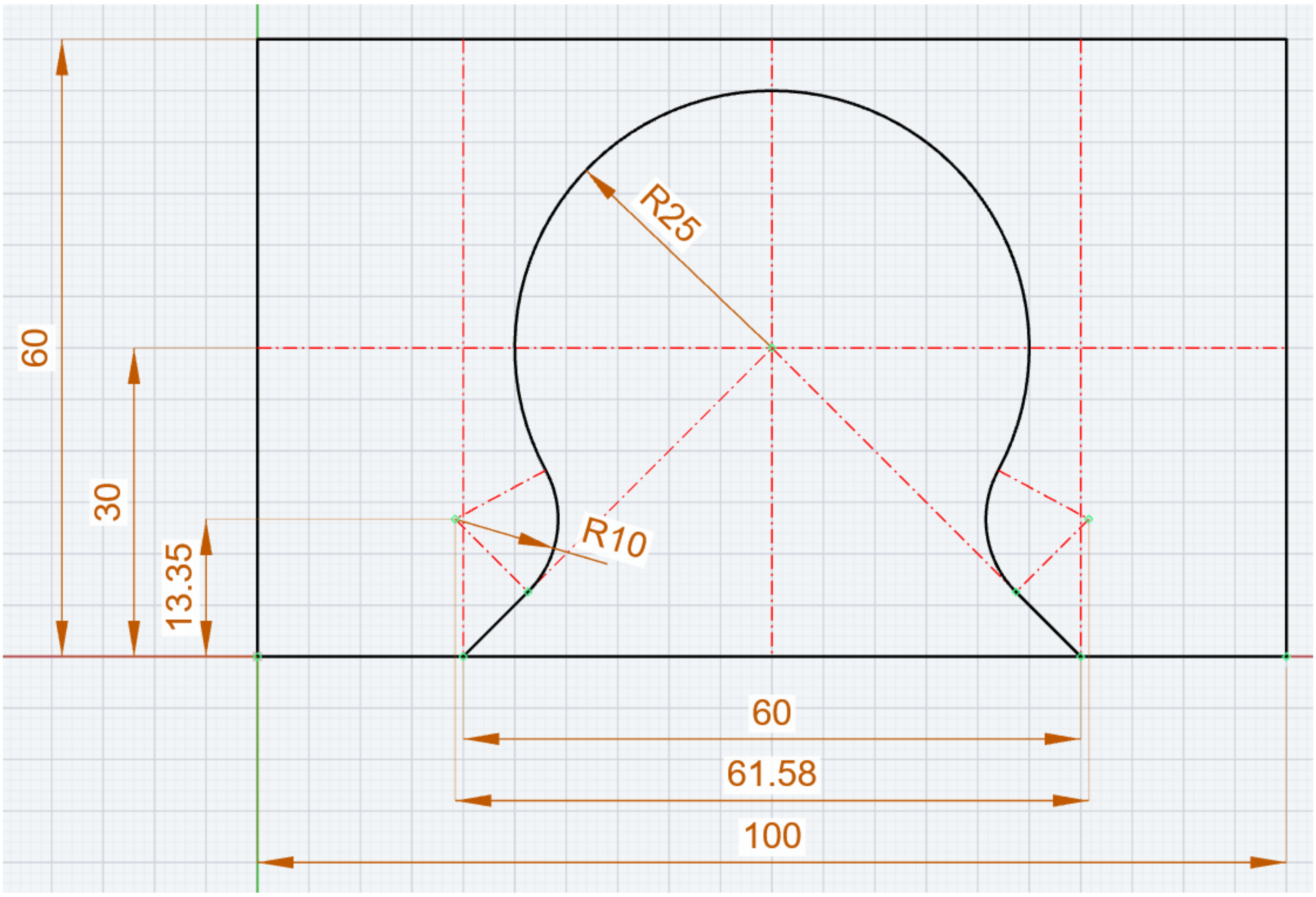
activate layer: **SICHTBAR**.
With the function **Line** and the snap functions [**E**] endpoint and [**Z**] center, draw both lines from the **tangent connection** onto the **center** and on to the other **tangent connection**.



Function: **Line**
(watch the corresponding snap options!)

Now dimension everything quite normally with the Dimension function as follows...

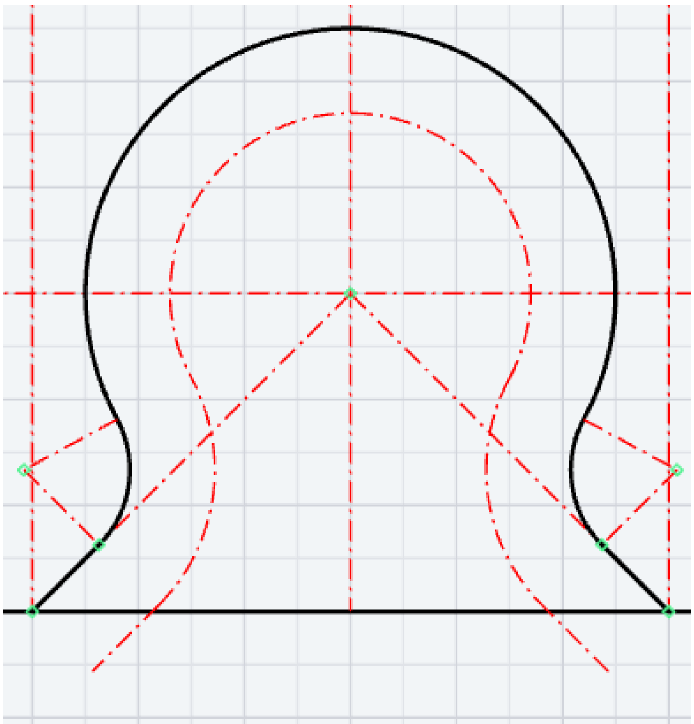
This is what the omega should look like now:



Use the dimensioning to check whether everything is correct...

48.7 Constructing the cutter coordinates

To obtain the cutter coordinates for a **16 mm cutter** we have to create the corresponding parallel **offset** elements (equidistants):



Activate layer: **ACHSEN**.

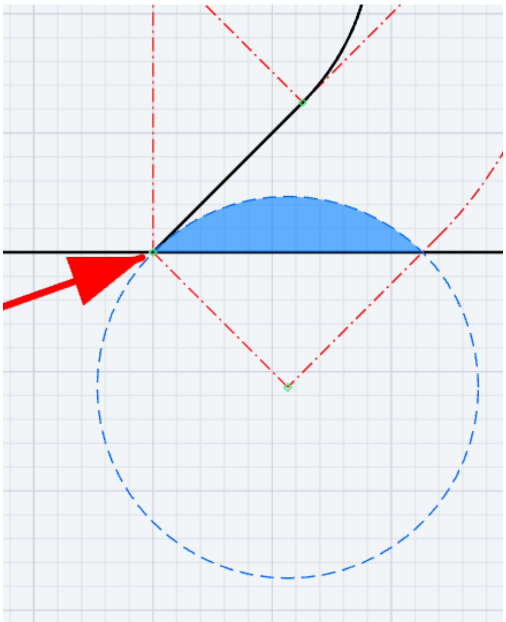
Function: **Offset**

Snap: [**F**] for Free...

Distance: **8 mm**

Create all the offset elements as shown on the right →

That is the center of the 16 mm cutter.



If the **16 mm cutter** now moves straight onto the endpoint and plunges vertically into the part, it stands exactly on the start point and **collides** in the area marked **blue**!!!

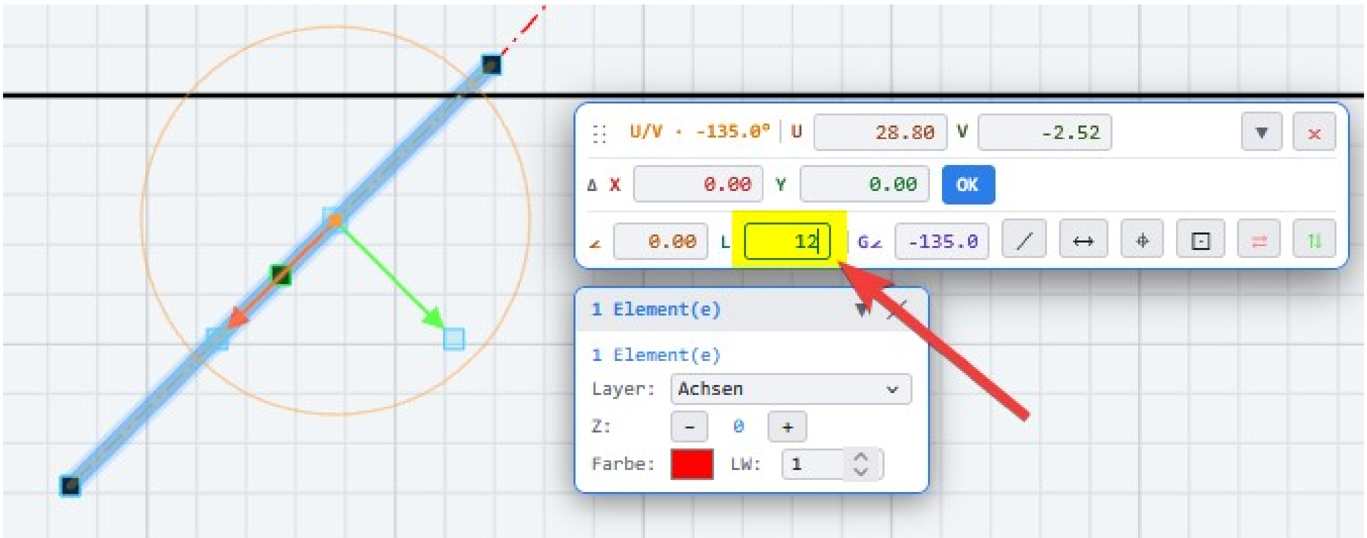
The same would happen on the opposite side.

That is why the two 45° lines are extended by about 10 mm as follows.

Function: **Select [V]** → and select the **line**

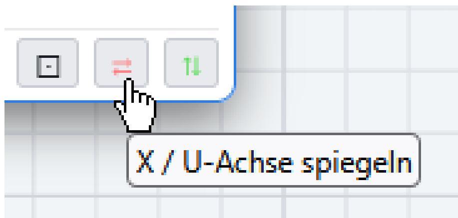
Our "**gizmo**" is displayed. In the field marked yellow, enter the new length **14**.

The line is extended to the L dimension **14** in the direction of the **red** arrow.

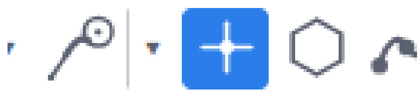


12 mm is too little. The cutter would still collide. Try it out.

If the line is to grow in the other direction, simply mirror that axis in the gizmo:

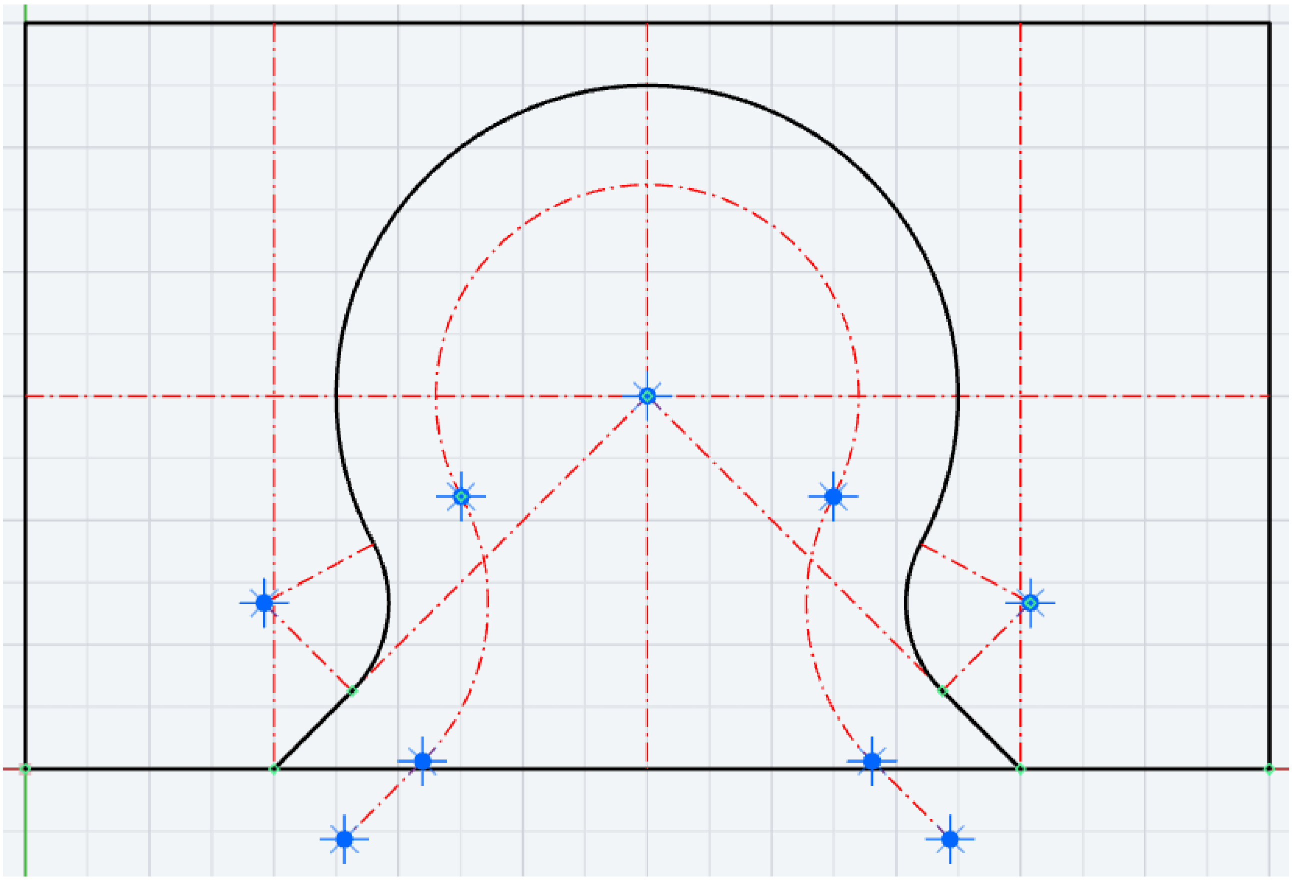


To obtain the coordinates you want in a clean table, point objects have to be placed at the corresponding positions:

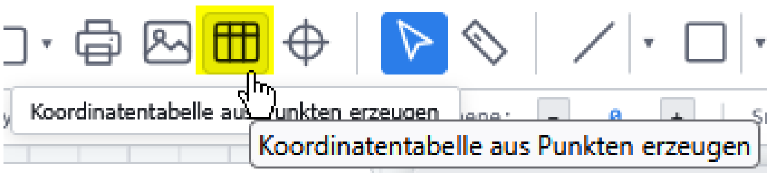


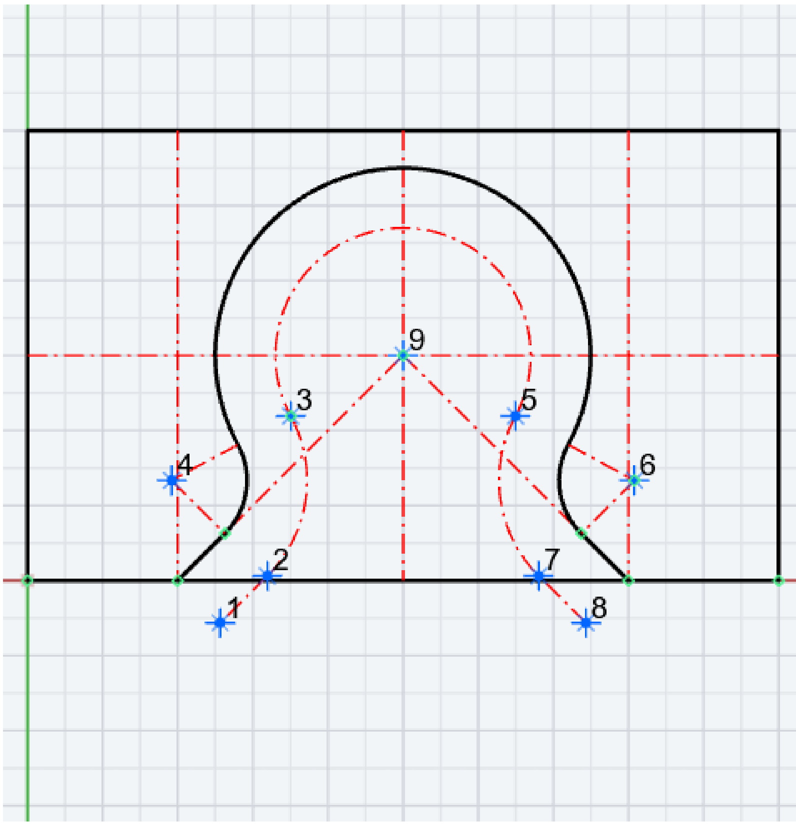
The easiest way is to place these points with the automatic snap [**A**] or the endpoint snap [**E**]:

48 Drawing the omega shape



Insert the coordinate table with:



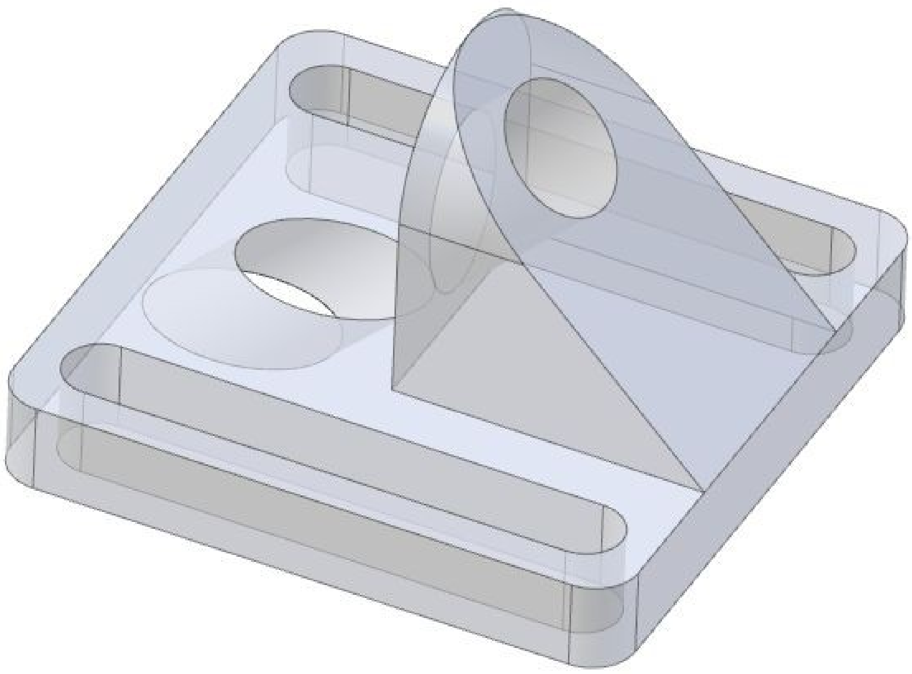


Koordinaten		
Pos	X	Y
1	25.657	-5.657
2	31.940	0.626
3	35.046	21.915
4	19.212	13.354
5	64.954	21.915
6	80.788	13.354
7	68.060	0.626
8	74.343	-5.657
9	50.000	30.000

Done :-)

49 Task: drawing a drill jig

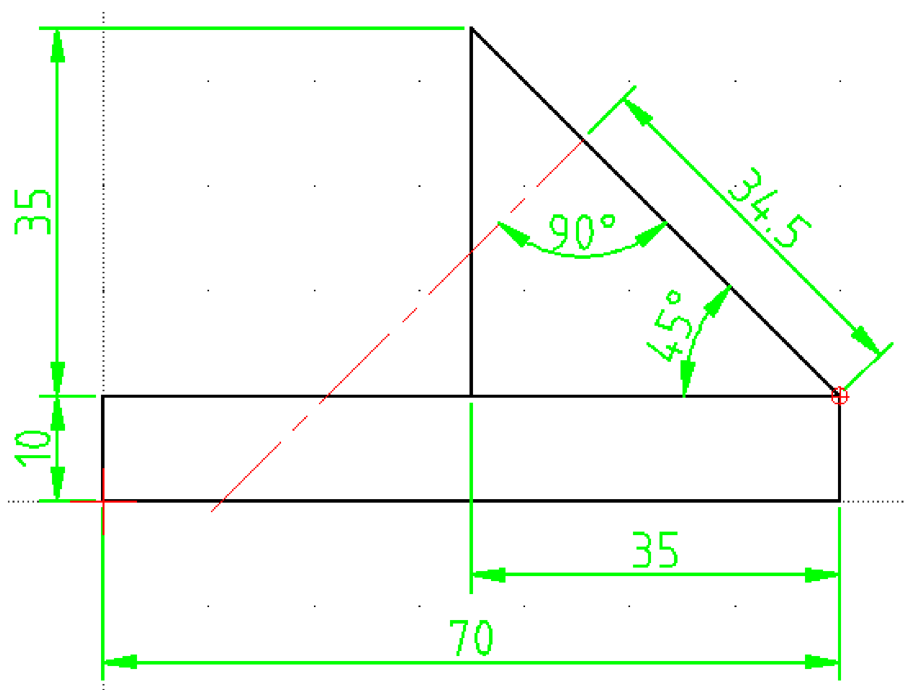
We only have the 3D print of a drill jig left, but there are no drawings of it any more.



Task:

The task now is to create a 2D drawing with all three views plus the section through both holes, including the hatching.

49.1 A hint towards the solution



First the front elevation is drawn.

The black edges are easy.

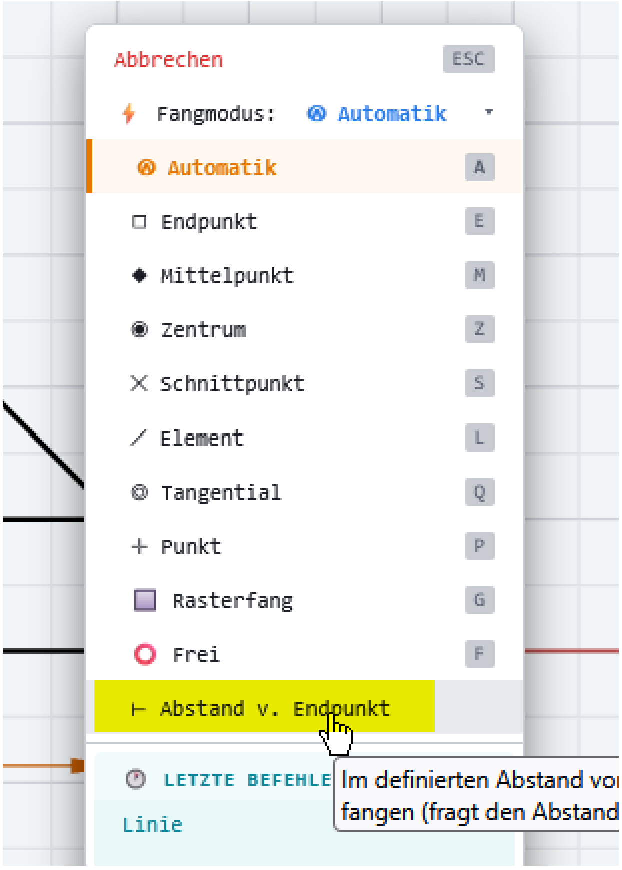
The red axis is created starting from the right-hand red corner point as follows:

Layer: **ACHSEN**

function: **Line → right-click → snap mode Distance from endpoint**

Distance: **34.5 → OK**

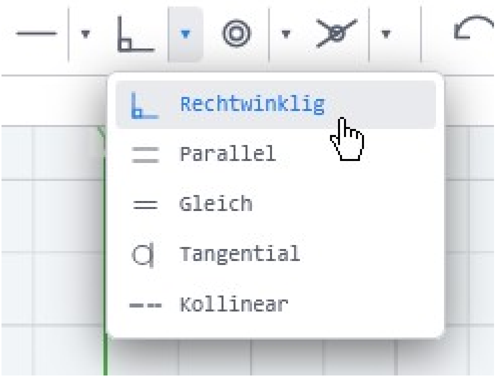
Snap mode: **Endpoint**



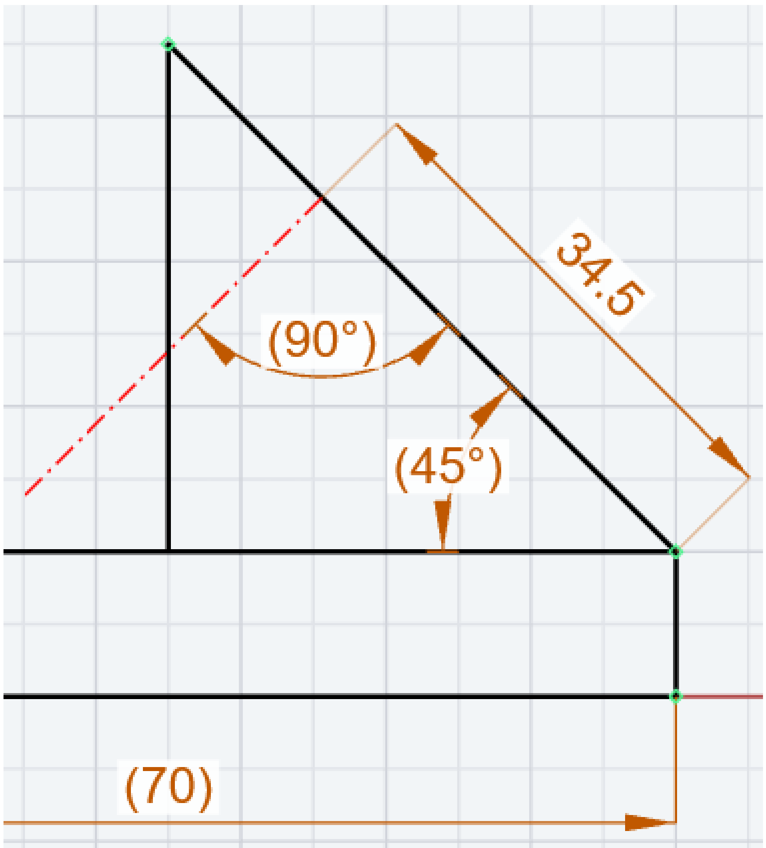
Move the mouse over the 45° line.
The distance of 34.5 mm is snapped.

Draw the line and enter **-135** for the angle. The length does not matter.

Or draw the line at an arbitrary angle and assign a **perpendicular dependency** with:



The axis is finished.



49.2 45° hole D15

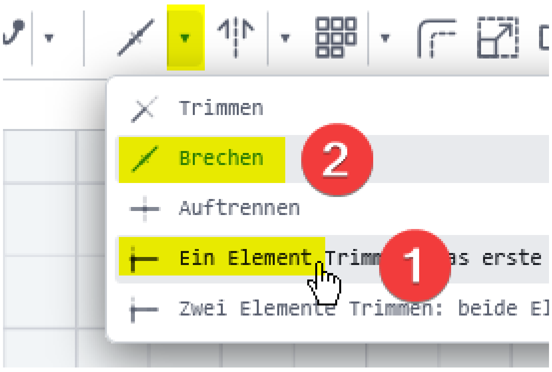
Now the 45° hole with hidden lines can again be drawn using the parallel lines at the correct distance.

Activate layer: **Hilfslinien**.

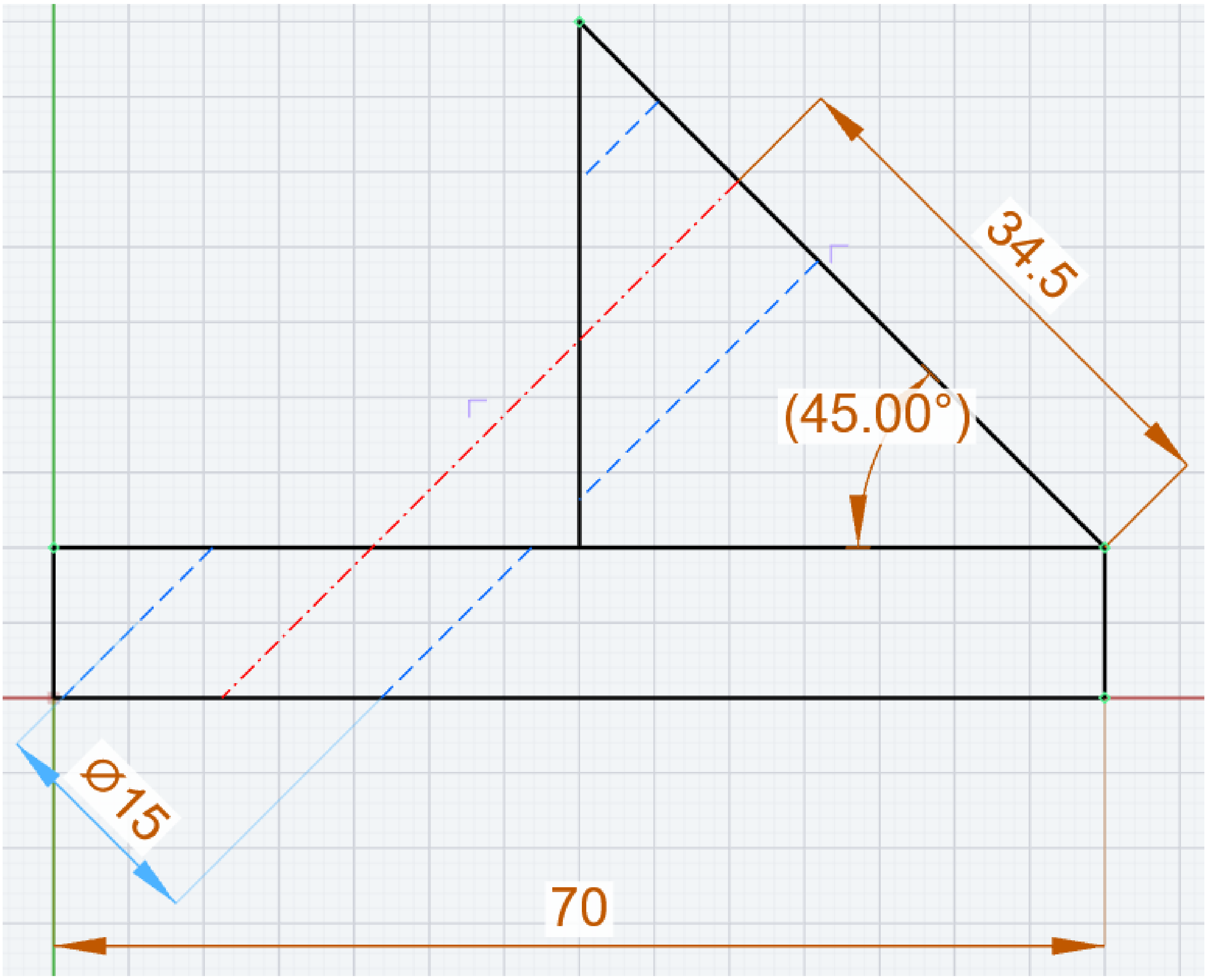
Function: **Offset → distance 7.5** on both sides of the axis.

After that you need for this:
(1) **Trim 1** and (2) **Break** …

Tip: With **Trim 1**, always click the element to be modified first.



Produce the drawing as follows:



49.3 Infinite guides

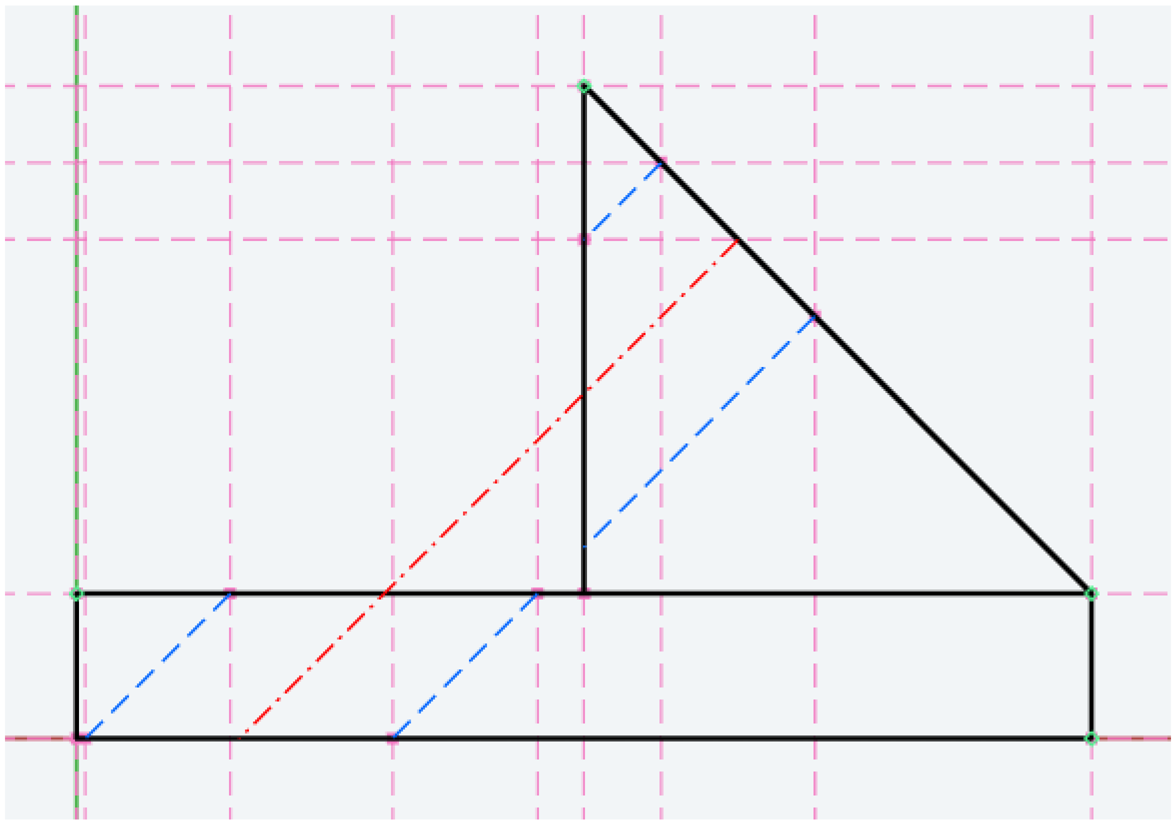
Now create all the guides. They are displayed across the whole drawing area.

In the panel on the right →

Function: **GUIDES → [+ Place]**

Snap: **Automatic**

Now place all the guides for **horizontal 0°** and **vertical 90°**.



This view is finished and we can concentrate on the further views.

Info: guides are always shown across the whole visible screen.

Next we draw the side view to the right of it.

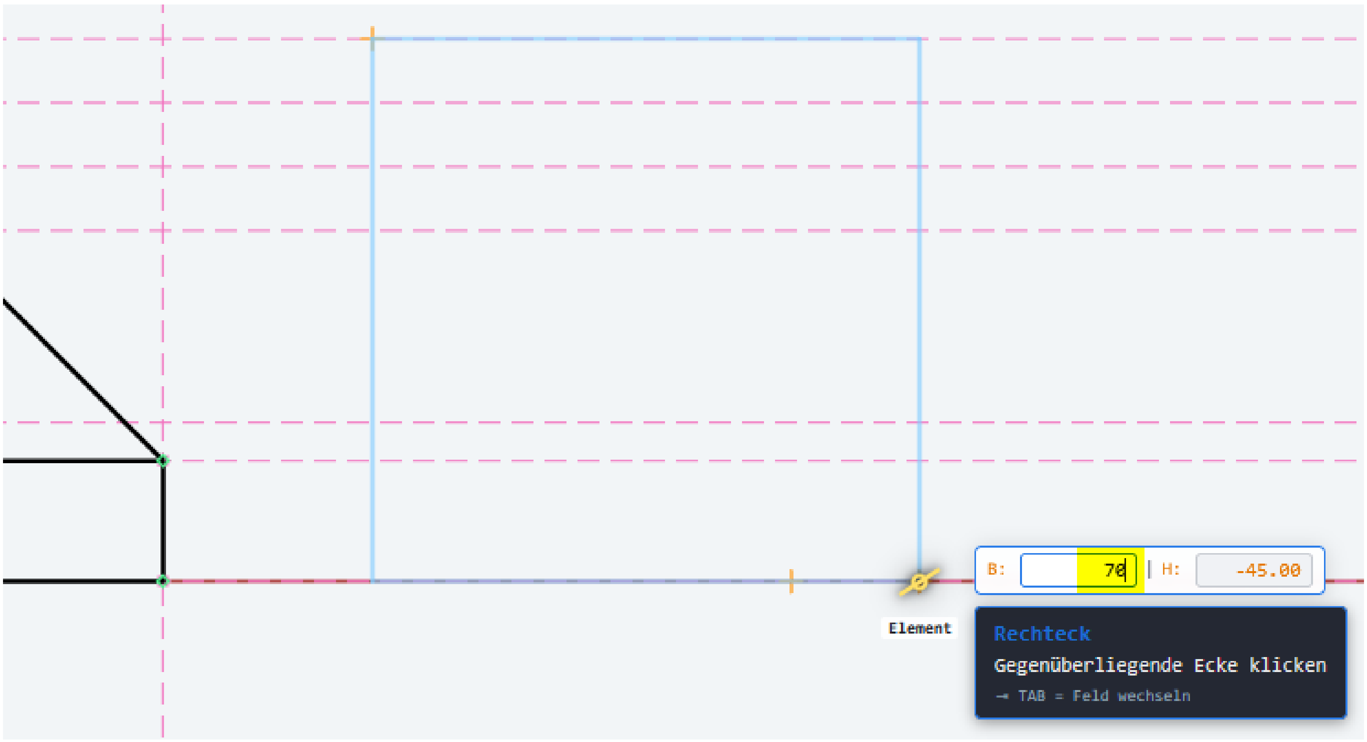
Layer: **SICHTBAR**

Snap: **Element [L]**

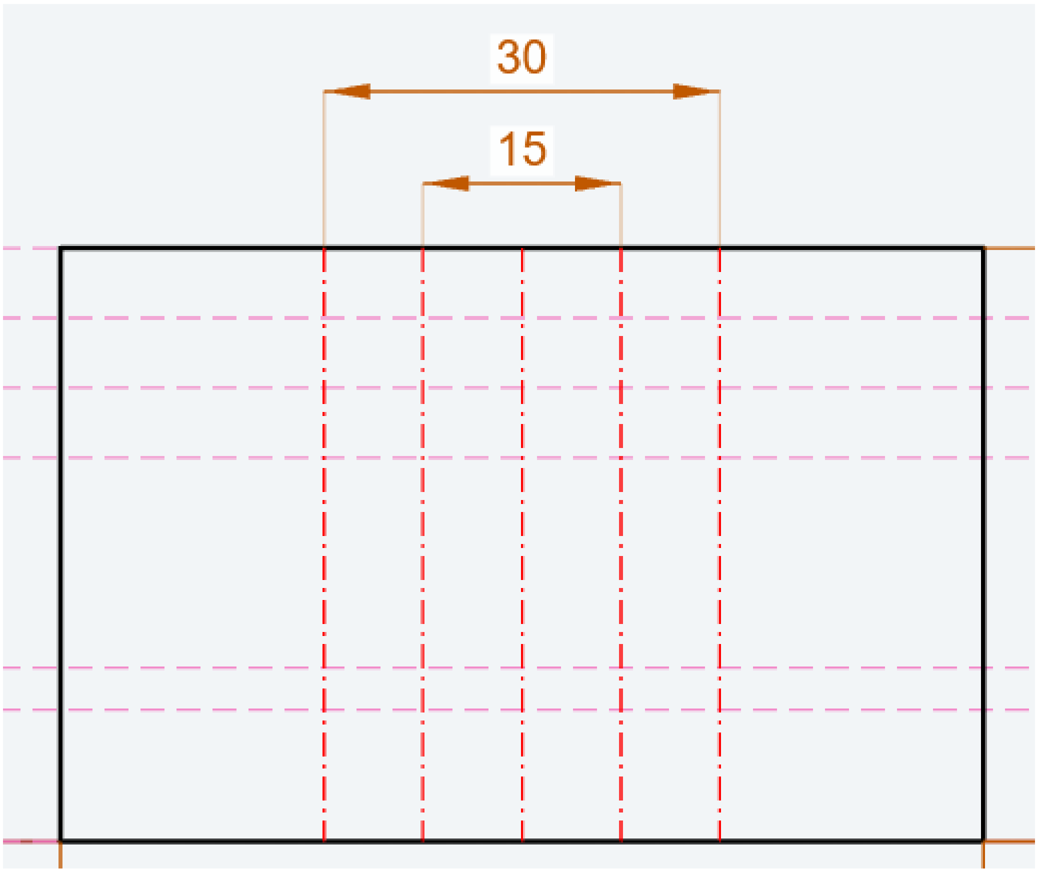
Function: **Rectangle**

Value entry **W: 70** (switch polar/cartesian with the **spacebar**)

H: -45 the value comes from snap mode [**L**] on the guide.



Draw further lines: first the **center line** and then 4 **offset lines...**



The hole is drawn with two ellipses:

function: **Circle → Ellipse**

Layer: **SICHTBAR**

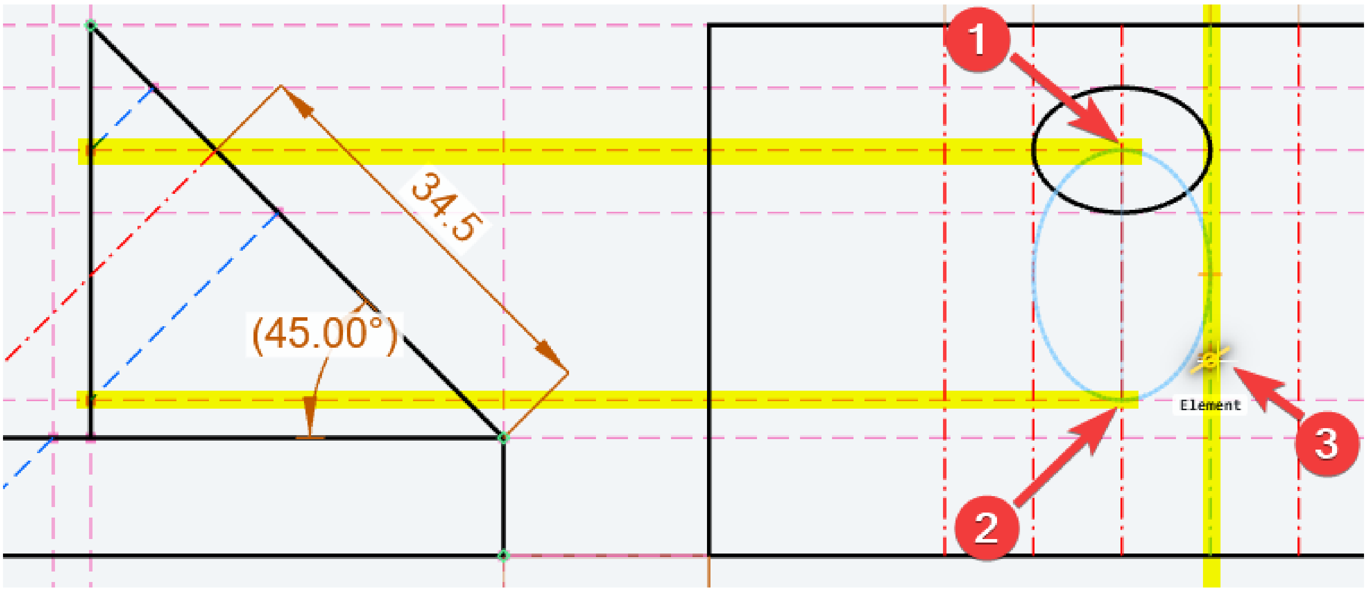
Snap: **[A]utomatic** for the small one and **[A]utomatic + e[L]ement** for the long ellipse.

The upper ellipse goes on layer **Hilfslinie** and is easy.
The long ellipse goes on layer **SICHTBAR** ; it is explained below...

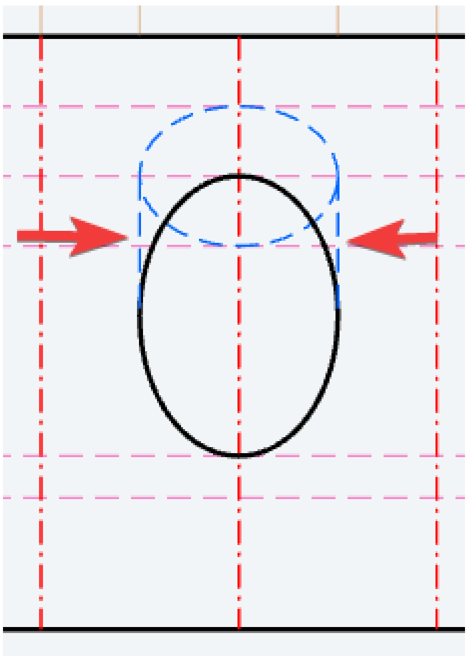
For the long ellipse:

- with snap [**A**], points **1** and **2**

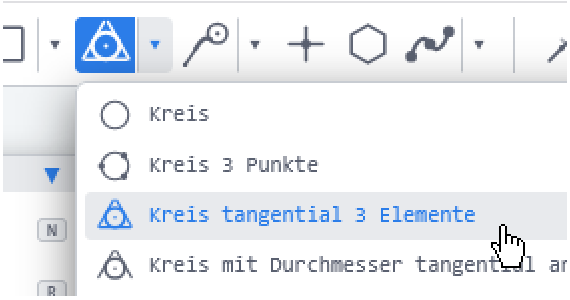
- and with snap [**L**], click at **3**...



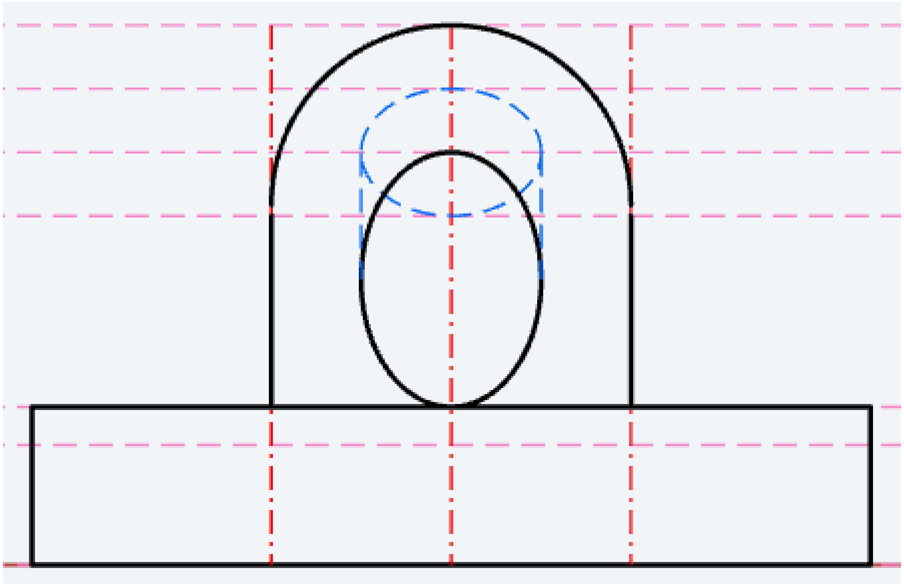
Then draw the two hidden lines:



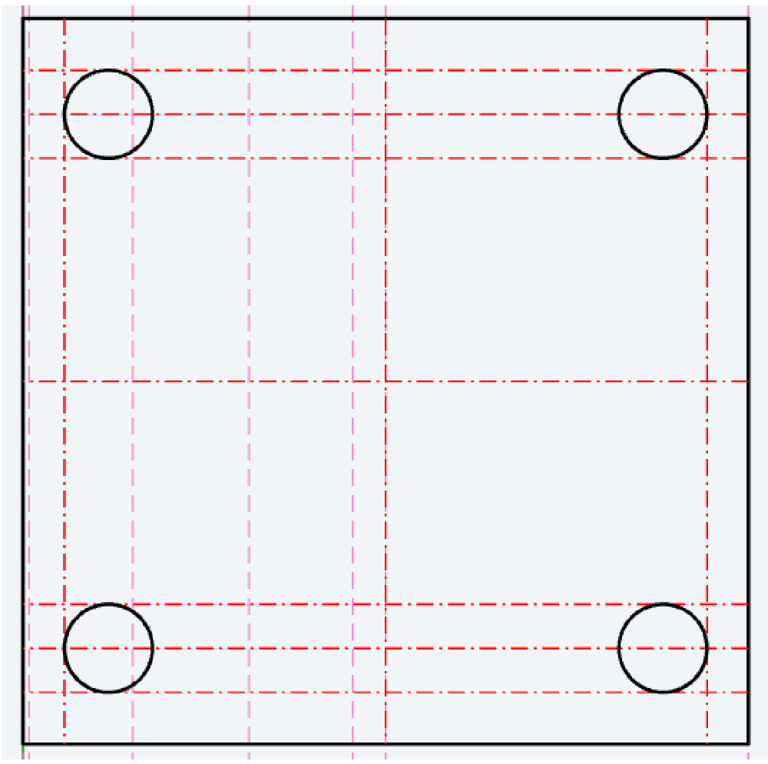
Now with



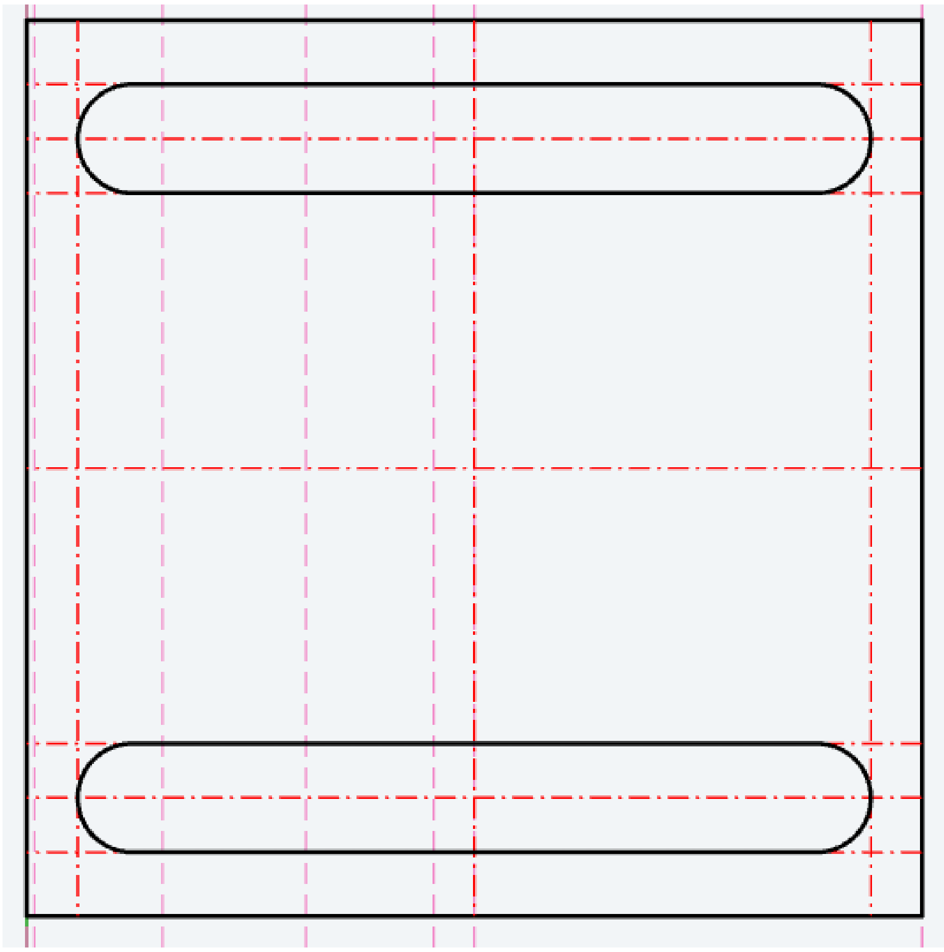
draw the arc... BUT **first explode** the rectangle, since it is still handled as a "block"...

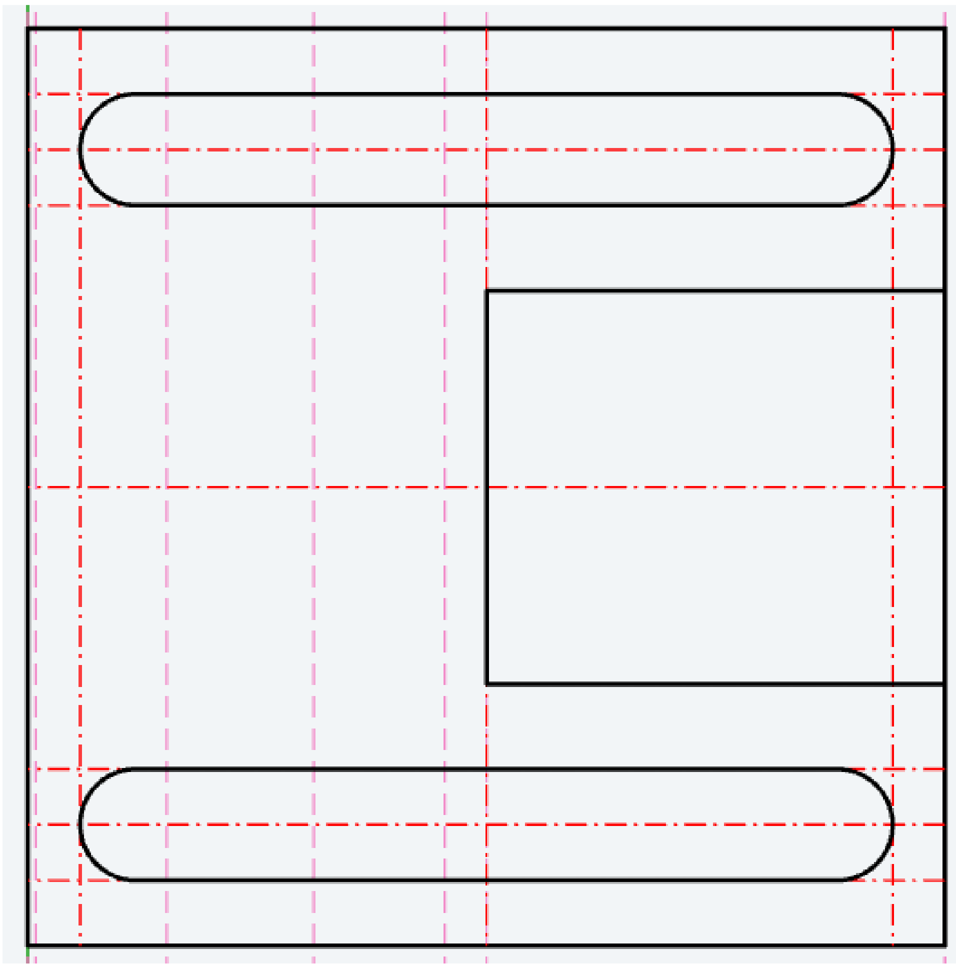


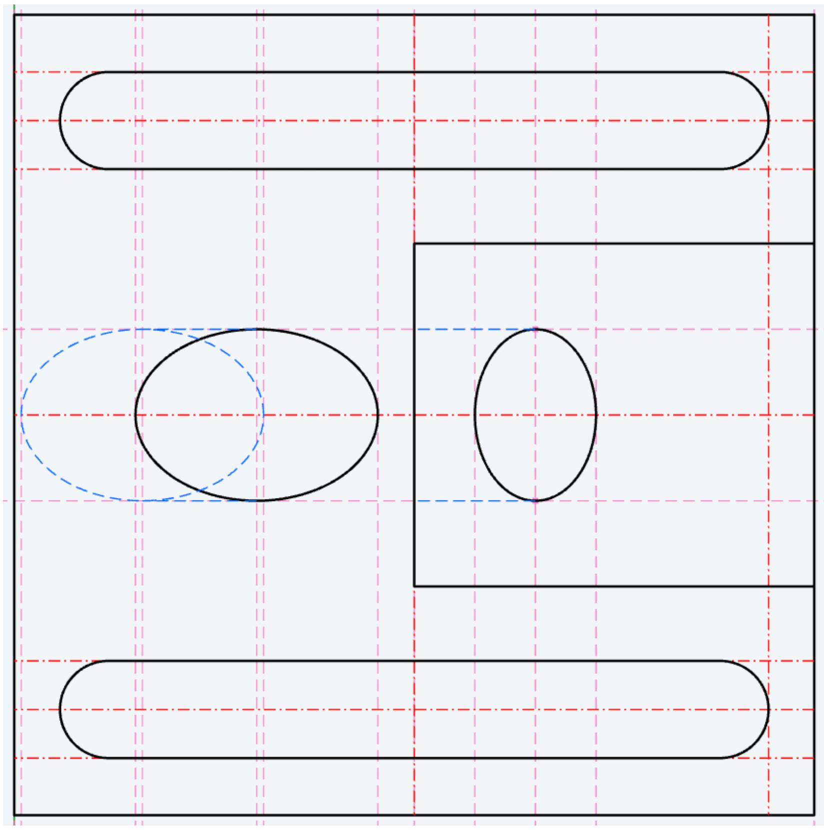
Draw further lines and break them until it looks like this.



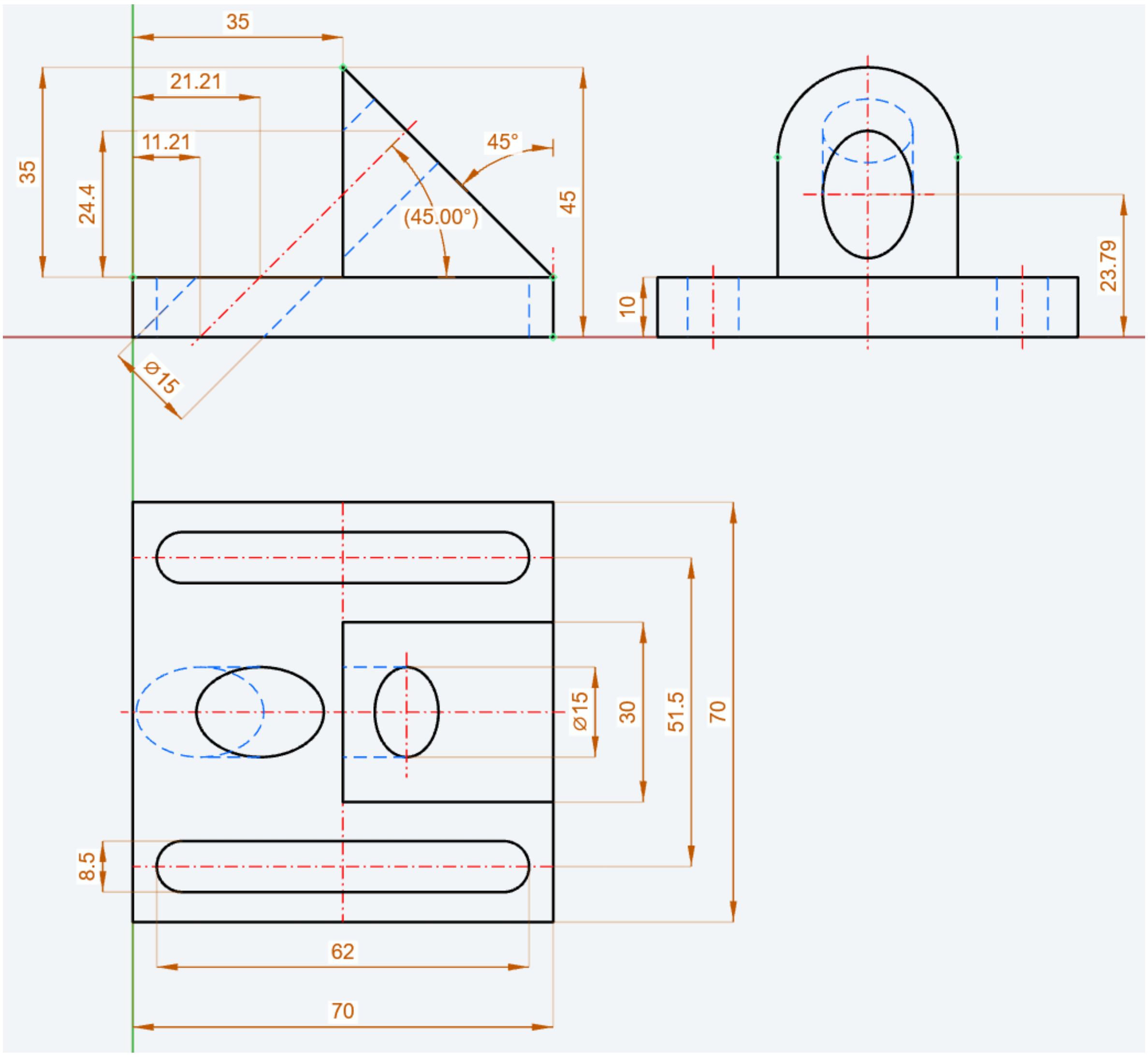
Draw everything else yourself, using the figures...







Hide the guides and dimension everything nicely…



50 Standard parts

STANDARD PARTS in LiberoDraft are ready-made collections of blocks, symbols or other elements that can be used in drawings. They serve to make drawing easier and faster by being available as reusable elements.

Using libraries in LiberoDraft has several advantages:

- Time saving
  By using ready-made blocks and symbols, they do not have to be created anew every time but can simply be inserted from the library.
- Consistency
  If the same elements are used again and again in a project, they can be stored in a library and inserted in the same form at any time. This ensures a consistent appearance in all drawings.
- Easy updating
  If an element in a library is changed, all instances of that element in all drawings can be updated by simply updating the library.
- Collaboration
  When several people work on a project, libraries can be used to ensure that everyone uses the same elements. That makes collaboration easier and avoids confusion.

There are a number of standard libraries, such as a library with architectural symbols, electrical symbols and mechanical parts.

Users can also create their own libraries and share them with others.

Your own standard part libraries should always be stored in a central place where everyone has access to them.
More on this in the chapter 50.2 Saving the standard part as M8

50.1 Creating your own standard part (M8)

An M8 thread is used to show how this is done and what possibilities there are.
The tapping drill hole for the M8 thread is 6.8 mm and the thread line is approximately shown as a ¾ circle.

Open a new file in LiberoDraft.

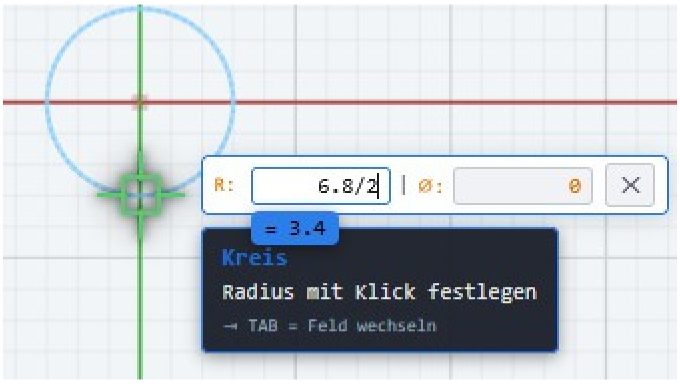
<u>**Circle for the tapping drill hole** at the origin of the drawing</u>

Activate layer: **SICHTBAR**.

Function: **Circle → center, radius**: **6.8/2**

INFO: In all numerical fields the basic arithmetic operations including brackets can be entered. :-)

You may have to activate snap mode **Grid** [**G**] so that the origin is snapped precisely.

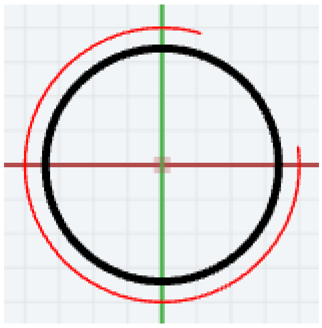


<u>**¾ circle for the thread diameter** as follows</u>

Activate layer: **KOSMETISCHE LINIE**.

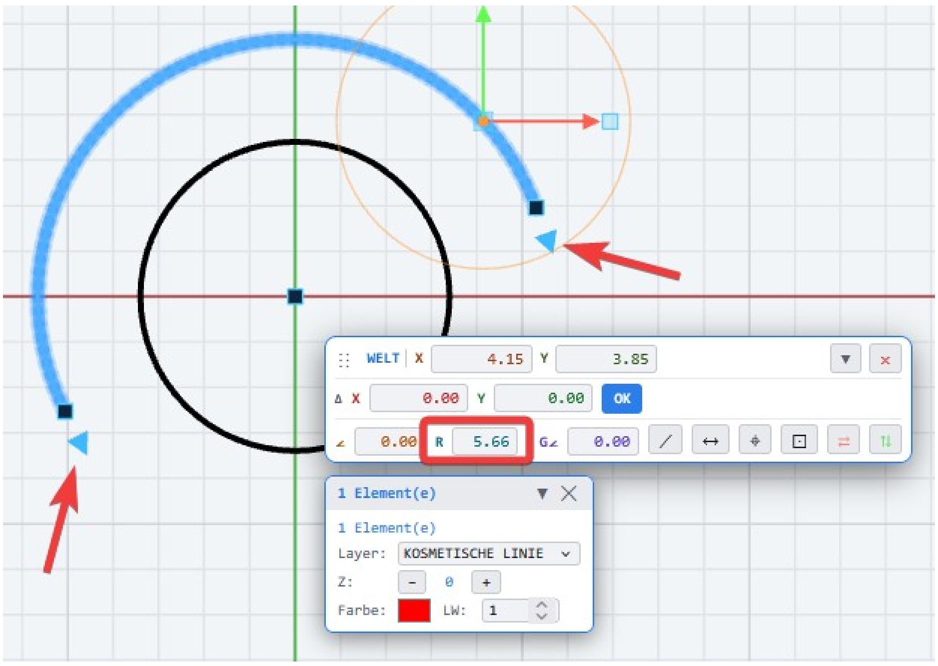
Function: **Arc → center, radius**: **4 → Enter**

Define the center, the start and the end of the arc.



Tip: If you have made a complete mess of the arc, no problem. In **select mode**, simply select the arc and drag the **blue** triangles until you have roughly a ¾ arc. Then enter the correct radius and you are done.

That is how easy it is in LiberoDraft :-)



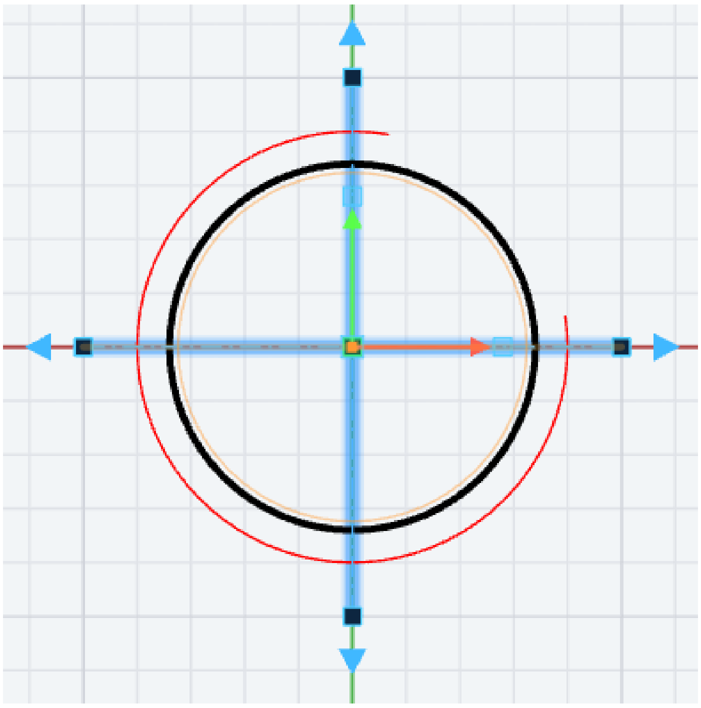
Drawing the axes

Activate layer: **ACHSEN**.

Function: **Line → choose 2 points**.

Draw both axes on the **grid**. The axes should protrude evenly beyond the thread line on all sides.

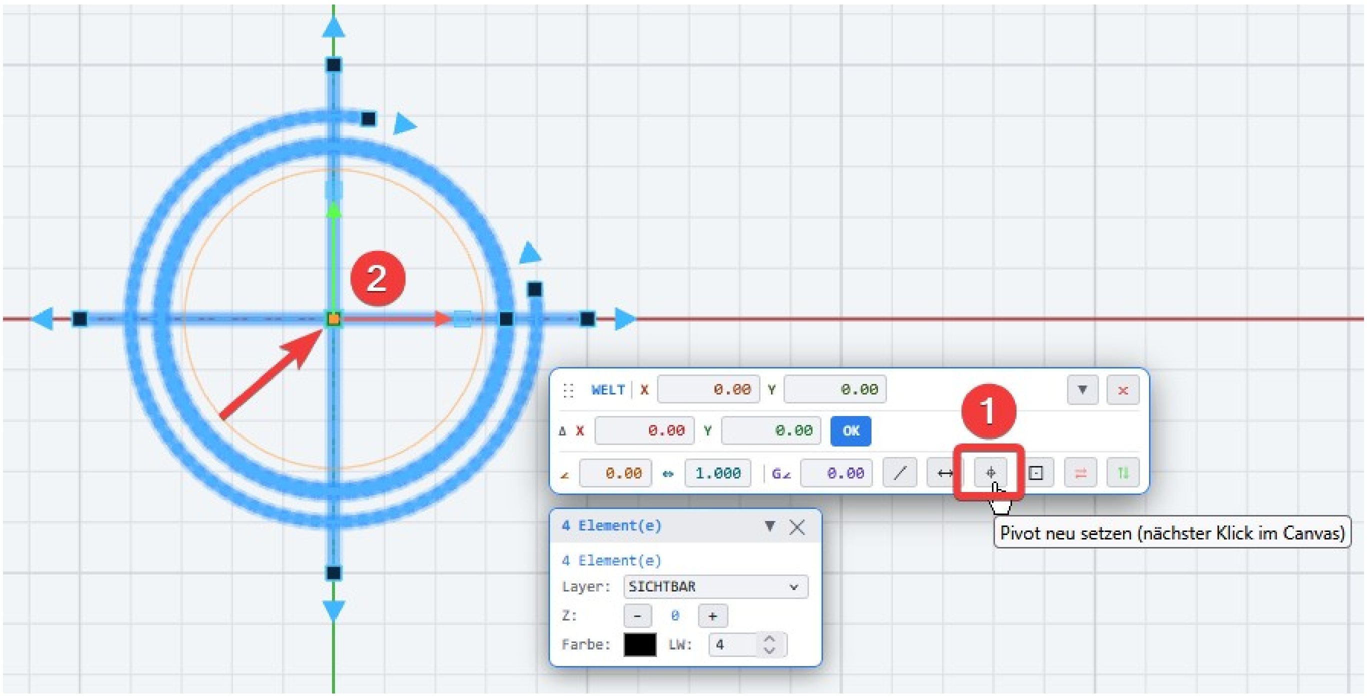
If they do not, simply select the element and drag the blue triangles…



50.2 Saving the standard part as M8

It makes sense to store all standard parts in a central place that everyone can access.

Select all elements with the rubber band and place the pivot point (1) of the gizmo on the center (2). That becomes the default insertion point of the M8 standard part.



Right-click → Save as standard part and give it a name.
In the file window, store the M8 centrally on your NAS. Done :-)

50.3 The M8 thread in the side view

While we are at it, we can create the M8 thread in the side view right away. In the side view the question is how long the thread is drawn and whether it is a through hole or a blind hole.

So we have to create two standard parts for the M8 side view:

- 1x M8 with a **through hole**, initial length: 10 mm

- 1x M8 with a **blind hole**, initial length of the hole: 15 mm, thread depth: 10 mm

Below, create the two threads according to the dimensioned figure but **without dimensions**:

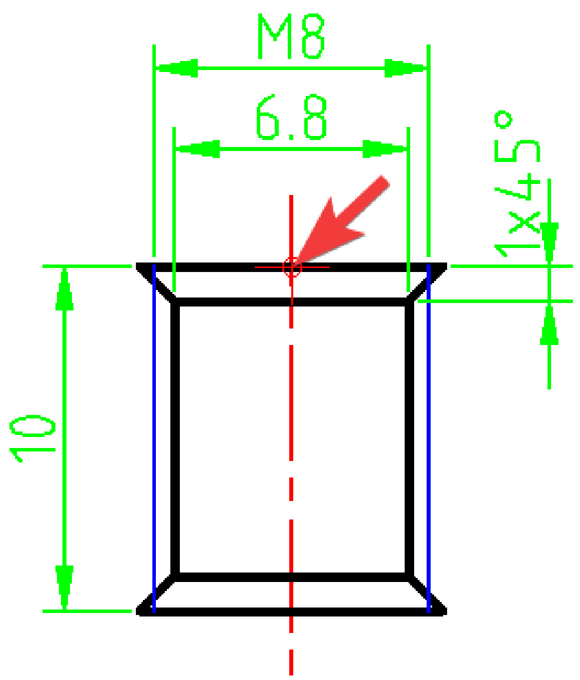


Diagram 3: Save as: ***M8-D.dxf***

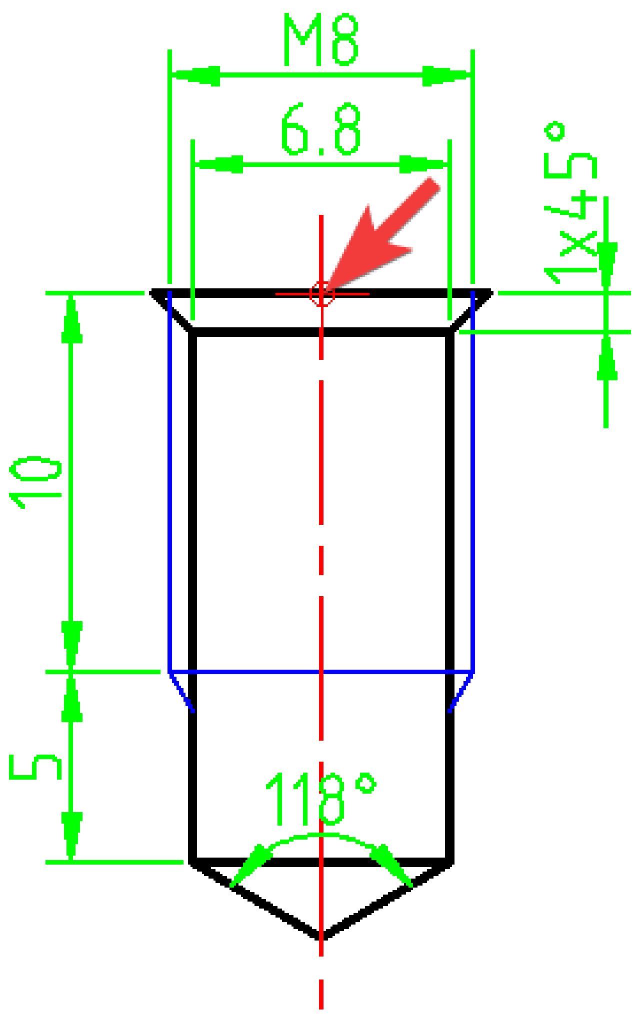


Diagram 2: Save as: ***M8-SLB.dxf***

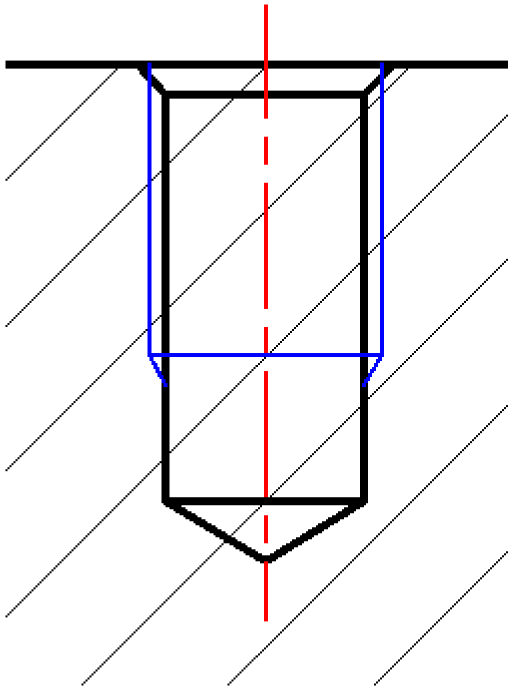
Saving the standard part

Now the thread can be saved as «M8-**D**.dxf» and «M8-**SLB**.dxf».

«**D**» stands for durchgehend (through), «**SLB**» for Sacklochbohrung (blind hole).

Save it as described in chapter 50.2 Saving the standard part as M8

If we now draw a hatched rectangle and insert the blind hole, for example, the following happens, as you can see on the right:



The hatch lies over the blind hole.

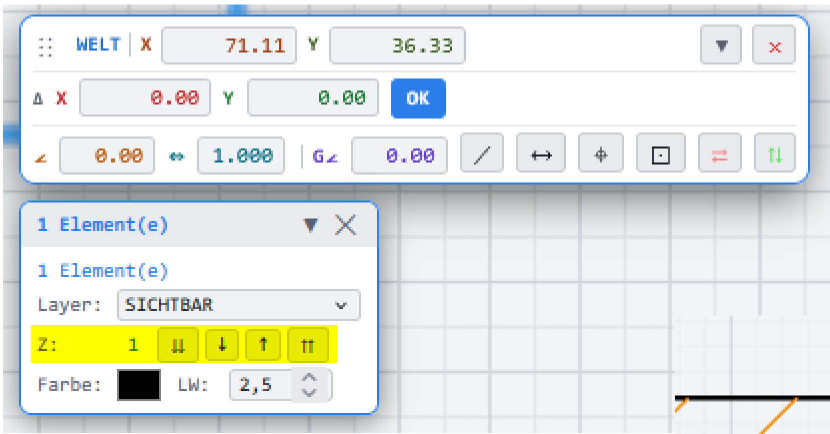
There are situations in which that is intended.

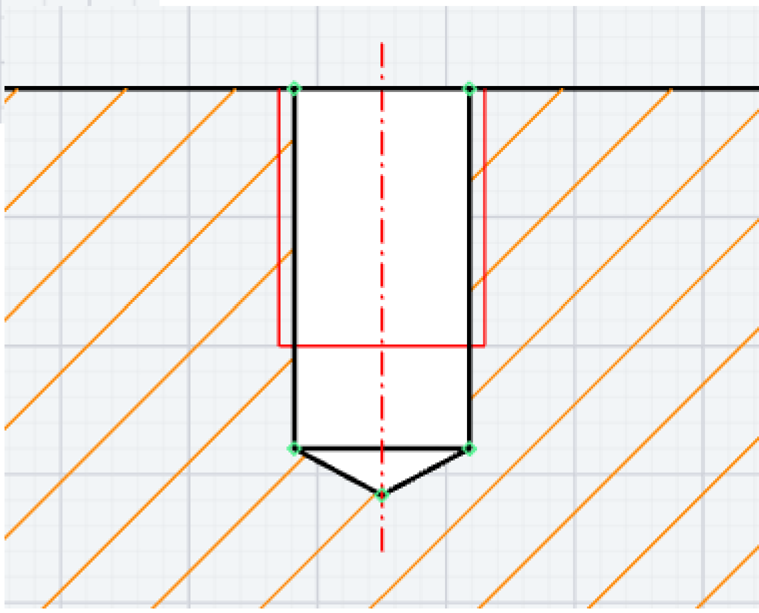
But usually this hole is shown in a section, and then the hatch is interrupted in the area of the tapping drill hole.

There is a trick for this: fill the area of the tapping drill hole and of the chamfer with a solid, white hatch or create the hatch anew.

When filling the tapping drill hole with a white, solid hatch, remember to bring the lines of the hole to the foreground so that they are not covered by the white area.

Select the lines and bring them right to the top in the Z level with the button [⇈].





Perfect! :-)

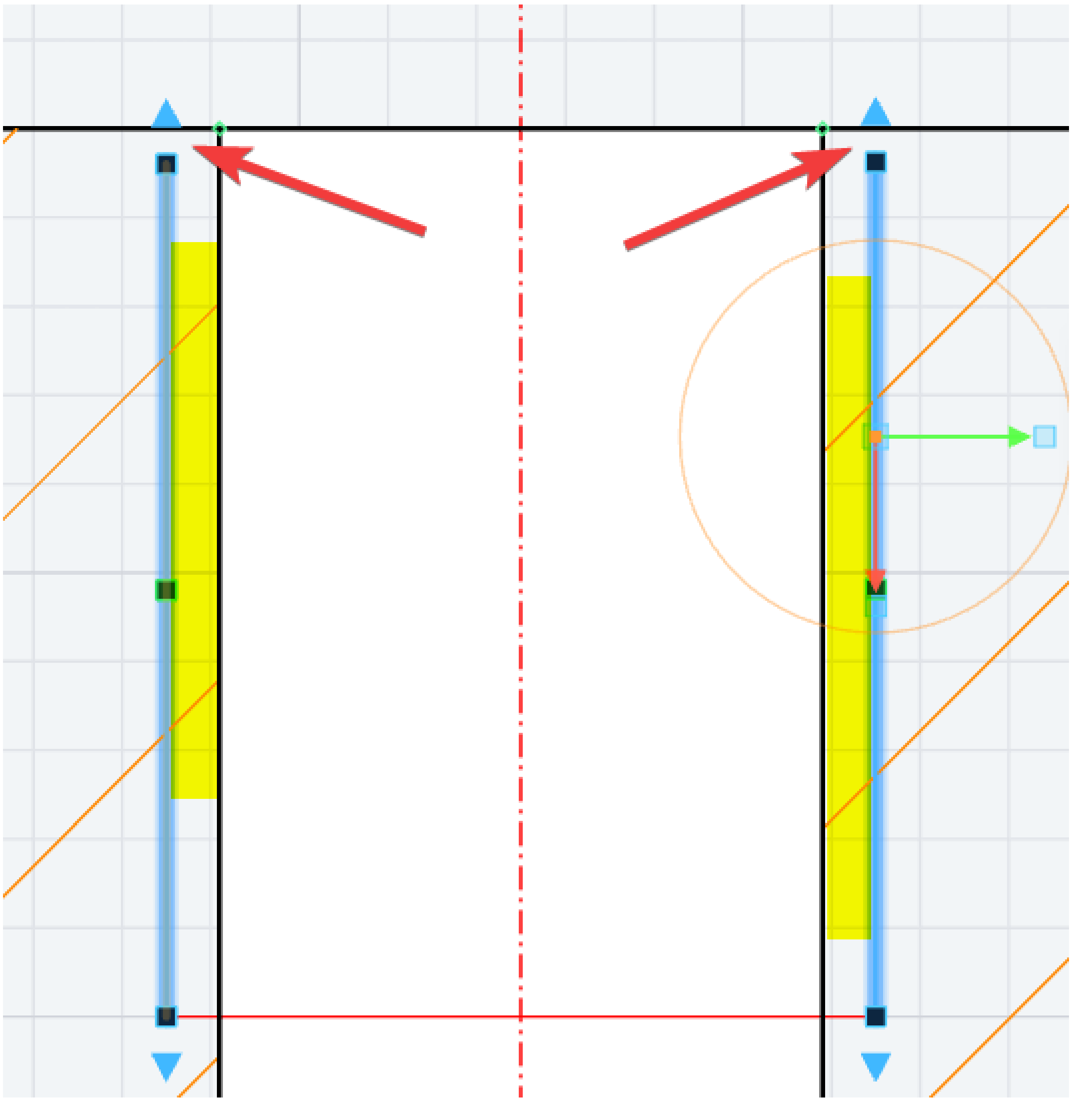
50.4 Hatch observation

With threads, the hatch has to run up to the tapping drill hole, as in the figure on the previous page.

But if the M8 blind hole is placed first and the hatch is only created **AFTERWARDS**, the hatch only runs up to the red thread line and not up to the tapping drill hole.

The trick to solve it:

Select both red thread lines and push the **blue triangles** down so that there is, so to speak, a small gap into which the hatch algorithm can penetrate and extend the hatch cleanly up to the tapping drill hole:



Splendid! :-)

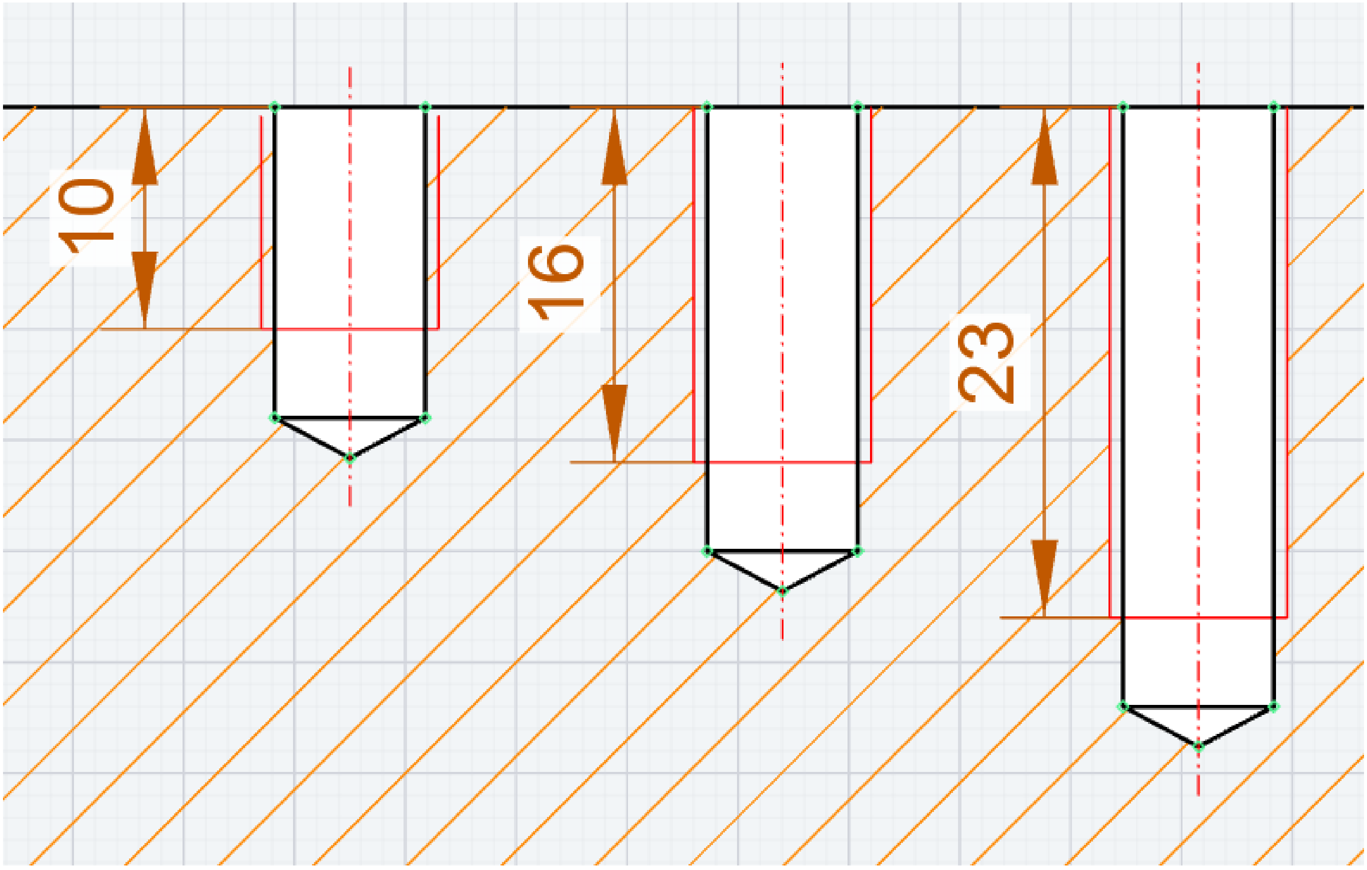
50.5 Stretching the blind hole thread standard part

That works very well with the Stretch function

Simply frame the **area to be stretched**.

Define a **base point** (it may also be outside on the grid [G])

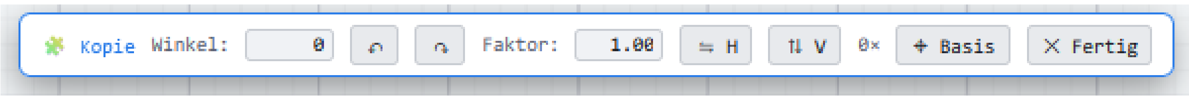
Click the **target point**.



The 10 mm blind hole was inserted as a standard part and then everything was selected, so that it can be placed as often as you like with Ctrl+C and Ctrl+V.

Tip: while placing with Ctrl+V, a new base point can always be snapped with [B].
With **Ctrl+mouse wheel** it is rotated.

You find the rest at the bottom **above the status bar**:


Kopie
Winkel:
0
Faktor:
1.00
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V
0×
Basis
Fertig

51 Blocks – reusable symbols and components

Blocks are among the most important tools for drawing efficiently and with few errors. This chapter explains what blocks are, what they are suitable for, how you create, insert and change them and provide them with parts-list data – and what you should watch out for while doing so.

51.1 What is a block?

A **block** is a named, reusable compilation of drawing elements (lines, circles, arcs, hatches, dimensions and so on). What makes it special: the block is stored **only once** as a so-called **definition**. In the drawing you place as many **instances** (copies) of that definition as you like.

The decisive advantage follows from this definition-and-instance model: if you change the definition **a single time**, **all** the placed instances update automatically.

> **Tip:** Think of a rubber stamp: you make the stamp once (the definition) and stamp it onto the sheet as often as you like (the instances). Re-carve the stamp and in LiberoDraft even the impressions already made change with it.

51.2 What are blocks good for?

- **Recurring components and symbols** – screws, bearings, hole patterns, welding and surface symbols, title block components or frequently needed detail parts, for example.
- **Define once, use as often as you like** – that saves drawing work and keeps the appearance uniform.
- **Central change** – if a component is adapted, you only change the definition; all occurrences follow automatically. That prevents forgotten places and consequential errors.
- **Parts lists** – every block can carry parts-list data (article number, material, supplier, price …). Blocks thereby become countable items of a parts list.
- **A clearer drawing** – many instances refer to a single definition; the drawing stays lean.

51.3 Block, group or standard part – the difference

These three terms are easily confused. The following overview helps to place them:

Term	Meaning
Group	A mere combination of several elements in the current drawing so that they can be selected and moved together. No central definition, no multiple instances, no parts-list data.
Block	A named definition with as many instances as you like in the current drawing. Centrally editable, with parts-list data. Ideal for recurring components.

Term	Meaning
Standard part	A part stored in the library that is available across projects. A block can be taken over into the library via "Save as standard part".

51.4 Creating a block

Proceed as follows:

1. **Select** all the elements that are to belong to the block. You may include dimensions and hatches in the selection.
2. Optionally define the **base point** (see the next section). Without a specification of your own, LiberoDraft uses the center of the selection.
3. Call up the command: **right-click** the selection and choose **"Create block"**. Alternatively, in the **"BLOCKS"** panel click the **"Create block from selection"** button.
4. A small input field appears. Enter a **name** ("Platte-1", for example) and confirm with **Enter**.

The block now appears in the **"BLOCKS"** panel. The selected elements are from now on a block instance.

51.5 The base point

The **base point** is the "handle" of the block: while inserting, the block hangs at exactly that point on the crosshair, and rotating and scaling take place about it. A well-chosen base point makes placing precise.

This is how you determine it:

- Default: if no point of your own is set, LiberoDraft uses the center of the selection.
- Your own point before creating: place the rotation/reference point (gizmo pivot) on a distinctive point – a corner, a midpoint or the center of a hole, for example.
- Your own point while placing: during insertion you set it anew with the key **B** (with object snap).

> **Tip:** Choose the base point where you will most often attach the block later on – the center of a hole or a reference corner, for example. That saves you having to nudge it into place afterwards.

51.6 Inserting and placing blocks

In the **"BLOCKS"** panel, click the **plus button "+"** of the row you want or simply the block name. The block now hangs on the crosshair. Every **click** places an instance – so you can place several one after another.

While placing, these keys are available to you:

Key / input	Effect
Click	Place the block at the current position (several times

Key / input	Effect
	possible)
R	Rotate the block
+ / –	Enlarge / reduce the block (scale)
Ctrl + mouse wheel	Change the rotation angle in steps of 1°
B	Set the base point anew (with snap)
ESC or a right-click	Finish placing

Note: The object snap also acts on the geometry of the block – so you can snap to the corner points, midpoints and so on of a block.

51.7 The "BLOCKS" panel at a glance

Every row in the panel stands for one block definition and offers the following buttons:

Element	Function
Name (puzzle symbol)	Clicking it inserts the block
N×	Number of instances of this block in the drawing
+	Insert the block (hangs on the crosshair)
Angle symbol	Edit the block geometry (block editor)
Pencil symbol	Edit the parts-list data
Wastebasket symbol	Delete the block definition (only if unused)

51.8 Changing the block geometry (block editor)

With the **angle button** in the block row (or via the right-click menu **"Edit block"**) you open the **block editor**. LiberoDraft thereby switches into an isolated mode: only the geometry of the block is visible, with the base point lying at the origin.

In the block editor:

- Edit the geometry with all the usual tools (drawing, deleting, moving, dimensioning …).
- **"Save"** writes the changes back into the definition and increases its version – which makes **all** the placed instances update.
- **"Cancel"** discards the changes; the definition stays unchanged.

Careful: A change in the block editor affects **all** instances of that block. If you want to design just one single occurrence differently, first dissolve that instance (see "Explode") and then change the free elements – or create a new block.

Note: A block cannot contain itself. So you cannot insert the same block inside a block.

51.9 Parts-list data (designation, article number …)

With the **pencil button** (or the right-click menu **"BOM data…"**) you open a dialog with nine fields. At the top right the dialog also shows how many times the block occurs in the drawing (*"N× in the drawing"*).

Field	Content / example
Designation	Plain-text name of the part ("Base plate", for example)
Article number	Internal or supplier article number
Standard	DIN 933, ISO 4017, for example
Material	S235, aluminum, 1.4301, for example
Unit	Unit of quantity (default: "pcs")
Weight	Weight per piece
Supplier	Source of supply
Price	Price per piece
Remark	Free text for additional information

The parts-list data belongs to the **definition** and therefore applies to all instances. It forms the basis for a parts list in which the items are counted and quantities, weights and prices are totalled.

51.10 Dissolving a block (explode)

Right-click an instance and choose **"Explode"**. The instance is converted back into its individual elements – including the dimensions and hatches it contains.

Careful: When exploding, the link to the definition is lost. Later changes to the block definition no longer affect the exploded elements.

51.11 Deleting a block definition

The **wastebasket button** removes the definition from the panel. That is only possible when **no** instance is left in the drawing. So first delete or explode all occurrences.

51.12 Dimensions and hatches in a block

When creating a block, the following are included:

- the dimensions and hatches selected along with it, as well as
- additionally those dimensions whose reference elements lie completely inside the selection.

These dimensions and hatches are displayed on screen and printed correctly with every instance.

51.13 What you should watch out for

1. **Choose the base point deliberately** – it determines how the block can be inserted, rotated and scaled.
2. **Changes act globally** – the block editor changes all instances. For a one-off deviation, explode the instance first.
3. **Give unambiguous names** – that makes selection in the panel and in the parts list easier.
4. **Delete only without instances** – otherwise first remove or explode all occurrences.
5. **Exploding is final** – the link to the definition is severed permanently.
6. **No block inside itself** – a block cannot contain its own definition.
7. **Rotated instances** – automatically created rectangle width/height dimensions show the axis-parallel bounding dimension on rotated instances. In such cases place the dimensions by hand if required.

51.14 Quick reference

Action	Operation
Create a block	Selection → right-click "Create block", or the panel button "Create block from selection"
Insert a block	"BLOCKS" panel → click "+" or the block name
Rotate while placing	R or Ctrl + mouse wheel
Scale while placing	+ / −
Set the base point anew	B (while placing)
Finish placing	ESC or right-click
Change the geometry	Angle button (block editor) → Save
Parts-list data	Pencil button or "BOM data…"
Dissolve a block	Right-click → "Explode"
Delete a definition	Wastebasket button (only if unused)

51.15 The block manager

The block manager is a window of its own that you can move freely while you carry on drawing. It shows all the block definitions of the drawing as a tree: blocks contained in other blocks appear indented under their parent block.

[FIGURE: block manager with the tree expanded]

At the top level, the number next to a block means the number of placements in the drawing; for a sub-block it means how many times it is contained in the parent block.

What you can do here:

• Search — also by details from the parts-list data, not only by name.

• Hide via the eye symbol. That acts on the definition, and therefore on all placements; the dimensions and hatches of the block disappear with it.

• Jump to it: a click on a row makes the placements concerned flash briefly in orange in the drawing.

• Edit, rename, delete. If a block cannot be deleted because it is still in use or is contained in another block, the button is pale and states the reason.

The panel on the right and the block manager show the same details; whatever is selected in the drawing is marked in both lists:


BLÖCKE
Block aus Auswahl erstellen
Block laden (.ldblk)
Stückliste (BOM)
Blockverwaltung
Zusatzprogramme
1×
1×
1×
1×
1×
1×
1×
1×

52 Bolt circle with the first hole at 75°

Seven M8 tapped holes are to be positioned evenly on a pitch circle with a diameter of 80 mm. The first hole lies at 75° on the intersection with the pitch circle and is patterned in a circle 8 times from there.

Activate layer: **ACHSEN**.

<u>Switch</u> on the grid snap [G].

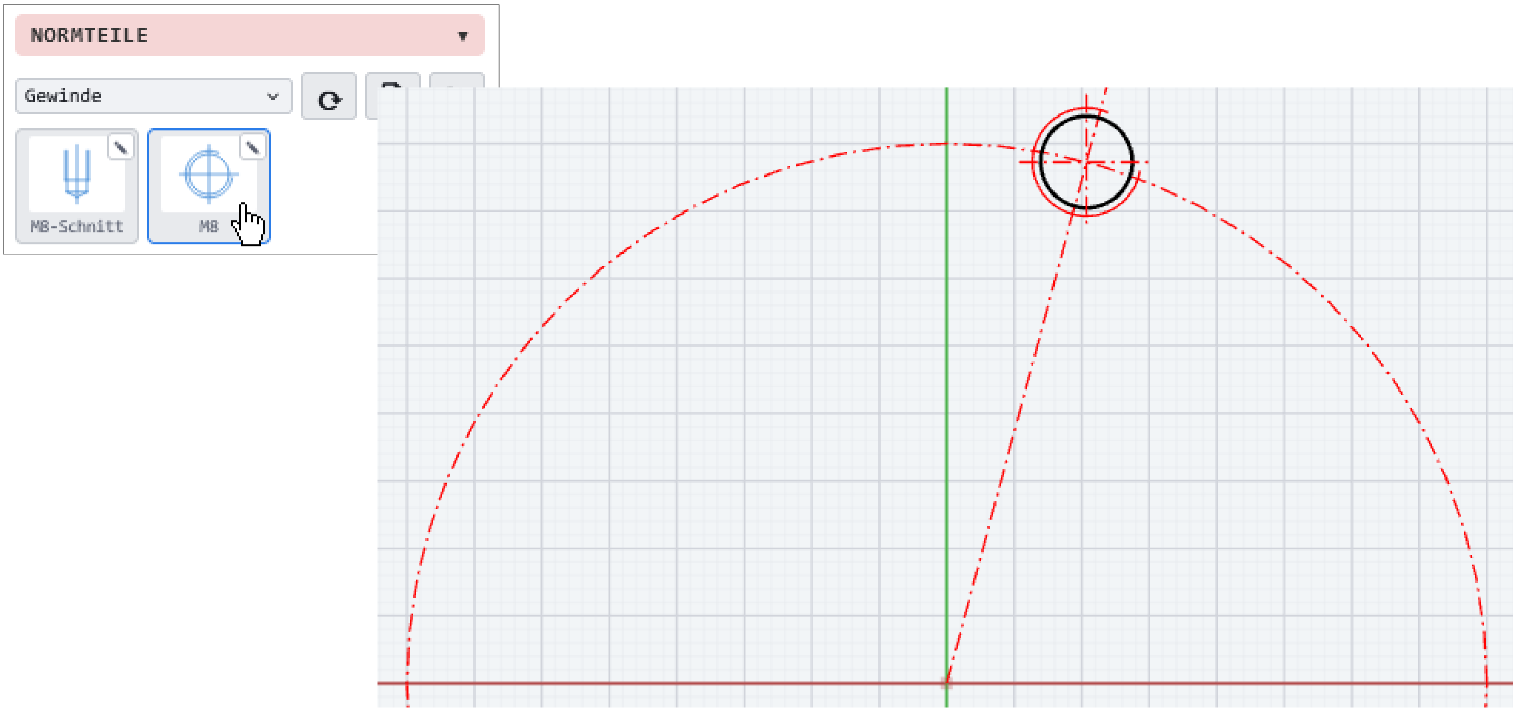
Function: **Circle → grid snap [G] → radius (R40)** placed at the origin (NP).

Activate the **automatic snap [A]**

Function **Line →** from the **circle center**

→

STANDARD PARTS → thread → M8 → place on the intersection. → 2x [**Esc**]



LiberoDraft has a **pattern function**. Functions for rotating elements:

1. Rotate **with self-rotation** of the elements being patterned.

2. Rotate **without self-rotation** of the elements being patterned.

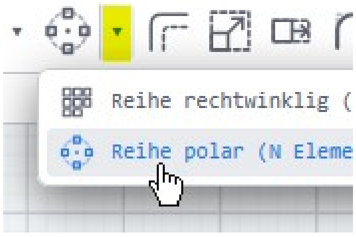
Below are two examples of what that means…

Related terms: array rectangular, array polar, pattern, circular pattern, rotation pattern

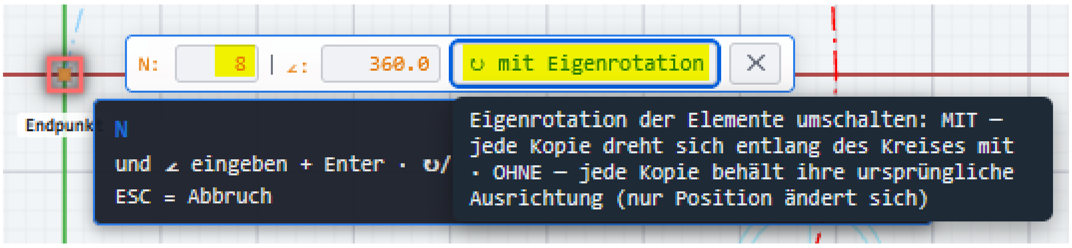
Info: Before the function can be carried out, the elements to be patterned have to be selected! In select mode, drag a rubber band over the thread.

Function: **select mode →**  select the thread

Function: **Array polar →**



Snap: **Automatic [A] →** click the center of the pitch circle

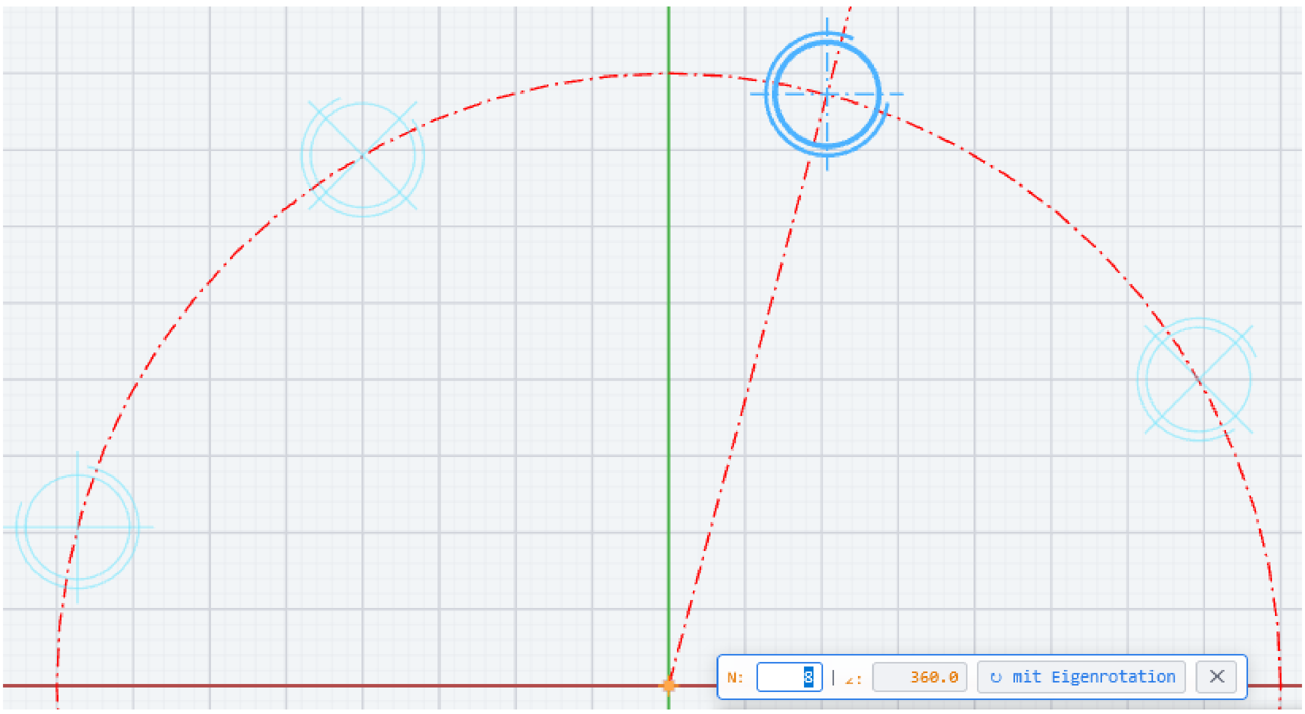


Enter the number N: **8**, distribute over an angle of **360°** and **with self-rotation.**

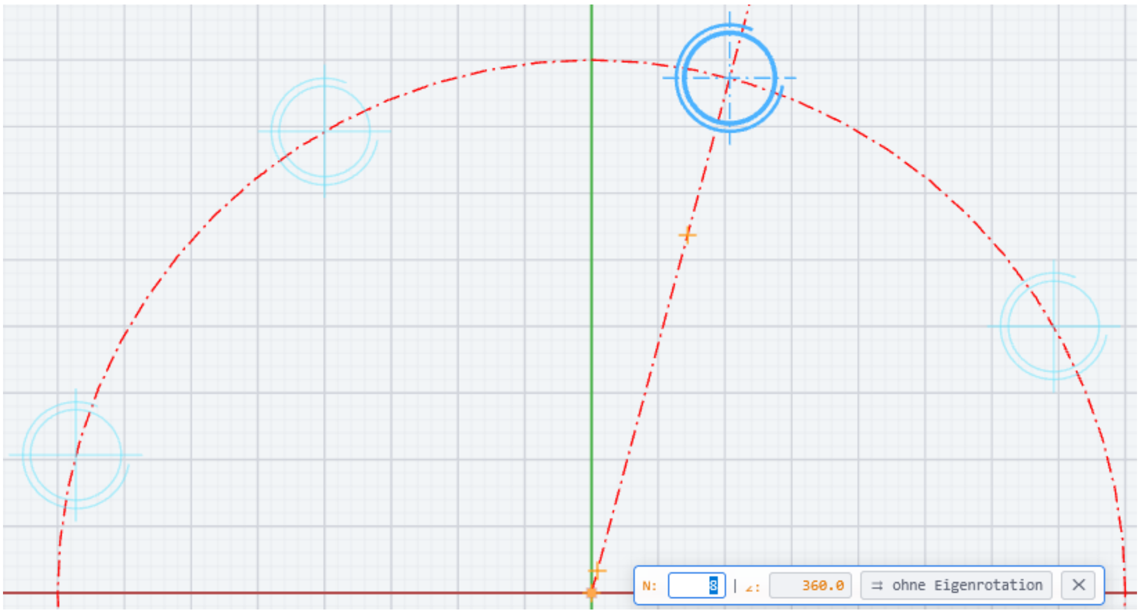
Switch the **with/without self-rotation** button over and watch what happens.

Entries are shown directly in the preview and carried out with **2x [Enter]**.

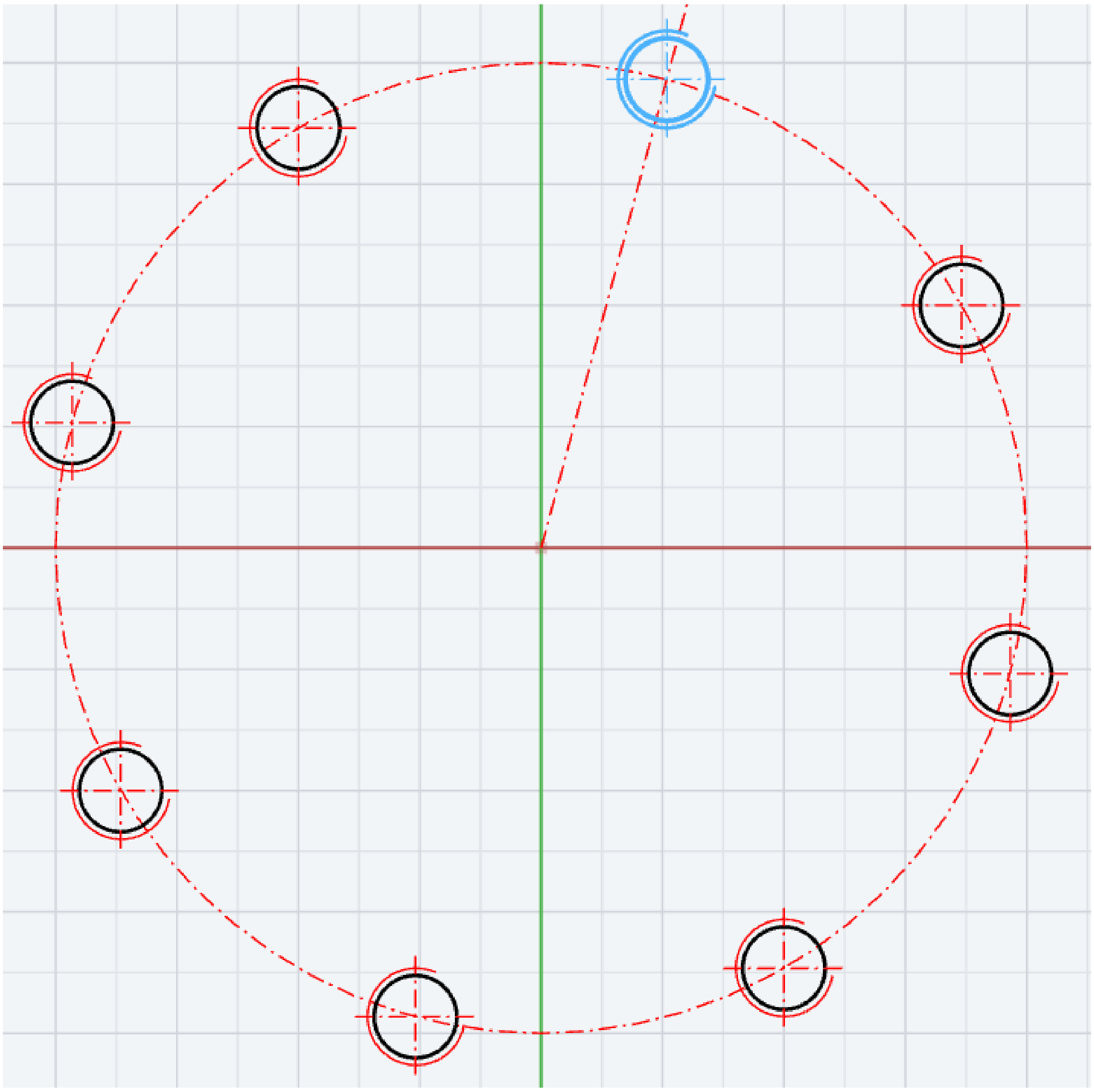
1. Circular pattern <u>**with**</u> self-rotation of the elements:



(2) Circular pattern **without** self-rotation of the elements:



In our case this variant is perfect!



53 A draw hook as an example

Using the example of a draw hook, we will show how to draw a workpiece with circular transitions properly.

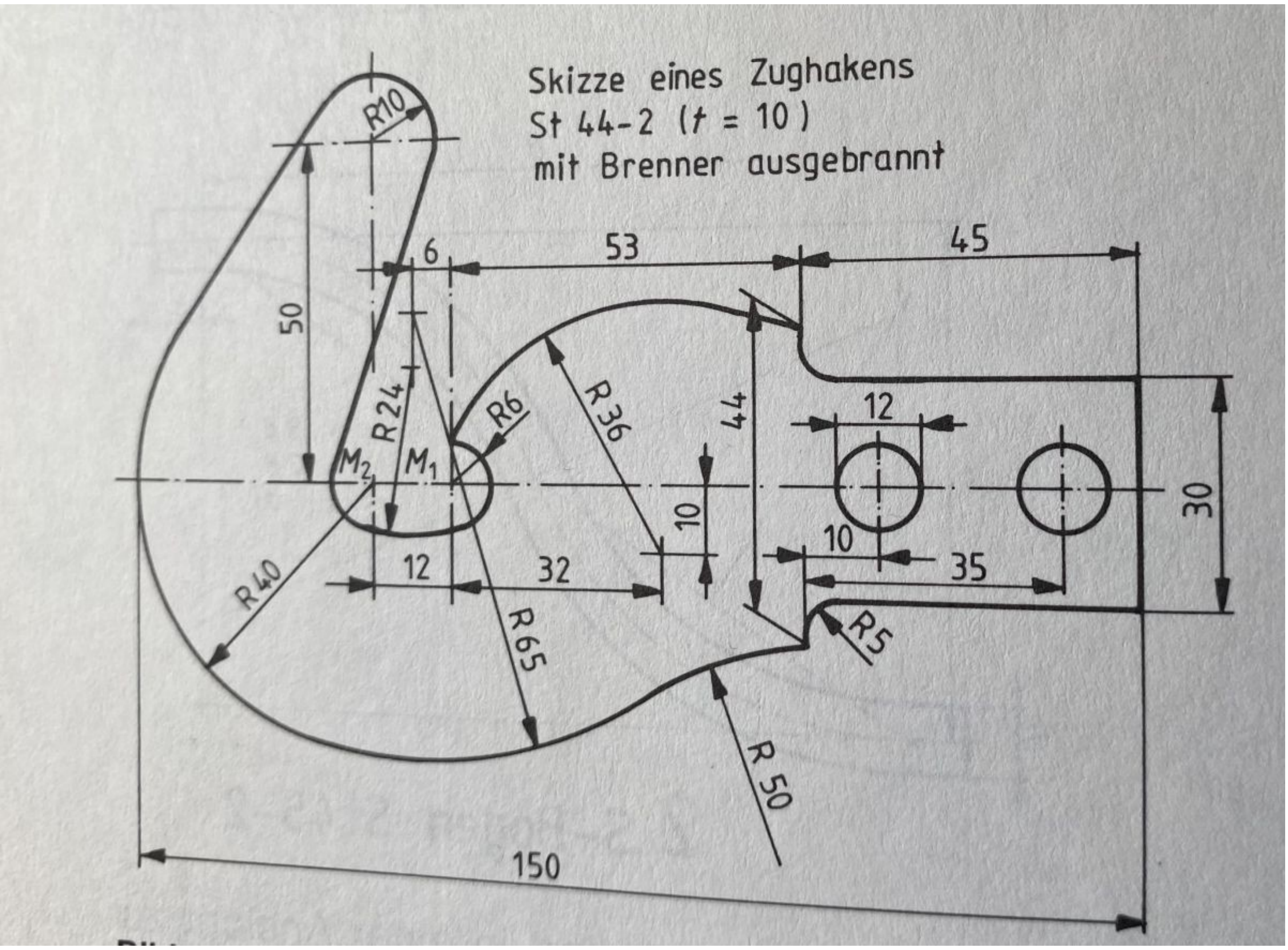


Diagram 4: source: Zeichnen für technische Berufe HT 3214

The starting point for the dimensions are the center lines running through M_1. That is why all dimensions related directly or indirectly to the axes are first defined by center lines or guides...

1. It is best to start with the horizontal axis and create a pair of axes through M_1.
2. Define all dimensions measured horizontally by means of center lines or guides.
3. Define all dimensions measured vertically by means of center lines or guides.

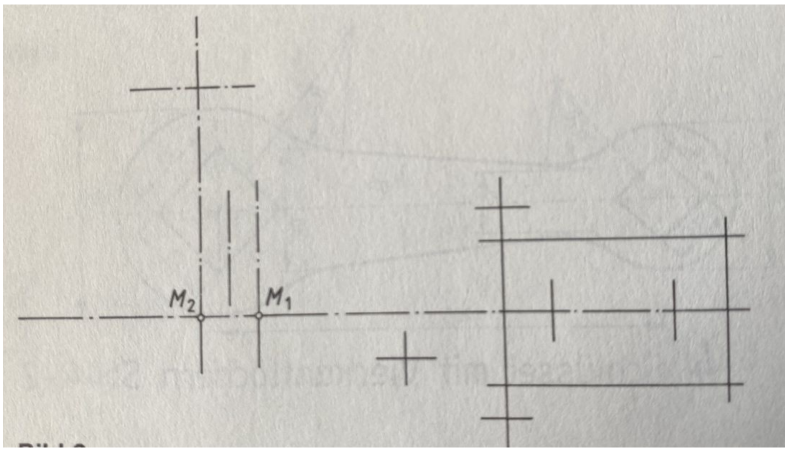


Diagram 5: source: Zeichnen für technische Berufe HT 3214

4. **Strike arcs with r=6 about M_1 and M_2.**

5. Determine M_3: set the compasses to 24-6=18 and strike arcs about M_1 and M_2.

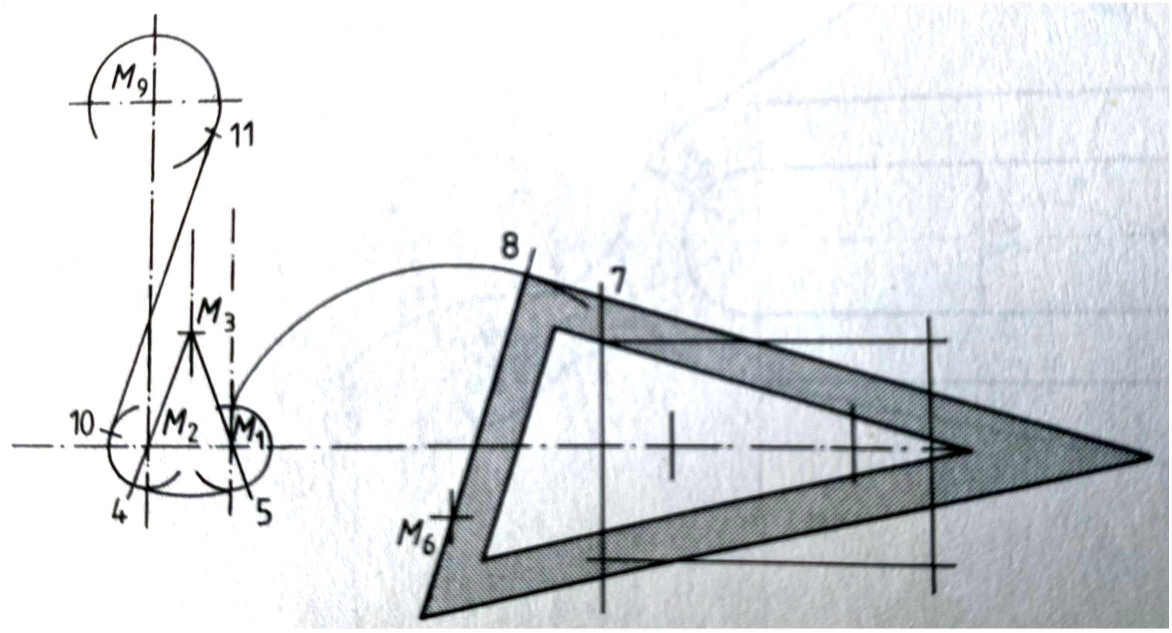


Diagram 6: source: Zeichnen für technische Berufe HT 3214

6. Determine transition points 4 and 5 by joining M_3 to M_1 and to M_2 and extending the line beyond M_1 and M_2 respectively.
7. Strike an arc with r=24 about M3.
8. Strike an arc with r=36 about M6.
9. Draw the tangent from point 7 to the circle and determine transition point 8.
10. Strike an arc with r=10 about M_9.
11. Draw the tangent to the circles M_2 and M_9.
12. Determine transition points 10 and 11, or apply Thales's theorem.
    See chapter 39.1 Thales's theorem

13. **Strike an arc with r=40 about M_2.**
14. Draw the tangent to the circles M_2 and M_9.
15. Determine transition points 12 and 13.
16. Determine M_{14} by striking the arc about M_2. Determine transition point 15.
17. Strike an arc with r=65 about M_{14}.
18. Determine M_{16} by striking the arc about M_{14} and the arc with r=50 about point 17.
19. Determine transition point 18.
20. Strike an arc with r=50 about M_{16}.

21. Determine M_{19} and M_{20} and the corresponding transition points by means of the parallel at a distance r=5, and strike the arcs.

22. Strike circles with r=6 about M_{21} and M_{22}.

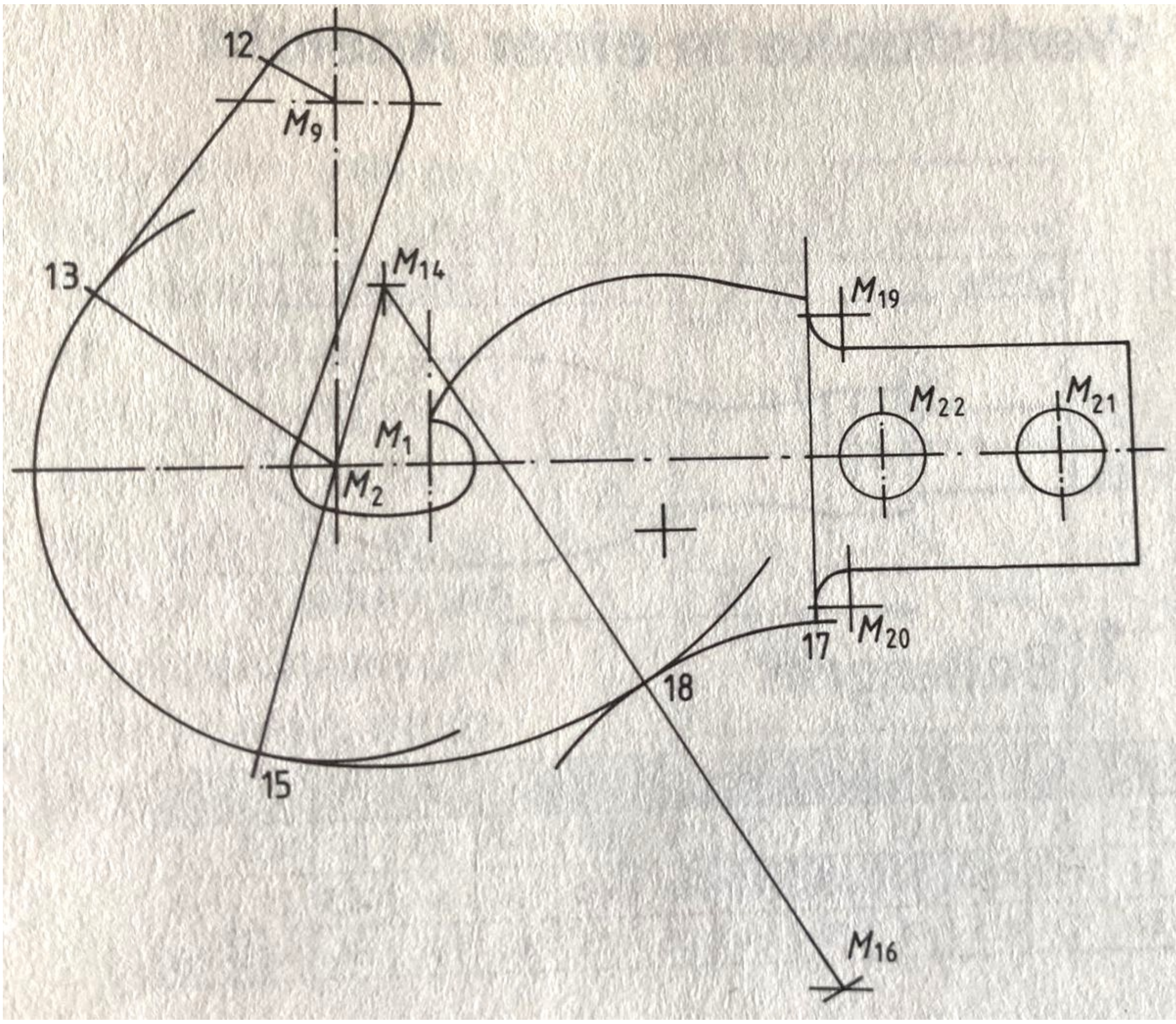


Diagram 7: source: Zeichnen für technische Berufe HT 3214

23. Trim away superfluous and protruding lines and arcs.

    Tip:

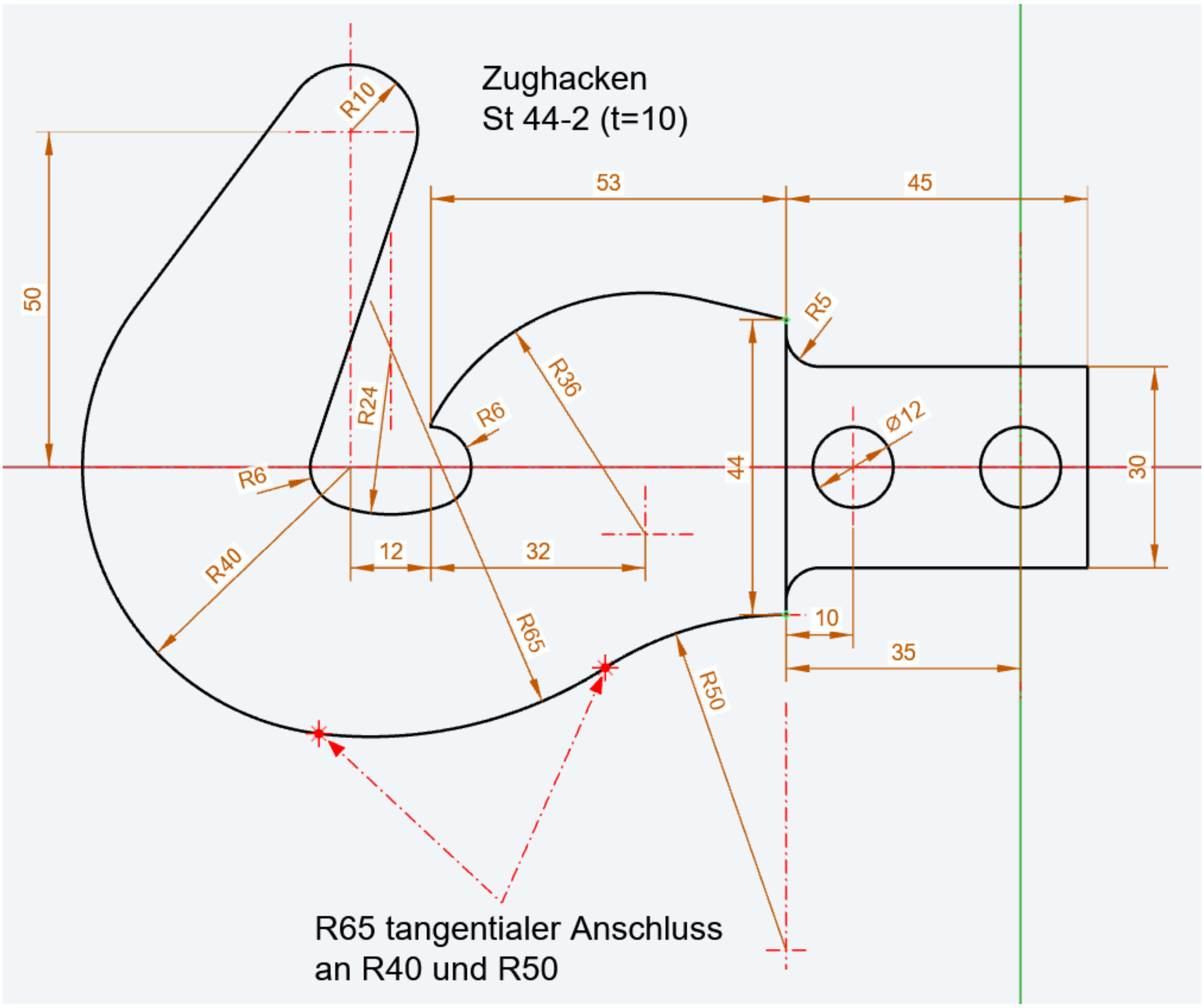
    In rare cases the trim or break function may not work.

    Should that be the case, we recommend drawing in an additional guide and then trying it again. It usually works then.

24. Draw in the extension lines, dimension lines and dimension arrows as in the template.

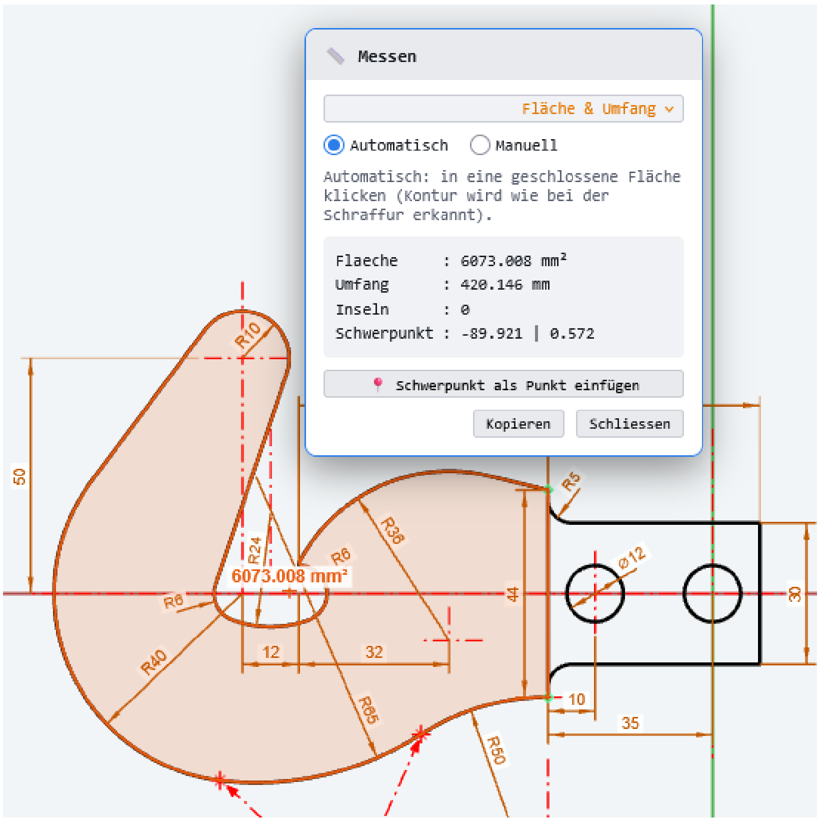
25. Enter dimension values 3.5 mm in size.

The drawing is now finished!



For completeness I quickly redrew the drawing in LiberoDraft.

Also interesting is the calculation of the centroid, which can be confirmed quite nicely at the projected position.



54 Chuck jaw plate with an M8 thread

What you have learned is practised right away in the following task. :-)

You have received this sketch:

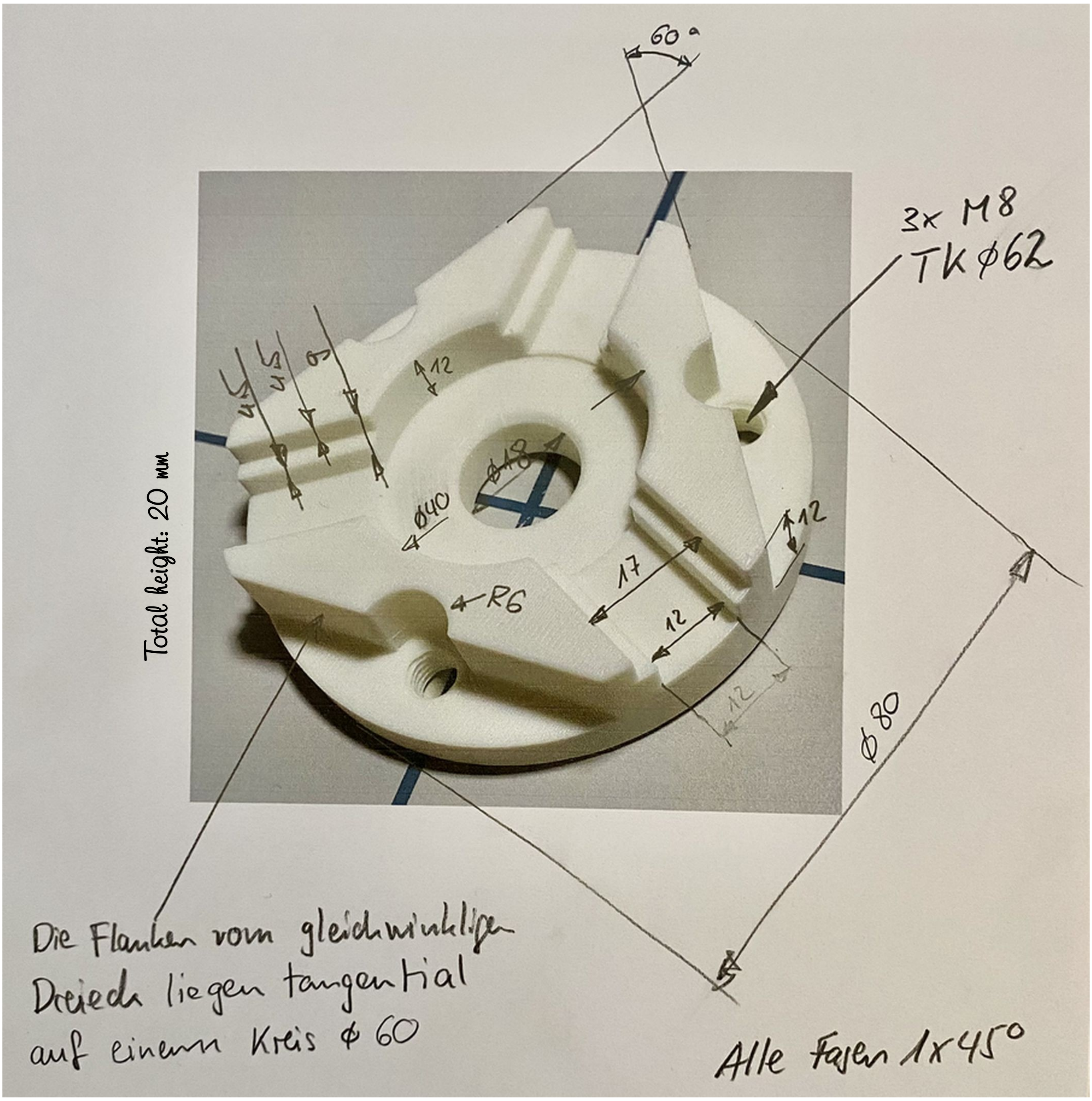


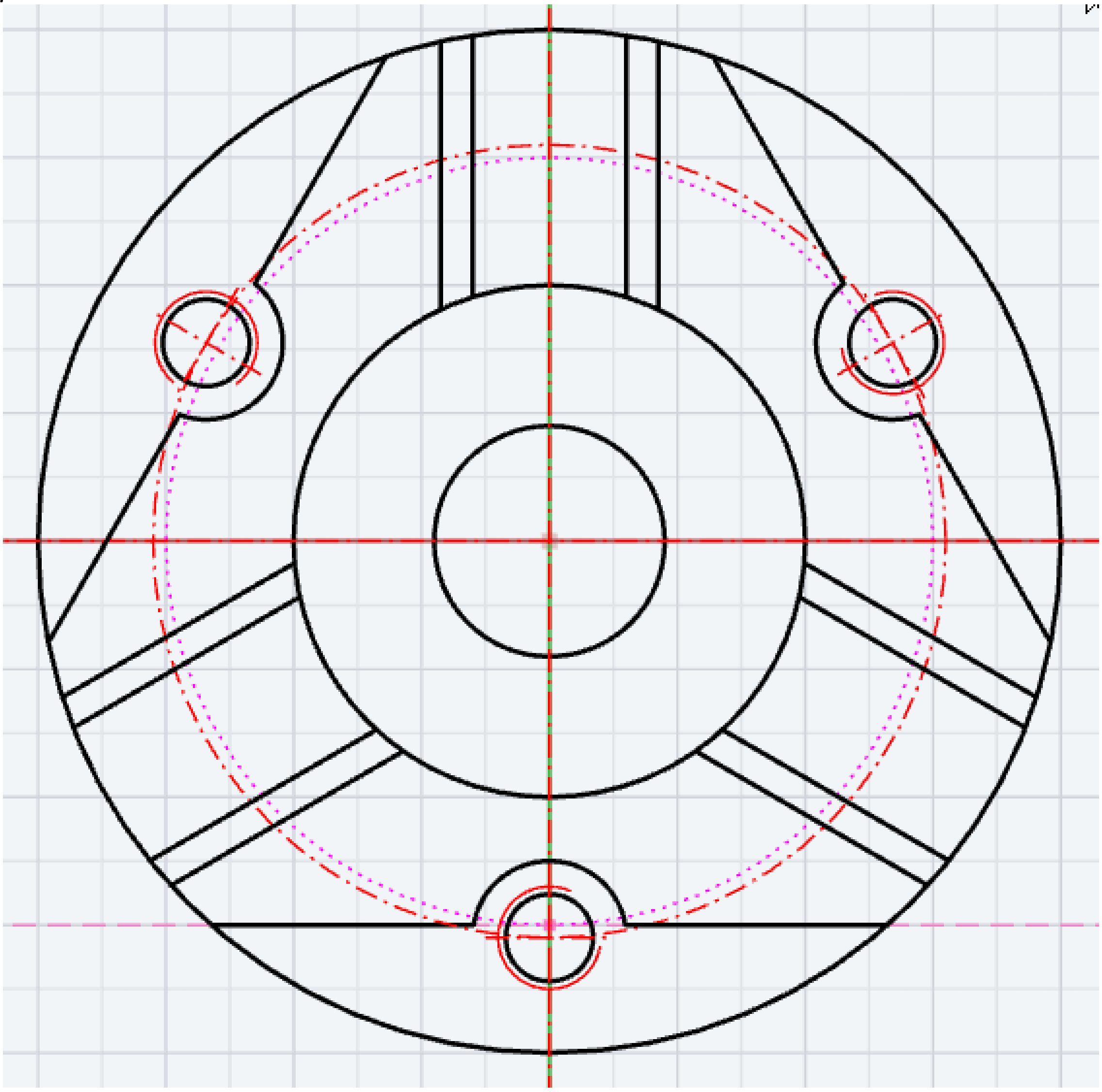
Diagram 8: photo of a part of which no drawing exists any more; the dimensions were sketched by hand.

Task:
Create a side elevation, a front elevation and a plan view, as well as a vertical section through the plan view. Place the section on the right-hand side of the plan view and hatch the cut faces.

My solution variant:

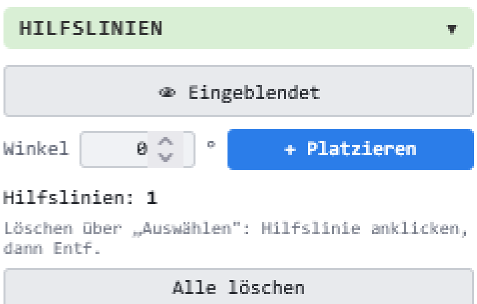
First you draw the plan view (top view) quite normally.

Tip:
*It makes sense to use the **Array polar** function several times here in order to duplicate elements in a rotation pattern:*



Carry on with the front elevation...

Remember that with the function **Guides** in the panel on the right you can and should project edges!



Based on the guides, you can draw further elements in other views.

54.1 Proceed in segments when projecting

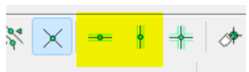
That is a good strategy for keeping an overview while projecting lines in LiberoDraft. If you project too many lines at once, things can quickly become confusing, especially when the lines lie close together.

By projecting only a few lines, you can make sure that you can place the lines precisely and that they are not accidentally joined to other lines. After drawing on the projected lines, they can be deleted to make room for the next ones.

This approach may require a little more time and effort, but it helps to avoid errors and to achieve a more precise result. It is also advisable to save regularly and to secure intermediate steps in order to make sure that you do not lose your work.

54.2 An alternative to projecting

Alternatively these two snap modes can be used:



With these two snap modes you can fix distant points in the horizontal or vertical alignment. This method makes projecting partly superfluous.

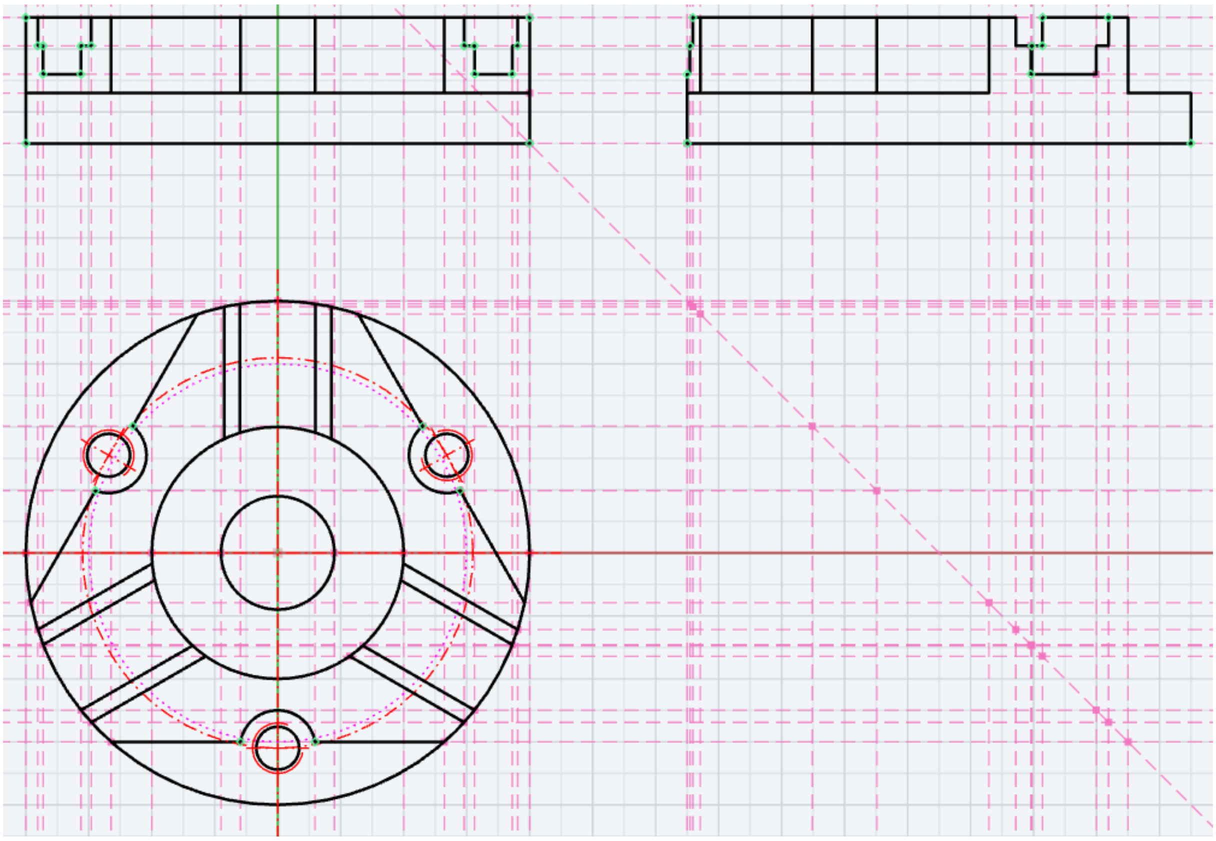
54.3 Orthographic projection

Orthographic projection is a method of representing three-dimensional objects on a two-dimensional surface. The three dimensions of the object are mapped onto two dimensions by viewing the object from a particular direction and projecting it onto a plane.

It is important to choose the projection lines carefully and to specify the direction of projection correctly in order to obtain an accurate representation of the object. It can also be helpful to use guides and construction lines to determine the position and orientation of the projections.

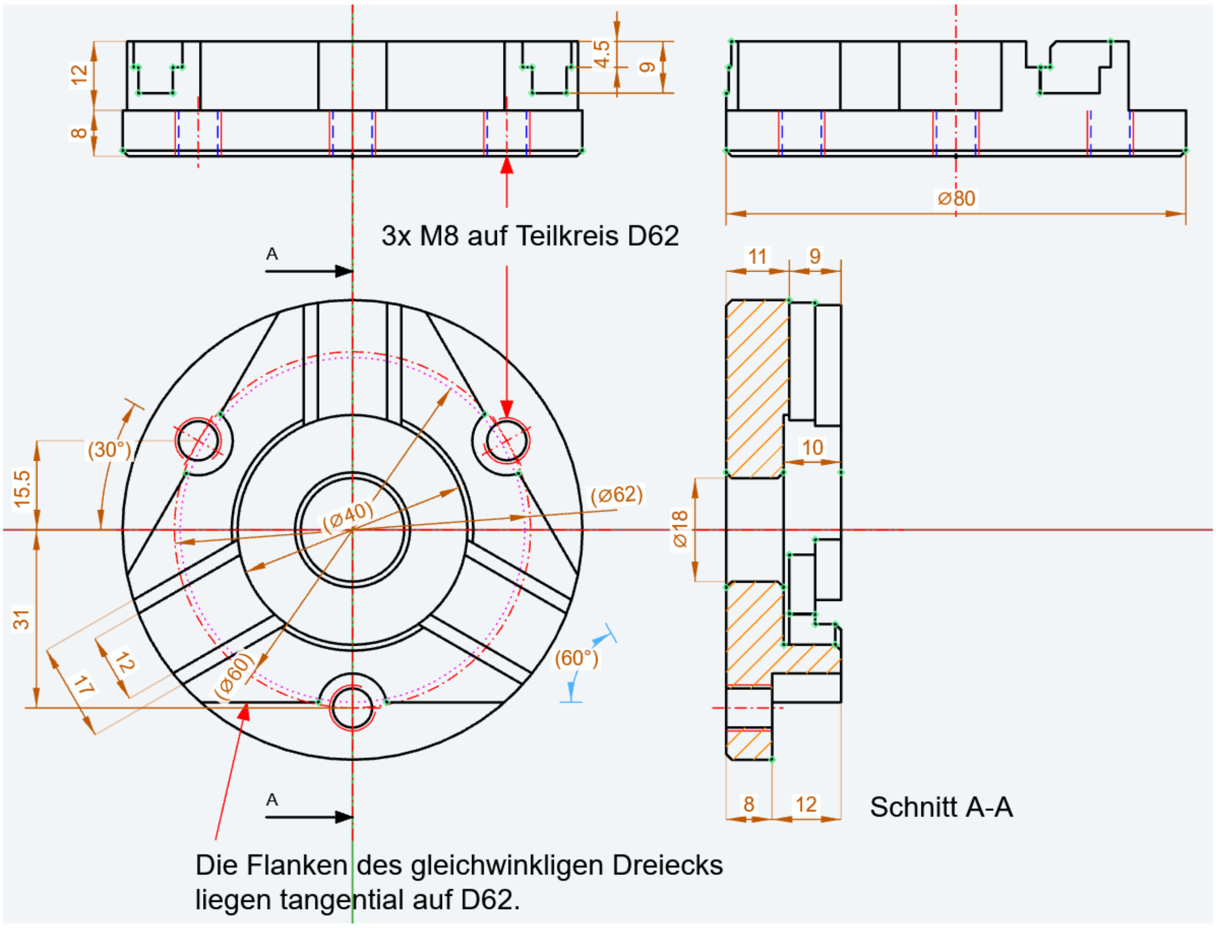
To sum up, orthographic projection is a method of representing three-dimensional objects on a two-dimensional surface. The three dimensions of the object are mapped onto two dimensions by viewing the object from a particular direction and projecting it onto a plane.

Orthographic projection of the visible edges:



54.4 Creating a section view

Vertical section A-A through the whole part with hatched cut faces:



55 Downloading standard parts

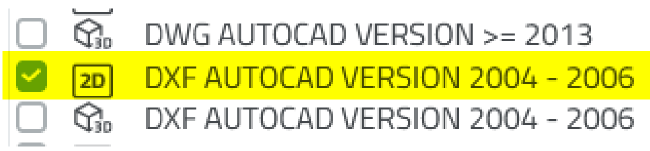
To download a screw as a 2D DXF standard part from the BOSSARD screw catalog with the web service www.3dfindit.com, you can carry out the following steps:

1. Open the website www.3dfindit.com in your web browser.
2. Register at the top right of the page.*
3. Submit the form and make sure you use a **valid e-mail address!***
4. Confirm your registration in your e-mail account *

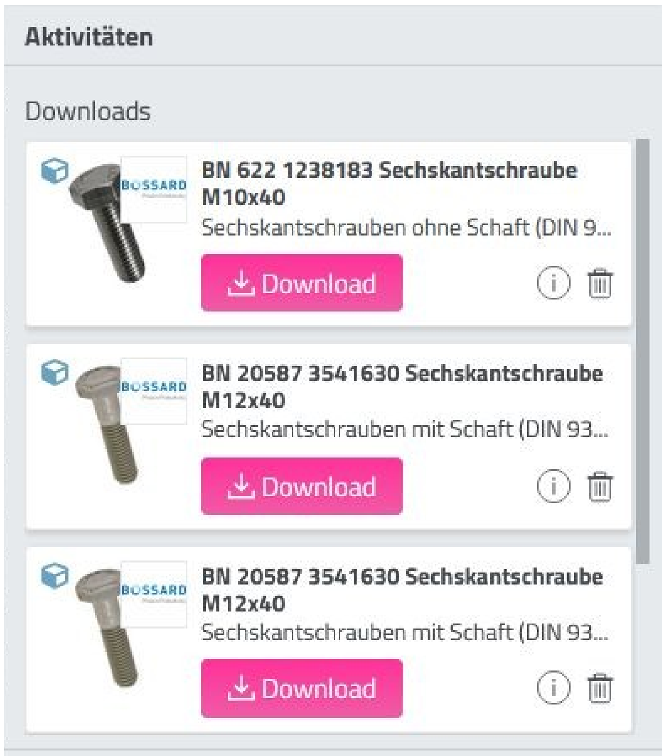
   * *These steps only have to be done on first contact.*

5. In the search field, enter the name of the screw you want, for example "**BOSSARD screw M10**".
6. In the search results, choose the screw you want and click the download button...

7. On the download page you find various formats. Choose **DXF AUTOCAD VERSION 2004-2006**



8. Click the button «**Generate CAD file**». The DXF files are generated and offered for download in the newly opened window...



9. Click the **Download** button. The 2D DXF file is now downloaded as a ZIP file.

   This ZIP file contains all three views of the screw as DXF files. With a double-click on the ZIP file you can extract the view you want.

   Standard part libraries should always be stored in a central place where everyone has access. More on this in the chapter 50.2 Saving the standard part as M8

56 Copying a bushing from a hand sketch

A hand sketch differs from a technical drawing in that all lines (including the circles) are drawn freehand and that no scale is used.

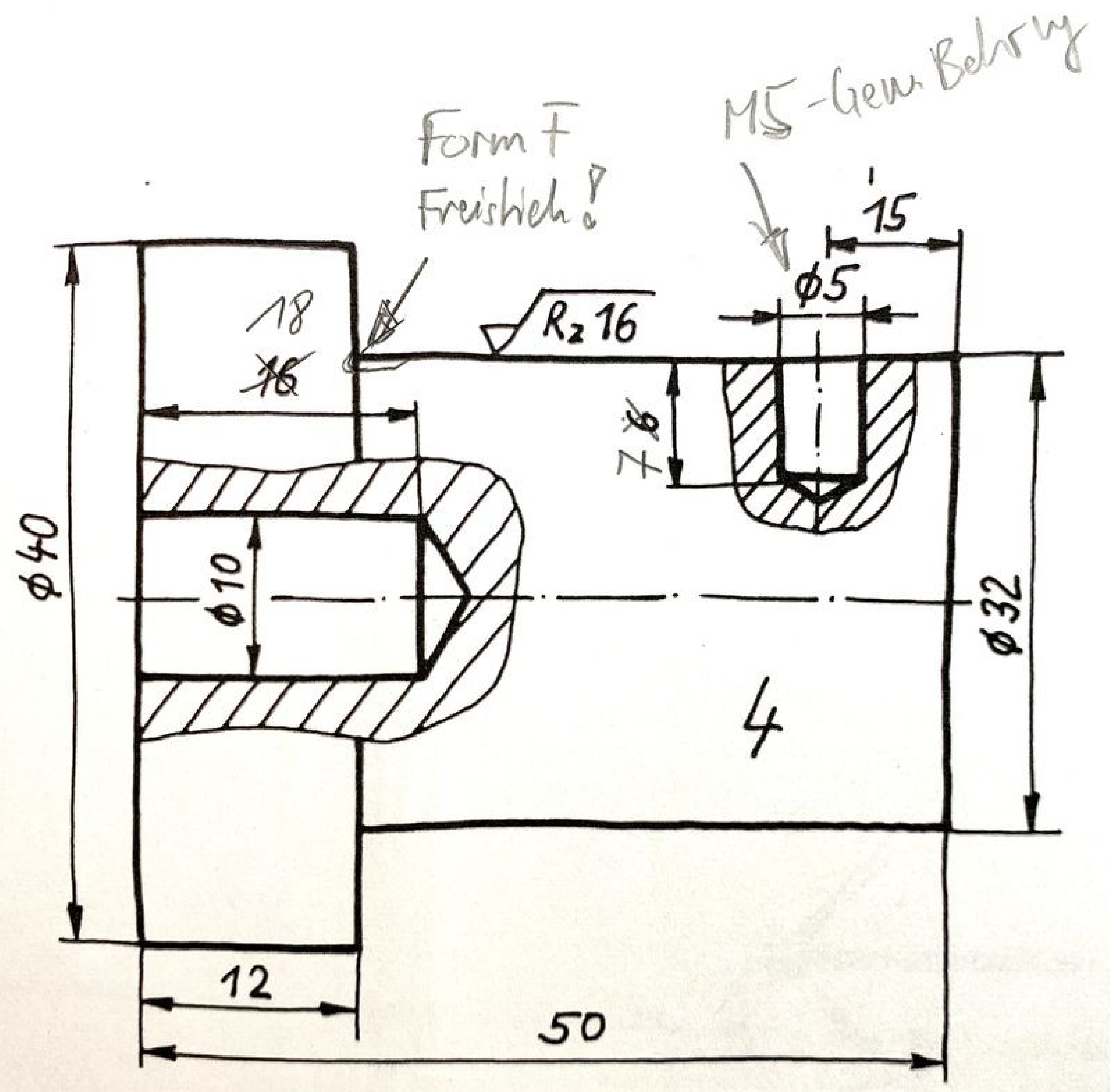


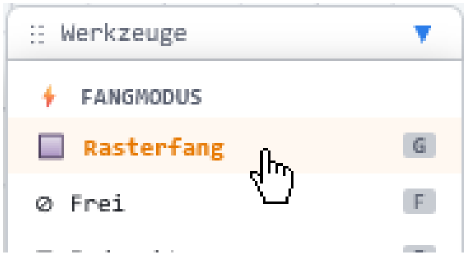
Diagram 9: a CAD drawing is to be created from this hand sketch below.

Open a new drawing in LiberoDraft and check whether the right template with the layer settings was preloaded correctly.

56 Copying a bushing from a hand sketch

This hand sketch shows a turned part.

With a turned part you always start with the axis.



Activate layer: **ACHSEN**.

Activate snap mode **Grid** [G].

Function: **Line →** draw 2 points

Draw the axis from the **origin** about 60 mm to the right.

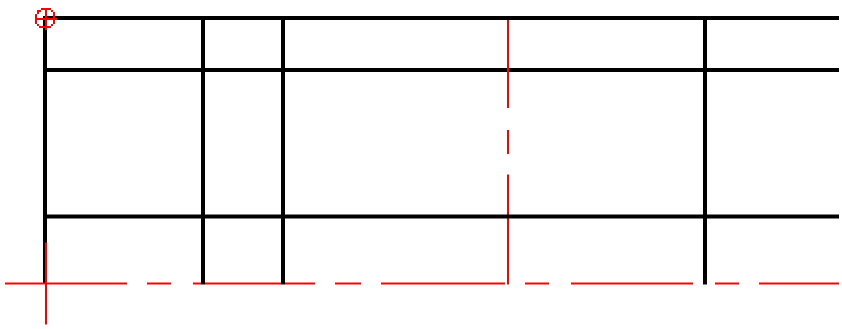
Activate layer: **SICHTBAR**.

Function: **Line →** draw 2 points

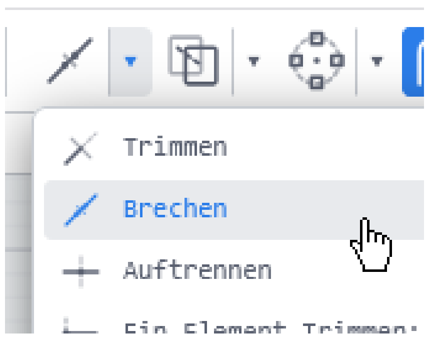
From the origin, drag a vertical line of 20 mm upwards on the grid .

Carry on with the function **Offset** and **draw all the parallels** . The **offset** works very well in **snap mode Free [F].** Take all the values from the sketch.

Select the lines and move to the corresponding side to create the parallel. Enter the value and press [**Enter**].



Now draw all the further **offset** lines until it looks as shown above.
All visible edges as well as the axis of the tapped hole are drawn.



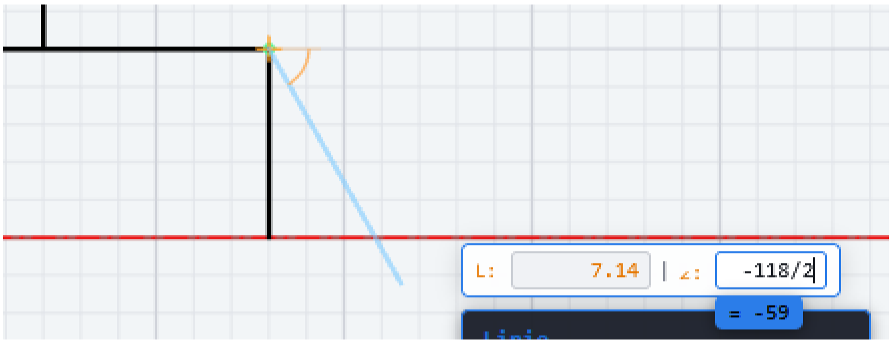
In the next step, superfluous lines are removed with the

Break [B] function...

... until it looks like this:

The hole with D10 is made with a drill that has a point angle of 118°. This is done with the function **Line → polar entry**.

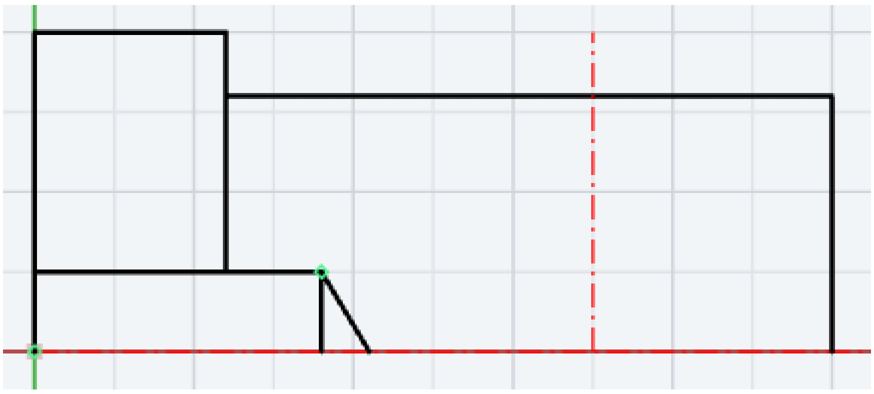
Switch between polar and cartesian with the **[spacebar].**



In the angle field, enter **-118/2** →

[Enter]

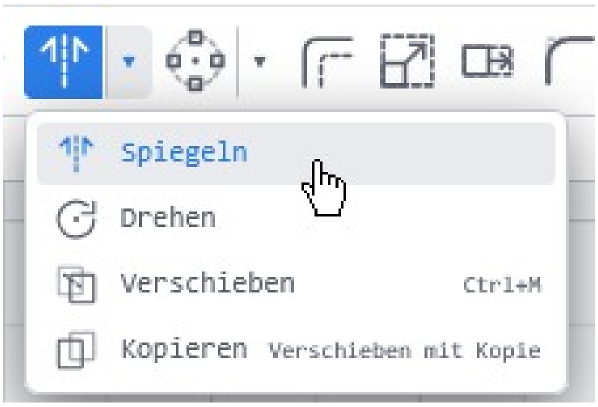
Function: **Break [B]** → break away the overhang.



Now we can mirror all the elements about the horizontal axis.

First **select** all the elements to be mirrored.

Function: **Mirror**



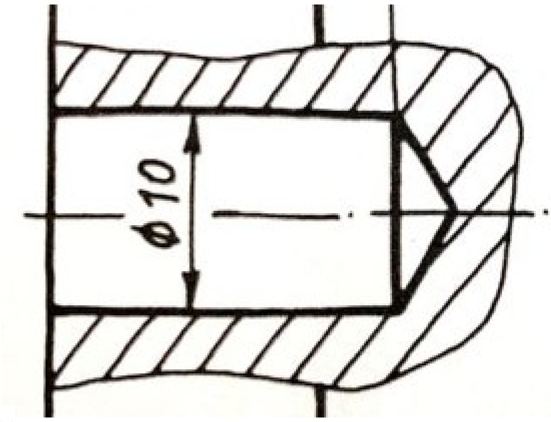
Specify the **first point** of the mirror axis

Click the origin, for example.

Specify the **second point** of the mirror axis.

Done.

56.1 Creating a hatched broken-out section



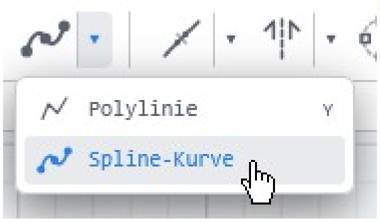
A «hatched broken-out section» at the D10 hole is almost child's play with LiberoDraft :-)

The boundary for the hatch is a spline.

Layer : **KOSMETISCHE LINIE**

Function : **Spline curve**

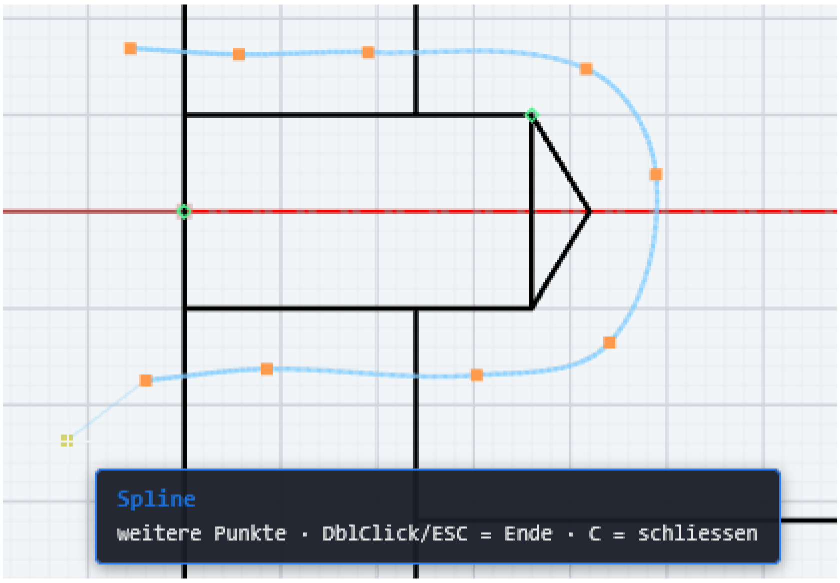
Snap : **Free [F]**



Draw the spline points freely:

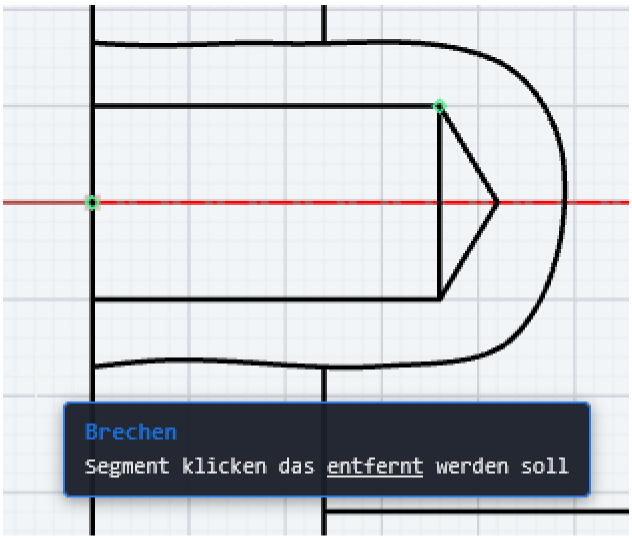
Finish the spline with **[Esc]** or a **right-click**.

Now the two line segments protruding into the hatch area have to be removed.



Function : **Break**

Simply click the segments to be removed in order to break them away cleanly.



Now the area can be hatched:

Function : **Hatch**

Pattern : **ANSI31**

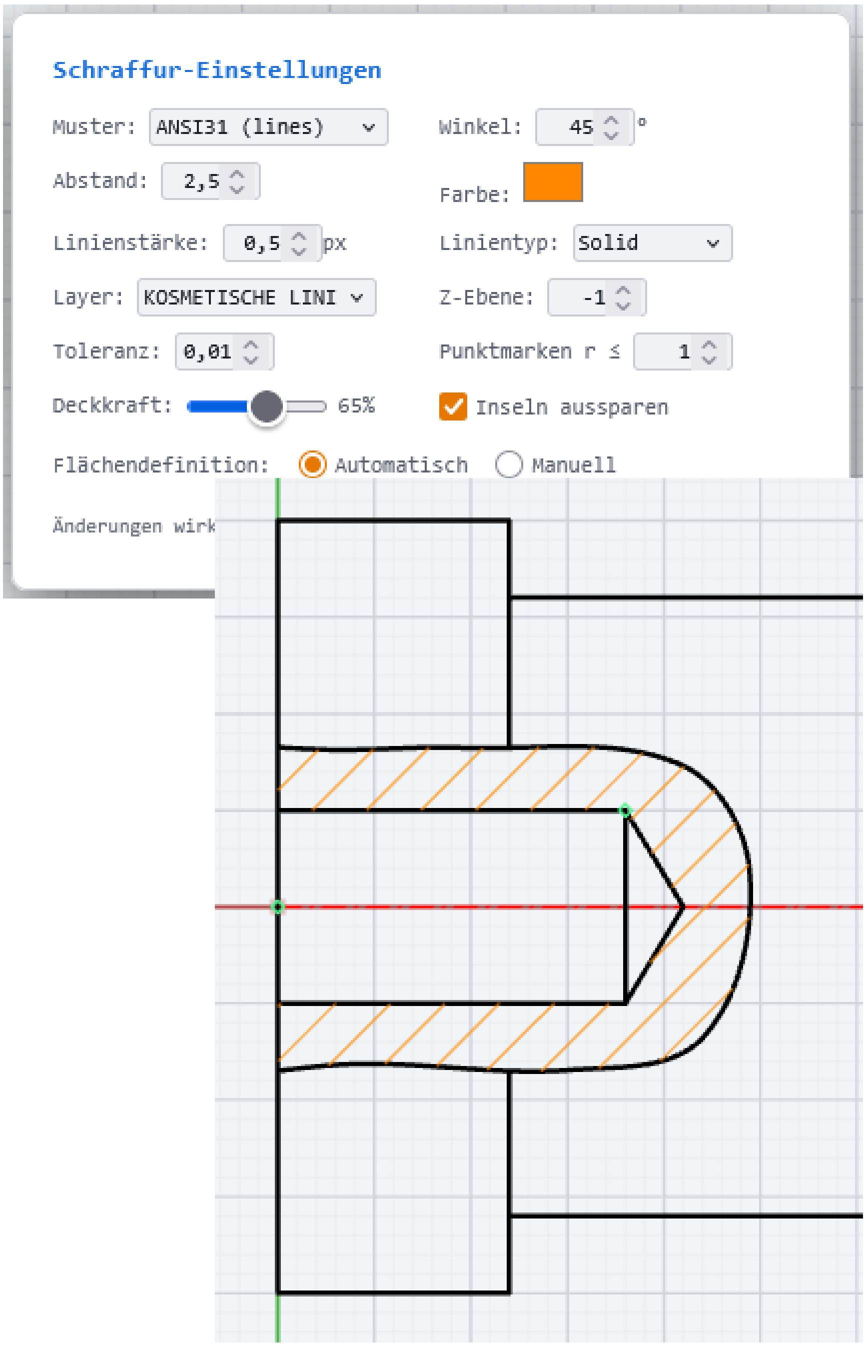
Spacing : **2**

Make further hatch settings to your own taste :-)

Area definition : **Automatic**

Now simply click inside the areas to be hatched.

A perfect hatch :-)



56.2 Inserting a blind tapped hole as a standard part

Now we implement the vertical blind tapped hole:

In the panel on the right, choose:

STANDARD PARTS → thread → M5-Schnitt

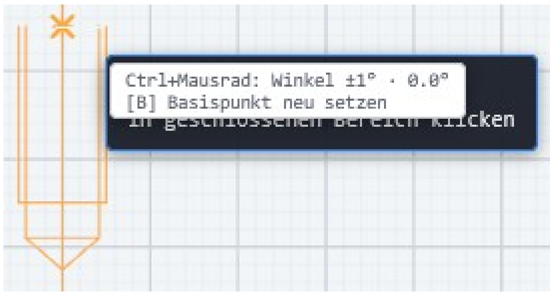
Here you can insert or edit your own standard parts from your NAS.

Choose M5-Schnitt

The thread hangs at the mouse and can be placed.

Snap: **[S]** for intersection or
[A] for automatic

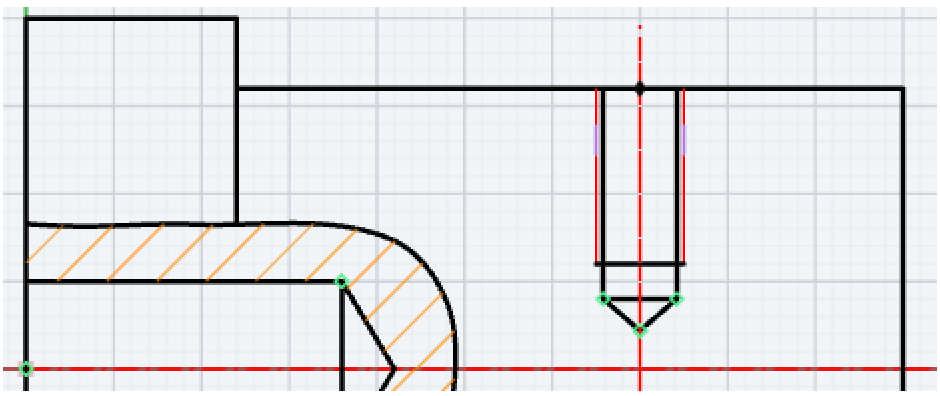
The standard part now hangs at the mouse and can be placed with many options:
[Ctrl] + mouse wheel rotates the elements.
[B] defines a new base point.



And further options are found at the bottom above the status bar:

Simply hover over the individual options with the mouse and read the tooltip :-)

Now place the thread on the intersection:



In the drawing, the drilling and thread depth are defined as follows:

Drilling depth: **7** mm

Thread depth: **5** mm

Our own blind hole threads always have the following initial dimensions:

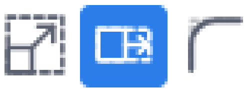
Drilling depth: **15** mm

Thread depth: **10** mm

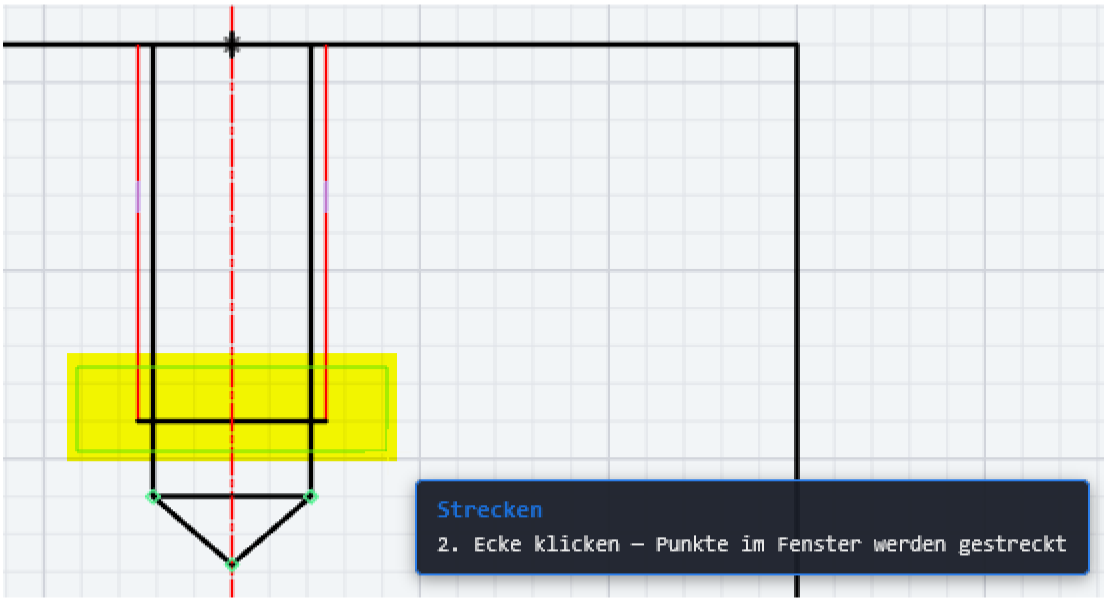
In the existing standard part library we have set the thread depth to 10 mm for all blind threads / blind holes with a thread.

That means we have to adjust the thread and drilling depth accordingly.

That is done very conveniently with:

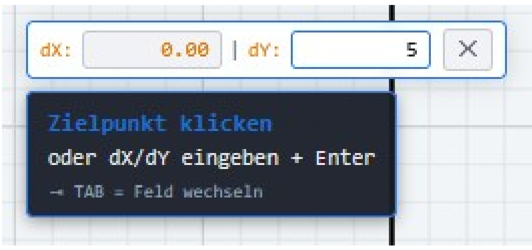
function: **Stretch** 

Snap: **Free [F]**



Frame the area to be stretched.

Define a **base point**. (It can be anywhere.)



For dY enter **5**. (With the **[Tab]** key = change field)

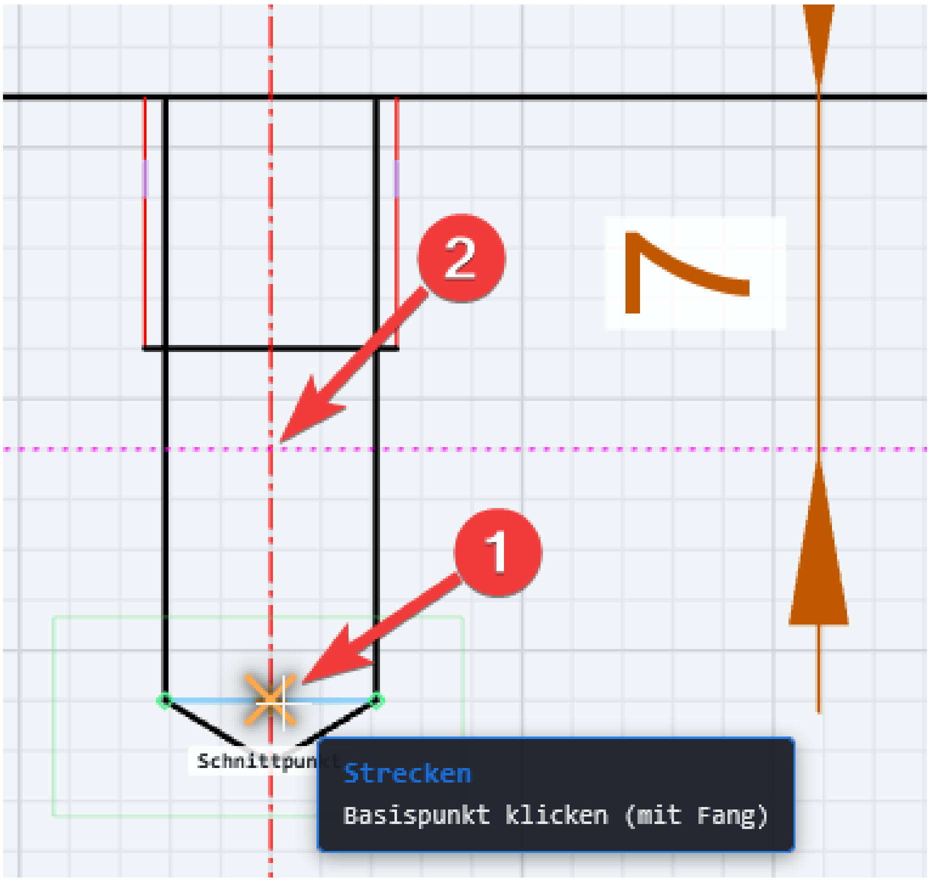
For the drilling depth, first draw a temporary **offset** guide at a distance of **7** mm.

56 Copying a bushing from a hand sketch

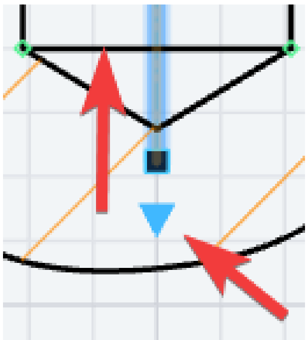
Function: **Stretch**

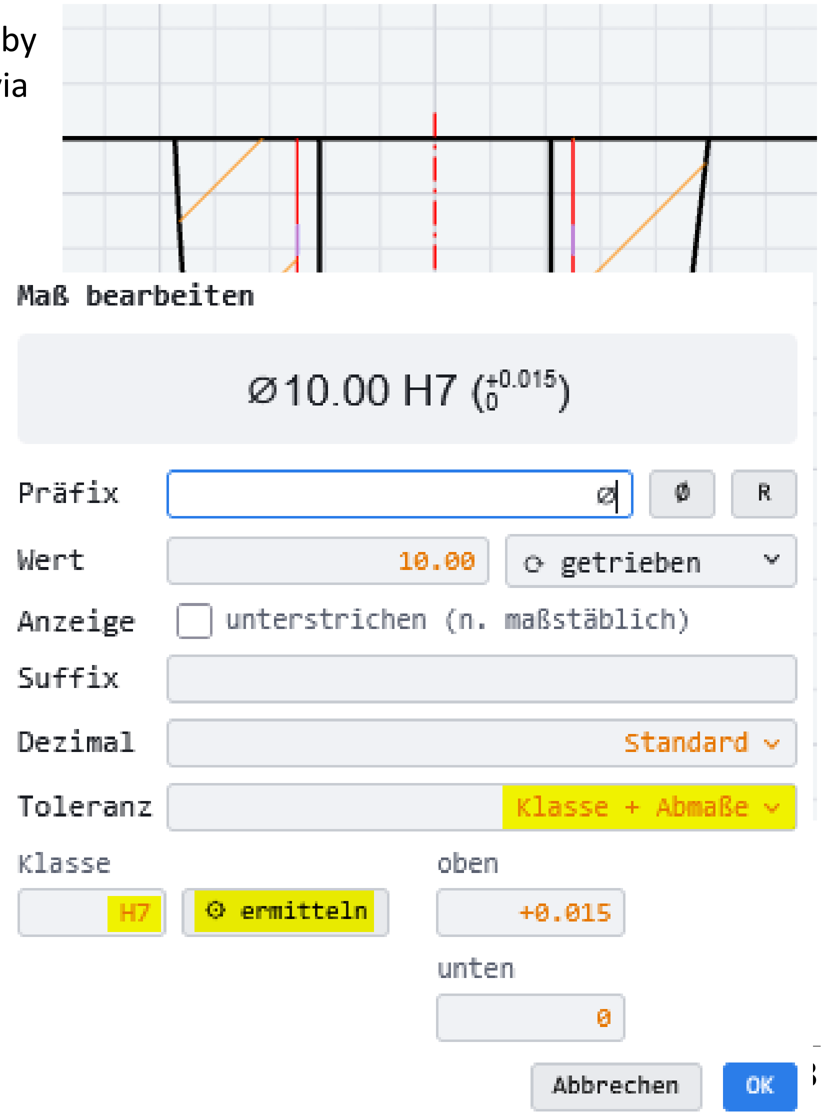
Snap: **Intersection**

For the procedure see the figure →



Shorten the **axis** of the tapped hole by selecting it and trimming it cleanly via the **blue triangles**:





Now draw another **spline** on layer **SICHTBAR** around the tapped hole, break away the overhangs and hatch the area.

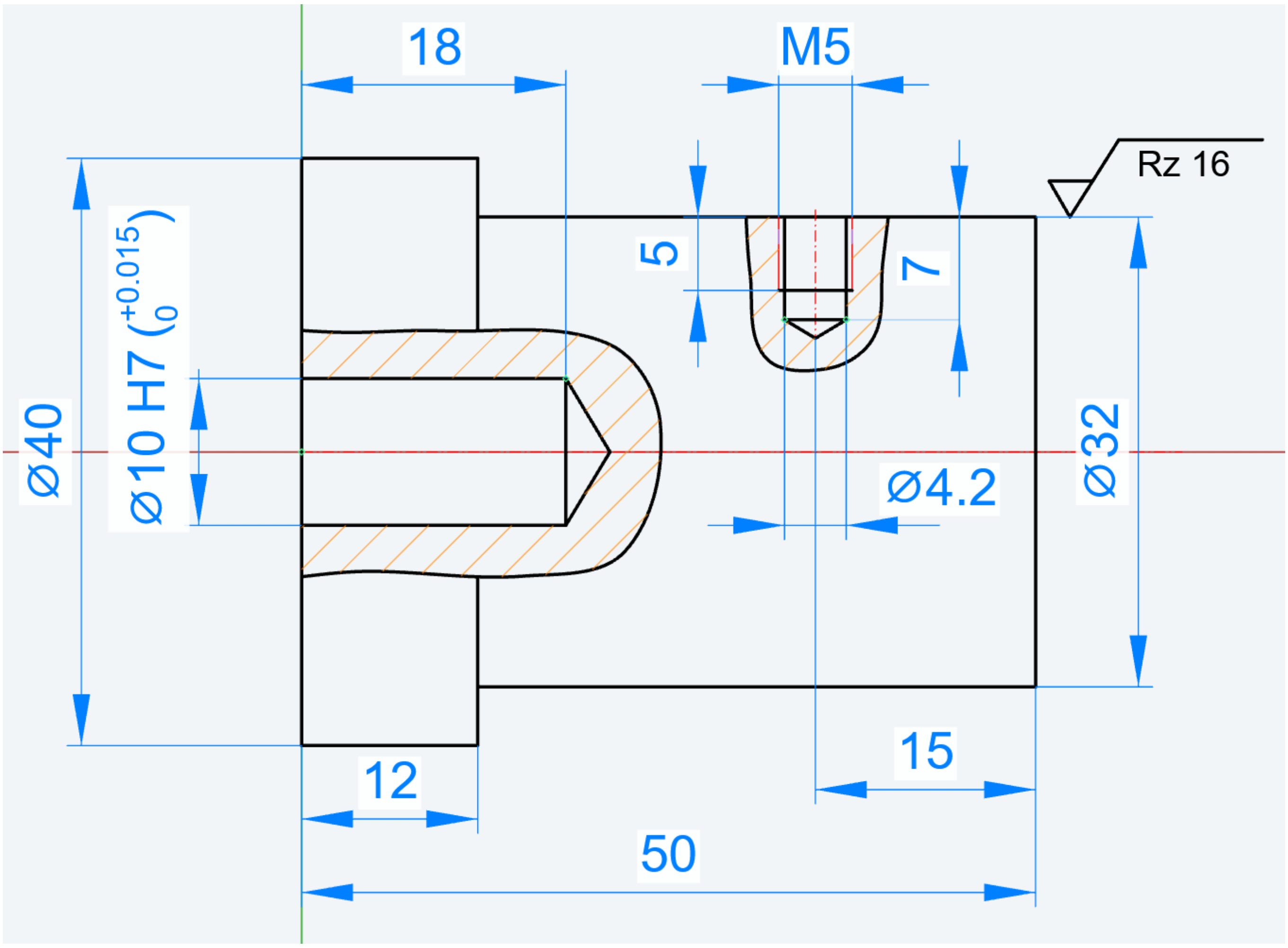
Finally, activate layer **BEMASSUNG** and dimension the drawing.

A **double-click** on a dimension →

Here you can set the properties of the dimension, in particular the tolerances.

A really nice feature is the **automatic** determination of the upper and lower deviations →

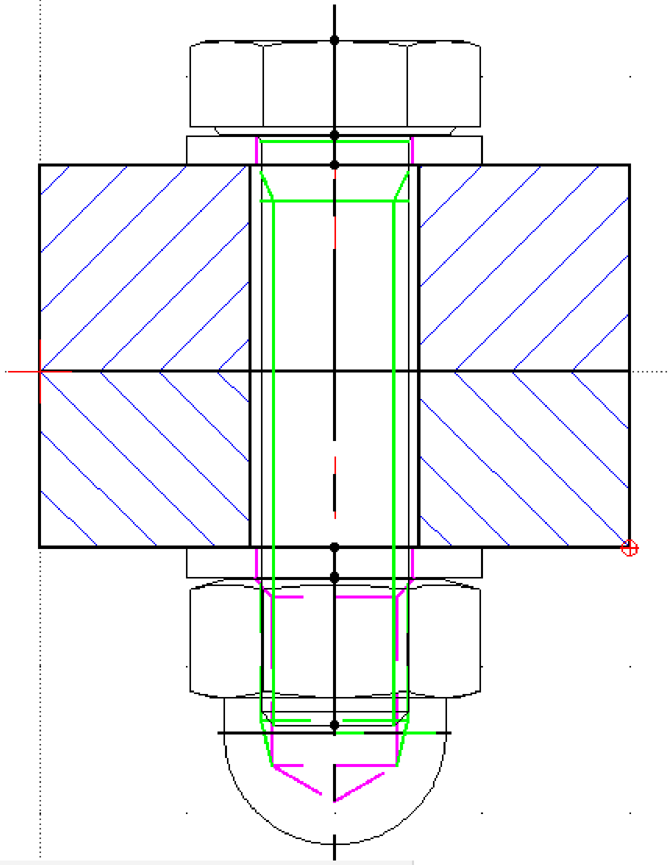
With **OK**, the dimension is supplemented accordingly. Cool!



Done! :-)

57 Creating a bolted joint

A bolted joint usually consists of a screw, one or more washers and a nut. These components are used together to join two or more parts and to ensure a firm connection.



With the **Bossard screw catalog (** www.3dfindit.com) we create the bolted joint shown alongside with two plates, each plate having a thickness of **14 mm**...

The screw used is:

BN 622 1238183

hexagon head screw M10x40

Two washers:

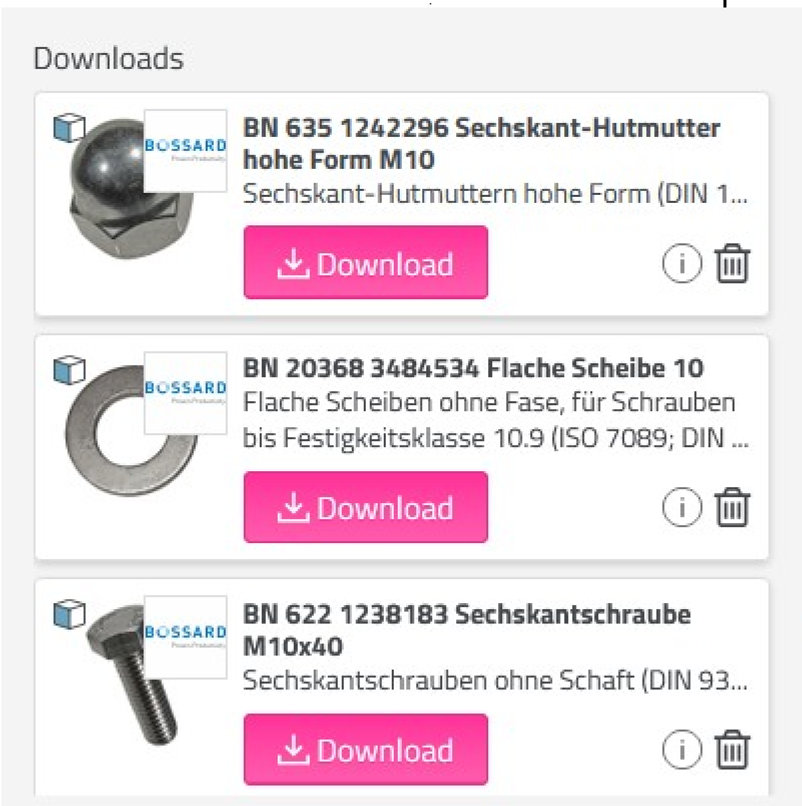
BN 20368 3484534

flat washer 10

Nut:

BN 635 1242296

hexagon cap nut, high type, M10



57.1 The bolted joint step by step

Open a new file and save it right away under the name **Verschraubung**.

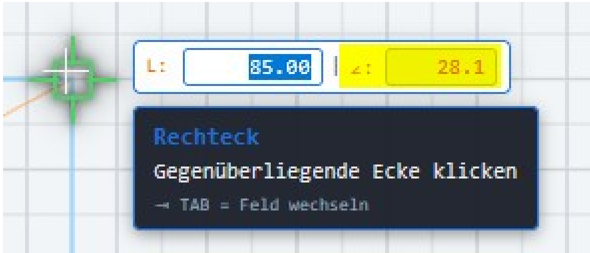
Draw two rectangles measuring 40x14 mm.

Activate layer: **SICHTBAR**.

Function: **Rectangle**

Snap: **Grid [G]**

Draw the rectangle with the **grid snap** and coordinate entry.



If the floating dimension menu is set to polar coordinates, you can switch it to cartesian coordinates with the **spacebar**.

Draw the axis for the hole through the middle and draw in the hole with a diameter of 11.5 mm.

Activate layer: **ACHSEN**.

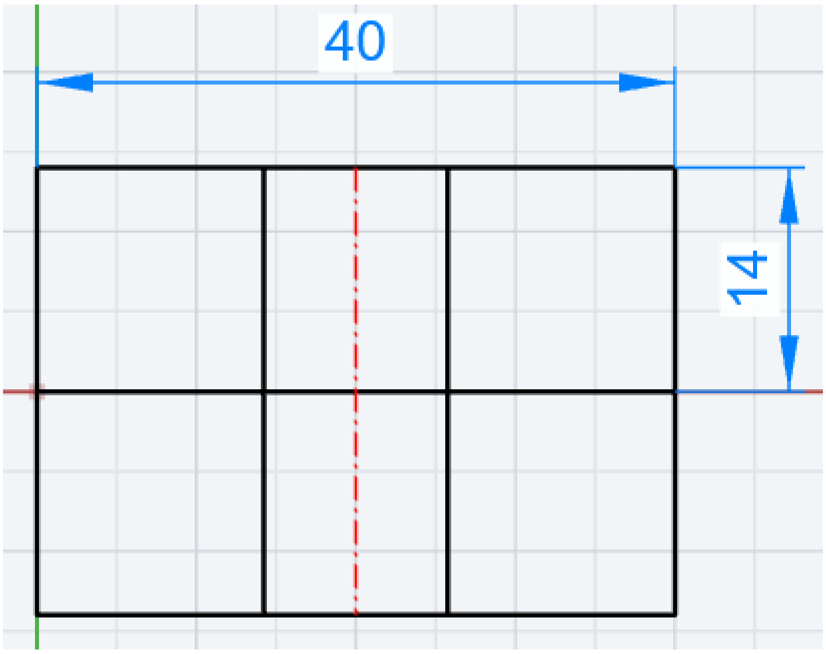
Function: **Line → automatic snap [A]**

Activate layer: **SICHTBAR**.

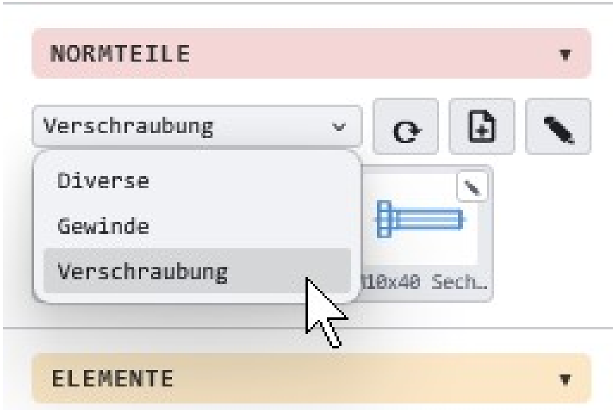
Function: **Offset** draw the hole D11.5.

Tip: *enter 11.5/2.*

Draw the hole on the left and on the right of the axis.



First insert the two washers.



In the **panel on the right**, click →

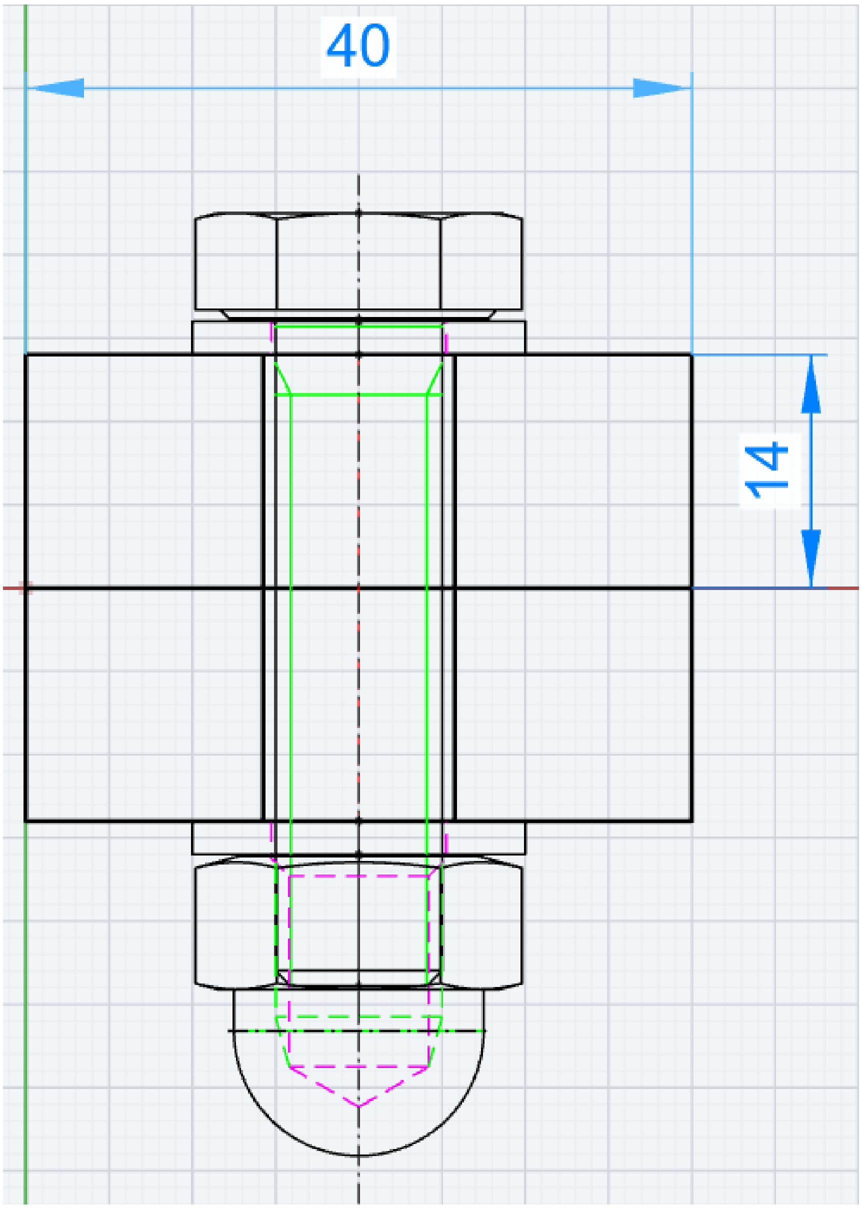
Then choose the **M10 washer** and place it 2x in the drawing.

The washer comes in vertically and has to be rotated by **90°**, either with **Ctrl+mouse wheel** or via the angle entry at the bottom above the status bar.

Set the snap mode to **Automatic [A]** and place the two washers.

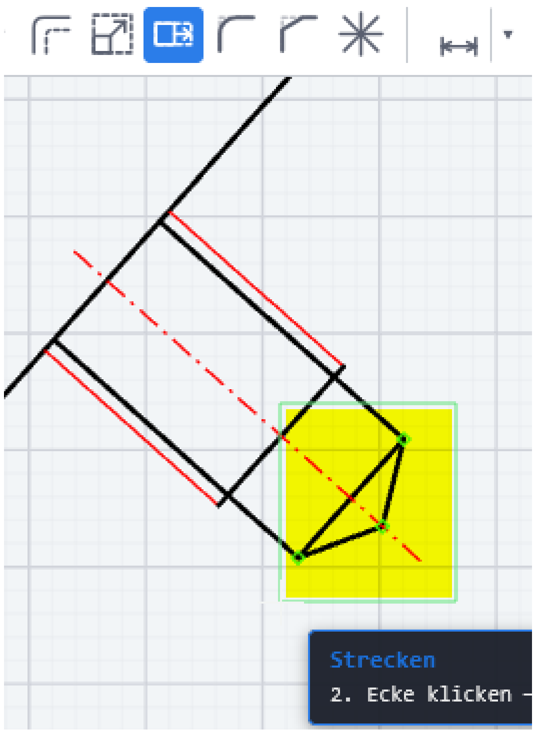
Then place the **M10 screw** and the **M10 cap nut**...

Done :-)



58 Extending an inclined hole

An inclined tapped hole is to be extended. The standard part library contains the tapped hole with: drilling depth **15** mm, thread depth **10** mm.



Now the task is to extend the tapped hole, which lies at a right angle on an inclined edge, and specifically by **12.5** mm.

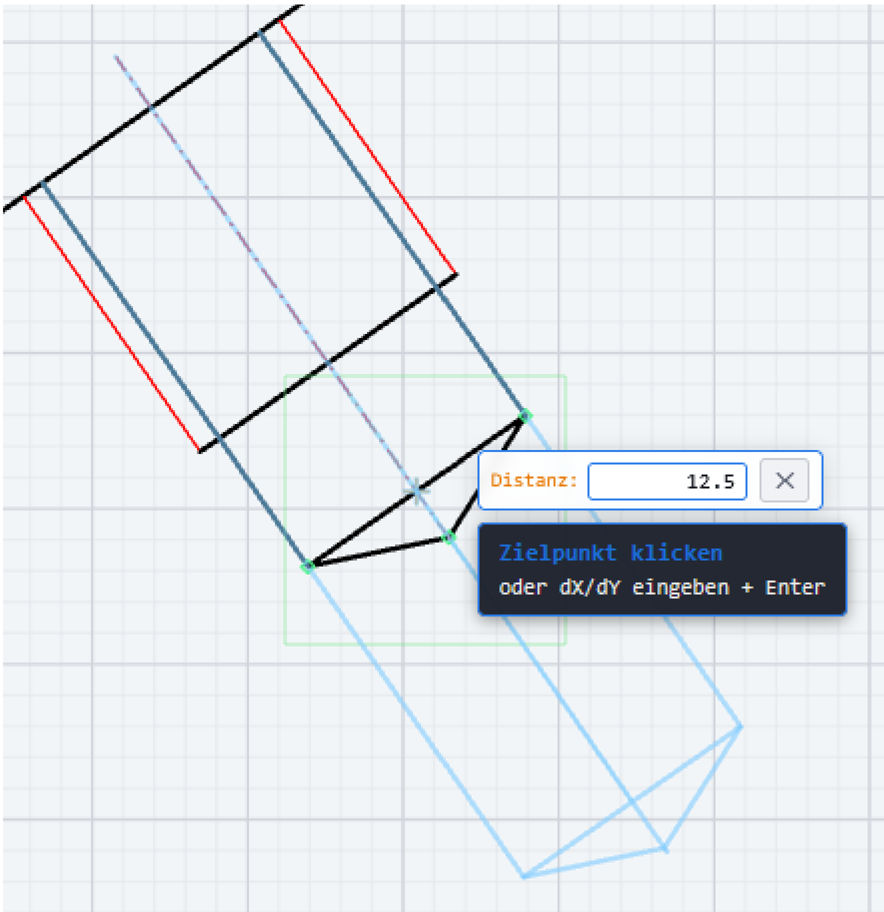
To do so, choose the **Stretch** function and use a rubber band to select all the points that have to move.

Then the **base point** is asked for.

Once the base point has been defined, click the blue **Direction** button and select a line for the stretch direction. In our case that would be the **axis** of the hole.

After that the stretch area can be moved along the axis and the distance value **12.5** entered → **[Enter]**

Perfect!



59 Creating a title block to DIN

Every technical drawing has a title block. It is located at the lower right corner of the drawing area; on drawings in A4 format (portrait) it lies below the drawing area.

If the drawing sheet has been folded to DIN 824, the title block should be legible at the lower edge of the sheet (see EN ISO 5457, which governs the division of the drawing sheet).

In contrast to the established DIN 6771-1, a large part of the requirements has been dropped or converted into mere recommendations, so that title blocks can now be adapted much better to individual needs.

But there have also been some changes, such as the width.

The most important differences from DIN 6771-1 are:

- width 180 mm
- height arbitrary
- only eight mandatory fields:
    1. legal owner (a company, for example)
    2. item number
    3. date of issue
    4. section/sheet number
    5. title
    6. approving person
    7. creator
    8. type of document
- additional fields optional
- division arbitrary, any number of rows
- column width arbitrary, only a recommended number of characters

The title block therefore fits exactly into the lower area of the drawing area on A4 drawings, which is 180 mm wide. On larger formats the title block goes into the lower right corner of the drawing area, whereby it must be noted that these may only be used in landscape format.

The drawing area is delimited by a thick continuous line at a distance of 10 mm (20 mm on the left-hand side) from the edge.

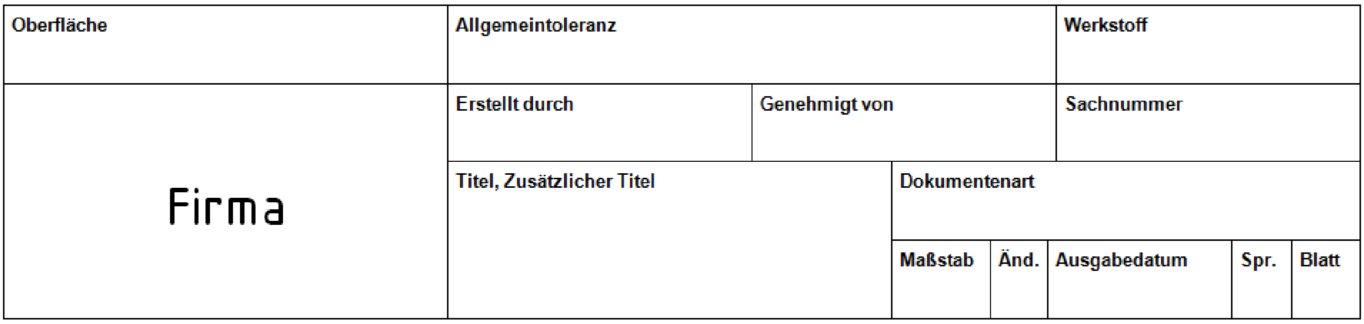
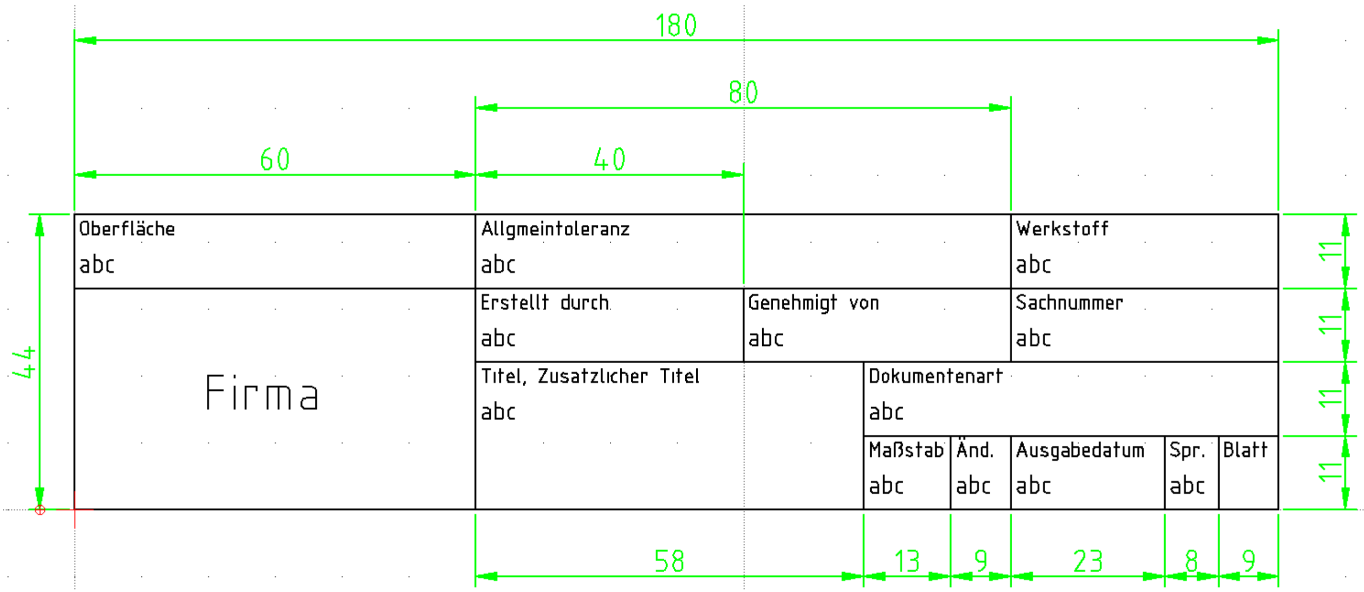


Diagram 10: Wikipedia title block to the standard ISO 7200.

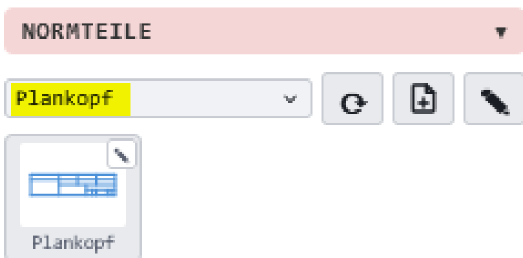
Source: Wikipedia - https://de.wikipedia.org/wiki/ISO_7200

59.1 Drawing a title block to ISO 7200 in LiberoDraft

Copy the following sketch. All dimensions comply with the standard.



We have already implemented the title block in the panel on the right, under **STANDARD PARTS**:



Tip:

*The text lines to be filled in were preset with «**abc**» and can be changed with a double-click, or deleted if not needed.*

60 Printing

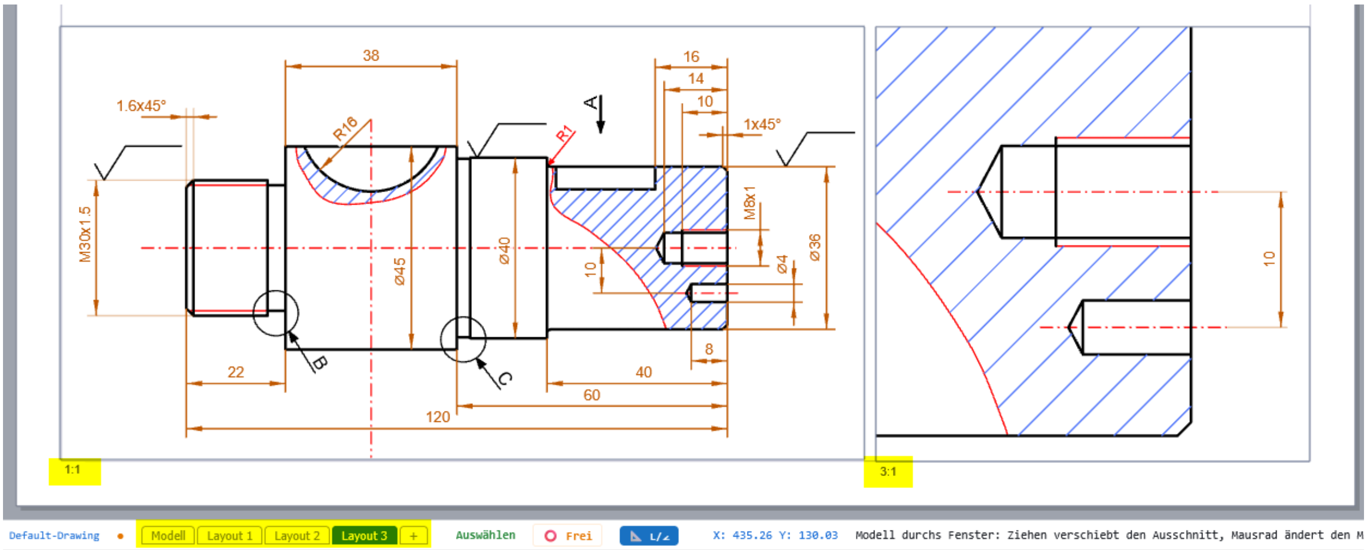
61 Paper space and viewports

LiberoDraft separates the model space, in which you draw at real dimensions, from the paper space — the sheet that is later printed. You find the tabs for this at the bottom in the status bar.

[FIGURE: status bar with the model and layout tabs]

61.1 Creating a viewport

On the sheet you create a viewport: a window that looks onto the model drawing. For each viewport you set its own scale — so you can bring the same drawing onto one sheet at 1:50 and a detail next to it at 1:5. Or, as in the following example, 1:1 and 3:1 in Layout 3:



[FIGURE: sheet with two viewports at different scales]

For each viewport, individual layers and whole object types can additionally be switched off — hiding dimensions in an overview, for example, while they stay visible in the detail viewport.

61.2 Dimensioning through the viewport

From the sheet you can dimension the model geometry directly. The dimension then lies on the sheet but reaches through the viewport onto the drawing and shows the real value — independently of the scale of the viewport. If you change the scale, the dimension stays stuck to its geometry.

Areas clicked through the viewport can likewise be hatched.

61.3 Printing and DXF

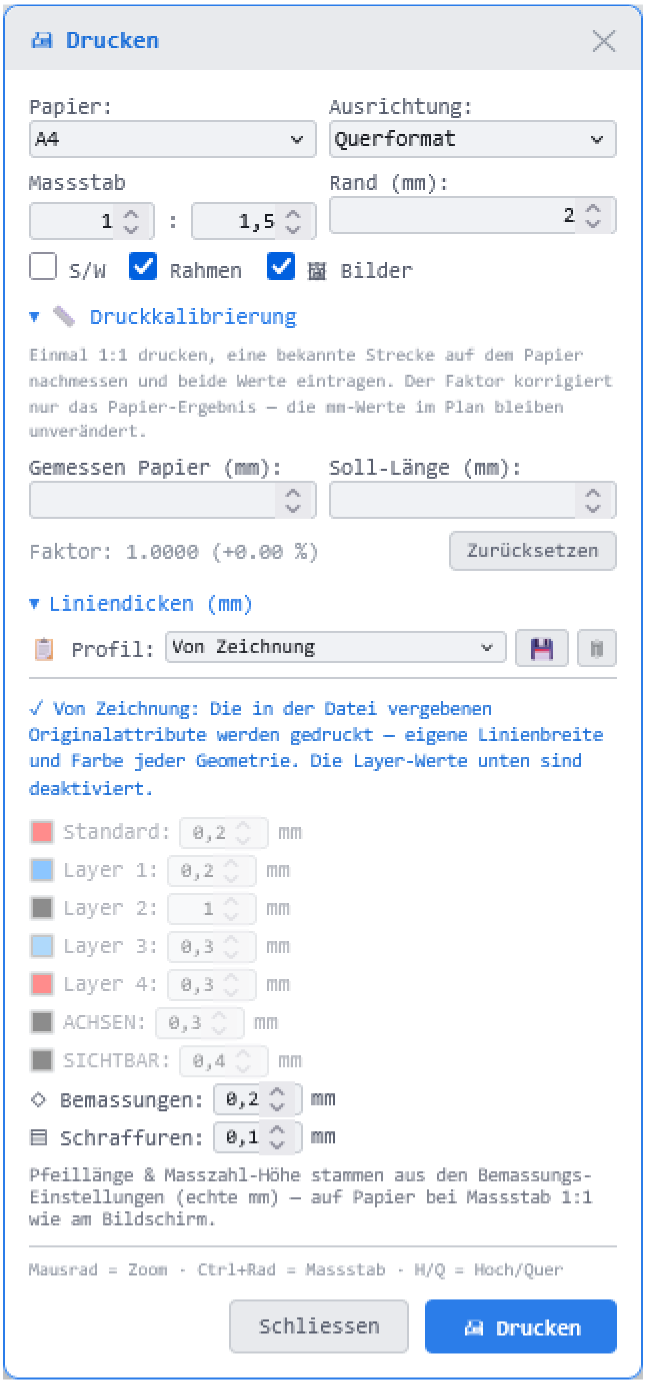
The sheet is printed with everything that lies on it. The DXF export can also output the paper space; the contents of the viewports are flattened and cleanly clipped at the viewport edge.

62 Printing

LiberoDraft outputs your drawing to scale via the **print dialog**. In the background a vector-based print template is generated and handed over to the normal print dialog of your browser. From there you can output it on a connected printer or save the drawing as a PDF. Because the template consists of vectors, the result stays razor-sharp at any size -> entirely without a pixel grid.

You open the print dialog in one of the following ways:

- via the printer symbol in the upper toolbar,
- with the key combination **Ctrl + P**,
- via the function search (the magnifier or **F3**) by entering "Print".



62.1 The print preview on the drawing area

As soon as the dialog is open, a **paper frame** is laid over the drawing area showing the chosen sheet at the scale that has been set. This lets you see immediately which part of the drawing fits onto the sheet. You can push the frame over the section you want with the mouse. Every change to the paper format, the orientation or the scale is shown in the preview immediately.

The following mouse and key commands are available to you for the preview -> they are also displayed as a short reminder line at the bottom of the dialog:

Field / button	Meaning
Mouse wheel	Zooms the view. This only changes the display on screen, not the print scale.
Ctrl + mouse wheel	Changes the print scale (see the section "The scale").
Key H / Q	Switches between portrait (H) and landscape (Q).

62.2 Page setup: paper, orientation and margin

In the upper area of the dialog you define the sheet:

Field / button	Meaning
Paper	The paper format. The DIN formats A4 (factory default), A3, A2, A1 and A0 as well as the US formats Letter and Legal are available.
Orientation	Landscape (factory default) or portrait. The orientation can also be switched over more quickly with the Q and H keys.
Margin (mm)	The space left free between the sheet edge and the drawing content, in millimeters. Factory default: 10 mm. A larger margin reduces the usable area but protects against clipping at the non-printable sheet edge.

62.3 The scale

You enter the scale as a **ratio X : Y** -> that is, in two fields separated by a colon. The left field stands for the paper, the right one for reality (the plan). The click arrows of the fields count in steps of 0.5; any other value can be typed in directly.

A few examples:

Field / button	Meaning
1 : 1	Original size. 1 mm on the paper corresponds to 1 mm in the plan.
1 : 2	Reduction to half size. A component 100 mm long is printed 50 mm long.
1 : 10	Reduction to one tenth -> typical for larger components or overviews.
2 : 1	Enlargement to twice the size. A small detail of 10 mm is printed 20 mm large.

> **Note:** Arrow length and dimension text height of the dimensions come from the **dimension settings** and are stored there as real millimeter values. At a scale of 1 : 1 they appear on the paper exactly as large as on screen; at other scales they are scaled with it accordingly.

62.4 Display options: B/W, frame, images

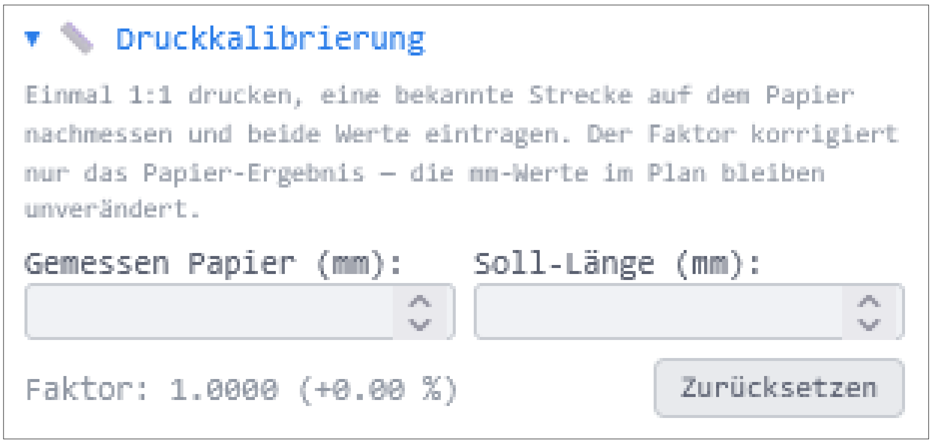
Directly below the scale fields you find three check boxes:

Field / button	Meaning
B/W	Prints the drawing in black and white instead of in color. Practical for black-and-white printers, master copies or to save toner. Factory default: off.
Frame	Draws a frame around the sheet. Factory default: on.
Images	Prints inserted images (underlaid photos or logos, for example) as well. If the box is empty, images are hidden when printing. Factory default: on.

62.5 Print calibration

Printers and browsers do not always reproduce dimensions exactly -> depending on the device, a printed distance may turn out slightly too long or too short. With the **print calibration** you compensate for this deviation once and for all. To do so, expand the "Print calibration" section and proceed as follows:

1. Print the drawing once at a scale of 1 : 1.
2. Measure a known distance on the paper with a ruler.
3. Enter the measured value in the field "Measured on paper (mm)" and the correct nominal value in the field "Nominal length (mm)".



From this LiberoDraft calculates a **factor** that is shown below the fields ("Factor: 1.0000 (+0.00 %)", for example). This factor corrects only the printed result on the paper -> the millimeter values in your plan stay unchanged. With the **Reset** button you set the factor back to 1.0000 (no correction).

62.6 Line widths (mm)

In the **Line widths (mm)** section you control how thick the lines are printed on the paper -> independently of the screen display. The control works via **profiles** that you can save with the symbols next to the selection field (the diskette symbol) or delete (the wastebasket symbol).

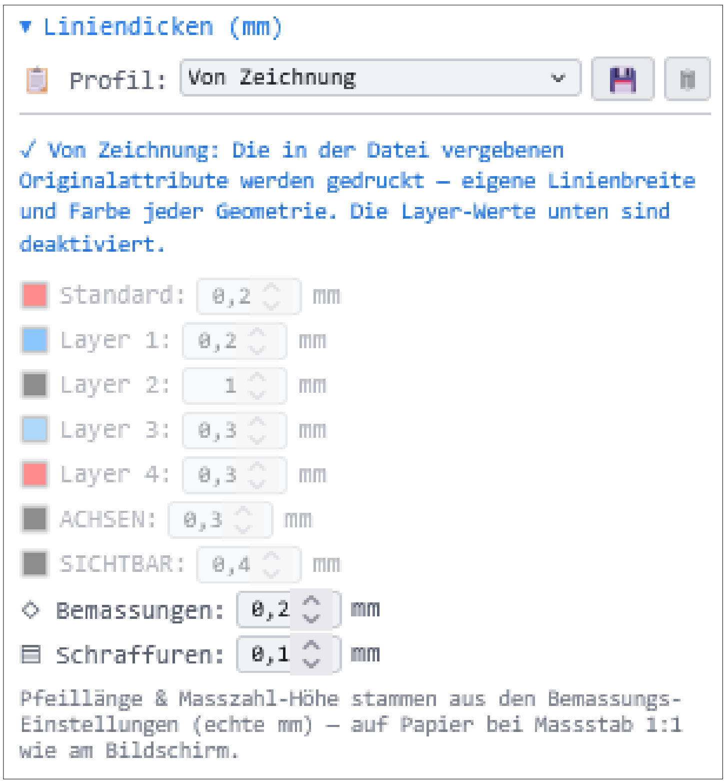
62.6.1The profile "From drawing" (factory default)

In the factory default the profile **"From drawing"** is active. The original attributes stored in the file are then printed -> that is, each individual geometry's own line width and color. The layer values in the list below are deactivated (grayed out) in this case.

62.6.2Your own line widths per layer

If you choose a profile of your own instead of "From drawing", the line widths are printed in millimeters per layer and element type. The following values can be set separately (the figures given are typical defaults):

Field / button	Meaning
Standard	Basic line width for elements without their own layer assignment. Default about 0.2 mm.
Layer 1 … Layer 4	Line width per user-defined drawing layer (layer 2 thicker for main contours, for example).
ACHSEN	Line width of the axis/center lines. Default about 0.3 mm.
SICHTBAR	Line width of the visible main edges. Default about 0.4 mm.
Dimensions	Line width of the dimension lines and extension lines. Default about 0.2 mm.
Hatches	Line width of the hatch lines. Default about 0.1 mm.



Tip: A proven profile arrangement according to drawing standards uses thin lines for dimensions and hatches, medium ones for axes and strong ones for visible edges. Once you have found a suitable profile, save it via the diskette symbol and you can select it again at any time.

62.7 Printing and closing

With the **Print** button, LiberoDraft hands the finished template over to your browser's print dialog. There you choose the printer or save as a PDF ("Save as PDF" or "Microsoft Print to PDF").

Important: In the browser's print dialog set the scaling to **100 % or "Actual size"** and avoid "Fit to page". Otherwise the browser changes the size afterwards and the scale set in LiberoDraft no longer holds on the paper.

The **Close** button closes the dialog again without printing; the print preview on the drawing area disappears.

62.8 Practical tips

- **Printing to scale:** choose a scale of 1 : 1, set the browser to "Actual size" and carry out the print calibration once. After that, distances measured on the paper are exactly right.
- **Large components:** a smaller scale (1 : 5 or 1 : 10, for example) or a larger sheet (A3, A2) brings the whole drawing onto the paper.
- **Copying and black-and-white printing:** tick the "B/W" check box so that colors appear as clear shades of gray.
- **Reproducible line widths:** create and save a line width profile of your own instead of relying on "From drawing" -> all drawings then print uniformly.

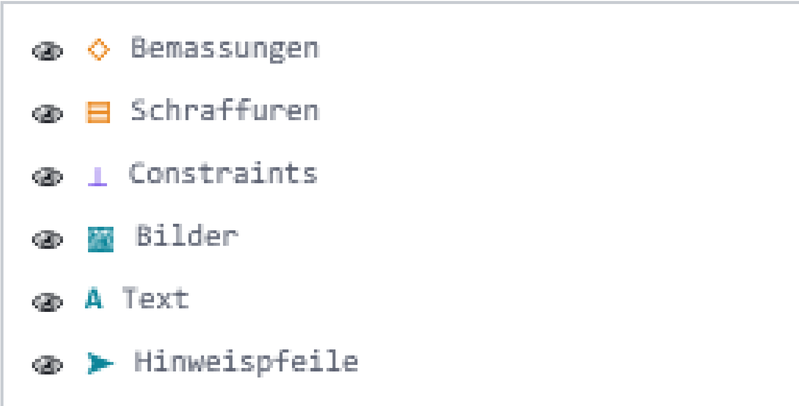
63 Tips and tricks with LiberoDraft

Over the time in which this book is extended we will collect useful tips and tricks about LiberoDraft here. Even if an additional program were needed for something and it serves the purpose, that is no problem. The main thing is that it takes LiberoDraft further.

63.1 Speeding LiberoDraft up

If navigating in LiberoDraft stutters or is very slow with large drawings, whole element groups can be hidden in the panel on the right.

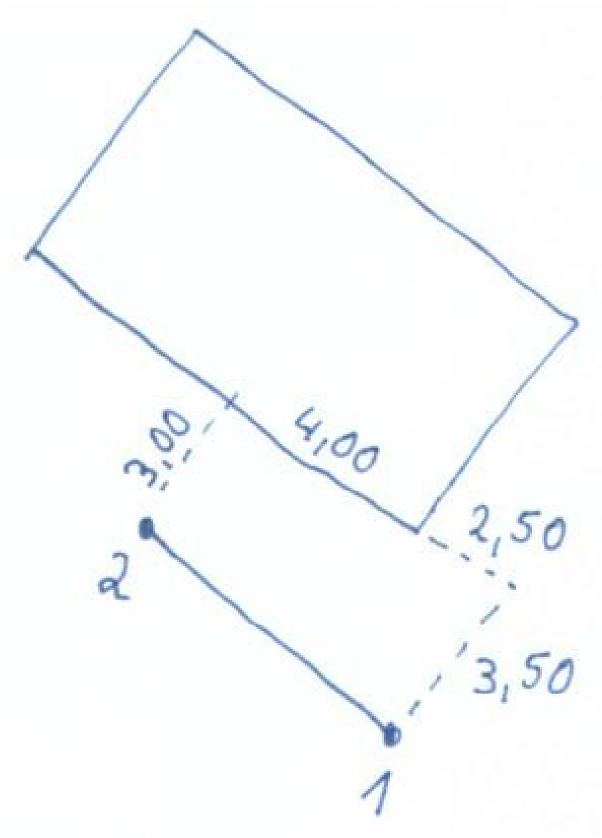
This hides hatches and texts, for example, which speeds up the display process...



63.2 Relative object snap

For example, I have a rectangle lying at 45° and I want to draw a line relative to the corner of the rectangle with a measurement parallel to the line (from point 1 to point 2, see the figure).

How can I determine the points at pos 1 and 2?



Short description:

Layer: **SICHTBAR**

Function: **Rectangle via 3 points**

Snap: **Grid [G]**

Line: L = **8**, angle = **-135°**, **Enter**

Width: W = **8, Enter**

Offset the lower line of the lower long wall by 3.00.

Offset the line of the lower short wall upwards by 4.00.

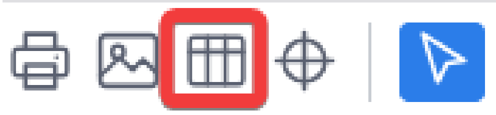
Trim the lines and you obtain point 2.

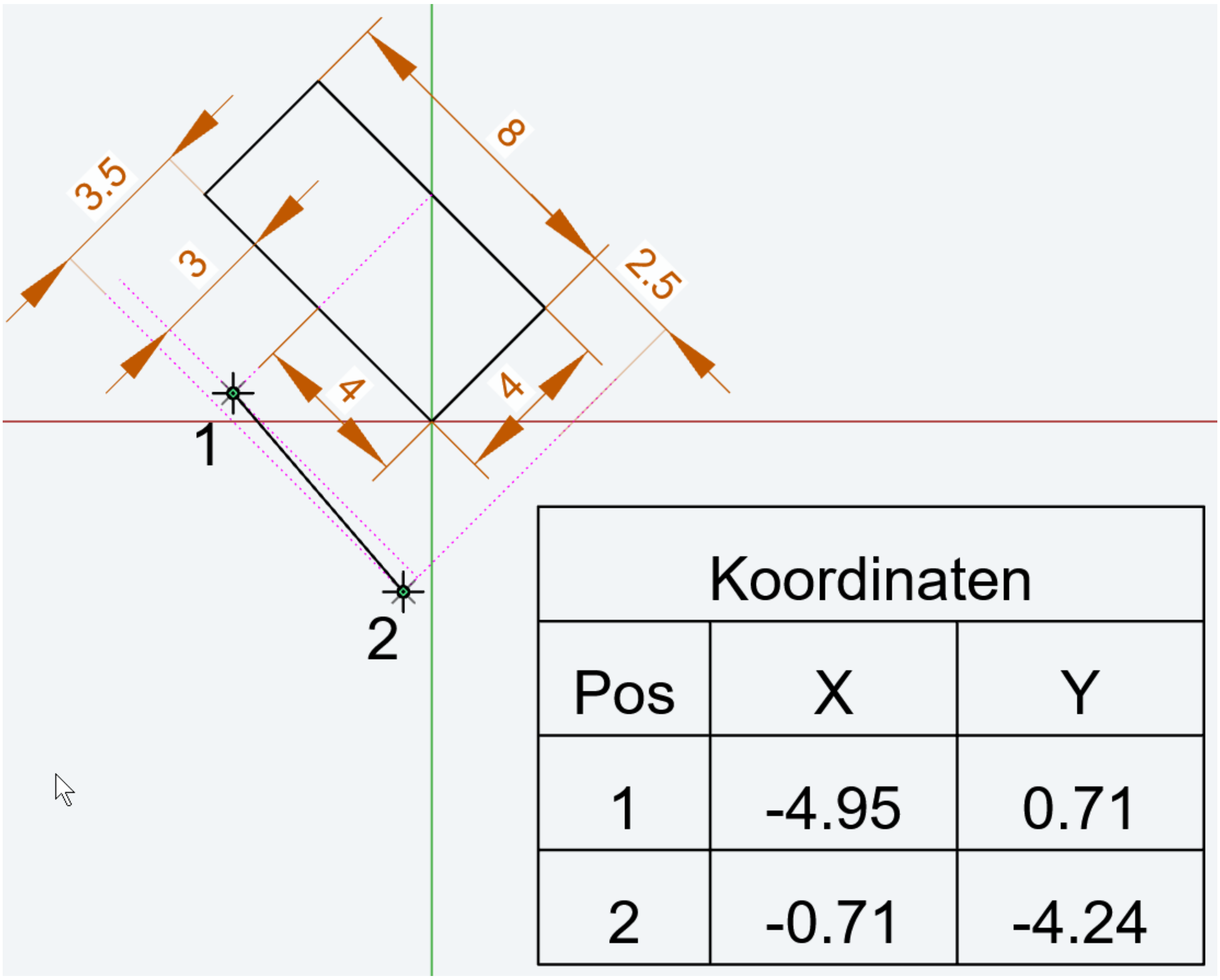
Offset the line of the lower long wall downwards by 3.5

Offset the line of the lower short wall downwards by 2.5.

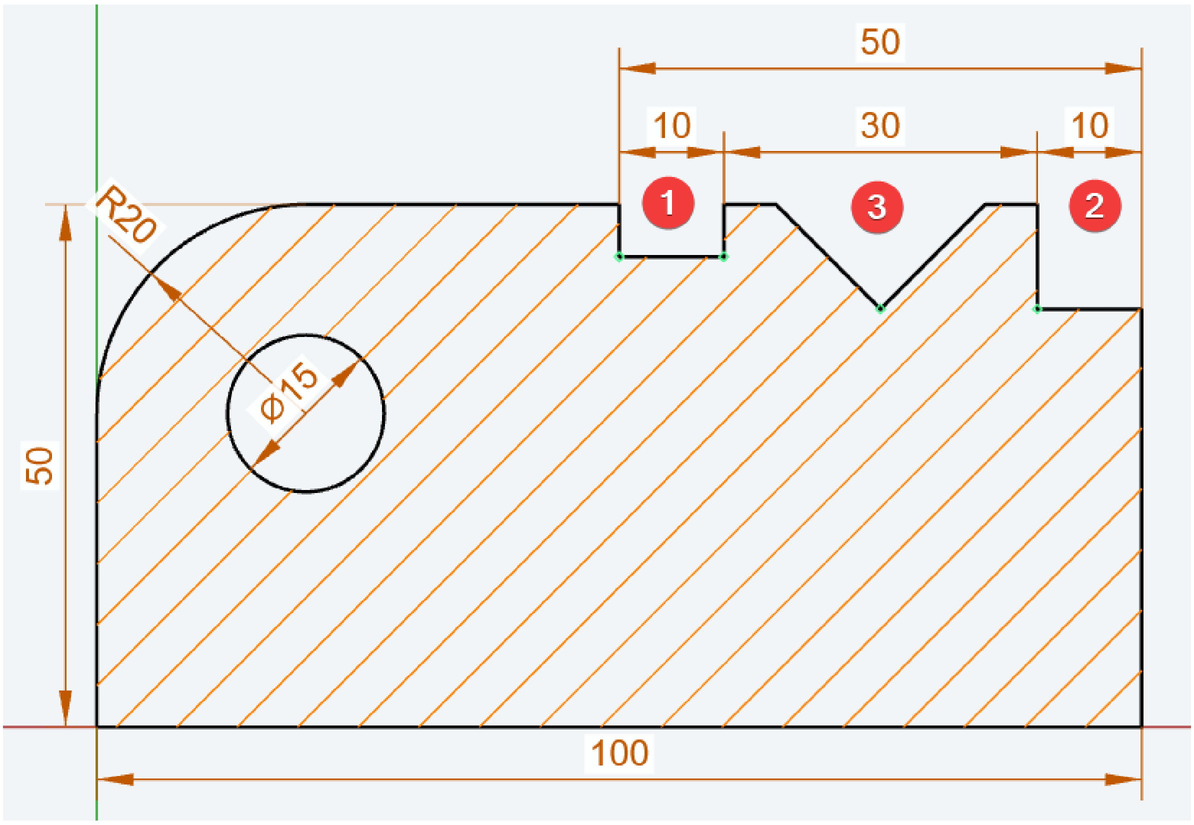
Trim further lines and you obtain point 1.

Insert **points** at **intersections 1** and **2** and click **Coordinate table →**



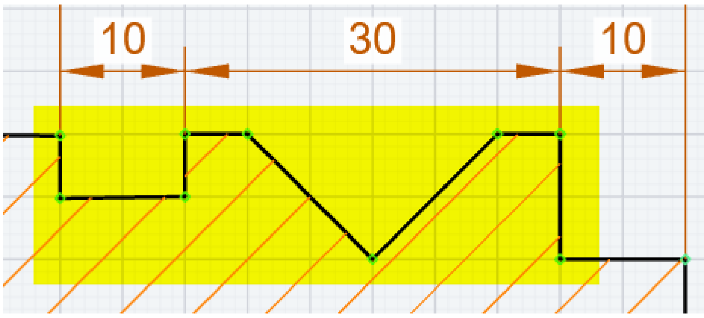


63.3 Correcting geometry with the Stretch function



In this drawing the geometry has to be corrected as follows:

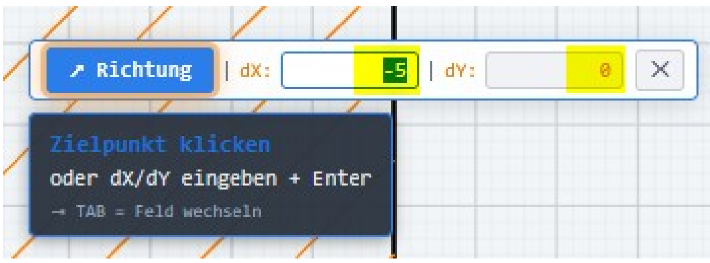
Dimension **(1)** is to become **20 mm** and dimension **(2)** **15 mm**.



Function: **Stretch**

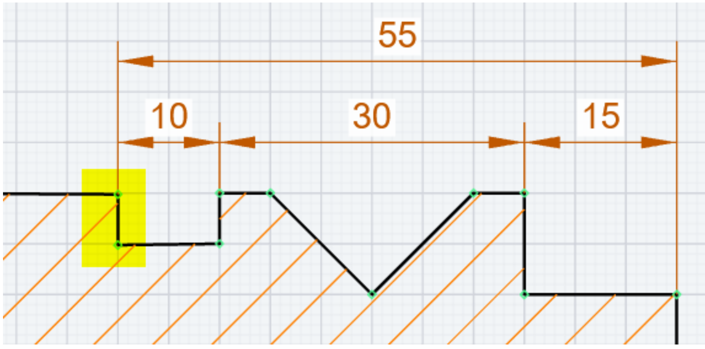
Frame the elements to be stretched…

Specify the **base point**	Click a point with the **LMB**.
Specify the target point	**dX: -5 / dY: 0**
[Enter]	



The geometry including the dimensioning
 is corrected cleanly. Excellent!

Function: **Stretch**

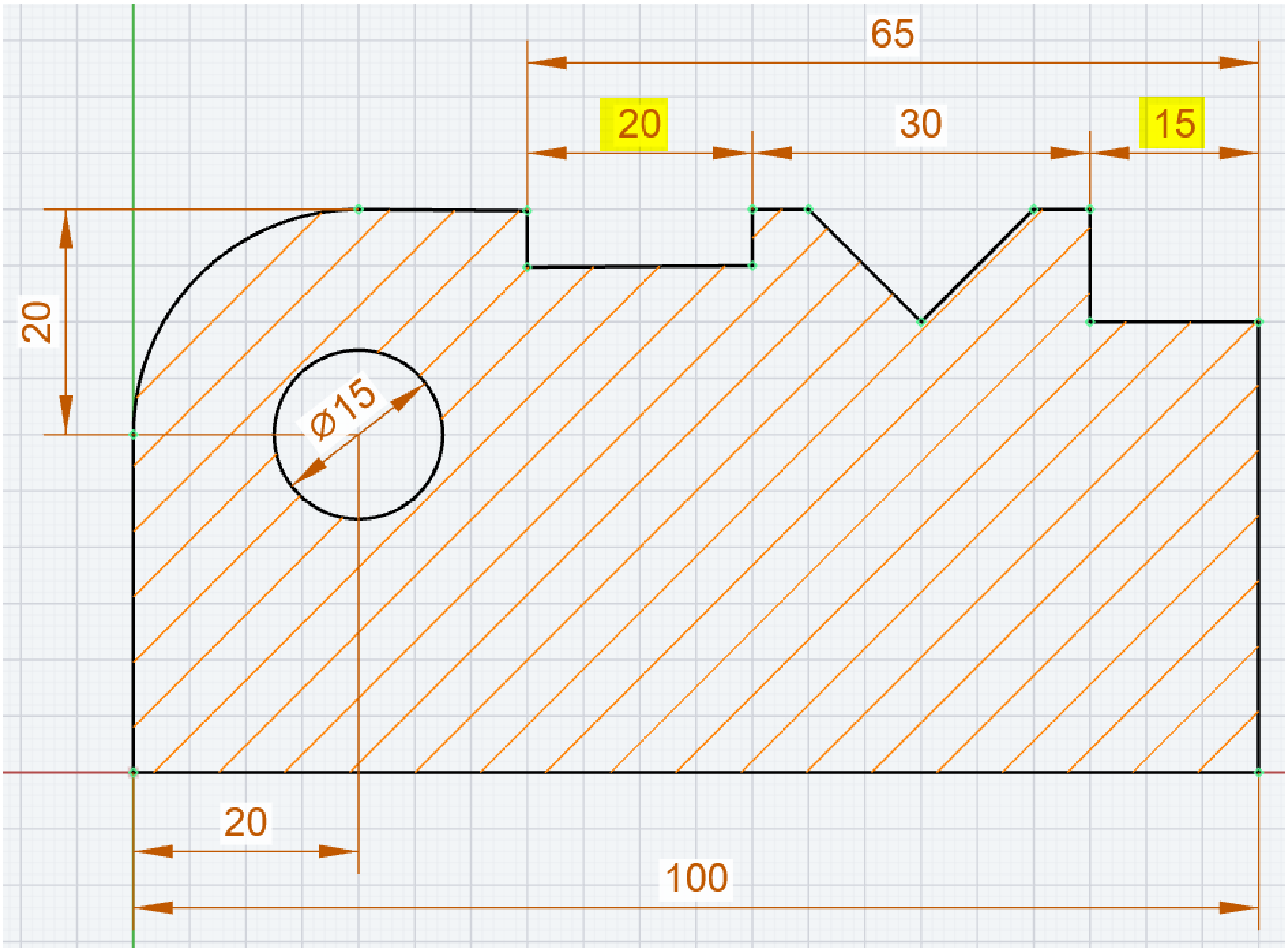


Frame the elements to be stretched...

Specify the **base point** Click a point with the **LMB**.

Specify the target point **dX: -10 / dY: 0**

[Enter]

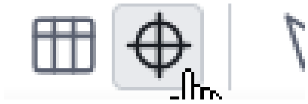


Perfect! New dimensions 20 and 15.

Now just move the hole D15 to the position X20, Y20...

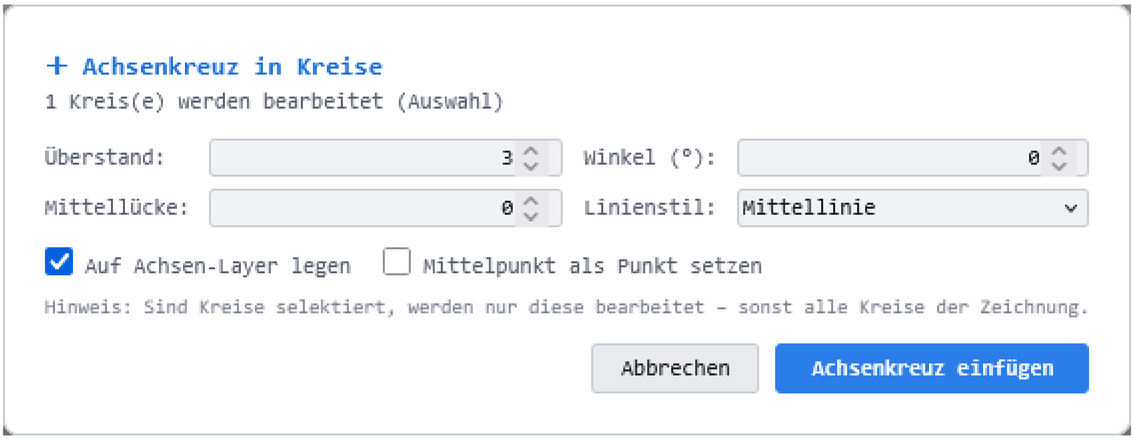
For that to work, a **hole** must always have an **axis cross**. That is done with the function:

Axis cross



Select the circle(s) and click **[Next]** .

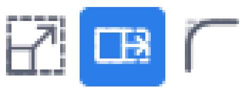
In the next dialog, define the design of the axis cross...



... and click the blue button → the axis cross is drawn.

So that the dimensions travel along cleanly during the stretch that follows, the two dimensions have to be drawn onto the axis cross anew.

Function: **Stretch**

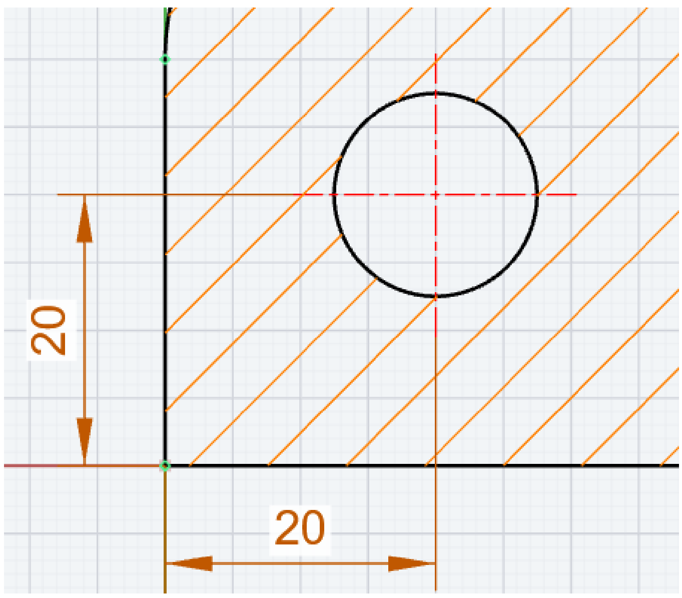


Frame the circle geometry including the axis cross...

Specify the **base point** Click a point with the LMB.

Specify the target point **dX: 0 / dY: -10**

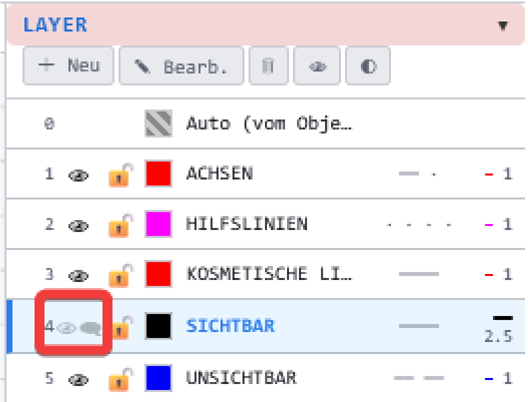
The circle is moved to the new position. The dimensioning and the hatch are corrected impeccably.



Done :-)

63.4 The element disappears while drawing!

Yes, that happens when a layer is hidden and you draw on it →



64 Check & clean up drawing

64.1 What this function is for

In the course of the work, elements sometimes accumulate in a drawing that do not visibly get in the way but are nevertheless unwanted: a line whose start and end lie on the same point; two circles lying exactly on top of each other; a dimension whose reference edge was deleted long ago. Such "legacy items" arise particularly often when importing DXF files or through repeated copying and pasting. They are treacherous because their symptom – an element that cannot be selected, or unexpected behavior when saving – lies far from the actual cause.

The **Check & clean up drawing** function searches the whole drawing specifically for such findings, lists them clearly and removes them on request. It is therefore the tool of choice for bringing an imported drawing, or one that has grown over a long time, into a clean and reliable state.

64.2 Calling it up

The function is opened via the **function search**: press **F3** and type "clean" (or "check", "tidy", "duplicates"), then choose the entry **Check & clean up drawing**. The check runs immediately and shows the result in a small window.

64.3 The dialog

The window lists only those categories in which something was actually found – each with a **check box** and the **number found**. By default all the categories found are ticked. The box lets you exclude each category individually from the cleanup, in case you want to keep certain elements.

A click on **Clean up** removes all the ticked findings in one step. **Close** leaves the window without changing anything. If the drawing is clean, the window simply reports that no problems were found.

Three things are important here and give you confidence: the cleanup affects **only the problem cases named** – valid geometry is never touched. **Display, export and measurements stay untouched.** And every cleanup can be undone completely with **Ctrl + Z**.

64.4 The categories checked

Category	What is found and removed
Zero-length lines (start = end)	Lines whose start and end point coincide. They are invisible but can block the selection of neighboring elements.
Circles/arcs with radius 0	Circles and arcs without extent – likewise invisible artifacts.
Degenerate polylines (< 2 points)	Polylines that do not form a real segment because they have too few or only coincident points.
Duplicate elements (exactly coincident)	Identical elements lying on top of each other several times. Exactly one copy of each geometry is kept (see below).
Empty hatches (without a valid boundary)	Hatches whose boundary no longer yields a valid area and which therefore show nothing.
Dimensions without valid reference geometry	Dimensions whose associated edge, circle or line no longer exists.
Orphaned weld groups	Welds whose members were deleted or which no longer join two valid points.
Element groups with missing members	Groups that refer to elements that no longer exist. The dead references are removed; completely empty groups are dissolved.
Orphaned constraints	Geometric conditions that refer to elements that no longer exist.

Each of these categories addresses a concrete error case that occurs in practice. The first four concern the geometry itself; the rest concern references that point into the void after the element originally referenced was removed.

64.5 Duplicate elements: what exactly is detected

With duplicate elements the function deliberately detects only **exactly coincident** geometry. The decisive factor is a very fine tolerance of 0.0001 mm, so that only genuinely identical elements count as duplicates. The following are covered:

- **Lines** – independently of direction: a line and the same line with its start and end point swapped count as duplicates.

- **Circles** – with the same center and radius.
- **Arcs** – with the same center, radius and angular range. Here the **direction of rotation** counts: an arc with swapped angles describes a different section and therefore does not count as a duplicate.
- **Points** – at the same position.
- **Polylines** – likewise independently of direction (the same sequence of points forwards or backwards).

From every group of identical elements, **one copy is always kept**; only the surplus copies are removed.

Quite deliberately, **partly overlapping or collinear segments are not** treated as duplicates. A short line lying on a longer one may well be intended – removing it automatically would be risky. Likewise **texts and hatches** are excluded from the duplicate check, since here multiple copies may rather be intentional.

64.6 A typical workflow

It is advisable to run the check **directly after a DXF import** and occasionally with extensive drawings that have grown over a long time. The procedure is always the same: press F3, call up the function, look through the result, deselect individual categories if necessary and click Clean up. Should the result not be to your liking after all, pressing Ctrl + Z is enough to restore the previous state.

Since the function only removes unambiguous cases that can be reliably identified as faulty and never changes valid design geometry, it can be used regularly without hesitation. It is the fastest way to free a drawing of invisible ballast and thus to avoid later problems that are hard to track down.

65 Whats new? Updatehistory...

v1.3.35 Clearer wording on the Show tooltip (Improved)

When you clean up a drawing, the Show button explains itself whenever a finding draws nothing visible. The parenthetical remark it used to carry caused more confusion than clarity and has been removed. The message now reads simply: "This finding draws nothing in the drawing – there is no place to point to."

v1.3.34 Texts stay visible inside viewports (Fixed)

Texts sitting on a very low Z level disappeared from viewports and sheets in the layout – the opaque white paper covered them up. In paper space all texts now go to the upper layer, model space is unchanged, and printing was never affected. In addition, the Show button gives its explanation as a tooltip right at the button, and the settings export no longer carries the clipboard along.

v1.3.33 Clean up drawing (New)

Large files can now be slimmed down deliberately: the "Check & clean drawing" dialog works in four steps with a progress bar and plain language, and gives every finding a Show button that jumps to the spot and makes it flash orange for about three seconds – so you decide what stays before anything is removed. The persistent clipboard no longer travels inside the project file either; opening and saving slims down older drawings automatically (roughly two of 3.9 MB in our test). Dimensions whose reference is missing are now recognised as orphaned as well.

v1.3.32 Mouse wheel steps while placing (New)

When a selection hangs from the cursor while inserting or placing, Alt+wheel now scales it in fixed steps (factory setting 0.1) and Alt+Shift+wheel in steps of 1 – the factor field in the secondary bar follows along. For rotation, Ctrl+wheel still turns 1°, and Ctrl+Shift+wheel now turns 5°. The step size can be set in the Settings dialog.

v1.3.31 Images inside blocks mirror correctly (Fixed)

A mirrored block insertion showed its image the wrong way round: "H" came out mirrored and upside down at once, while "V" had no effect at all. The image surface is now flipped about its own axis in addition to being rotated, and both directions are correct. Hidden and isolated-away texts carrying a hyperlink no longer react to the cursor either.

v1.3.30 Images inside blocks are drawn (Fixed)

Turning elements including a bitmap into a block made the image vanish – the block definition took it over correctly, but it was silently dropped while drawing. Images inside blocks are now drawn with their position, size and rotation. A second finding along the same path: a locked image sat offset inside the block, which has been fixed as well.

v1.3.29 Parallel-distance dimensions selectable by rubber band again (Fixed)

Dimensions of the parallel-distance type could be clicked, but the selection window missed them in both modes – which meant they could not be hidden either. Their bounding box is now derived from the dimension line, so window and crossing selection catch them again.

v1.3.28 Hidden blocks really do disappear (Fixed)

Blocks inserted more than once stayed visible when hidden or isolated away, and merely stopped responding to the cursor. The cause was an internal shortcut in the drawing pass that bypassed the visibility check. It now applies the same check as every other path.

v1.3.27 Hidden objects stay out of the export too (Fixed)

Isolate and Hide have applied everywhere since v1.3.13 – the DXF and SVG export was the only path that ignored this and wrote everything out. All export paths now carry the same gate, for elements, dimensions and hatches, in model space as well as on the sheet.

v1.3.26 Hide button next to Isolate (New)

Next to "Isolate" there is now a "Hide" button – both in the HUD and in the context menu. It hides the selected objects, either on its own or on top of an existing isolation; "Show all objects" brings everything back in one step. Hiding applies to the display, selection, snapping, printing and export, and is not saved with the drawing.

v1.3.25 Tolerance class updated automatically (New)

When the value of a dimension using "class + deviations" or limit dimensions changes, the deviations are now recalculated on their own – the same calculation the "determine" button performs. Values entered by hand win and stay untouched, as does the content of loaded drawings. Angular dimensions are excluded, and unknown classes are left alone.

v1.3.24 Dimension dialog and attributes side by side, deviations aligned to DIN (Improved)

The "Attributes..." button in the "Edit dimension" dialog no longer closes it – both windows stay open next to each other and step out of each other's way. Stacked deviations now align their digits flush underneath one another, with the sign right-aligned in a column of its own in front: on screen, in print and in the dialog preview.

v1.3.23 Deleting a layer now asks first (Improved)

The waste bin in the layer list used to move the contents onto the active layer without a word. It now opens a dialog, names what the layer holds and offers a choice: move to any other layer – locked and hidden ones included – or delete along with the layer. If locked or hidden elements, hatches or dimensions sit on it, a second confirmation follows that spells out the quantities.

66 LiberoDraft 1.3.22 -> what is new since version 1.3.11

66.1 Slot and curved slot (1.3.21 – 1.3.22)

Slots are part of the daily craft in mechanical engineering — and are tedious to draw by hand: two parallel lines, two semicircular arcs, and everything has to fit together exactly. Since version 1.3.21 two dedicated tools in the shapes flyout, directly next to the rectangle, take this over.

You draw the straight slot via its two semicircle centers and the width. **The curved slot** is created via the arc center, the radius, the start and end angle (counter-clockwise) and the width. At every stage you can click or type in the value and confirm it with Enter.

Every slot is created as a finished group of four contour pieces — so it can immediately be moved or copied as a whole, or removed again with a single Ctrl+Z. The width last used is remembered and is offered as a suggestion for the next slot. Impossible entries — a width that would be larger than the arc allows, or an angular range close to zero — are caught by LiberoDraft with an understandable message.

66.2 The live workflow of version 1.3.22

Version 1.3.22 turned the two tools into a continuous live workflow. At every stage a floating input field travels along with the mouse, and orange guide lines show at all times what is currently being created:

- With the straight slot, the L/∠ field appears after the first center point and shows the length and angle of the axis live — just as when drawing a line. A length typed in and confirmed with Enter also sets the second center point numerically.
- With the curved slot, the orange guide line appears immediately after the first click. Radius and start angle run together in the R/∠1 field — a single click sets both at once. After that an orange line shows the end angle live.
- In the width stage, an orange line runs from the axis (with the curved slot: radially from the arc) to the mouse, and the W field shows the width live.

The most important trick lies in the width stage: you do not have to know the width. A click takes it off directly in the drawing — also with a snap onto an existing element. The edge of the slot then passes exactly through the snapped point; this lets you relate the width directly to existing geometry. Typing it in and pressing Enter of course still work, and a click exactly on the axis (width zero) is not accepted but is guided kindly by the status bar.

66.3 The double line (1.3.18 – 1.3.20)

Walls, grooves, profiles: wherever two parallel runs are needed, the new double line saves twice the effort. You click the points as with a polyline — LiberoDraft guides both lines at once.

- The gap can be set in the floating field and is remembered for next time.
- Corners are cleanly mitred; the ends are closed with end strokes.
- On which side of the clicked points the second line lies is decided by you after the last point with a mouse click — with a live preview of the complete double line.

66.4 Operation tailored to fit

The two following versions polished the operation just as daily drawing suggested. Since 1.3.19, Enter adopts the typed values without ending the run — the dialog stays open and you simply carry on drawing. ESC completes the run and builds it, exactly like a double-click; an unwanted run is gone again immediately with Ctrl+Z. And since 1.3.20 the length and angle in the dialog run live with the mouse, as you know it from drawing a line — values typed in by hand stay untouched.

66.5 Tangents in any position (1.3.16 – 1.3.17)

Constructing tangents by hand is compass geometry — LiberoDraft now calculates them exactly. Version 1.3.16 began with the perpendicular tangent: you choose a circle or arc and a line, and LiberoDraft erects the tangent that stands at a right angle on the line — the point of contact and the foot of the perpendicular are determined exactly, and the order of the two clicks does not matter.

Version 1.3.17 completed the family:

- Tangent from a point to a circle or arc — of the two possible solutions LiberoDraft takes the one nearest to your click.
- Common tangent to two circles — up to four solutions (external and internal); here too your click decides.

With arcs, only tangents are created whose point of contact actually lies on the arc segment as drawn. And for all drawing tools the following applies since 1.3.17: the status bar names the first step as soon as the tool is chosen — you see what to do next without having to try.

66.6 Isolate and Show all objects (1.3.13 – 1.3.15)

Large drawings get crowded. Since version 1.3.13, isolating creates calm: make a selection, choose Isolate in the context menu or in the header bar — and only the chosen objects are visible, together with their dimensions and hatches. Show all objects brings the rest back. The state is purely a working aid: the complete drawing is always saved, and after loading everything is visible again.

Version 1.3.14 made sure the effect takes hold immediately — previously the view was only updated after a zoom. Version 1.3.15 refined the operation: isolating without a selection shows a brief note directly at the button, and "Zoom to fit" fits the view to the isolated objects only while isolation is active — exactly to what you are working with.

66.7 A visible crosshair (1.3.12)

Small, but noticeable in daily use: since version 1.3.12 the crosshair used while drawing has a white fringe around the black cross. It can therefore be clearly recognized on light and dark backgrounds alike — and for the first time it also appears on screenshots, which previously used a barely visible pointer image depending on the Windows settings. Instructions and support images thereby become considerably easier to understand.

67 LiberoDraft 1.3.11 -> what is new since version 1.0.0

From version 1.0 to version 1.3.11 · as of 11 August 2026

This chapter summarizes all the innovations since the release of version 1.0 — ordered by topic, the newest at the top. The names in quotation marks are the labels as they appear in the interface.

67.1 Working faster: the new keyboard shortcuts (1.3.5 – 1.3.9)

The most frequently used view commands now sit on single keys — and every key does exactly the same as its button in the toolbar:

H — Zoom out (1.3.9): with every press the visible section grows by half a screen width; the center of the view stays put, and pressing repeatedly zooms further out. H is a pure view function: a drawing operation in progress continues, exactly as with the mouse wheel. Hatching, which previously sat on H, remains reachable via its tool button and the function search.

A — "Zoom to fit" (1.3.8): the view jumps to the entire drawing content, exactly like the button — including its cancel behavior. Ctrl+A ("Select all") stays untouched.

Q — automatic snap (1.3.8): the most important snap mode now sits on Q; the tangent snap remains reachable via right-click → "Snap mode".

Ctrl+F — "Search functions" (1.3.8): the function search opens with the shortcut you know from practically every application. F3 is thereby free. The messages after a DXF import now also refer to Ctrl+F.

W — window zoom (1.3.5): frame an area and the view jumps to it; ESC ends it. "Split", which previously sat on W, remains reachable via its button and the function search. And as everywhere: if a line is hanging at the mouse, the keyboard belongs to the input field — first ESC, then the shortcut takes effect.

The shortcut help (F1), the snap menus, the function search and all the tooltips have been updated in both languages.

67.2 The right-click menu: cut and order (1.3.6)

"Cut" (Ctrl+X) now stands in the right-click menu directly before "Copy" — in the model space as on the sheet, because it is one and the same menu. Cutting copies the selection to the clipboard and deletes it; Ctrl+V pastes it again at the cursor and Ctrl+Z brings everything back. When right-clicking over a viewport, a line now separates the viewport block from the rest of the menu — the viewport commands thus stand clearly on their own.

67.3 Viewports and through-hatches on the sheet (1.3.1 – 1.3.2)

A selected viewport can be recognized at a glance: a delicate area tint, four large corner grips and a floating note say what is currently selected — confusing it with a selected through-hatch is a thing of the past (1.3.1).

The seed point of a through-hatch is now visible on the sheet as well: it can be moved (the hatch searches for its boundary through the viewport again) and copied with the Ctrl key held down — exactly as in the model (1.3.2).

67.4 The paper space, rounded off (1.2.2 – 1.2.7)

The six rounds of this series completed the paper space:

Viewports have their own right-click menu: "Edit (view + scale)", "Fit to content", "Viewport settings…", "Move", "Resize", "Lock", "Copy", "Delete" — in both languages (1.2.6 / 1.2.7).

A viewport stays operable even when a through-hatch fills it: the frame is a grip of its own, and a viewport that is already selected wins against its own hatch (1.2.5).

Through-hatches are clipped cleanly and without loss at the viewport edge — on screen, in print and in the DXF export; the original boundary in the model stays untouched

Dimension values always lie on top when the sheet is printed — no semi-transparent hatch covers a value any more (1.2.3). And the paper space became a module of its own, cleanly cut (1.2.2).

67.5 Dimensioning by your rules (the 1.1 series)

The dimension styles have matured: the factory styles "Mechanical engineering (mm)", "Timber construction" and "Construction dimensioning" now define every setting completely — switching over is predictable. With "★ Save as default" your own style becomes the initial state. The active style travels with the drawing: if a customer opens your file, they see your dimensions exactly as you created them — without their own settings being lost.

And the clear principles that go with it: OK in the dimension dialog acts exclusively on new dimensions; existing dimensions are changed only by the "Apply" button. And changes in the "Properties" panel act exclusively on the elements currently selected.

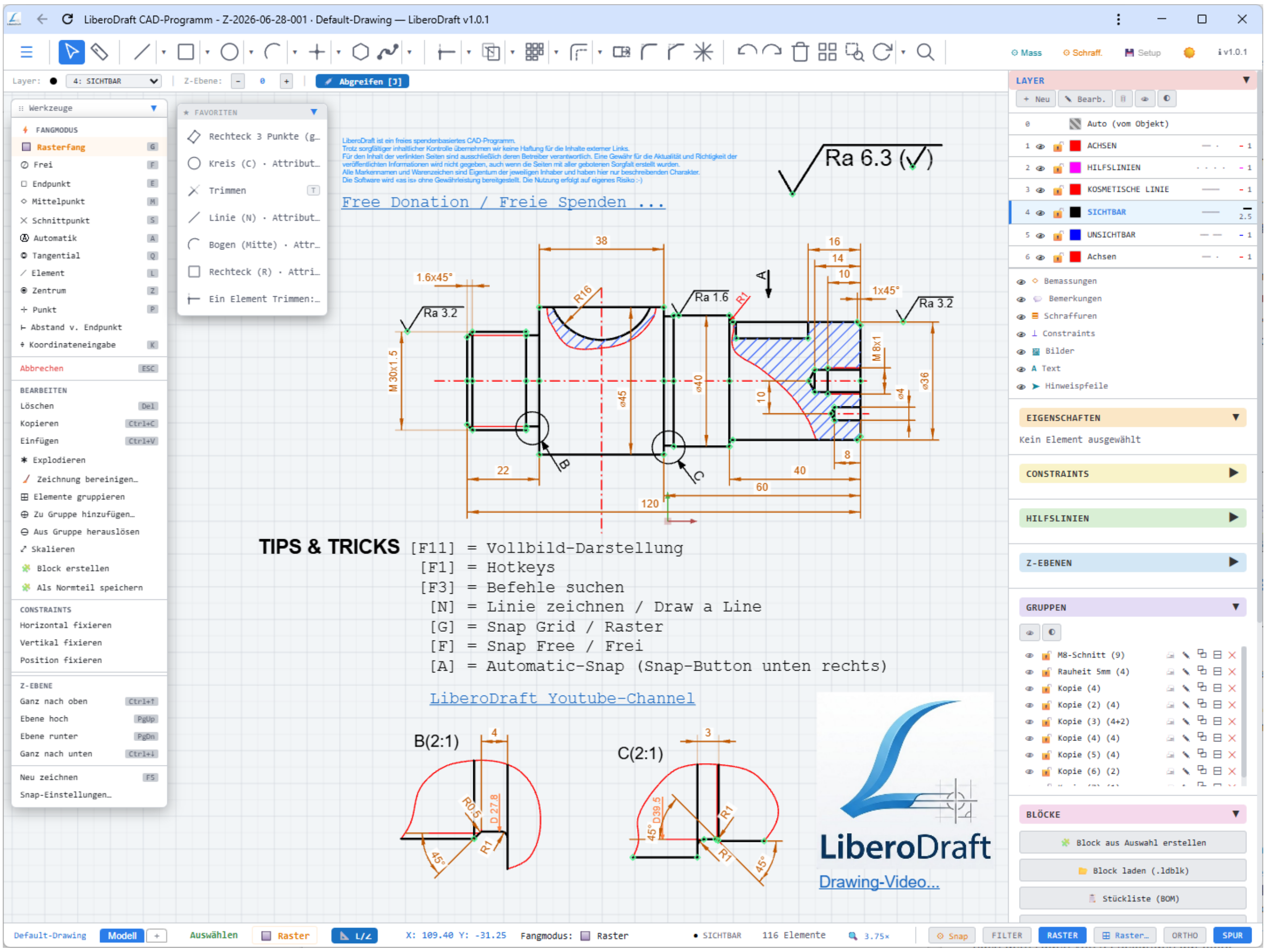
67.6 Under the hood: order, speed and guards (1.0.5 – 1.3.11)

Invisible to you, decisive for every future update: the entire source code was divided into 60 thematic modules (completed with 1.3.0) and fully disentangled in several tidying-up rounds.

68 LiberoDraft 1.0.1 -> what is new since version 0.9.211

As of version 1.0.1

With the update to version 1.0.1, LiberoDraft leaves the 0.9 series. Since v0.9.211 around seventy intermediate versions have appeared - new drawing and dimensioning functions, a fundamentally faster display, a clean DXF exchange and many improvements in daily use. The following list gives the changes, the newest first.



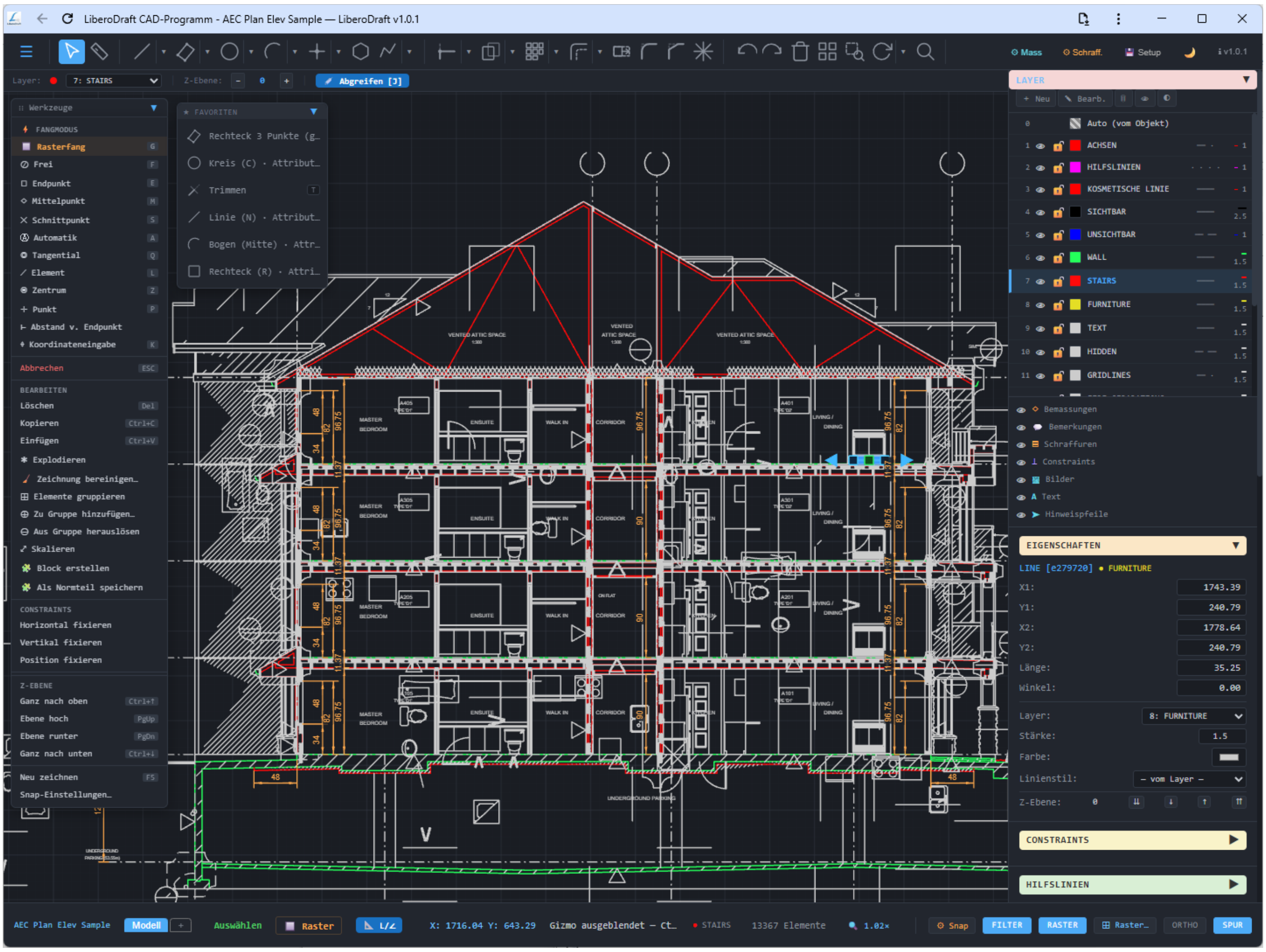
[Figure: overall view of LiberoDraft 1.0.1]

68.1 The most important points in brief

68.1.1 Considerably faster with large drawings

[Figure: considerably faster with large drawings]

A scene cache, block patterns and a targeted hatch update have brought work on very large files down from seconds to milliseconds: drawing an image of a drawing with thousands of elements from about two seconds to about 0.14; cancelling a complex



function with huge data sets from 30 seconds to 0.12; the first working step after loading from 39 seconds to 0.08.

68.1.2Dimensioning identical on all three output paths

Screen, print and DXF draw dimensions along separate paths. In a round of its own, angular and arc dimensions, arrow shapes, the slash style of the construction standard and the settings of every single dimension were aligned on all three.

[Figure: dimensioning identical on all three output paths]

68.1.3Blocks: hide, nest, evaluate

Blocks can be hidden -> together with their dimensions and hatches and across all nesting levels. The block manager shows a tree, and on request the parts list also counts hidden blocks.

[Figure: blocks: hide, nest, evaluate]

68.1.4New drawing and hatching functions

Relative angle, window zoom, a visible and movable hatch seed point, hatching across blocks and manual area definition through the viewport.

[Figure: new drawing and hatching functions]

68.2 All changes since v0.9.211

The list is ordered by version, the newest at the top.

`v1.0.1` **First release** (New)

After the stabilization phase was completed, LiberoDraft appears as version 1.0.1. In the final check before release, two mute buttons of the properties panel were found and fixed: "Edit properties" for images and "Edit text" for text now reliably open their dialogs.

v0.9.285 Leader properties are adopted (Fixed)

In the properties dialog of a leader, the text, arrow size, text size, text offset, fill and arrow side had no effect. All six fields now take effect immediately, can be undone and are saved with the drawing.

v0.9.284 Instructions and tooltip right from the tool call (Improved)

The angle bisector, the perpendicular and the relative angle showed no hint after being called up, neither in the instruction field nor in the tooltip flying along at the mouse pointer. Both now guide you through the function step by step -> in German and English. With the relative angle, the object snap switches to automatic after the reference line has been chosen.

v0.9.283 Locking the angle with the Tab key (Improved)

With the relative angle, the angle typed in can be locked with TAB: the direction stays put and the mouse only determines the length. In addition, the context menu now shows plain-text names under "Recent commands" instead of internal abbreviations.

v0.9.282 New line function "Relative angle" (New)

A line can be drawn at an angle to a freely chosen reference line: click the reference line, set the start point, click the endpoint -> or type in the angle and the length. The angle is measured from the direction of the reference line, positive counter-clockwise.

v0.9.281 Window zoom stays active (Improved)

The window zoom no longer ends after the first frame. You can work your way towards a spot with several frames one after another; it is ended with ESC or the right mouse button.

v0.9.280 A changed hatch appears immediately (Fixed)

If you drag a line straight through a hatch, the hatch area now adapts immediately. Previously the change only became visible after a zoom.

v0.9.279 No more pause for thought with very large files (Faster)

In drawings with many blocks and hatches, the first working step after loading could take almost forty seconds. Measured on a file with 7,680 elements and 180 hatches: 39 seconds become 0.08 seconds -> with a point-for-point identical result.

v0.9.278 Grid and preview clearly visible again in day mode (Fixed)

After the switch to the second text level, the grid and the drawing preview appeared almost invisible in light mode. The cause was the alpha blending of the drawing area; it has been corrected.

v0.9.277 Texts can lie behind the drawing (New)

Until now, texts and dimension values always lay above the geometry. With a second text level below the drawing, lines can now be placed over text as well -> controlled via the Z level.

v0.9.276 Z level also applies to images and annotations (Fixed)

The order of the Z levels applied to the geometry but not to images, remarks, item balloons, datum features and feature control frames. Now it applies to everything.

v0.9.275 Panel switches take effect immediately (Fixed)

Showing and hiding dimensions, hatches, texts and leaders only took effect after a zoom. In addition, dimensions can no longer be attached to text blocks or other annotations by accident.

v0.9.274 DXF adopts the settings of every single dimension (Fixed)

The DXF export read the arrow shape and text height once globally. Drawings with differently set dimensions were therefore exported uniformly. Now every dimension travels with its own settings -> including the slash style of the construction and timber construction standard.

v0.9.273 Angular dimensions in DXF as on screen (Fixed)

In the DXF, the arrows of an angular dimension always stood inside and overlapped, and the degree value always lay horizontally. Both now follow the screen display. In addition, the dimension value of an arc dimension sits on the arc again in the printout.

v0.9.272 Arc arrows in print and in DXF too (Improved)

The arrowheads adapted to the dimension arc now apply on all three output paths -> screen, print and DXF.

v0.9.271 The dimension arrow follows the dimension arc (Fixed)

The arrowheads of angular and arc length dimensions sat on the tangent as a straight triangle and lay visibly beside the arc at small radii. They are now built up along the arc.

v0.9.270 Angle arc and dimension arrows stay proportional (Improved)

An arrow takes up at most a quarter of the dimension line, and the angle arc no longer wanders while zooming.

v0.9.269 Through-dimensions no longer wander (Fixed)

Dimensions pointing through a viewport onto the drawing shifted when the viewport scale was changed. They now stay stuck to the geometry.

v0.9.268 Double-click on dimensions over images (Fixed)

If a dimension lay over an inserted bitmap, the double-click opened the image dialog instead of the dimension editing.

v0.9.267 Dimensioning over inserted images (Fixed)

Over a bitmap -> a scanned cadastral plan, for example -> lines could not be dimensioned because the image intercepted every click.

v0.9.266 Through-dimensions follow the viewport scale (Fixed)

If you change the scale of a viewport, the dimensions reaching through stay at their geometry.

v0.9.265 Dimension attributes reachable faster (Improved)

The settings of a dimension can be opened via the context menu, with Ctrl+double-click and via a button in the dimension dialog.

v0.9.264 Zooming out further (Improved)

The lower zoom limit was reduced to 0.001 -> useful with very large cadastral plans.

v0.9.263 Construction dimensioning: whole meters without decimal places (Improved)

With "Hide zeros" active, 2.00 becomes a plain 2. Values such as 2.40 or 2.36^5 stay unchanged.

v0.9.262 Hidden blocks take their dimensions with them (Fixed)

If you hide a block, the dimensions and hatches it contains disappear as well -> with nested blocks likewise.

v0.9.261 Annotations respond immediately again (Fixed)

The datum feature, feature control frame, item balloon and remark respond without delay again when their grips are dragged. In addition, an edited element occasionally remained colored blue.

v0.9.260 Changes via the grips appear immediately (Fixed)

After releasing a grip or the gizmo, the old position remained until something else refreshed the image.

v0.9.258 Radius and diameter dimensions survive the DXF round trip (Fixed)

A LiberoDraft DXF read back into LiberoDraft lost the radius and diameter dimensions. On import they now find their element again via its position.

v0.9.255 Dimensions and hatches of nested blocks visible (Fixed)

If dimensioned blocks were combined into a new block, their dimensions disappeared from the drawing. The resolution now works across all nesting levels.

v0.9.254 Paper space with a scene cache (Faster)

The sheet area now uses the cache as well. Measured on a very large file: 673 milliseconds per layout image become 42.

v0.9.253 Drawing bundles merged (Faster)

Capturing a very large drawing drops from 3.4 to 2.0 seconds.

v0.9.252 Block patterns instead of individual build-up (Faster)

Every block definition is captured once and merely offset for each placement. Capturing a very large drawing drops from 6.0 to 3.4 seconds.

v0.9.251 Zooming without a rebuild at every step (Faster)

When turning the mouse wheel, the scene is no longer rebuilt for every step but only once the view has stood still for 0.3 seconds. Five notches together cost 0.36 seconds instead of several seconds per notch.

v0.9.250 Coloring on hover and a steady dimension line (Fixed)

Elements turn blue again when hovered over. And a dimension line that has been grabbed no longer jumps away by a few pixels.

v0.9.249 Moving a dimension value, and undo per individual dimension (Fixed)

After moving a dimension value, the old dimension line remained. In addition, undo now takes back each dimension individually instead of all of one pass at once.

v0.9.248 Dimensions and texts in the cache (Faster)

Dimensions and texts also run through the scene cache; the image of a very large drawing takes about 50 milliseconds.

v0.9.247 Scene cache (Faster)

The drawing is captured once into graphics buffers and then output in one go. The image of a very large file drops from about two seconds to about 140 milliseconds.

v0.9.246 Tooltips legible in light mode (Fixed)

In day mode the title of the tooltips stood dark on a dark background.

v0.9.245 A guard for the language file (Improved)

If the language file does not match the program version, this is reported bilingually at startup.

v0.9.244 Copying the seed point, and a drag hint (New)

With the Ctrl key held down, the seed point of a hatch can be copied; while dragging, an explanatory hint appears.

v0.9.243 Hatches are updated selectively (Faster)

A hatch is only searched for anew when its surroundings have really changed. Cancelling a drawing function in a very large file drops from 30 seconds to a good tenth of a second.

v0.9.242 Object snap with the mouse at rest (Faster)

In very large drawings the snap lagged behind while hovering. It now latches on as soon as the mouse stands still briefly; a click always snaps immediately.

v0.9.241 DXF blocks keep their dimensions and hatches (Improved)

A block with its own dimensions and hatches stays a real block in the DXF and is restored completely when read back in.

v0.9.240 Mirroring blocks and ellipses correctly (Fixed)

When mirroring, block placements and elliptical arcs tipped over incorrectly.

v0.9.239 A hint at the seed point (Improved)

If you move over the seed point square of a hatch, a tooltip explains what it is for.

v0.9.238 A steady frame during the window zoom (Faster)

The dragged frame lies as a screen overlay over the drawing and costs practically no time to update.

v0.9.237 Window zoom (New)

An area can be framed with two clicks or by dragging and zoomed to fill the view.

v0.9.236 Object snap considerably faster (Faster)

The snap in large drawings drops from 57 to about 1 millisecond.

v0.9.235 Defining an area by hand through the viewport (New)

A hatch area can also be traced by hand when you are pointing at the model through a viewport.

v0.9.234 In the viewport the sheet takes precedence (Improved)

A click in the viewport first hatches the sheet geometry; only if there is nothing there does it reach through to the model.

v0.9.233 Boundary search in dense drawings (Faster)

Creating a hatch in a dense drawing drops from 170 seconds to a few seconds -> with the same result.

v0.9.232 Hatches in the viewport (Faster)

Creating one through the viewport drops from 1.0 seconds to 0.2, and recalculating from 0.7 to 0.04 seconds.

v0.9.231 Seed point visible and movable (New)

With a hatch selected, a black square shows the starting point of the area search. If you drag it into another chamber, the area is searched for there anew.

v0.9.230 Creating a hatch accelerated (Faster)

While creating one, LiberoDraft searches within a growing box instead of the whole drawing: 1.1 seconds become 0.23.

v0.9.229 Boundary search with blocks (Faster)

If the drawing contains blocks, their edges are no longer resolved in advance; islands are only searched for inside the area that was found.

v0.9.228 Deleting in the block manager explains itself (Improved)

The button shows before the click whether it can act -> and otherwise names the reason in the window.

v0.9.227 Quantity and structured BOM (Improved)

Both views are necessarily identical as long as no assembly is marked for breaking down. Only with "Break down assembly" does the evaluation descend into the sub-blocks.

v0.9.226 The parts list asks about hidden blocks (Improved)

If the drawing contains hidden blocks, LiberoDraft asks once per call whether they should be counted.

v0.9.225 Larger click areas in the block tree (Improved)

The expand triangle is considerably larger; blocks contained in other blocks are marked more clearly.

v0.9.224 Tree view of the block list (New)

The block list shows the nesting as a tree. The search goes beyond the name, and a click on the drawing area also clears the marking set via the list.

v0.9.223 List rows follow the selection (Improved)

Whatever is selected in the drawing is marked in the panel and in the block manager.

v0.9.221 Hiding blocks (New)

Blocks can be hidden via the panel and the block manager -> nested sub-blocks included.

v0.9.220 Hatching across blocks (New)

For the area search, blocks are broken down; a hatch can therefore end at block edges.

v0.9.219 Calmer button appearance (Improved)

The special coloring of the direction button in the numerical field is gone -> it looks like every other one.

v0.9.218 Buttons in stationary fields (Fixed)

Numerical fields that do not follow the mouse let their buttons be clicked reliably.

v0.9.217 The numerical field follows the mouse (Improved)

The floating numerical field accompanies the pointer while drawing, and every button in it can be reached with the Tab key.

v0.9.216 A visible Apply button (Improved)

The floating input fields have a button of their own; until now a name could only be confirmed with the Enter key.

v0.9.215 Negative scaling mirrors the block content (Improved)

A block with a negative scale is genuinely mirrored -> the content tips over with it.

v0.9.214 Mirroring blocks while inserting (New)

Horizontal and vertical mirroring already take effect in the insertion preview.

v0.9.213 A steady status bar (Fixed)

The fields of the status bar no longer wrap within themselves, and the layout bar can be scrolled. Previously ten layouts pushed half the status bar away.

69 Useful links

Here is a list of some useful links on the subject of LiberoDraft:

69.1 Further CAD programs

- 2D } 4MCAD: https://www.cadtec.ch/4MCAD/
- 3D } **Alibre Atom3D**: https://atom3d.alibre.ch
- 3D } **Alibre Design EXP**: https://www.alibre.ch/
- 3D } ZW3DCAD and ZW3DCAM: https://www.zcad.ch/
- 3D } Kompas 3D: https://www.k3d.ch/

69.2 CAM program

- 2x and 3x milling, turning, drilling, cutting, engraving }

  SimplexCAM: https://www.simplexcam.ch/
- NC-Corrector: https://www.cadtec.ch/NC-CORRECTOR/

70 Contributors

There are many people who have contributed to the LiberoDraft user manual. Some contributions came via the LiberoDraft forum, the wiki or the source code.

Some of those who have contributed directly or indirectly are:

André Hurler (administrator)

David Fischer

Patrick Auberson

70.1 Corrections, improvements, errors found

Please report them to the following e-mail address: info@cadtec.ch

71 Index

Alphabetisch sortiert:

4

A

B

C

D

E

K

L

M

N

O

P

R

S

T

U

The free LiberoDraft book

From beginner to professional - discover the functions and secrets of this powerful CAD tool!

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