

Plan 8.0

Welcome to sPlan 8.0	4
New in version 8.0	6
Main window	9
Drawing of circuit diagrams	11
Page settings	12
Select, Move and Delete Elements	13
Sizer and Changer	15
Grid settings	18
The lower Statusbar	21
Drawing functions	24
Drawing functions	25
Lines	26
Connections	28
Rectangles, rounded Rectangles	30
Circles, Ellipses or Circle segments	32
Polygons (closed areas)	34
Bezier curves	36
Freehand Lines	38
Special Forms	40
Dimensions	42
Text	45
Extended Text Edit	47
TextRect (multiline)	49
Bitmaps	51
Clipboard functions	53
Copy a Drawing to the Clipboard	55
Rotate, Mirror, Set to front / back	56
Align Elements	60
Colorize Elements	61
Build / Split Groups	62
Magnetic Lines	63
Using Forms	65
Multi-Page Projects	68
Zooming	71
General Settings	73
General Settings	74
Display	76
Directories	79
Library	80
Grid	82
Autosave	85
New page	86
Hotkeys	87
Components	88
Components	89
What is a Component?	90
Component Properties	91
The Component-Editor	95
Contacts	97
Create own Components	99

Split Components	100
Change Component inscription	101
Automatic Component Numbering	102
Parent-Child Relations	105
Component List	113
The Component Library	119
The Component Library	121
Representation of the Library	123
Edit a Library page	124
Variables / Text Constants	128
Fixed Variables	129
User Variables	136
Text Constants	138
Miscellaneous Functions	140
Active Text-Links	141
Convert Line <> Polygon	144
Printing	147
Export	151
Bitmap-Explorer	154
Paste Table of Contents	155
Scale	157
Search / Replace	158
Keyboard Control	162
Translation into other languages	164
Software-Registration	166
Tips and Tricks	167
The "sPlan-Viewer"	168

Welcome to sPlan 8.0



Welcome to sPlan 8.0

sPlan enables you to create drawings (or other drawings) easily and conveniently with the help of your computer. All components are simply dragged onto the drawing area and aligned. With the help of the saved snap grid, precise work is guaranteed at all times.

All components and drawing elements are included as independent elements on your drawing. You can group, move, rotate, copy, cut, paste, delete ...

In addition to the functions for drawing drawings, sPlan also provides you with other interesting options such as automatic component numbering or integrated component list management.

Much emphasis was also placed on the print version. In the convenient print preview you can freely set the scaling, position the printout, etc. All changes are immediately visible in the print preview.

The supplied component library can be expanded as required. You can set up completely new libraries and thus adapt and expand your component library according to your wishes.

The software was designed in such a way that it can be operated as simply and intuitively as possible. After a short training period, you will notice how easy it can be to create drawings quickly and conveniently.

Further information and updates are available on our homepage:

www.abacom-online.de

www.electronic-software-shop.com

Copyright © ABACOM GbR, Germany. All Rights Reserved.

New in version 8.0

[Welcome to sPlan 8.0](#) >>



For those switching from version 7.0, there is a list of all the main innovations in sPlan 8.0:

New graphics engine

sPlan 8.0 now uses the modern Windows graphics engine (Direct2D). This results in a much better and also a considerably faster graphic output. Now you can zoom and navigate smoothly and smoothly even in more complex drawings, which makes working with sPlan much more convenient.

High resolution monitor support (high DPI scaling)

sPlan 8.0 now directly supports scaling on high-resolution monitors. sPlan adopts the scaling set in the Windows system and scales itself to it.

Properties panel

With the new properties panel on the right edge of the window, all properties of elements can now be changed or viewed directly at any time. It is no longer necessary to call up a dialog by double-clicking. Any changes to a property are also immediately visible on your drawing. In this way you work even more effectively and intuitively.

New component library

The newly revised component library is now multi-level (with sub-categories). This allows the library to be sorted in a much more structured and clear manner. The entire library can now be backed up or a backup can be restored at the push of a button.

Unicode

sPlan now supports the full Unicode character set. This means that sPlan can display all characters, such as Greek letters or special mathematical characters.

Numeric entry

Size specifications of individual elements such as circles or rectangles or position specifications can now also be specified numerically directly in the properties if required.

New component lists

Component lists can now be defined more precisely. In addition, if desired, component lists can be inserted directly into your page in a kind of table.

Table of Contents

If desired, a table of contents can be inserted directly into your drawing as a kind of table.

Angle snap when drawing lines

This function allows the lines to snap into place at 45 ° angles when drawing or editing. This enables you to automatically and intuitively create precisely aligned connections, even outside the set snap grid.

Bitmap functions

The resolution (dpi) of inserted bitmaps can now also be adjusted (reduced) directly in sPlan. A reasonable reduction can save a lot of storage space. There is also a bitmap explorer with which you can list all the bitmaps inserted in a project. This enables you to track down unnecessary "memory hogs".

Text links / page references

There are now some new variables for automated designations available for creating text links or page references.

Conversions line <> polygon

Each line can now be converted into a polygon and each polygon can be converted into a line. That sounds unspectacular at first, but it is a very valuable aid for a lot of drawing work.

Hatching

sPlan 8.0 now uses its own hatching technique to fill elements. Hatching can now also be precisely defined with line width and line spacing. This new technique also eliminates any problems with hatching when printing.

Freehand lines

A new drawing mode allows you to easily draw freehand lines.

Page list

A new optional vertical page list makes it easier to navigate through the pages, especially for very large projects with more than 20 pages.

Hotkeys for the drawing modes

With the (editable) hotkeys for the individual drawing modes, you can switch between the modes even faster if necessary.

Full screen mode

With the full screen mode you can display your drawing in full format without any window frames or other elements on the entire monitor. This can be particularly useful for presentations.

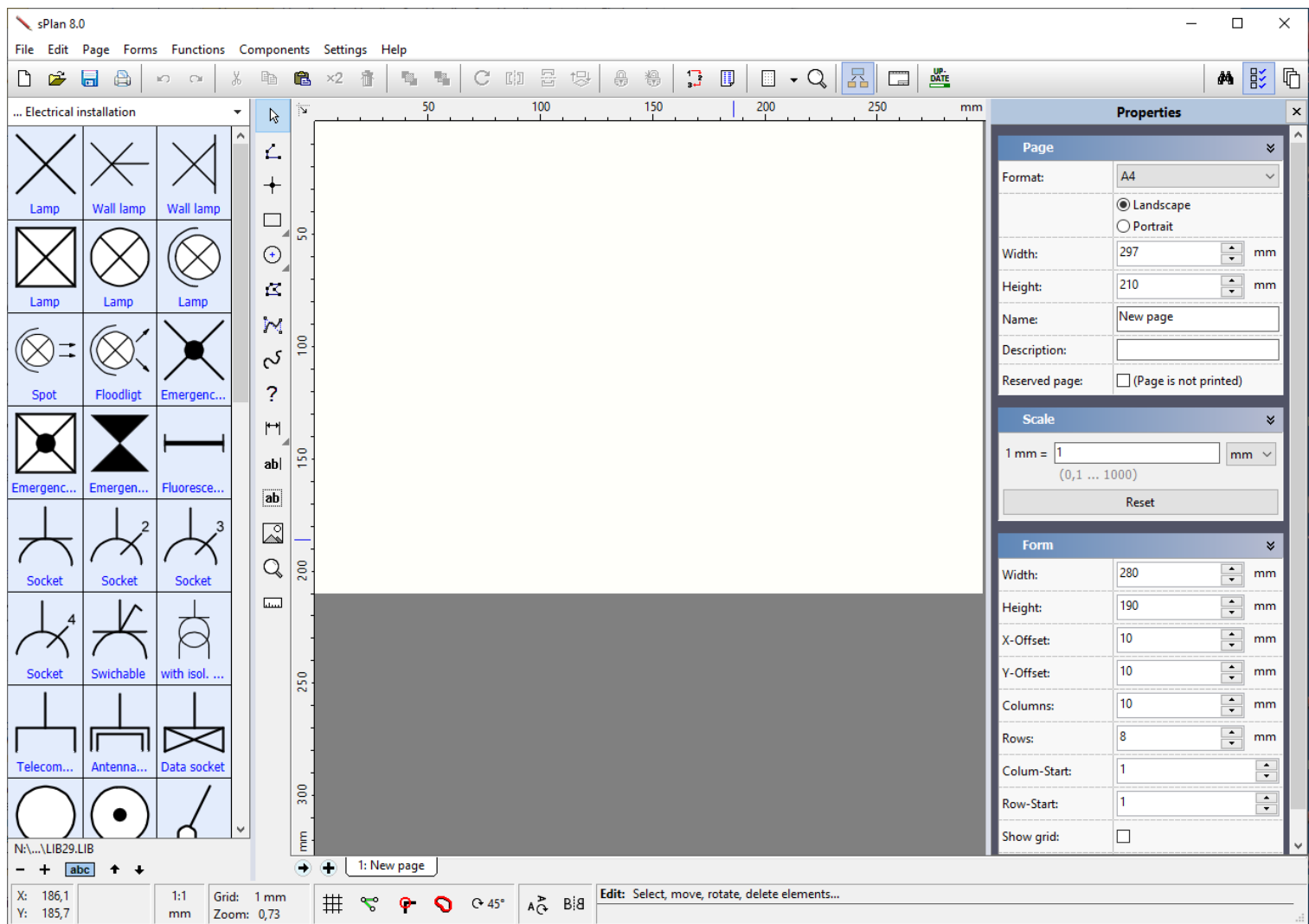
... and many other functions and improvements

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Main window

[Welcome to sPlan 8.0](#) >>

The main windows of sPlan consists of different areas:



The toolbar is located below the main menu. Here you can call up many of the program's functions with a single click. If you hover the mouse over a button for a while, a short text for the toolbar button appears at the mouse position.

The component library with a light blue background is located on the left edge of the window. Here you can select the individual library pages and drag the components onto the drawing. You can determine the width of the component library with the mouse. To do this, move the mouse to the right border of the library. The mouse pointer will now turn into a double arrow. You can now determine the width of the library by holding down the mouse button. Directly below the library you will find some buttons with which you can influence the display of the library. Here you can, for example, set the number of columns or hide the component names.

The mode toolbar for the individual drawing modes is located next to the component library.

To the right of it is the light yellow drawing area. Use this to create your drawing.

The properties panel is located to the right of the drawing area. Here you can change the properties of individual elements or the page directly.

The tab-list for the pages is located under the drawing area. Here are the "tabs" for the individual pages of the drawing. A click on the corresponding tab is sufficient to display the respective page.

The status bar is located at the bottom of the window. Information such as coordinates is displayed here. You also make basic settings here, such as the grid size or the snap mode.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Drawing of circuit diagrams



Copyright © ABACOM GbR, Germany. All Rights Reserved.

Page settings

[Drawing of circuit diagrams](#) >>


At the beginning of the work it makes sense to determine the format of the drawing.

The properties of a page are displayed in the properties panel on the right edge of the window if no other element is currently selected on the plan:

Page	
Format:	A4
	<input checked="" type="radio"/> Landscape <input type="radio"/> Portrait
Width:	297 mm
Height:	210 mm
Name:	New page
Description:	
Reserved page:	<input type="checkbox"/> (Page is not printed)

Here you can set the page size and the page format.

The name names your page. This name also appears as text in the page tab-list below.

You can also enter a more detailed description under Description. This extended description appears as a tooltip on the page tab-list when the mouse is over it.

With the Reserved page option you can define pages as placeholders. Sometimes it is important that the page numbers do not change when a new page is inserted. If you know this beforehand, you can insert reserved pages in advance. When printing, these reserved pages are simply skipped so that they do not use up paper unnecessarily.

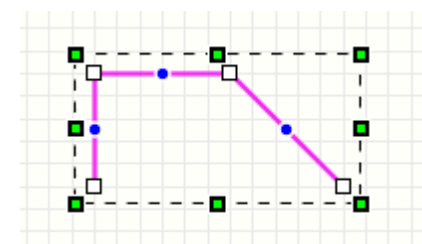
Select, Move and Delete Elements

[Drawing of circuit diagrams](#) >>



Select

After you have drawn the elements you want, you can change them at any time. You can move them, change their size or shape, choose new colors, etc. To change an element, you must first select it. If you have not already done so, switch back to standard mode. Then simply click on the desired element with the mouse. If the element has no filling, e.g. a hollow circle, then please click on the edge of the element.



You can recognize selected elements at any time by their color scheme (magenta) and the small green/black box [sizers](#) around them.

Select several elements

Sometimes it is necessary to select several elements at once. To do this, please click with the mouse on an empty space in the drawing. Then draw a frame with the mouse button still held down. When you let go of the mouse, all elements that lie within or only partially within the frame are selected.

For particularly tricky selections, you can also work with the <SHIFT> key. If you keep this pressed, you can select all the desired elements one after the other with the mouse without losing the previous selection. You can also deselect a selected element by clicking on it. With these functions you can easily create complicated selections.

Move

To move elements, they must first be selected. Then click with the mouse on one of the selected elements (please not on the sizer or changer) and move it with the mouse to the desired position. All other selected elements are also moved.

Tip:

You can also move the selected objects with the cursor keys on the keyboard. The elements are shifted by one grid point in the desired direction. With the <CTRL> key you can refine the grid by a factor of 10 and thus position it more finely using the keyboard.

Delete

To delete elements, they must first be selected. Then execute the "Delete" command. You can reach this either via the "Edit" menu or directly with the "DEL" key on the keyboard. Alternatively, you can also click the corresponding toolbar button. All selected elements will be removed from your drawing.

Scaling, rotating and shearing

Selected elements can be scaled (changed in size), compressed and stretched, rotated or sheared as required. You can do all these functions easily with the mouse using the [sizer and changer](#).

Copyright © ABACOM GbR, Germany. All Rights Reserved.

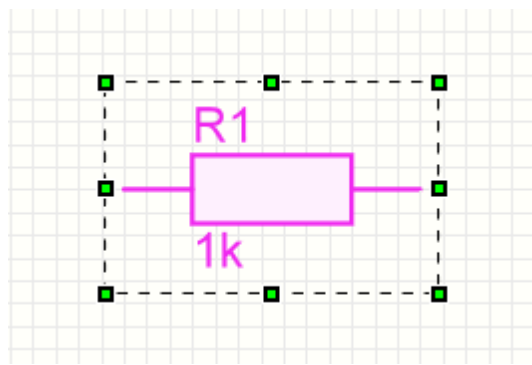
Sizer and Changer

[Drawing of circuit diagrams](#) >>



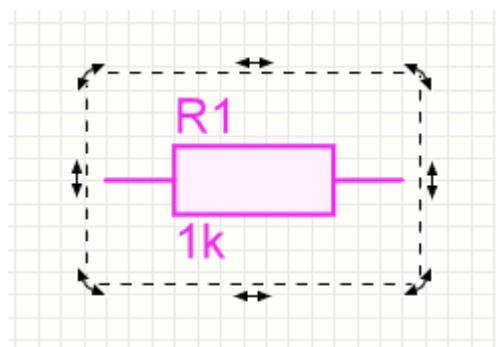
Sizer

If an element (or several) is selected, you can recognize this by the fact that it is colored magenta (pink) and surrounded by 8 black/green boxes, the so-called sizers.

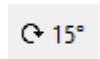


With these sizers you can freely scale the element. If you move the mouse over one of these sizers, you will recognize the possible stretching direction from the changing mouse pointer. Just click on the sizer you want and drag the mouse in the direction you want. The element can be enlarged, reduced, compressed or stretched as required.

If you click again with the mouse on a selected element, the sizers change to arrows.



With the arrows you can now turn and shear the element as you like. Use the 4 semicircular corner arrows to turn. The angle rotation always locks into the angle of rotation snap you set. You set the rotation angle snap in the lower toolbar:

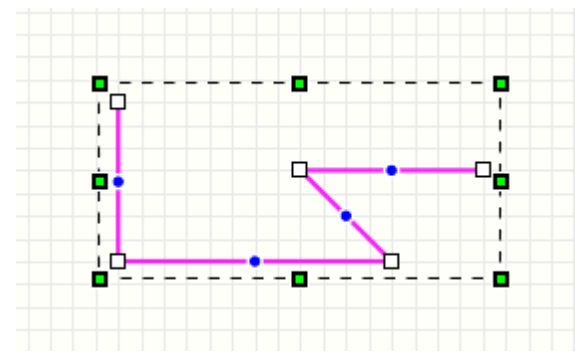


Tip:

With the CTRL key you can temporarily switch off the angle of rotation snap at any time, i.e. as long as you keep the key pressed.

Changer

Some drawing elements can also be changed with the help of the changer. These are only visible if the corresponding element is selected on its own.



The changers are small white boxes that lie on top of the element. Click on a changer and move the mouse until you reach the change you want. With the changers you can change the following drawing elements:

Lines:

You can move the start, stop and node of the line. With a right-click on a node, you can delete and insert nodes here, or split the line into two individual lines.

Polygons:

You can move all of the nodes of the polygon. With a click with the right mouse button on a node you can delete or insert nodes here.

Bezier curves:

You can move the start and stop point as well as the control points of the bezier curve.

Circles:

You can freely determine the start and stop point of the circular outline. This gives you the option of creating arcs or segments.

Rectangles:

You can determine the rounding of the corners. With this you can easily create oval shapes or rounded rectangles, for example.

Dimensions:

You can redefine the positions of the extension lines.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Grid settings

[Drawing of circuit diagrams](#) >>

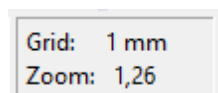


When drawing with sPlan, a saved snap grid is normally always activated. With this snap grid you can no longer move to all positions on the drawing, but only "jump" from one grid point to the next grid point. This function is of great help for the exact positioning and connection of components. You can see and change the grid width of this stored snap grid at any time in the lower status bar.

On the screen, the grid becomes larger or smaller as you zoom in or out. If you zoom out too far so that the individual grid lines would almost touch each other, the grid is automatically hidden.

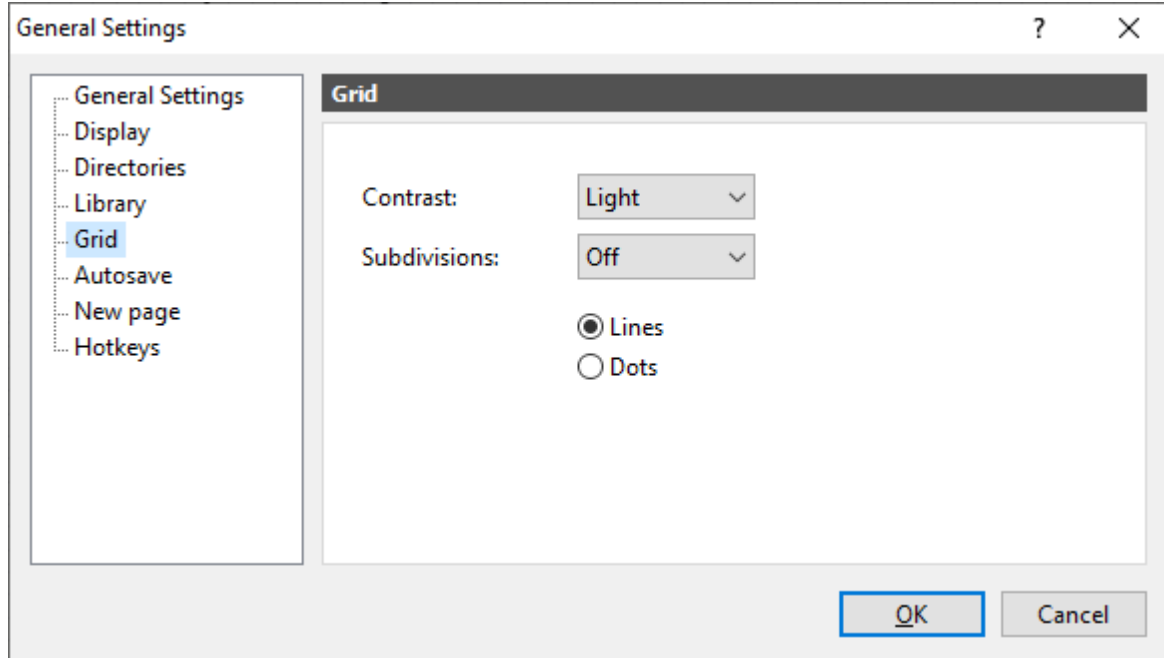
Incidentally, the grid is only ever visible on the monitor. This is not included in the printout.

The current grid is always displayed in the lower status bar:



With a click on this grid value you can set it directly.

To change the optical properties of the snap grid, call up the General Settings dialog either via the "General Settings ..." menu item in the "Settings" menu and then select the "Grid" entry on the left, or simply click on the corresponding toolbar button.



Contrast:

Here you enter the display strength of the grid.

Depending on your monitor or your habits, you can use it to adapt the display of the grid to your requirements.

Subdivisions:

This allows you to subdivide the grid again.

With a setting of "5", for example, every 5 grid lines are displayed a little stronger. This makes orientation on the grid much easier, e.g. when counting.

Lines / Dots

Determines whether the grid is displayed in lines or dots.

Tip:

With the <CTRL> key you can temporarily switch off the grid snap at any time, i.e. as long as you keep the key pressed. This is sometimes helpful, especially in tricky areas.

If you want to work without the grid snap for a longer period of time, you can deactivate it with the button in the lower status line.



Your grid settings are retained, only the snap is deactivated.

To reactivate the grid snap, simply click the same button again.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

The lower Statusbar

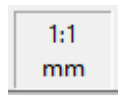
[Drawing of circuit diagrams](#) >>



The statusbar is located at the bottom of the sPlan window:

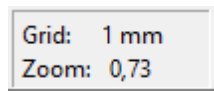


Here you can read the current mouse coordinates at any time, but also make some settings:



The current Scale.

Click here to change the scale in the properties panel.



The current grid and the current zoom factor.

Click here to change these values.



Here you can switch the grid snap on / off.

You can also temporarily deactivate the grid snap at any time with the <CTRL> key.



Here you can switch the angle snap on / off when drawing lines.

With the <SHIFT> key you can temporarily deactivate the angle snap when drawing lines at any time.



Here you can switch the connection snap on / off.

With the <ALT> key you can deactivate the connection snap at any time for a short time.

The connection snap is a very helpful function of sPlan. In addition to the grid snap, this function activates an automatic snap to all nodes of lines as well as the start and end points of circular arcs and bezier curves are to be understood as connection points here.

As soon as the mouse position approaches a connection point during corresponding actions, the connection snap captures the mouse position. A small red circle will then be displayed around the connection point to indicate that the mouse position has been captured by a connection point. With the connection snap, exact connections can be made quickly and safely.



Here you can switch the rubber band function on / off.

The rubber band function is intended to help hold connections in the drawing together when individual components or elements are moved. If a connected component is moved and the rubber band function is activated, all connection lines are taken along by the component.

sPlan can only clearly recognize lines on the components if they are correctly connected. If the lines overlap the component connections are not recognized as "connected" and are therefore not included. Likewise, the rubber band function cannot work if 2 components are connected without a cable. Then there is no line that could take the rubber band function with it.

After a component and its connections have been moved, the connection lines often have to be reworked. If you select the line, you can edit the nodes of the line with the changers. You can also remove or add individual nodes. To do this, right-click on a changer that is on the line. Then you get a small menu in which you can remove the point or insert another point.



Here you can set the rotational angle snap.

The rotational angle snap determines the grid of the rotation angle when you freely rotate a component or an element with the mouse.

With the <CTRL> key you can deactivate the angle of rotation snap at any time briefly and thus rotate the element freely.



With this you determine the behavior of texts when they are rotated together with other elements.

The text is either always completely rotated, or only the text position is rotated.



With this you determine the behavior of texts when they are mirrored together with other elements.

The text is either always completely reflected, or only the text position is reflected.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Drawing functions

[Drawing of circuit diagrams](#) >>



With sPlan you have all the elementary drawing functions you need to create sophisticated drawings:

[Lines](#)

[Connections](#)

[Rectangles, rounded rectangle](#)

[Circles, circle segments](#)

[Polygon \(closed area with or without filling\)](#)

[Bezier curves](#)

[Freehand lines](#)

[Special forms \(polygons, stars, tables, sine curves,...\)](#)

[Dimensions](#)

[Text \(simple text label\)](#)

[TextRect \(multiline text for larger texts and descriptions\)](#)

[Bitmaps \(for inserting logos or other graphics\)](#)

A separate mode is available for each of these elements. Use the button bar (between the library and the drawing area) to select the desired mode.

To select, edit and move the elements, please switch back to standard mode again. A click with the RIGHT mouse button on the drawing or with the <ESC> key is sufficient.

Tip:

The drawing area has its own local menu (context menu). You can do this by moving the mouse to the drawing area and clicking the RIGHT mouse button once. The local menu gives you quicker and more convenient access to many commands.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Drawing functions

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



With sPlan you have all the elementary drawing functions you need to create sophisticated drawings:

[Lines](#)

[Connections](#)

[Rectangles, rounded Rectangles](#)

[Circles, Ellipses or Circle segments](#)

[Polygon \(closed area with or without filling\)](#)

[Bezier curves](#)

[Freehand lines](#)

[Special forms \(polygons, stars, tables, sine curves,...\)](#)

[Dimensions](#)

[Text \(simple text label\)](#)

[TextRect \(multiline text for larger texts and descriptions\)](#)

[Bitmaps \(for inserting logos or other graphics\)](#)

A separate mode is available for each of these elements. Use the button bar (between the library and the drawing area) to select the desired mode.

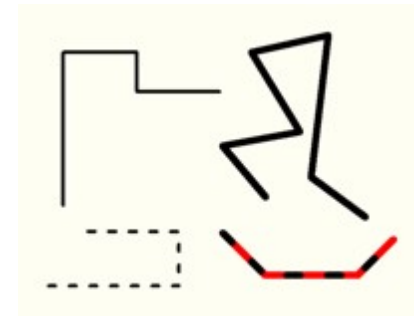
To select, edit and move the elements, please switch back to standard mode again. A click with the RIGHT mouse button on the drawing or with the <ESC> key is sufficient.

Tip:

The drawing area has its own local menu (context menu). You can do this by moving the mouse to the drawing area and clicking the RIGHT mouse button once. The local menu gives you quicker and more convenient access to many commands.

Lines

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Drawing a new line



Select the appropriate drawing mode.

Click with the mouse on the drawing to determine the starting point of the line and then release the mouse button again. Now click on the next desired corner point of the line and release the mouse button. You can "lay" your line over any number of corner points.

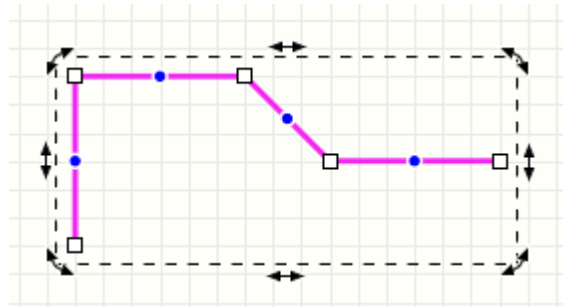
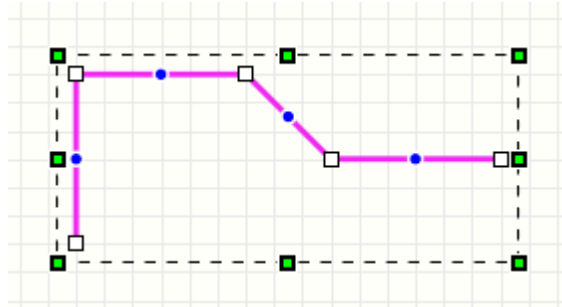
To stop drawing the line, click the right mouse button once. You can now start a new line or switch back to standard mode by clicking the right mouse button again.

The line is initially always created with the preset properties for a line (width, color, etc.).

If you have selected the line mode but have not yet started drawing, you can change these preset properties here directly.

Change an existing line

If you select a line on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time. With the [changers](#) you can move all nodes of the line. So you can easily change the line afterwards.



You can also create new additional nodes at a later date. For this, there is a so-called virtual node in the middle of each line segment, shown as a blue dot. If you click on it and move it, this virtual node becomes a new real node. So you can interactively expand your line with new nodes.

You can also remove individual nodes or split the line into 2 parts. To do this, right-click on a node. Then you get a small menu in which you can remove the node, among other things. You can also split the line at this position here and thus get 2 individual lines.

Change the properties of lines

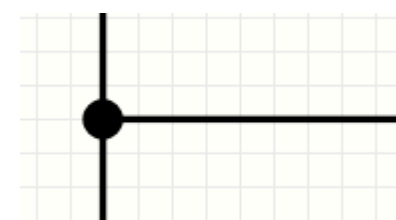
If you select a line, you can set its properties in the properties panel on the right edge of the window. You can also select several lines at the same time and thus change their properties together.

Connections

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Connections are the connection points in a drawing. They indicate whether 2 crossing lines are electrically connected.



Place a new connection



Select the appropriate drawing mode.

Click with the mouse on the drawing to set a new solder point at this position. With every further click you can now set further solder points. To exit the soldering point mode, click once on the right mouse button.

A connection is initially always created with the preset properties for the solder point.

If you have selected the solder point mode, you can change these preset properties directly in the properties panel.

To delete a solder point, select it and press the <ENTF> key.

Change properties of connections

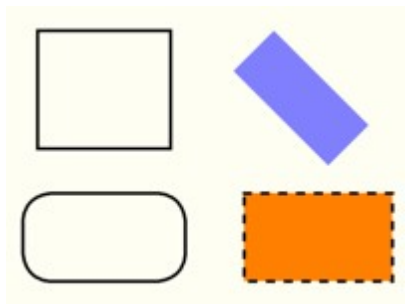
If you select a connection, you can set its properties in the properties panel on the right edge of the window. You can also select several connections at the same time and thus change their properties together.

Automatic

The automatic function is a special setting. If this function is activated for a connection, the connection automatically adapts to the line on which it lies in terms of size and color. So if you change the line width or the line color, the connection also changes automatically. With Auto-Size you can define the adjustment of the connection to the line width.

Rectangles, rounded Rectangles

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>

Drawing a new rectangle



With a click on the small triangle you can choose whether you want to draw out the rectangle as a frame (starting point above / left) or from where the starting point determines the center of the rectangle.

Now click with the mouse on the drawing to determine the starting point of the rectangle. Now, with the mouse button held down, draw a frame with the mouse to determine its size and shape. When you release the mouse button, the rectangle is created as an element on your drawing.

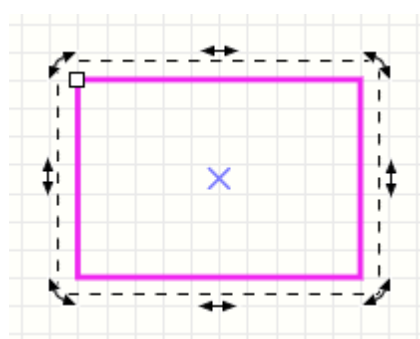
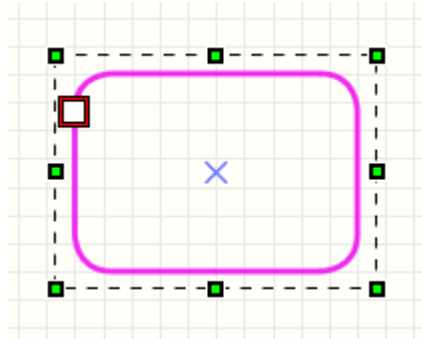
If you hold down the <SHIFT> key while drawing, you will force a proportional aspect ratio, i.e. a square.

The rectangle is initially always created with the preset properties for an area (outline and filling).

If you have selected the rectangle mode but have not yet started drawing, you can change these preset properties here directly.

Modifying an existing rectangle

If you select a rectangle, you can use the [sizers](#) to change its size, shape and direction of rotation at any time.



With the [changer](#) you can adjust the roundness of the rectangle and thus create rounded rectangles or ovals.

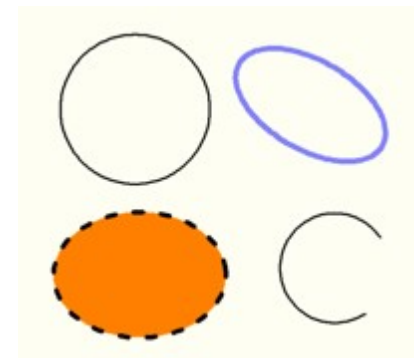
Change the properties of rectangles

If you select a rectangle, you can set its properties in the properties panel on the right edge of the window. You can also select several rectangles at the same time and thus change their properties together.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Circles, Ellipses or Circle segments

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Drawing a new circle or ellipse



Select the appropriate drawing mode.

With a click on the small triangle you can choose whether you want to draw the circle as a frame (starting point above / left) or from where the starting point determines the center of the circle.

Click with the mouse on the drawing to determine the starting point of the circle. Now, with the mouse button held down, draw a frame with the mouse to determine its size and shape. When you release the mouse button, the circle is created as an element on your drawing.

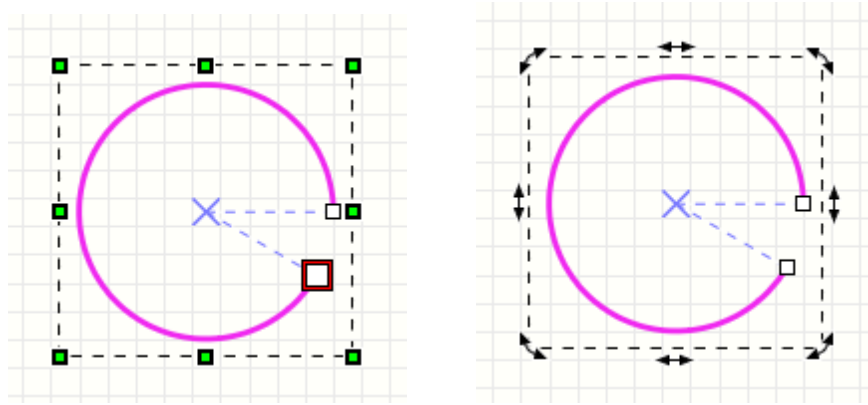
If you hold down the <SHIFT> key while drawing, you will force a proportional aspect ratio, i.e. an exact circle instead of an ellipse.

The circle is initially always created with the preset properties for an area (outline and filling).

If you have selected the circle mode but have not yet started drawing, you can change these preset properties here directly.

Change an existing circle / arc

If you select a circle on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time.



With the [changers](#) you can determine the start and stop point of the circular outline. In this way you can create arcs or segments (with filled circles).

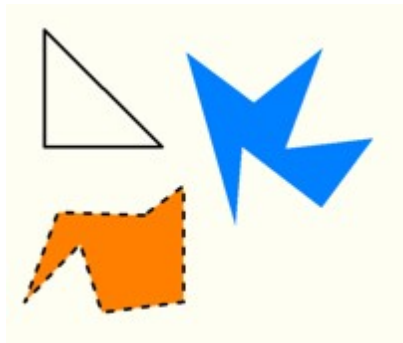
Change the properties of circles / arcs

If you select a circle, you can set its properties in the properties panel on the right edge of the window. You can also select several circles at the same time and thus change their properties together.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Polygons (closed areas)

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>

Draw a new polygon

 Select the appropriate drawing mode.

Click with the mouse on the drawing to determine the starting point of the polygon and then release the mouse button again. Now click on the next desired corner point of the polygon and release the mouse button. This is how you create your polygon using any number of corner points.

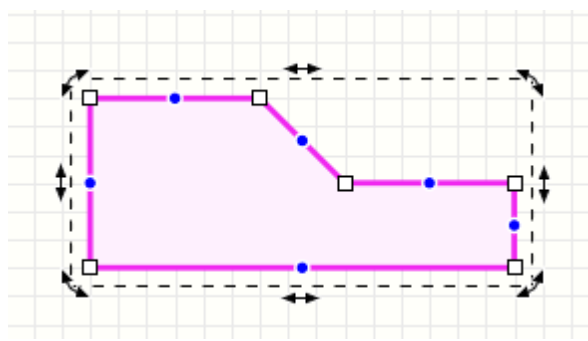
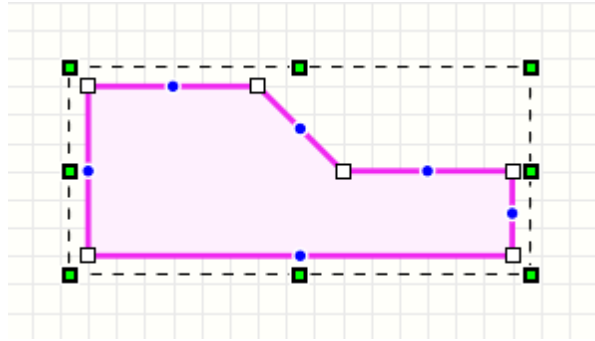
To end the polygon drawing, click once on the right mouse button. The polygon is automatically closed. You can now start a new polygon or switch directly back to standard mode by clicking the right mouse button again.

The polygon is initially always created with the preset properties for an area (outline and filling).

If you have selected the polygon mode but have not yet started drawing, you can change these preset properties here directly.

Modify an existing polygon

When you select a polygon, you can use the [sizers](#) to change its size, shape and direction of rotation at any time. With the [changers](#) you can move all nodes of the polygon. So you can easily change the polygon later.



You can also create new additional nodes at a later date. For this, there is a so-called virtual node in the middle of each line segment, shown as a blue circle. If you click on it and move it, this virtual node becomes a new real node. So you can interactively expand your line with new nodes.

You can also remove individual nodes. To do this, right-click on a node. Then you get a small menu in which you can remove the node, among other things.

Change the properties of polygons

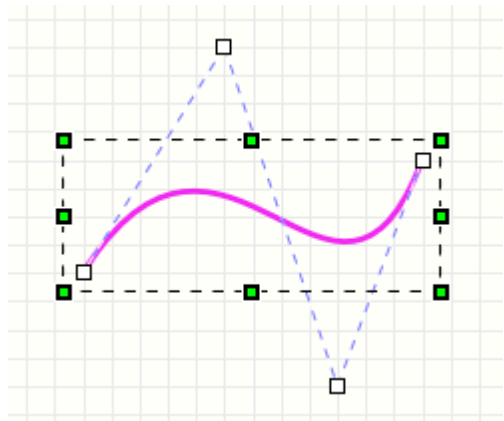
If you select a polygon, you can set its properties in the properties panel on the right edge of the window. You can also select several polygons at the same time and thus change their properties together.

Bezier curves

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Bezier curves are special curves that each consist of a start point, 2 control points and an end point. The curve begins at the start point and ends at the end point. The two control points determine the shape of the curve.



A simple bezier curve always consists of exactly 4 points. You can also define several curves in one go. The end point of the last curve is then automatically used as the starting point of the next curve, so that 3 points (2 control points and 1 end point) are sufficient for each additional curve. bezier curves therefore always consist of at least 4 points and 3 points for each additional curve, i.e. either 4,7,10,13, ... points in total.

Draw a new bezier curve



To draw a bezier curve, select the appropriate drawing mode.

Click with the mouse on the drawing to determine the starting point of the curve and then release the mouse button again. Now determine the two control points of the curve with further mouse clicks and then the end point. For another curve you then define 2 further control points and the end point. To end the curve drawing, click once on the right mouse button. You can now start a new curve.

When setting the control points for a bezier curve, the curve cannot yet be displayed on the screen because the final shape of the curve still depends on the end point. But this is not a big deal, because the control points can then be moved as desired, and the desired shape of the curve can be precisely set.

The bezier curve is initially always created with the preset properties for a line (width, color, etc.).

If you have selected the bezier curve mode but have not yet started drawing, you can change these preset properties here directly.

Change an existing bezier curve

If you select a bezier curve on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time. With the [changers](#) you can move all nodes of the curve. So you can easily change the line afterwards.

Change the properties of bezier curves

If you select a bezier curve, you can set its properties in the properties panel on the right edge of the window. You can also select several bezier curves at the same time and thus change their properties together.

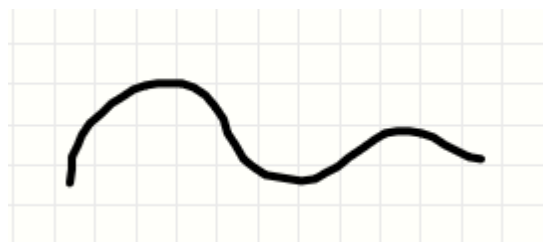
Copyright © ABACOM GbR, Germany. All Rights Reserved.

Freehand Lines

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Freehand lines are special lines that are drawn "freehand", so to speak, with the mouse.



Draw a new freehand line



Select the appropriate drawing mode.

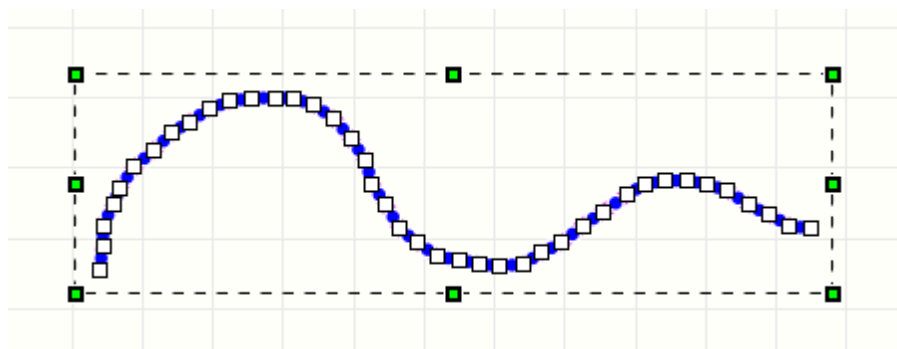
Click with the mouse on the drawing to determine the starting point of the freehand line. Hold down the mouse button and now draw the freehand line. Release the mouse button and the freehand line is done.

The freehand line is always initially created with the preset properties for a line (width, color, etc.).

If you have selected the freehand line mode but have not yet started drawing, you can change these preset properties here directly.

Modify an existing freehand line

If you select a freehand line on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time. With the [changers](#) you can move all nodes of the line. You can also change the freehand line later. Since a freehand line usually consists of a large number of nodes, you can zoom into these and edit the individual nodes more precisely if necessary.



You can also create new additional nodes at a later date. For this, there is a so-called virtual node in the middle of each line segment, shown as a blue circle. If you click on it and move it, this virtual node becomes a new real node. So you can interactively expand your line with new nodes.

You can also remove individual nodes or split the line into 2 parts. To do this, right-click on a node. Then you get a small menu in which you can remove the node, among other things. You can also split the line at this position here and thus get 2 individual lines.

Change the properties of freehand lines

If you select a freehand line, you can set its properties in the properties panel on the right edge of the window. You can also select several lines at the same time and thus change their properties together.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

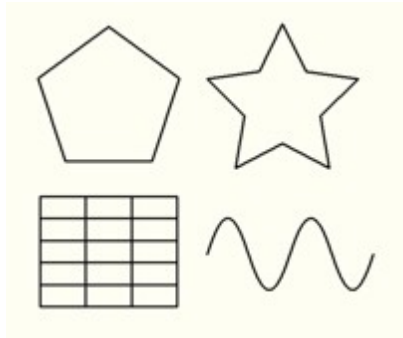
Special Forms

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



The special forms are simple but often required and sometimes difficult to create elements.

Often these forms can also serve as a construction aid. For example, you can use an equilateral polygon as a construction aid to arrange solder points in an exact circular shape. To do this, simply create the polygon with the number of corners you want. Now place a soldering point on each corner and then delete the polygon again. You have already aligned your soldering points exactly in a circular shape.



Create a new Special form



Select the appropriate drawing mode.

A pop-up menu opens and you can select the special form you want from the list.

With the first 4 special shapes, a dialog for setting special properties also appears:

Special shapes

?

×

Polygon

Star

Grid

Wave

Edges:

5

▲

▼

Angle offset:

0

▲

▼

°

☐ As line

☒ As polygon

OK

Cancel

With polygon and star you can choose whether the element should be created as a line or as a polygon. If the element is to be given a fill color, you must select "as polygon".

After selecting the special shape (and possibly confirming the dialog with OK), you can place the selected special shape on the drawing with the mouse. To do this, click with the mouse on the drawing to determine the starting point. Now, while still holding the mouse button down, draw a frame with the mouse to determine its size. When you release the mouse button, the special shape is created on the drawing. With some special shapes (such as arrows) you can influence the shape with the direction in which you open the frame (to the left / right or up / down).

Change an existing special shape

After creating a special shape, it is created from the basic elements of sPlan (line, polygon, etc.).

So changing these elements is the same as changing these simple basic elements.

The special properties that were selected when the special shape was created are no longer available after creation and cannot be changed afterwards. If necessary, the special shape can simply be deleted and a new one created.

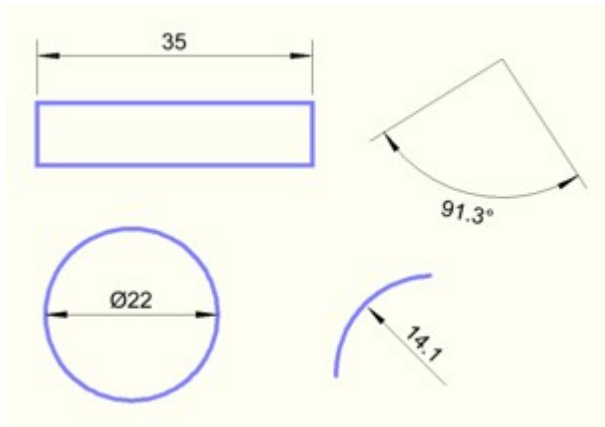
Copyright © ABACOM GbR, Germany. All Rights Reserved.

Dimensions

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>




With the dimensioning functions of sPlan you can easily create various dimensions:

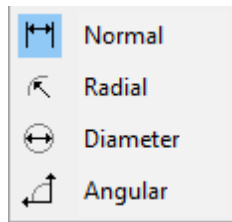


Depending on the type, a dimension consists of 2-3 control points. These control points are set one after the other when creating a new dimension. The control points can then be moved again at any time with the help of the [changer](#).

Draw a new dimension

 To create a new dimension, select the appropriate drawing mode.

Click on the small red triangle to open a small list from which you can choose a different dimension type:



You can now set the individual control points of the new dimension on the plan with a simple mouse click, thus creating the new dimension. Move the mouse after each click to see the effect of the control points. You will quickly see its effect. After the 2nd or 3rd click (depending on the dimension type) the dimension is finished.

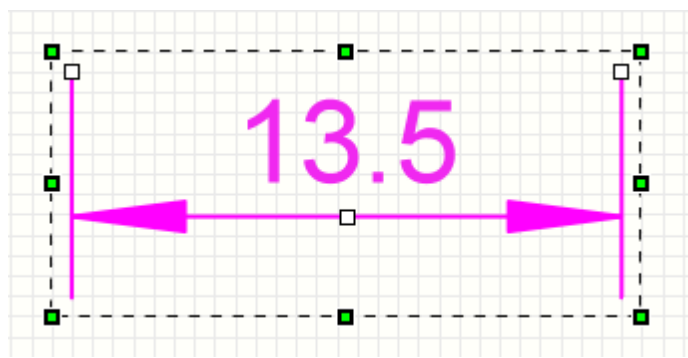
You can cancel the creation of a dimension at any time with the right mouse button.

The dimensions are initially always created with the preset properties for a dimension (color, text type, arrows, etc.).

If you have selected the dimension mode but have not yet started drawing, you can change these preset properties here directly.

Change an existing dimension

Select the desired dimension.



With the [changers](#) you can now reposition all nodes of the dimension. So you can easily change the dimensions afterwards.

Change properties of dimensions

If you select a dimension, you can set its properties in the properties panel on the right edge of the window. You can also select several dimensions at the same time and thus change their properties together.

Some special properties of a dimension are briefly explained here:

Show diameter:

If activated, the diameter symbol appears in front of the numerical value.

Prefix:

Any text that comes before the numeric value.

Suffix:

Any text that appears after the numerical value (e.g. a "°" sign for angular dimensions).

Upper tolerance / Lower tolerance:

Here, as is sometimes necessary for special dimensions, the tolerances for a dimension can be specified. The two tolerance specifications for positive and negative values are then, according to the standard, at the top and bottom after the dimension figure.

Auto-Measure:

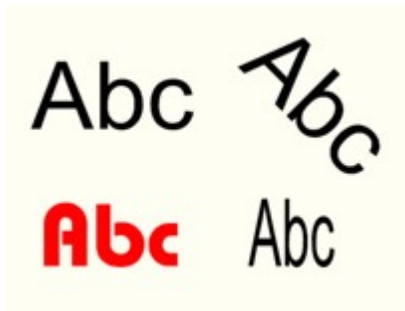
If Auto-Measure is activated, the numerical value is always set automatically when the dimension is changed.

You can also enter your own fixed numerical value. The auto function must then be deactivated for this.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Text

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>

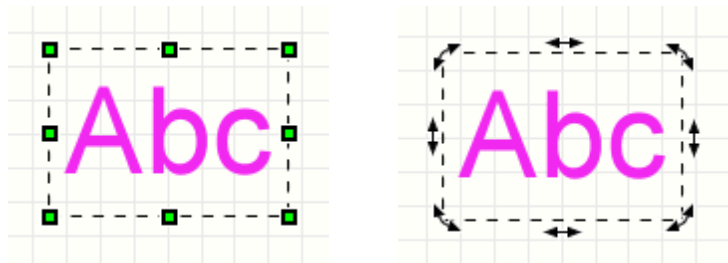
Create a new text label

 Select the appropriate drawing mode.

Click with the mouse on the drawing to place a text. You can now place further texts directly with further mouse clicks or exit the text mode with the right mouse button. Now you can directly adjust the properties (text, font, color, etc.) of the new text.

Change an existing text

If you select a text on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time.



Advanced text entry

With a double click on the text, or with the head "..." next to the text input in the properties panel, you can call up the [extended text edit](#)

Change the properties of texts

If you select a text, you can set its properties in the properties panel on the right edge of the window. You can also select several texts at the same time and thus change their properties together.

Text linking

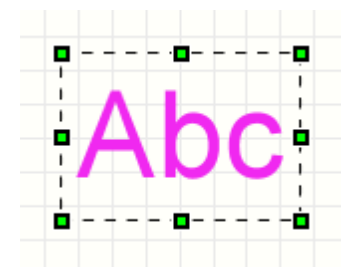
In the properties you can also make the settings for internal or external text links.

[Here you can find more information about the text links.](#)

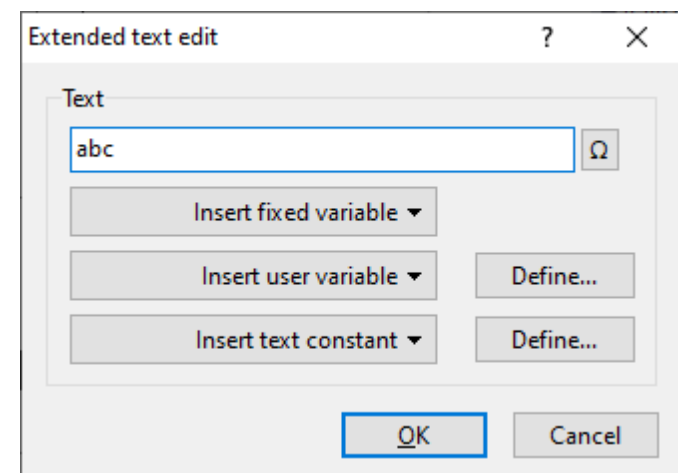
Copyright © ABACOM GbR, Germany. All Rights Reserved.

Extended Text Edit

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >> [Text](#) >>

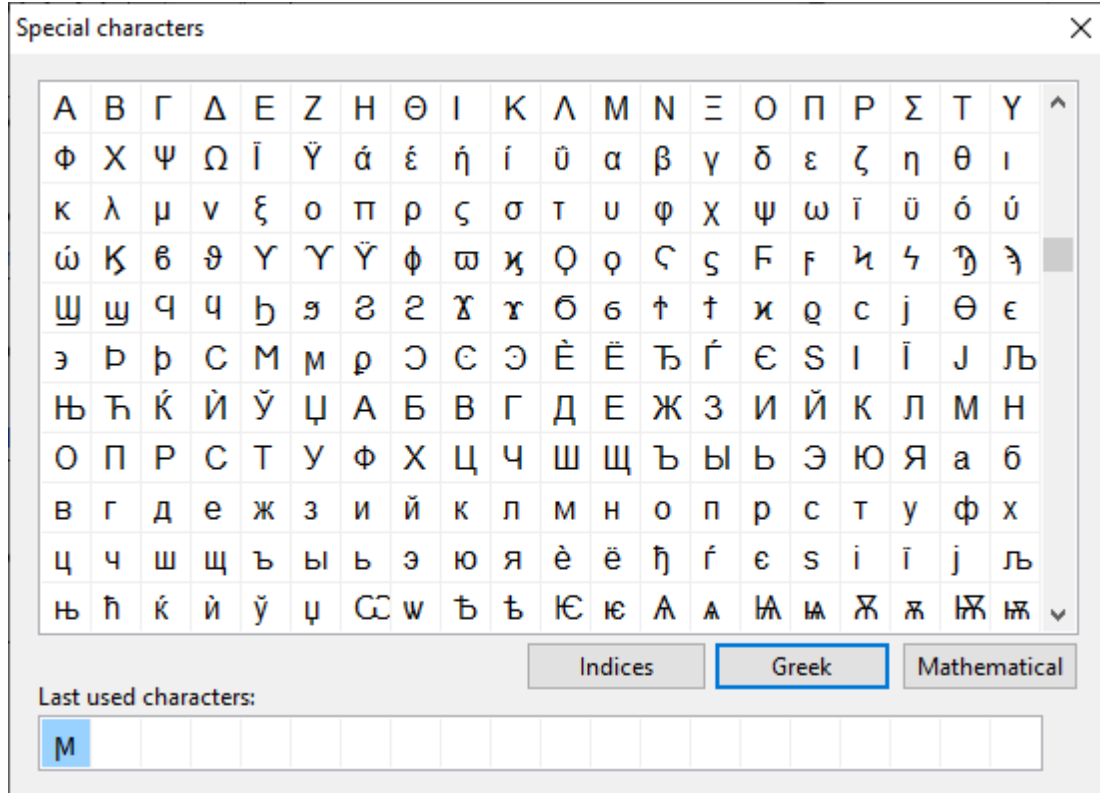


With a double click on the text, or with the button "... " next to the text input in the properties panel, you can call up the Extended text edit:



Here you can change the text or insert [variables or text constants](#) into the text.

With the button " " you can call up the special characters dialog:

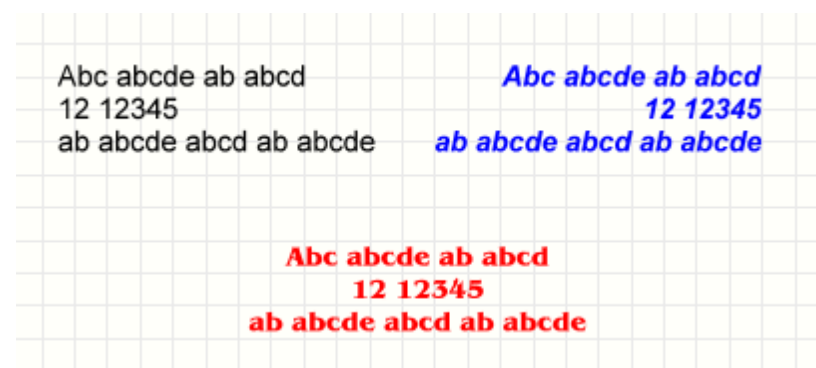


Here you will find all characters that are available in the currently selected character set (Unicode), including those that cannot be accessed directly via the keyboard.

If you double-click on a character, it will automatically be included in the text input.

TextRect (multiline)

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>

Create a new TextRect

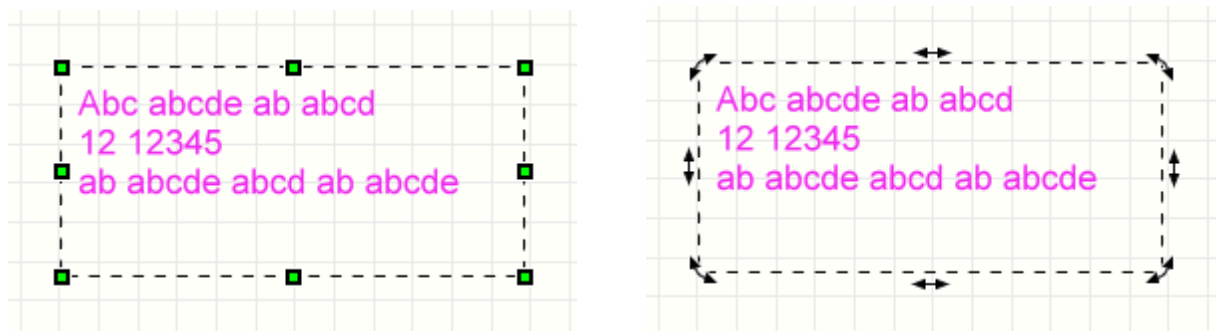


Select the appropriate mode switch.

Click with the mouse on the drawing in order to determine the starting point of the TextRect. Now, while keeping the mouse button pressed, draw a frame with the mouse to determine its size. When you release the mouse button, the TextRect is placed on your drawing. You can now directly create further TextRects or exit the TextRect mode with the right mouse button. Now you can directly adjust the properties (text, font, color, etc.) of the new TextRect.

Change an existing TextRect

If you select a TextRect on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time.



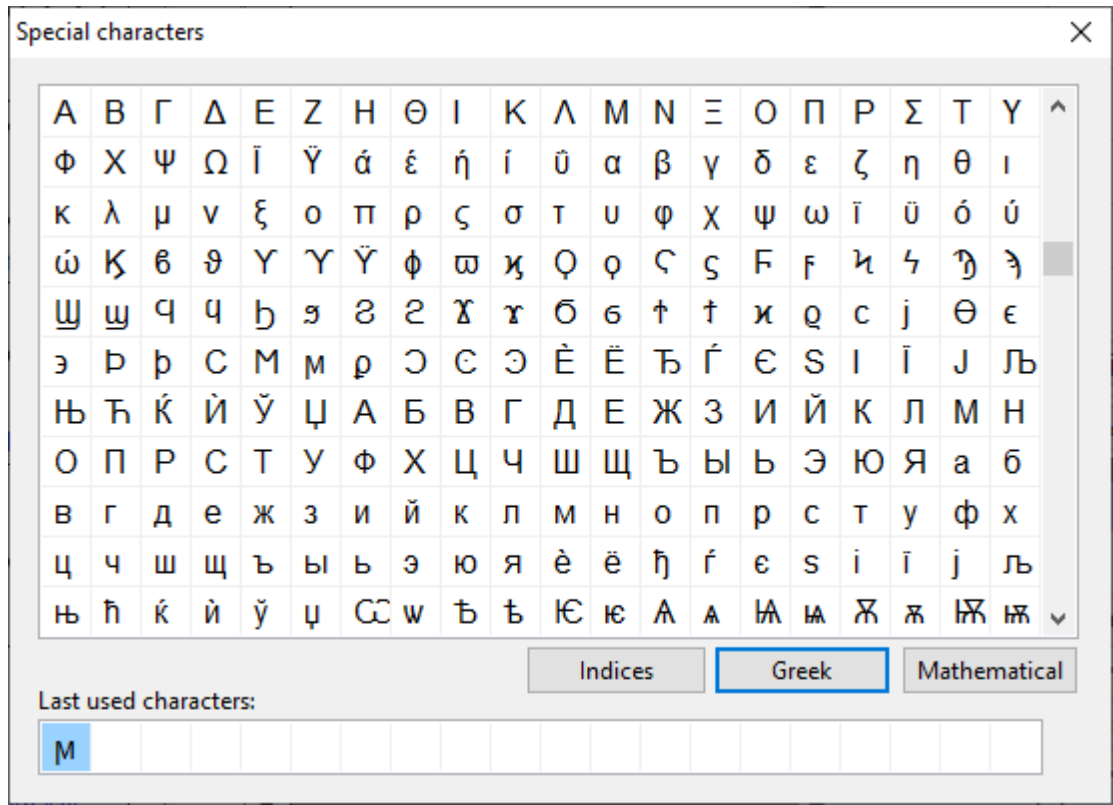
Change properties of TextRects

If you select a TextRect, you can set its properties in the properties panel on the right edge of the window. You can also select

several TextRects at the same time and thus change their properties together.

Tip:

Press the key combination <Ctrl> + <Ins> in the input field of the paragraph text to call up the special characters dialog.



Here you will find all characters that are available in the currently selected character set (Unicode), including those that cannot be accessed directly via the keyboard.

If you double-click on a character, it will automatically be included in the text.

Bitmaps

[Drawing of circuit diagrams](#) >> [Drawing functions](#) >>



Select the appropriate drawing mode.

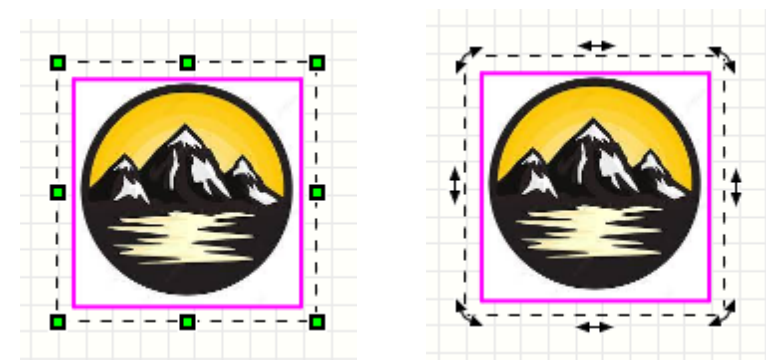
An "Open" dialog appears in which you can select the desired bitmap. sPlan supports bitmaps in * .BMP or * .JPG format. you would like to read in other formats (such as GIF, PNG or PDF), you must first convert them to the BMP or JPG format using an appropriate graphics program.

After you have selected a bitmap, the graphic is imported into your schematic.

As an alternative to this procedure, you can also execute the "Import picture from file..." command from the "File" menu.

Change an existing bitmap

If you select a bitmap on the drawing, you can use the [sizers](#) to change its size, shape and direction of rotation at any time.



Properties of bitmaps / special functions

When you select a bitmap, you can see its properties (memory requirements, size, resolution, etc.) in the properties panel on the right-hand side of the window.

There are also some special functions here:

Reduce resolution

This allows you to reduce the internal resolution of the integrated bitmap by a factor of 1.4.

Especially when a bitmap is shown relatively small on the plan, its resolution is often much too large.

If the resolution of your bitmap is more than 300 dpi, you can usually reduce the resolution of the integrated bitmap here in order to avoid unnecessarily high memory usage. If necessary, you can also perform this function several times in a row. Note how much the memory requirement decreases!

If the resolution of the bitmap is very high, this button is highlighted in red to indicate this directly.

Rotate bitmap 90°

This allows you to rotate the internal bitmap directly (physically, so to speak) by 90° clockwise.

You could simply rotate the bitmap on the drawing, but the internal bitmap remains untouched (i.e. not rotated) and sPlan must first rotate the bitmap "software-wise" in order to display it correctly every time the bitmap is displayed on the monitor and also on the printer . This costs unnecessary computing power and can sometimes even lead to problems when printing.

Lighten up

If a bitmap appears too dark, you can use this to lighten it a little. If necessary, you can also perform this function several times in a row.

Normalize

With this you can bring a possibly "distorted" bitmap back to its original proportions.

Bitmap-Explorer

This opens the [Bitmap-Explorer](#).

All bitmaps used in your project are listed in the Bitmap-Explorer. In this way you can quickly get an overview of all bitmaps, their resolution and their memory requirements.

Here you can directly specify a maximum value for the resolution (DPI) of the bitmaps and automatically adjust all bitmaps with a higher resolution to this maximum value. Depending on the bitmaps used, this can save a lot of memory and thus make your sPlan project narrower and faster.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Clipboard functions

[Drawing of circuit diagrams](#) >>



The clipboard is a useful tool in almost all Windows programs. You can also use these functions with sPlan. The clipboard is like a container in which you can store elements of your drawing and access them again later. You can use the clipboard with the following commands:

Cutting out

Copy

Paste

Duplicate

You can find these commands in the "Edit" menu or in the local pop-up menu (right-click on the drawing). A corresponding button in the toolbar is dedicated to each command. The individual commands have the following meanings.

Cut

Copies all selected elements from your drawing to the clipboard. The selected elements are then deleted from your drawing.

Copy

Copies all selected elements from your drawing to the clipboard. In contrast to cutting, however, the selected elements remain on your drawing.

Insert

Inserts the contents of the clipboard onto your drawing. An image of the elements to be inserted "sticks" at the mouse position. With a click of the mouse, the elements are inserted at the corresponding point in the drawing.

Duplicate

Duplicating copies all selected elements to the clipboard and inserts them again from there.

For further processing in other programs (e.g. for documentation in word processing), sPlan can copy the drawing as a graphic to the clipboard. For more information, see [Copy a Drawing to the Clipboard](#).

[Copy a Drawing to the Clipboard](#)

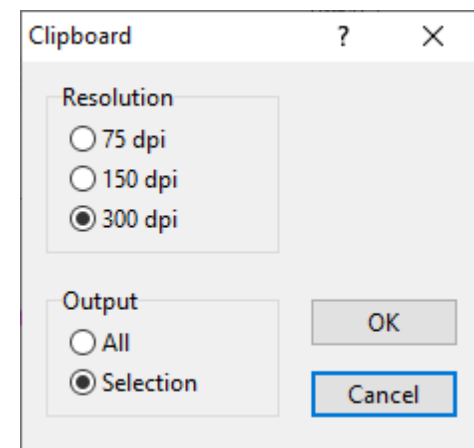
Copy a Drawing to the Clipboard

[Drawing of circuit diagrams](#) >>



For further processing in other programs (e.g. for documentation in word processing), sPlan can copy the drawing as a graphic to the clipboard. Other programs can then import this graphic using the Paste command.

To copy your drawing to the clipboard as a graphic, select Clipboard... from the File menu.



Resolution

Here you select the desired resolution.

The higher the resolution, the better the quality of the graphic. However, the internal memory requirements also increase. Since Windows limits the size of the clipboard for graphics, problems can arise with very large drawings and high resolutions. In this case, reduce the resolution.

Selection

Here you can choose whether you want to copy the entire drawing or only the selected part of your drawing to the clipboard.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Rotate, Mirror, Set to front / back

[Drawing of circuit diagrams](#) >>



All elements on the drawing can be rotated, mirrored or changed in their Z-order (above / below one another). The following commands are available for this:

- Rotate
- Flip horizontally
- Flip vertically
- Set to front
- Set to back

You can find these commands in the "Functions" menu, in the local pop-up menu (right-click on the drawing) or in the toolbar.
First select the elements on which you want to apply these functions and then call the function.

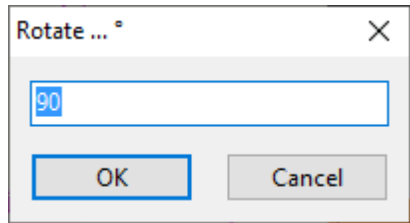
Rotate (90°)



With the button in the toolbar, all selected elements are rotated 90 ° clockwise.

Rotate

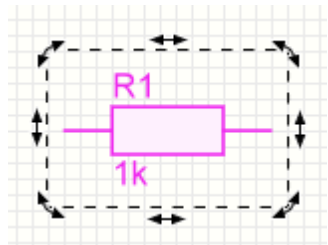
With the menu item "Rotate ..." in the "Functions" menu, or in the local pop-up menu (right mouse click on the drawing), all selected elements are rotated clockwise by an adjustable angle.



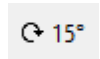
Individual elements as well as complete drawing parts can be rotated.

You can also simply rotate selected elements with the mouse (see also [Sizer and Changer](#))

If you click again with the mouse on an element that has already been selected, the sizers at the 4 corners change to arrows. Now you can simply click on one of these arrows and rotate the element with the mouse button still held down.



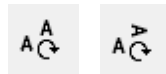
The angle of rotation always locks into the angle of rotation snap you set. You set the rotation angle snap in the lower toolbar:



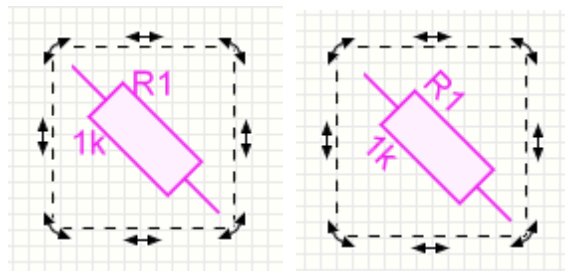
Tip:

With the <CTRL> key you can temporarily switch off the angle of rotation snap at any time, i.e. as long as you keep the key pressed.

If texts are also rotated, you can determine the rotation behavior of these texts with the button in the lower toolbar. With a click on this button you can change the turning behavior:



Either the texts always stay in the same text orientation when rotating, or the text orientation is rotated together with the text:



Mirror

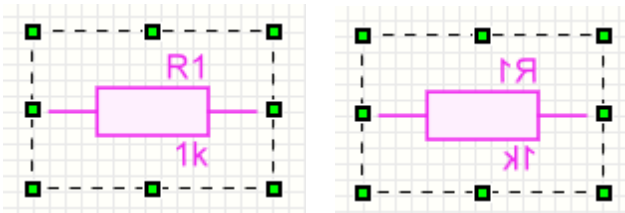


With the two toolbar buttons or with the menu items "Mirror horizontal / Mirror vertical" in the "Functions" menu or in the local pop-up menu (right click on the drawing), all selected elements are mirrored either horizontally or vertically.

If texts are also mirrored, you can determine the mirroring behavior of these texts with the button in the lower toolbar. With a click on this button you can change the mirror behavior:



Either the texts themselves always remain the same when mirrored, or the texts are also mirrored:

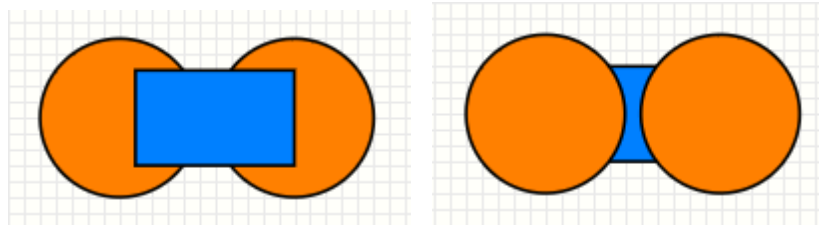


Set to front / Set to back



With the two toolbar buttons or with the menu items "Set to front / Set to back" in the "Functions" menu or in the local pop-up menu (right click on the drawing), all selected elements are set to front or back.

This allows you to determine the Z-order of the individual elements. The Z-order determines which elements are on top of which other elements. So if an element is completely or partially covered by another element, and this element should lie above the other element, you can bring it to the foreground with the "Set to front" function. It is then automatically on top of all other elements. You can do exactly the opposite with the "Set to back" function. This function sets the selected element all the way back so that it is covered by all other elements. These functions also affect all selected elements.



Copyright © ABACOM GbR, Germany. All Rights Reserved.

Align Elements

[Drawing of circuit diagrams](#) >>



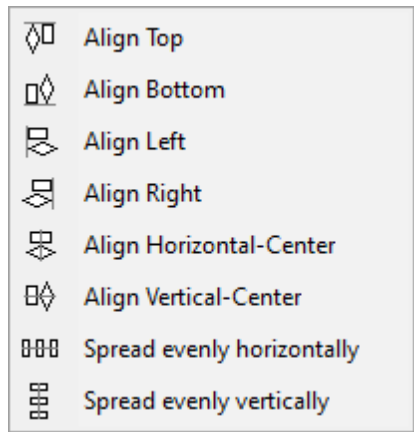
You can use this function to automatically align several elements with one another.

First, select the elements that you want to align. At least 2 elements must be selected.

Then select the "Align" button from the toolbar:



A menu then appears in which you can select the desired orientation. You have various options here:



After you have chosen an alignment, all selected elements are aligned as required

Colorize Elements

[Drawing of circuit diagrams](#) >>



With this function you can color certain areas of your drawing without having to change the colors individually for each element.

To colorize, first select the desired elements on your drawing and then select the entry "Colorize elements..." from the "Functions" menu.

A color selection box then appears in which you can select the desired color. After you have confirmed the color selection, the selected elements are colored. You do not see this immediately because the corresponding elements are still selected and are therefore still displayed in magenta. However, if you deselect it, you will see that the elements have been colored in the selected color.

All lines and fills are set to the selected color. An exception is a white filling. This is always retained. This makes sense because many more complex components have e.g. a white box in which there are further elements or texts. If the white filling would also be colored, you would not be able to see anything here. When components are filled, they usually have a white filling.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Build / Split Groups

[Drawing of circuit diagrams](#) >>



You can combine several elements of the drawing into a group. This has the advantage that you can completely select, move or otherwise edit all elements of the group with just one click. In addition, the elements contained in the group are protected against accidental changes. Individual elements cannot be deleted from a group. Each group always consists of at least two elements. Multi-level groupings are also possible, which means that different groups can be combined again to form one group.

It may be necessary to break down an existing group into its components, e.g. to delete or change a single element from it. When a group is dissolved, only the top level of the grouping is canceled, i.e. any previously existing subgroups are retained. If necessary, repeat the process for the subgroups.

The two necessary functions are available to you with the "Build group" and "Split group" commands.

You can find these commands in the "Functions" menu, or in the local pop-up menu (right-click on the drawing) or in the toolbar.



First select the elements on which you want to apply these functions and then call the function.

Tip:

If you click on an element in a group while holding down the <ALT> key, you can also select it individually. This can be very helpful in some cases. For example, you can change the properties of an element within a group without breaking the grouping beforehand.

Magnetic Lines

[Drawing of circuit diagrams](#) >>



You have the option of positioning so-called "magnetic lines" on the drawing. These can be helpful when creating the drawings. Magnetic lines are purely construction aids and are neither printed out, nor can they be edited in any way. You can create, move and delete these lines. The advantage of these lines is that they "magnetically attract" all elements. As soon as you move an element and with it come close to such a magnetic line, it will be attracted to it. This can be used to align several elements on exactly one line, for example. With several such lines, table-like structures could also be designed.

Insert new magnetic lines

With the commands "New vertical magnetic line" and "New horizontal magnetic line" in the submenu "Functions -> Magnetic lines" you can create a new magnetic line. From there, these auxiliary lines can be moved as desired.

You can also simply draw magnetic lines from the rulers more easily and quickly. Simply click on the ruler with the mouse, hold the mouse button down and drag a magnetic line onto the drawing.

Moving magnetic lines

If you move the mouse over a magnetic line, the mouse pointer changes into a double arrow. This is the sign that you can move the magnetic line here. Click with the mouse on the corresponding magnetic line and then drag it to the desired position while holding the mouse button down.

Deleting magnetic lines

To delete an magnetic line, click with the mouse once on the magnetic line to be deleted. This will then turn red. With the command "Delete" (menu "Edit" or toolbar) you can now delete this magnetic line again.

You can delete magnetic lines more easily and quickly by simply dragging them onto a ruler.

Fix magnetic lines

With this function from the submenu "Functions-> Magnetic Lines" you can fix all magnetic lines on the drawing. These are then automatically protected against being moved and deleted. You cannot add new magnetic lines in this mode either. You can exit the mode at any time by selecting this function again.

Hide magnetic lines

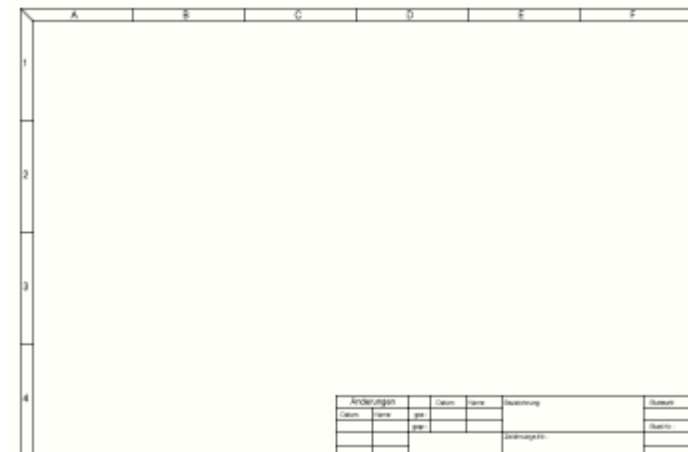
With this function from the submenu "Functions-> Magnetic Lines" you can hide all magnetic lines on the drawing. You cannot add new magnetic lines in this mode either. You can show the magnetic lines again at any time by selecting this function again.

Using Forms

Drawing of circuit diagrams >>



Each drawing in sPlan can have its own form. The form is, so to speak, a separate drawing level that lies below the actual drawing. In the normal working mode, the form is only visible visually, but you cannot select or edit any elements of the form. In this way, the form in the background does not interfere with normal work with sPlan.



You can create your own new forms or change existing forms according to your requirements.

Edit form level



If you want to edit the current form or create a new one, you must first release the form level for editing. To do this, select the Edit form layer command in the Forms menu, or click the corresponding button in the toolbar. The drawing is now hidden and the form is released for editing. You can now design the form as you like. While you are working on the form, all of the drawing functions of sPlan are available to you as normal.

When you have finished editing the form, deactivate the Edit form layer option again by clicking on Exit or by clicking the menu item again. Now the form is put back and the drawing is visible again. You are now back in the normal editing mode for the drawing.

Save form

You can save the current form as a separate file at any time. The form is then also available for loading for all other pages. To save forms, select the Save form to file... command from the Forms menu. All forms are automatically given the extension *.SBK.

Load form

If you want to load an existing form for a page, select the Load form from file... command from the Forms menu. Please note that when a new form is loaded, any existing form will be lost.

Remove the form

If you want to remove a form from a schematic, select the command Remove form... from the Forms menu. Here you can also remove forms from several or from all pages of your project.

Generate form automatically

If no element is selected, the properties of the page are displayed in the properties panel on the right edge of the window. Here there is the Form section with which you can generate a form automatically:

Form	
Width:	<div><div>280</div><div></div></div> mm
Height:	<div><div>190</div><div></div></div> mm
X-Offset:	<div><div>10</div><div></div></div> mm
Y-Offset:	<div><div>10</div><div></div></div> mm
Columns:	<div><div>10</div><div></div></div> mm
Rows:	<div><div>8</div><div></div></div> mm
Column-Start:	<div><div>1</div><div></div></div>
Row-Start:	<div><div>1</div><div></div></div>
Show grid:	<input type="checkbox"/>
<div>Generate...</div>	

The form is first divided into a kind of grid. Here you can specify the total size of the form as well as the offset and the number of columns and rows in the grid. If you activate the Show grid option, this grid will be displayed on your page, so you can also visually check the individual parameters here.

This grid also determines the content of the following variables:

<COLNUM>

<COLCHAR>

<ROWNUM>

<ROWCHAR>

With these variables, components can automatically display their position (column and/or row on the form). This is particularly common for electrical installation plans.

When you have made the required settings, click on Generate... and after a further dialog for defining the frame labeling of the form, it will then be generated automatically.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Multi-Page Projects

[Drawing of circuit diagrams](#) >>



A project can consist of several single pages under sPlan. For example, it makes perfect sense to split up a large drawing, if possible, into several small drawings, which are then all placed on a single page. For example, the circuit of a complex audio amplifier could be divided into the following individual plans:

Power adapter

Entrance level

tone control

power amplifier

The 4 individual pages would then all be stored in a sPlan file called "Amplifier.spl8", for example.

You can create any number of single pages within a project (i.e. in one file). Each page has its own name and the corresponding drawing. Each page has its own "tab" with the name of the page at the bottom of the drawing area. You can then switch to the corresponding page by clicking on the respective tab.

You can also determine the order of the pages afterwards, e.g. to sort the individual pages according to priorities or similar.

You can save and load individual pages from a drawing separately. This gives you the option of transferring certain pages from a complex circuit to other projects.

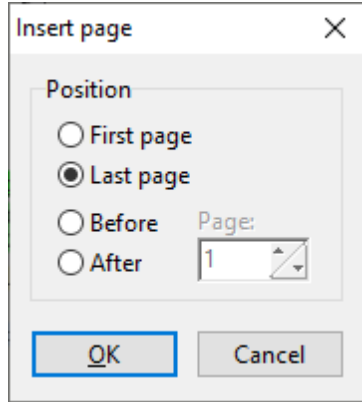
The "tab-list" below the drawing area has its own local menu (context menu). You can do this by moving the mouse over this tab-list and clicking the right mouse button once. The local menu gives you quick and easy access to many commands:

Page Properties...

Directly calls up the properties of the page in the properties-panel on the right edge of the window. See also [page settings](#).

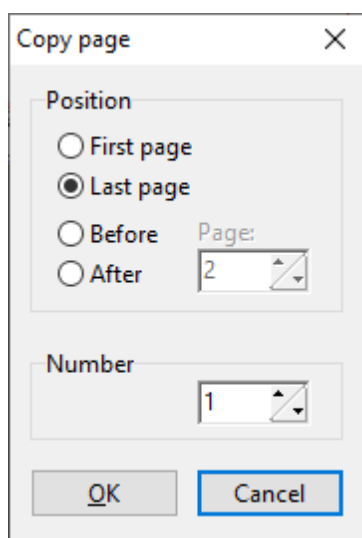
Create new page...

This allows you to add a new page. You can freely determine the position of the new page in a small dialog:



Copy page...

With this you can copy a page within a project. You can freely determine the position and number of copies in a small dialog:



Delete pages...

This deletes a page from the project. If you confirm the subsequent security question, the page will be irrevocably deleted.

By the way, if you only have one page in your project, you cannot delete it. Each project must consist of at least one page.

Sort pages...

If you want to change the order of the individual pages, you can use the "Sort pages..." command. Here you can easily re-sort the individual pages in a list using drag & drop.


Load page from file / Save page to file

You will not find these two commands in the local context menu of the page bar but only in the main menu under Page -> Load page from file... and Page -> Save page to file...

If you would like to have a single page in a different file, you can save it separately. You can then later reload the page from any other project file. Saved pages have the extension * .BLT.

 Goto page...

Navigating in the page tab-list can sometimes be tedious, especially in projects with many pages. To quickly go to a specific page, just click this arrow and select the page from the page list.

 Add page quickly

With this you can add a new page quick and easy, bypassing the extra dialog.

Vertical page list

Particularly for large projects with a large number of pages, a vertical page list can be clearer than the conventional arrangement of the pages in the tab-list below the plan. At the top/right in the toolbar there is a button to switch between the page-list modes:



With this you can easily switch back and forth between the conventional page tab-list and the vertical page list.

Zooming

[Drawing of circuit diagrams](#) >>



Zooming is a very important feature of sPlan. Only then is it possible to look at the complete drawing first, but then to edit a particularly tricky, narrow area in large format in the next moment.

Zoom with the mouse wheel

The easiest and most convenient way to zoom and navigate is to use the scroll wheel on your mouse.

When the mouse is on the work surface of your plan, you can zoom with the scroll wheel. When entering, the current mouse position is automatically placed in the middle of the work surface. So you can always zoom in and out to all desired areas of your plan with a little zooming in and out.

Panning (move visible view)

In panning-mode, you can move the visible view freely in all directions simply with the mouse. You can reach the panning-mode in 2 different ways:

- Double-click on an empty space on the page.

- Simply double-click on an empty spot, hold down the mouse and move the visible view with the mouse.

- With the middle mouse button (mouse wheel).

- Simply click with the middle mouse button, hold down the mouse and move the visible view with the mouse.

The panning-mode ends as soon as you release the mouse button.

The zoom mode

If you cannot or do not want to zoom with the mouse wheel, sPlan also has its own zoom mode:



The mouse then turns into a magnifying glass over the drawing. With a click on the left mouse button you can now zoom into the drawing, with a click on the right mouse button you zoom out again.

When zooming in, you can determine the center of the image for the next zoom level with the left-click position. In this way you can move to the desired area quickly and precisely.

You can also "frame" the area you want. To do this, click with the left mouse button and while still holding the button, drag the frame accordingly. After releasing the mouse button, the framed area is displayed zoomed.

When you have reached the largest or smallest zoom level, further zoom attempts have no effect.

There are 3 special zoom functions:

Zoom page (zooms so that the whole page is visible)

Zoom elements (zooms to all elements)

Zoom selected elements (zooms to all selected elements)

You call up these zoom functions with the magnifying glass symbol in the upper toolbar.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

General Settings



In the General settings you make general settings about the appearance and operation of sPlan.

You call this dialog in the main menu under Settings -> General settings...

Use the individual entries on the left to select the respective category:

[General Settings](#)

[Display](#)

[Directories](#)

[Library](#)

[Grid](#)

[AutoSave](#)

[New page](#)

[Hotkeys](#)

Copyright © ABACOM GbR, Germany. All Rights Reserved.

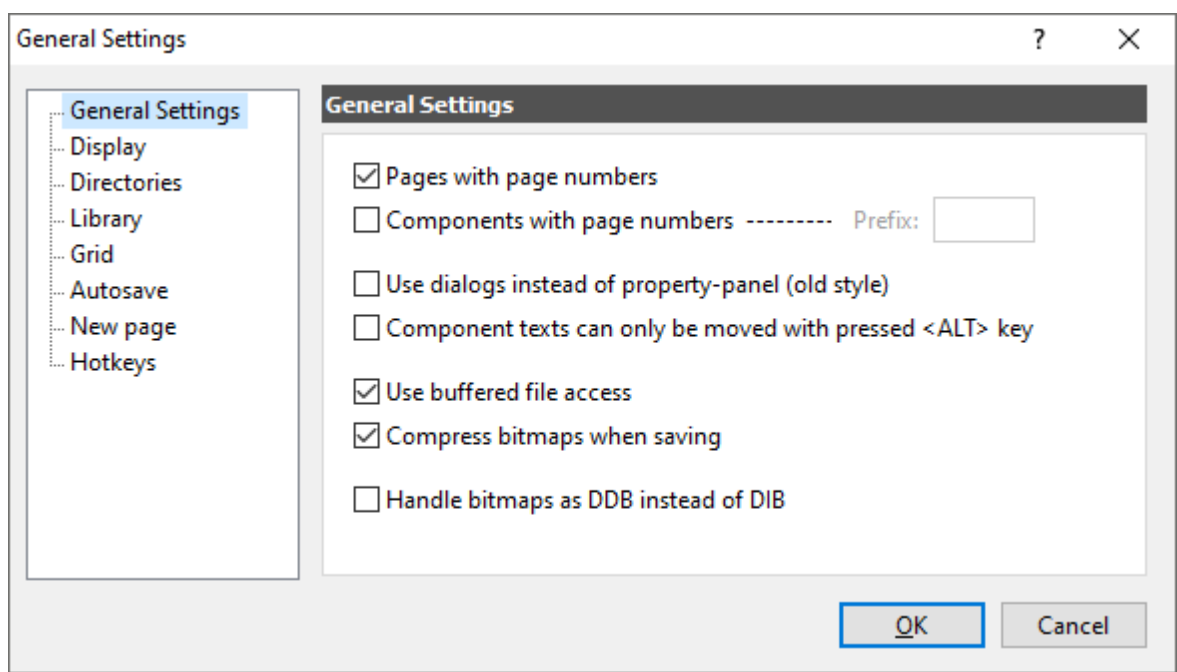
General Settings

[General Settings >>](#)


Here you can make general settings for working with sPlan.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry General Settings from the left bar.



Pages with page numbers

If this option is selected, all pages are automatically prefixed with the page number.

Components with page numbers

If this option is selected, all component identifiers are automatically preceded by the page number.

Prefix

The prefix, if specified, is preceded by all component identifiers.

Use dialogs instead of the property-panel (old style)

If this option is selected and the properties-panel on the right is hidden, the properties of an element are only displayed in a dialog with a double click on the element, as was always the case with earlier versions of sPlan. There is actually no reason for this option, unless you absolutely cannot get used to the new permanent property bar.

Component texts can only be moved with pressed <ALT> key

Normally you can click the component texts (identifier, value, contacts) directly on the plan and move them independently of the component (provided that the component itself is not selected). If you activate this option, this is only possible in connection with the <ALT> key. Otherwise, when you click on a component text, the entire component is selected.

Use buffered file access

This option can speed up saving and loading files, especially when working on network drives.

Compress embedded bitmaps when saving

If this option is selected, inserted bitmaps (BMP or JPG) are saved in compressed form when they are saved.

This drastically reduces the file size of the SPL files depending on the number and size of the bitmaps used.

Handle bitmaps as DDB instead of DIB

If there is a memory problem with sPlan files with too many and too large bitmaps, this option can bring an improvement on some systems.

In general, however, an sPlan file should never contain so many bitmaps that such memory problems occur at all. Make sure that the inserted bitmaps do not have an unnecessarily high resolution. A resolution of 300 dpi is sufficient in most cases. You can use the [Bitmap Explorer](#) to examine the bitmaps in your file.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

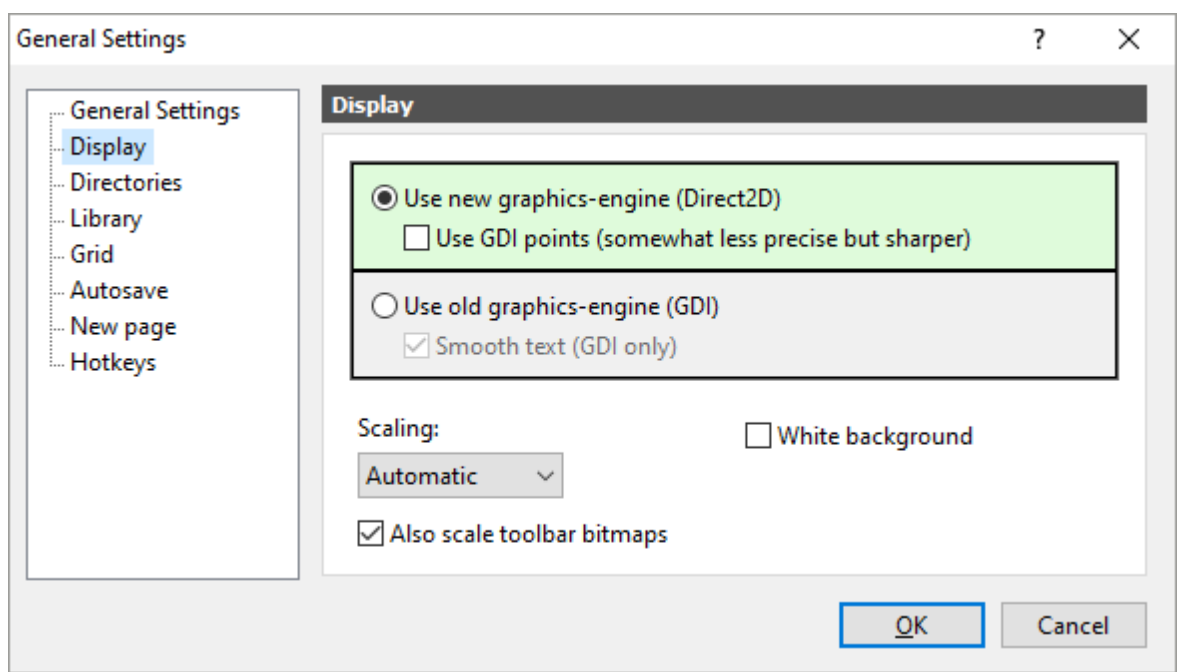
Display

[General Settings](#) >>


Here you make basic settings for the graphics and scaling of sPlan.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Display from the left bar.



Graphics engine

Use new graphics engine (Direct2D)

This is the standard setting of sPlan and should only be changed in the event of problems. The new Direct2D technology is many times faster and more effective than the old GDI technology.

Use GDI points

With this option, some lines and edges are sometimes displayed a little sharper, especially with plans that have been zoomed in, thanks to a special grid. However, this is partly at the expense of accuracy, so some lines may look slightly offset. If you are bothered by the slightly blurred display when the plans are zoomed in, then you can give this option a try.

Use old graphics engine (GDI)

This option should really only be activated in the event of problems with Direct2D and is no longer recommended.

Smooth text

With this, the text is displayed smoothly when the display is zoomed in small.

Scaling

The complete sPlan interface can be displayed on your monitor in various scales (sizes).

Automatically

This is the standard specification of sPlan. The degree of scaling is based on the value set in your Windows system.

None (100%)

This prevents any scaling in sPlan. This setting can result in sPlan being displayed very small on your monitor.

Since not all elements can be scaled independently in Windows, different scaling may occur.

125%, 150%, ...

This will scale sPlan to the specified value regardless of the value set in your Windows system.

Since not all elements can be scaled independently in Windows, different scalings may occur.

Also scale toolbar bitmaps

If this option is activated, the small images on the toolbar buttons are also scaled and displayed larger. This also makes them a bit more "pixelated".

White background

If you are bothered by the slightly colored background in sPlan, you can use this option to set the background color to pure white.

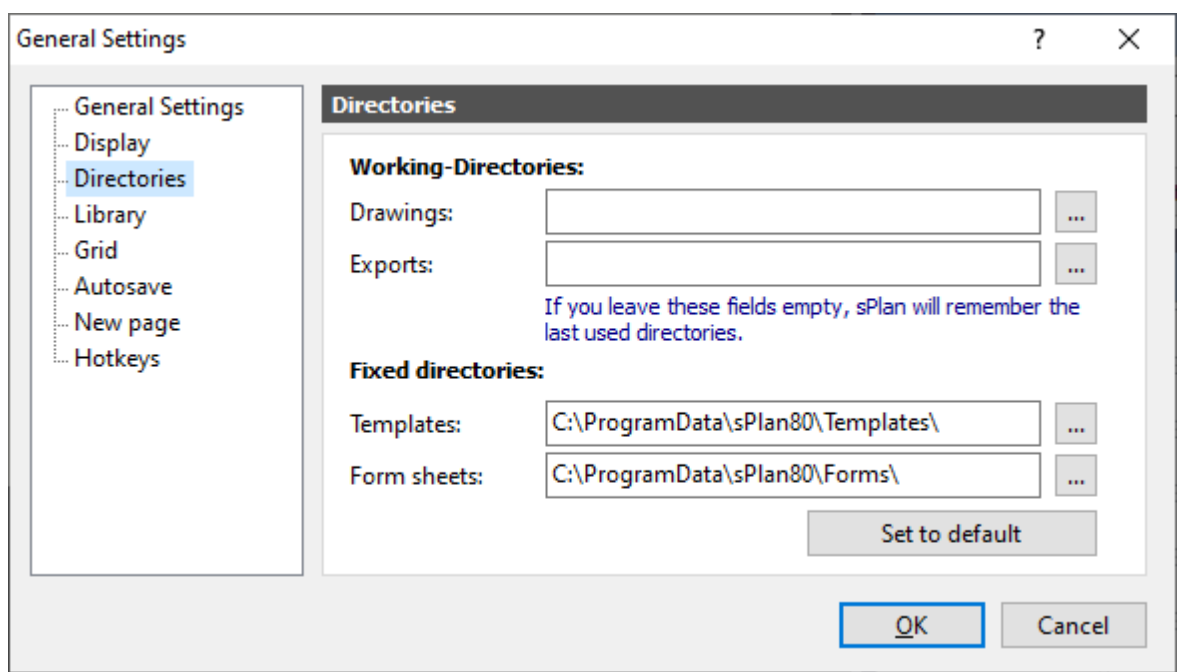
Directories

[General Settings](#) >>

When working with sPlan, you can make certain specifications for the working directories and other special directories.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Directories from the left bar.



Working directories

If necessary, you can specify fixed directories for the file dialogs here.

If you leave the fields empty, sPlan remembers the last directories used and uses them again the next time.

Fixed directories

If necessary, you can change the default directories for templates and forms here.

With the button Set to default these directories are reset to the default sPlan directories.

Library

[General Settings](#) >>

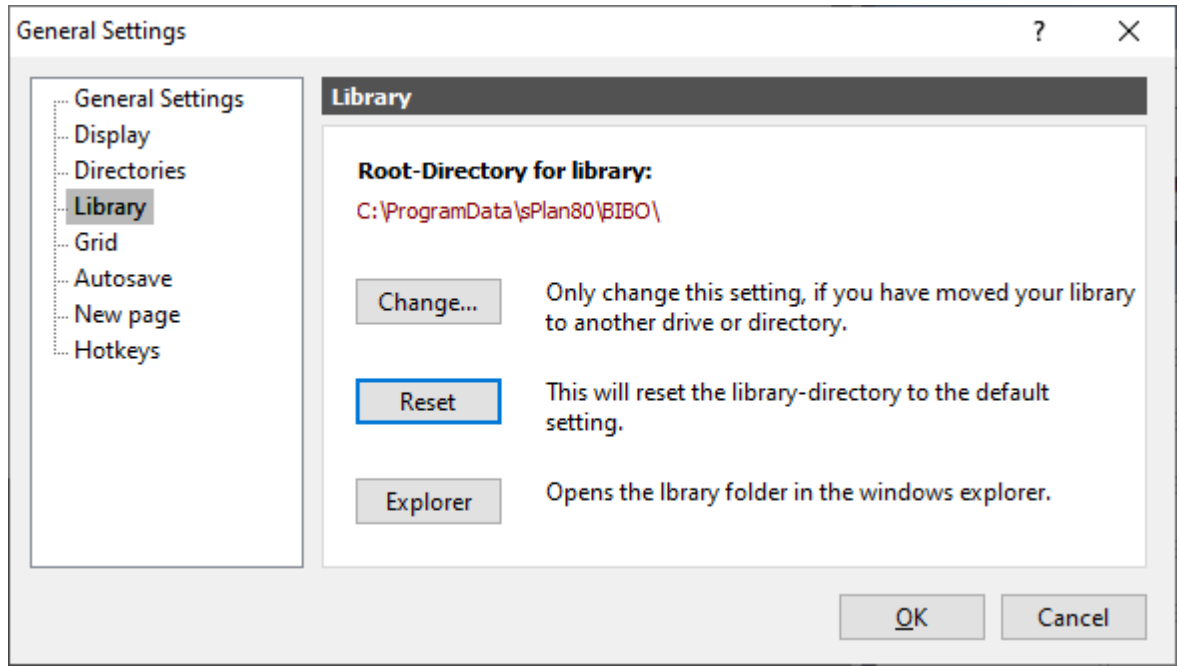

Here you can change the library root directory of sPlan.

The entire component library of sPlan is stored in a root directory.

This directory contains all of the sPlan library files, sorted in subdirectories. Each library file here corresponds to a library page. The library files have the extension * .lib and can contain any number of component symbols.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Library from the left bar.



Change...

Here you can change the library root directory.

This makes sense, for example, if you want to move your library to a network drive.

The new selected directory must exist and should of course also contain the desired library files.

Reset

This will reset the library root directory to its default value.

Explorer

This opens the library root directory directly in a Windows file window.

Here, experienced users can edit the library at file level and thus also adapt the directory structure of the library to suit their own requirements.

[More information about the component library](#)

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Grid

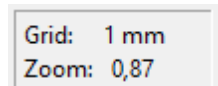
[General Settings](#) >>


When drawing with sPlan, a snap grid is normally always activated. With this snap grid you can no longer move to all positions on the drawing, but only "jump" from one grid point to the next grid point. This function is of great help for the exact positioning and connection of components. You can see and change the grid width of this stored snap grid at any time in the lower status bar.

On the screen, the grid becomes larger or smaller as you zoom in or out. If you zoom out too far so that the individual grid lines would almost touch each other, the grid is automatically hidden.

Incidentally, the grid is only ever visible on the monitor. This is not included in the printout.

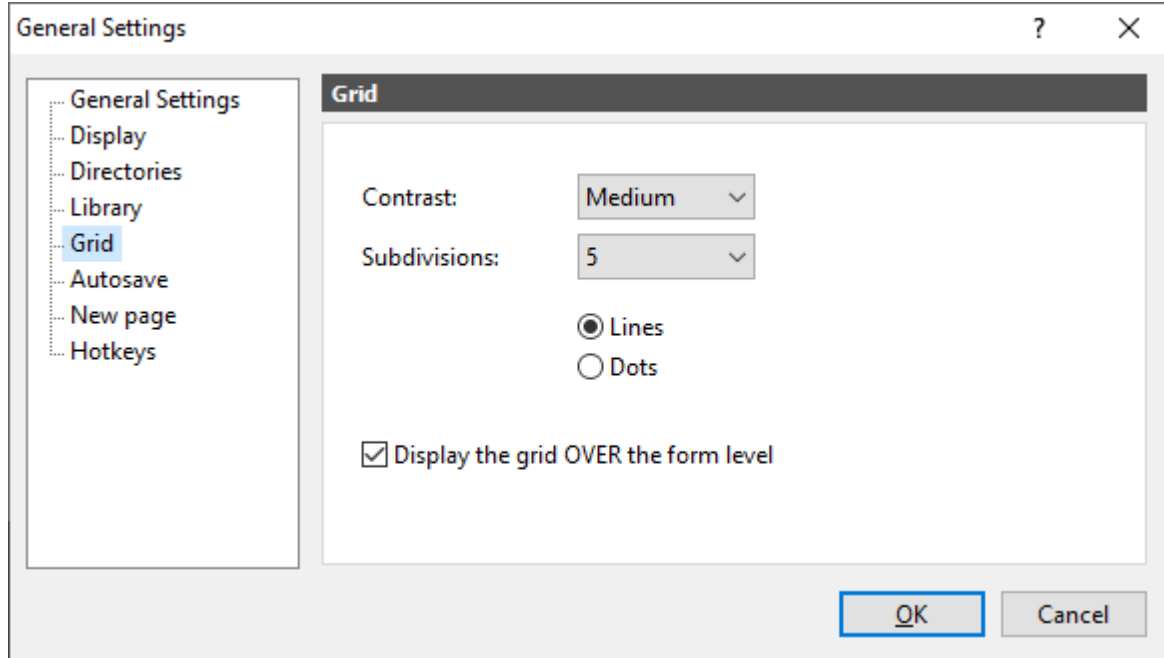
The current grid is always displayed in the lower status bar:



With a click on this grid value you can set it directly.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Grid from the left bar.



Contrast:

Here you enter the display strength of the grid.

Depending on your monitor or your habits, you can use it to adapt the display of the grid to your requirements.

Subdivisions:

You can use this to subdivide the grid again.

With a setting of "5", for example, every 5 grid lines are displayed a little stronger. This makes orientation on the grid much easier, e.g. when counting.

Line / Dots

Determines whether the grid is displayed in lines or points.

Display the grid OVER the form level

When this option is selected, the grid is drawn OVER the form level.

This is particularly helpful if, for example, you have stored a large bitmap as a background on the form that would otherwise completely cover the grid.

Tip:

With the <CTRL> key you can temporarily switch off the grid snap at any time, i.e. as long as you keep the key pressed. This is sometimes helpful, especially in tricky areas.

If you want to work without the grid snap for a longer period of time, you can deactivate it with the button in the lower status line.



Your grid settings are retained, only the snap is deactivated.

To reactivate the grid snap, simply click the same button again.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

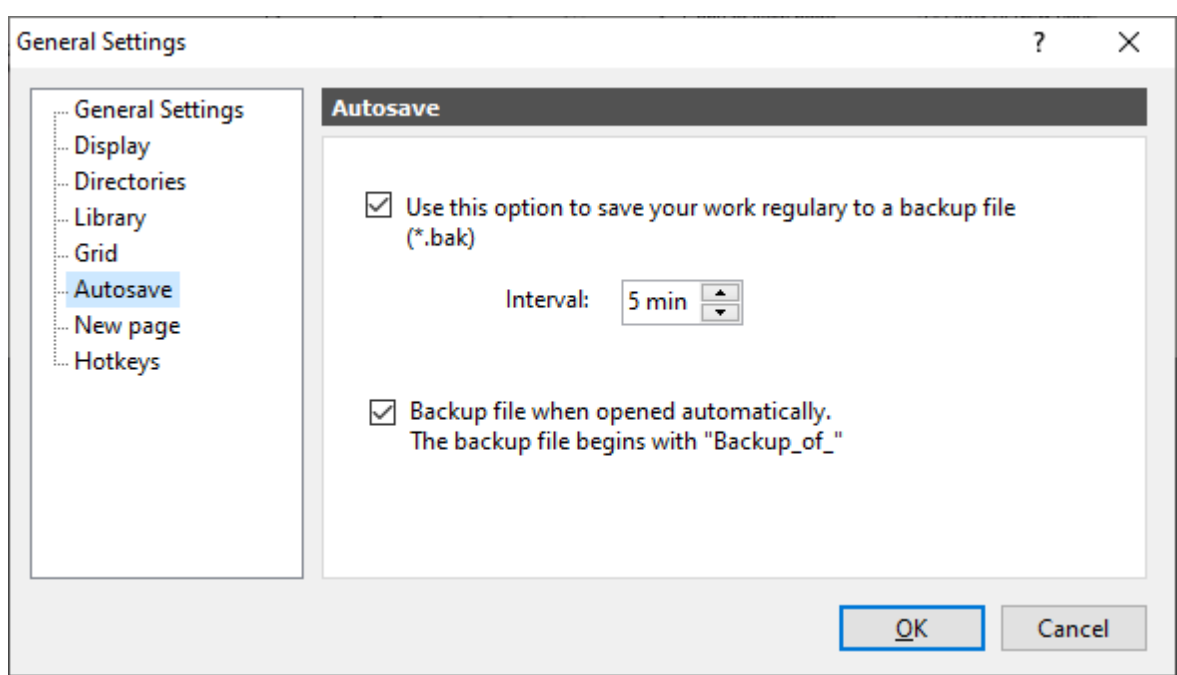
Autosave

[General Settings](#) >>


Here you can activate 2 additional security mechanisms that can protect against data loss in certain cases.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Autosave from the left bar:



With the first option you can activate a function in which the currently open file is automatically saved in a further backup file at regular intervals. The backup file has the same file name as the original file, only the extension is ".bak" instead of ".spl8".

If for whatever reason the original file should be defective, rename the backup file to ".spl8" and try to open it.

The second option provides even more security against data loss. Every time a file is opened, a copy of the original file is automatically saved as a backup file. This backup file always has the status of the file when it was last opened. These backup files always have the name "Backup_of_" followed by the original name of the file.

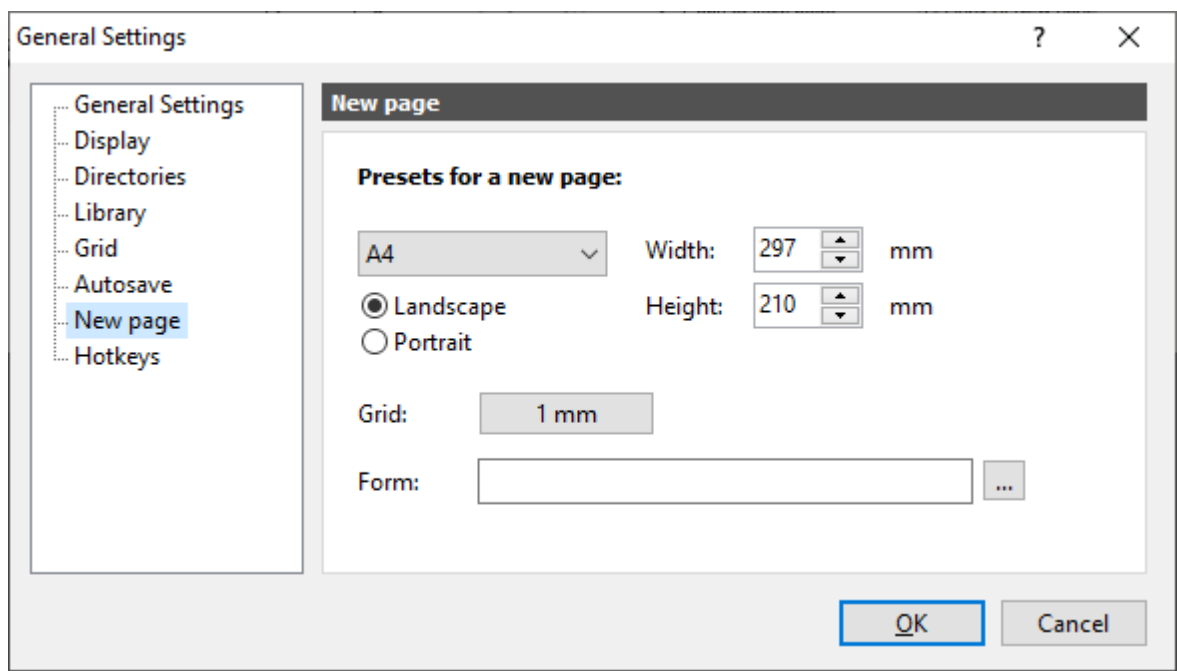
New page

[General Settings](#) >>


Here you can make specifications for the properties of a new page. Whenever you create a new page in sPlan, these defaults are used.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry New page from the left bar.



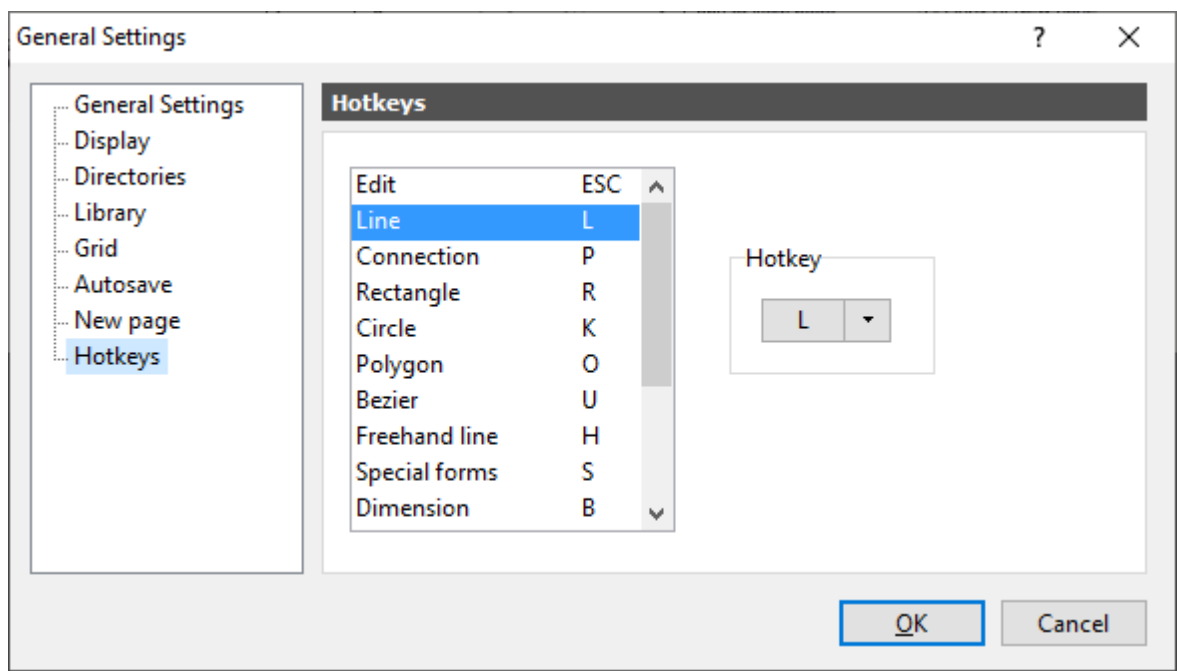
Hotkeys

[General Settings](#) >>


Here you can define hotkeys for the individual tool modes in sPlan. Each button in the vertical tool bar can be assigned its own button. You can then select the corresponding tool mode with the keyboard instead of the mouse.

You call up the settings dialog from the main menu under Settings -> General settings...

Then select the entry Hotkeys from the left bar.



Components



The components are of course the most important elements on a drawing. That is why the processing and use of components occupy an important place at sPlan.

The [component library](#) is located on the left side of the program window. This contains all the components available in this library, sorted by page. You can rearrange this library as you wish: set up new pages, copy components from one page to another, rename pages or delete them entirely, add new components, etc.

The individual pages of the library are arranged alphabetically in a drop-down list directly above the library. Here you select the library page you want.

The components are simply dragged from the library onto the drawing (drag & drop). To do this, click with the mouse on the desired component in the library. With the mouse button still held down, drag the component to the desired position on the drawing. There you release the mouse button again.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Components

[Components](#) >>



The components are of course the most important elements on a drawing. That is why the processing and use of components occupy an important place at sPlan.

The [component library](#) is located on the left side of the program window. This contains all the components available in this library, sorted by page. You can rearrange this library as you wish: set up new pages, copy components from one page to another, rename pages or delete them entirely, add new components, etc.

The individual pages of the library are arranged alphabetically in a drop-down list directly above the library. Here you select the library page you want.

The components are simply dragged from the library onto the drawing (drag & drop). To do this, click with the mouse on the desired component in the library. With the mouse button still held down, drag the component to the desired position on the drawing. There you release the mouse button again.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

What is a Component?

[Components](#) >>



A component is actually just a special group of drawn elements. For example, the component "resistor" consists in the simplest case only of a drawn rectangle with two lines at its ends, which mark the connections. When these drawing elements are then combined into a component, the following happens:

- 2 new text elements are automatically added (identifier and value)

- The drawing elements and the text elements are grouped

By definition, each component has the two text fields identifier and value. In our example, the identifier could be, for example, "R1" and the value "1k". These texts can be freely defined by the user.

The individual components therefore "only" differ in their external appearance. In principle, they all have the same structure.

The sPlan program also offers a few special options for components that make handling them easier and more flexible:

- The text element identifier can be hidden if desired

- The text element Value can be hidden if desired

- You can have the identifier numbered (incremented) automatically

- The value can be automatically queried when dragging the component onto the drawing

- You can determine whether the component should be included in the components list or not

- You can specify additional texts for the component when used in the components list

- You can enter a caption for the component in the library

The library does not necessarily have to contain only components as defined above. It is also possible to add a symbol to the library as a clip. A clip is not a component, just a simple group of drawing elements. Such a clip can be any symbol, a graphic or an entire circuit part. The imagination knows no limits. (Who says that sPlan can only be used to draw circuits?)

Component Properties

[Components](#) >>


You can change any component, whether in the library or on the schematic. If you want to make basic settings on a component, it is better to edit the corresponding component in the library. This is, so to speak, an original, i.e. all components that you drag from here onto the drawing have the same properties. Conversely, if you only want to change one component once, it is better to edit the corresponding component on the drawing. This leaves the "original" in the library untouched.

To change a component, call up its properties.

On the drawing, it is sufficient to select the component with a mouse click.

To edit a library component, double-click the component in the library. The component is now highlighted as "selected" in color.

The properties of the component are now displayed in the properties panel on the right-hand edge of the window:

Component - [1]

Identifier:

R

...

☒ visible

☒ automatic numbering

Value:

1k

...

☒ visible

☐ request value

Component list:


☒ Add to components list

- Add. text 1:

- Add. text 2:

- Add. text 3:

- Add. text 4:

 Component-Editor

Description for library - [1]

Identifier

The identifier is the identification letter of the component followed by a serial number (e.g. "R1"). You can either enter this identifier manually for each component, or you can use the very helpful automatic renumbering option.

visible

With this option you can show or hide the identifier of the component. This option can be useful for some components that have no identifier at all, or that are perhaps not actually components at all.

automatic numbering

With this option you leave the numbering completely to the program. As an identifier, please enter e.g. only the "R" (without a consecutive number!). The software then considers all components on the drawing that also have the identifier "R" (and have also activated the automatic renumbering option) as belonging together and automatically attaches the consecutive numbers to the identifier.

When you delete components on the drawing, there are gaps in the automatic component numbering. To close this again, you can have the software renumber the entire drawing at any time. To do this, select the Renumber Components... command from the Components menu, or click the corresponding toolbar button.

Value

The value is either the numerical value of a component (e.g. "1k") or a type designation (e.g. "BC547" or "SN7400"). With the option request value you have the option of entering the value directly when dragging the component onto the drawing.

visible

With this option you can show or hide the value of the part. This option can be useful for some components that are of no value or that are perhaps not actually components at all.

request value

If you activate this option, every time you drag this component from the library onto the drawing, a small dialog appears in which you can enter the corresponding value. With this option you can force yourself to enter the value of all resistors immediately, for example. If you do not do this, you have to add all the values at the latest when you have finished the drawing. A value can then be quickly forgotten.

Add to components list

If you deselect this field, the component will not be taken into account when generating a components list. This is useful, for example, if the component is not actually a component at all.

Additional texts 1..4 for components list

These additional texts only appear in the components list. Here you can enter special information such as design, materials or order number.

Description for library

If necessary, you can enter a text here, which will then be displayed in the library directly under the component. Please note that this text should not be too long.

Component editor

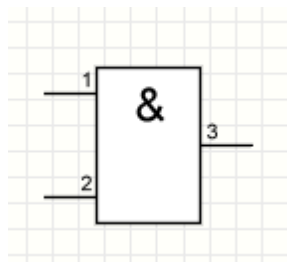
Use the Component editor button to open the [component editor](#) for the relevant component. You can edit the appearance of the component as you wish within the component editor.

Component-Contacts

A component can have any number of [contacts](#) or, better yet, contact descriptions. These are special names within a component. The advantage over fixed texts is that you can change these texts quickly and easily in the component dialog. So you don't need to go through the component editor first.

If contacts are defined in the component, a list of these contacts is displayed in the properties, and you can then view and edit the contact names here:

Component-Contacts:	
E1	1
E2	2
A	3



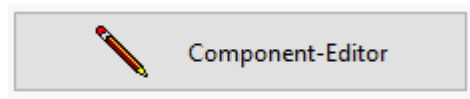
The Component-Editor

[Components](#) >>


To graphically change or expand a component, simply open the Component-Editor.

Either click on the Component-Editor button in the component properties, or simply double-click on the component in your plan.

To change a component directly in the component library, first double-click on the component in the library. the component is then highlighted in color and the component properties are displayed on the right edge of the window. Here you can call the editor with the component editor button.



If the Component-Editor is called up, the component is displayed directly on the work surface. In addition, the background color changes to light blue and a bar appears above the work area:



You can now freely edit the component here. All normal drawing functions are available to you.

The two fixed text elements [ID] and [Value] are shown here in square brackets. You can freely determine the position of these texts in relation to the component. You can also change the text properties (font, color, size) of these text elements. The only thing you cannot change here is the text itself. This is just a placeholder, so to speak, for the final texts in the drawing. These two items cannot be deleted either. If you do not want these to appear on your component, you can switch off the visibility of these text elements in the [component properties](#).

If you want to exit the component editor again, click either on Save and Exit or on Cancel.

You will then automatically find yourself back on your drawing.

Contacts

Another special feature of the component editor is the ability to define [contacts](#).

Contacts are actually nothing more than normal texts. However, contacts do not have a fixed text in the component, but you can edit the texts quickly and easily in the component properties.

Contacts are usually used to label or number the connections of a component. Of course, normal text can also be used for this. But if the connection names are not always the same, it makes sense to define them with contacts instead of normal texts. You only need to add the component to the library once. You can then change the texts of the connection designations directly in the properties of the component without having to open the component editor here.

You have to pay attention to one particularly important point when creating a component: the insertion point of the component.

The insertion point / Red dot

The insertion point is shown as a red dot in the component editor. With this point you determine how the component is inserted into your drawing. A snap grid is always stored on the drawing, and this insertion point of the component is always on a grid point. Only with the correct determination of the insertion point is it possible to connect the various components on the drawing directly with one another.

Normally you always set the insertion point of a component directly on a connector. Please make sure that the insertion point is really EXACTLY on the connection. The easiest way to achieve this is when the connection and insertion point lie on one grid point. Then the accuracy is 100% guaranteed.

The grid setting in which you edit the component has no influence on how the component will later behave on the drawing. You can therefore set the grid at any time during editing as required, just as you need it.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Contacts

[Components](#) >>



A special feature of the component editor is the ability to define contacts. Contacts are actually nothing more than normal texts. However, contacts do not have a fixed text in the component, but the texts can be edited quickly and easily in the [component properties](#).

Contacts are usually used to label or number the connections of a component. Normal text can of course also be used for this. But if the connection names are not always the same, it makes sense to define them with contacts instead of normal texts. You only need to add the component to the library once. You can then change the texts of the connection designations directly in the properties of the component without having to call up the component editor.



To insert a contact in the component editor, click on the corresponding button in the toolbar.

This button only appears in the [Component-Editor](#) at the bottom as the last button.

Click on the button and a contact "sticks" to the mouse pointer and you can position this contact as you like.

Edit properties of contacts

Whenever you select a contact, you can set its properties in the properties panel on the right-hand side of the window. You can also select several contacts at the same time and change their properties together.

Contact - [1]	
Name:	<input type="text" value="C0"/>
Text:	<input type="text" value="1.1"/> ...

Font - [1]	
Font:	<input type="text" value="Arial"/> ▼
Textheight:	<input type="text" value="18"/> [1/10 mm]
Bold:	<input type="checkbox"/>
Italic:	<input type="checkbox"/>
Color:	<input type="color" value="black"/>
Background:	<input type="checkbox"/> <input type="text" value=""/>
Alignment:	<input type="text" value="Right justified"/> ▼
<input type="button" value="Normalize"/>	

Name

Enter a name for the contact here. You can use this name to identify the contact when editing the contact list in the properties dialog of the component.

Text

Here you enter the text that is to be displayed in the component. This text can then be edited later in the properties of the component within the contact list.

With the button "... " you can call up an [extended text dialog](#). Here you can also insert variables or text constants, among other things.

Font

Here you can change the size and appearance of the text.

Create a contact list

In sPlan you can create a contact list. This list then contains all contacts with their designations and the associated components. To create this contact list, select the Create contact list... entry from the Functions menu.

Create own Components

[Components](#) >>



There are several ways to create your own components. The first decision you should make is the following: Should the component be created from scratch, or can it be better derived from an existing component?

Create component from scratch

If you want to create the component from scratch, right-click on the library and then select the New command.

When you execute this command, a new component is added directly to the bottom of the library. As a default, sPlan always creates a new component as a square with a side length of 20mm as a placeholder. The properties of the new component are displayed on the right edge of the window. You can now change this here or call up the [Component-Editor](#).

Create a new component from an existing component

If you want to create a component from another similar component, right-click on the component to be copied in the library and then select the Duplicate command. A copy of the component appears immediately at the bottom as the last component on the library page. The properties of the new component are displayed on the right edge of the window. You can now change this here or call up the [Component-Editor](#).

Create component from selection

There is a third way to create a new component. To do this, simply select the elements you want the new component to consist of on the drawing. Then choose the Create component from selection command from the Components menu. A component is immediately created from the selected elements.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Split Components

[Components](#) >>



You have the option of splitting individual components into their constituent parts. All properties of the component are lost in the process. After splitting a component, it is only available graphically on the drawing.

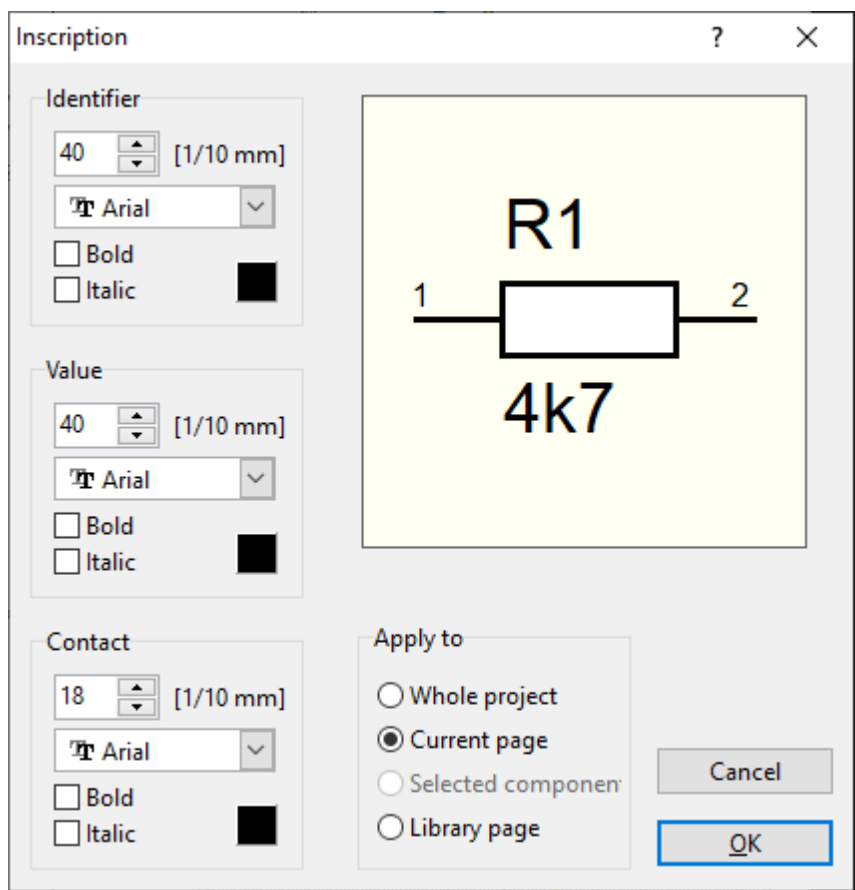
To split a component, first select the desired component. Now select the Split selected component command from the Components menu. You can also select and split several components at the same time.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Change Component inscription

[Components](#) >>


If you want to change the font, the color or the height of several or all component labels, you can do this easily with the following dialog. To do this, call the Inscription... command from the Components menu.



In this dialog you can redefine the text height and font for the components for the identifier, the value and the contacts. In the Apply to field, specify whether you want to apply the changes for the entire project (i.e. for all pages of your project), for the current page, only for selected components or for the current library page. The changes are carried out immediately after confirming the dialog.

Automatic Component Numbering

[Components](#) >>



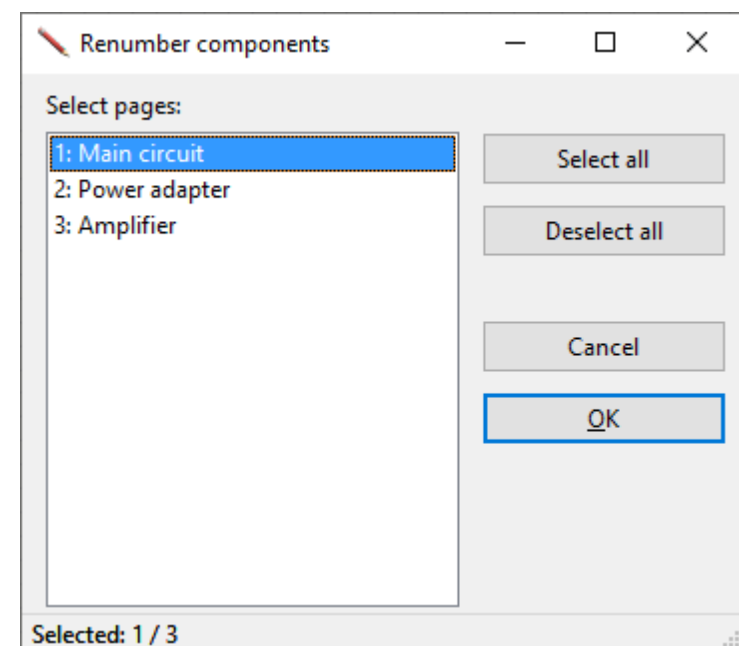
With this function, you no longer have to worry about the annoying numbering of the individual components. You can leave this process completely up to the software, so you can always be sure that all components are numbered correctly.

You can determine for each component individually whether it is eligible for automatic numbering or not. In the properties of a component you can activate the "automatic numbering" field to activate the function for this component. As an identifier, please enter e.g. only the "R" for resistors (without a consecutive number!).

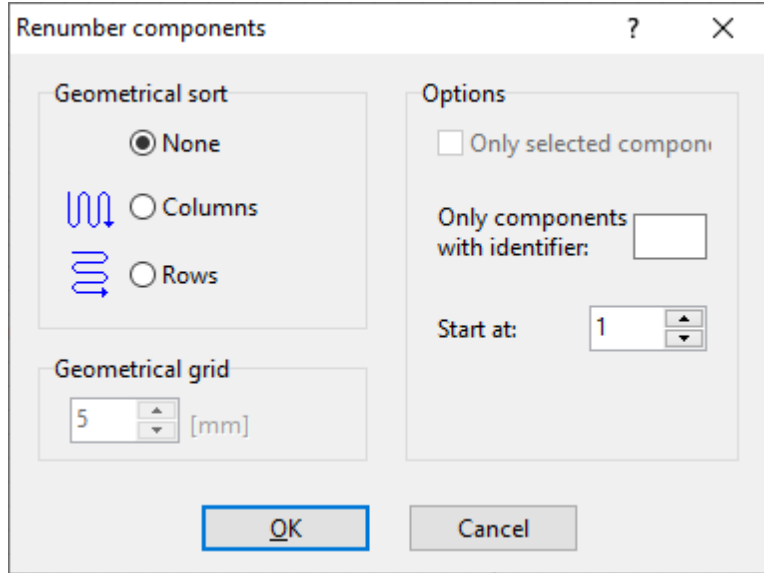
The software then considers all components on the drawing that also have the identifier "R" (and have also activated the option "automatic numbering") as belonging together and automatically attaches the consecutive numbers to the identifier.

Preferably make the changes to the components in the library and not to the individual components on your schematic. If you have changed the component accordingly in the library, all newly imported copies on the schematic will have the same properties as the original in the library.

When you delete components on the drawing, there are gaps in the automatic component numbering. To close this again, you can have the software renumber the entire drawing at any time. To do this, select the Renumber components... command from the Components menu, or click the corresponding toolbar button. The following window then appears:



Here you select the pages over which the automatic renumbering should extend. After you have confirmed this information, the following dialog appears:



Here you can have a certain influence on the component numbering.

Geometric sorting

None

If you choose this option, the components are simply numbered consecutively in the order in which they were created.

Columns

If you choose this option, the components are numbered column by column. This means that e.g. components lying one below the other are numbered in ascending order.

Rows

If you choose this option, the components are numbered line by line. This means that e.g. components lying next to each other are numbered in ascending order.

Geometrical grid

If you have selected a geometric sorting (columns or rows), you can use this to determine the "sample size". In the case of a geometric sorting, the drawing is scanned either column by column or line by line with an imaginary square of a certain side length. All components that then lie within this square are numbered first, then the square is shifted downwards (column by column) or to the right (row by row) and the process begins again.

You can determine the size of this square in the "Geometric grid" field. Depending on the size of the geometric grid, different numbering variants can arise.

Options

Only selected components

If you have selected components before calling up the dialog, you can determine here whether only these components should be renumbered.

Only components with identifier:

If you enter an identifier (e.g. "R") here, only components with this identifier will be renumbered.

Start at:

If the components are not to be numbered consecutively from 1, you can also specify a different starting value here if necessary.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

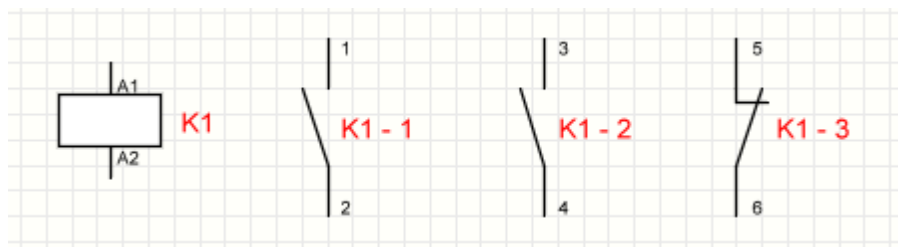
Parent-Child Relations

[Components](#) >>


Individual components can have a parent-child relationship to one another. A component can be declared as a parent and other components can then declare this parent component as their own parent and thus automatically become a child of the superordinate parent.

A parent knows all of their children, and a child always knows their parent. This relation enables the components to access the data of the associated component. For example, the identifier of a child can automatically contain the name of its parent. If the identifier of the parent then changes, the name of the child also changes automatically. Something like this is particularly advantageous when using contactors and contacts or also with ICs with several gates.

Here is an example with a contactor and 3 associated contacts:



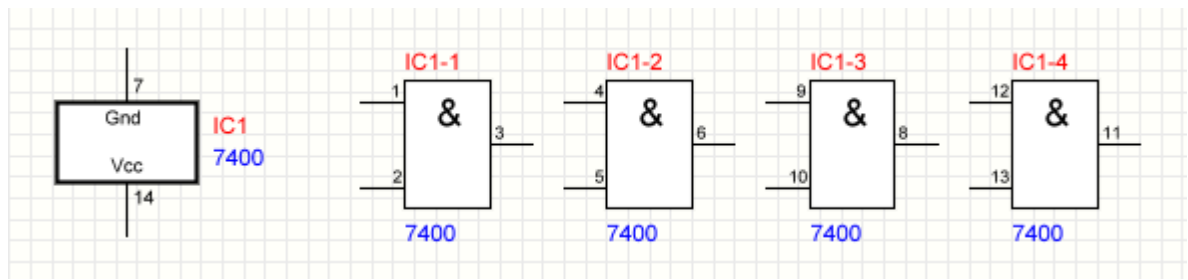
The contactor is a parent and has the identifier **K1**.

The contacts have assigned contactor **K1** as a parent and are therefore automatically its children. In this example, the children take the identifier **K1** from the parent and subsequently receive a sequential number.

If the identifier of the contactor changes, the identifiers of all contacts also change automatically.

The contacts can also be distributed over several different pages of the project.

Here is another example of a 7400 IC:



The main component on the left is the parent, and the 4 individual gates are the children.

In this example, the children take on the identifier **IC1** and also the value **7400** of the parent.

If the identifier or the value of the parent changes, all the identifiers and values of the individual gates change automatically. Here, too, the gates can be spread over several different pages of the project.

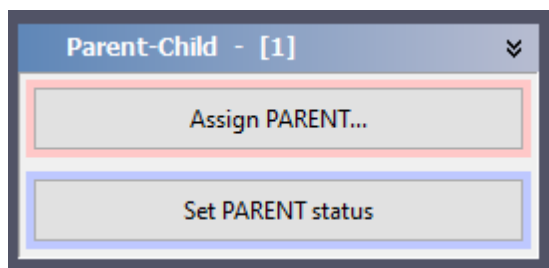
Create a Parent-Child relation

The creation of a parent-child relationship is simple: You simply select another component as the parent for any component. As soon as a component has a parent, it is automatically a child of this parent.

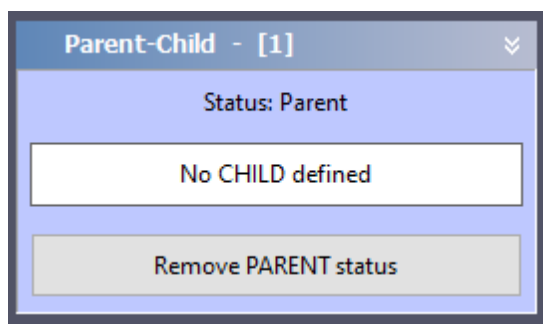
Define a Component as a Parent:

Only those components can be assigned as parents that have already been declared as parents.

To define a component as a parent, select it and then click the Set PARENT status button in the component properties.



The component then has the parent status.




Define a Child / Assign a Parent:

If a parent is assigned to a component, it automatically becomes a child.

To assign a component to a parent, select it and then click the Assign PARENT... button in the component properties.

Now select the desired parent from the list of available parents.


Parent-Child - [1] 

Assign PARENT...

Set PARENT status

You can also select several components at the same time and set the parent for all selected components together. So if you have a contactor and 3 contacts as in the example above, then select the 3 contacts and then assign all 3 components with the contactor as a parent.

After assigning a parent, the component is automatically a child.

Parent-Child - [1] 

Status: Child


Parent	on page	Identifier	Value
->	1: Main circuit	K1	

Assign PARENT...

Remove PARENT

Automatic jumping to a Child or Parent:

If you select a parent with subordinate children, all children are listed in the component properties:

Parent-Child - [1] 

Status: Parent

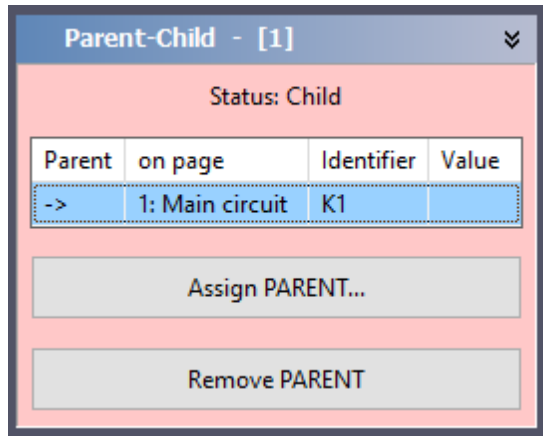
Child	on page	Identifier	Value
1	1: Main circuit	K1	
2	1: Main circuit	K1	
3	1: Main circuit	K1	

Create Childlist (Contact overview)

Remove PARENT status

You can now simply double-click on an entry within this list and you will be navigated to the corresponding child, even if this should be on a different page.

The whole thing works the same way with a child. The linked parent is always displayed here:



With a double-click on the entry you will be navigated directly to the parent, even if this should be on a different page.

Parent-Child-Variables

So that a child can automatically adopt the identifier and the value of its parent, special [variables](#) are used that are only available for the parent-child relationship:

<PARENT_ID>

Identifier of the parent

<PARENT_VALUE>

Value of the parent

<PARENT_PAGENO>

Page number of the parent

<CHILDNO>

Number of the child

<CHILDCHAR>

Alphabetical letter corresponding to the number of the child (a, b, c ...)

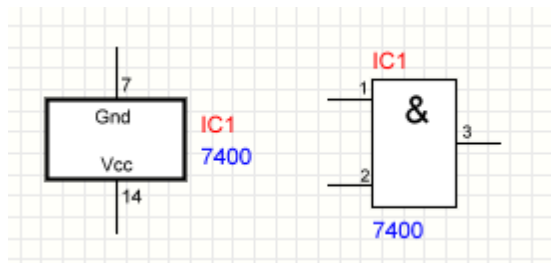
<CHILD_PAGENO>

Page number of the child

So that a child automatically adopts the identifier and the value of its parent, the corresponding fields of the child are pre-assigned with 2 variables when they are linked to a parent:

The identifier automatically receives the variable <PARENT_ID>

The value is automatically given the variable <PARENT_VALUE>



You can also change or extend this default as you wish.

<CHILDNO> and <CHILDCHAR>:

The parent manages its children and gives them a sequential number. The variable <CHILDNO> indicates this number. For example, if a parent has 3 children, then these 3 children have the numbers 1, 2 and 3. This number can e.g. be displayed in addition to the identifier. To do this, define the identifier of the child as follows: <PARENT_ID> - <CHILDNO>

Now the identifiers of the children on the plan will look like this: IC-1, IC-2, IC-3, ...

The <CHILDCHAR> variable specifies the serial number as a character, i.e. a, b, c ...

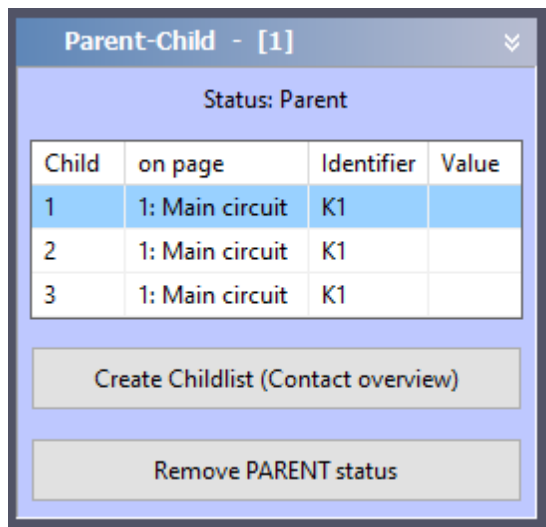
To do this, define the identifier of the child as follows: <PARENT_ID> - <CHILDCHAR>

Now the identifiers of the children on the map will look like this: IC-a, IC-b, IC-c, ...

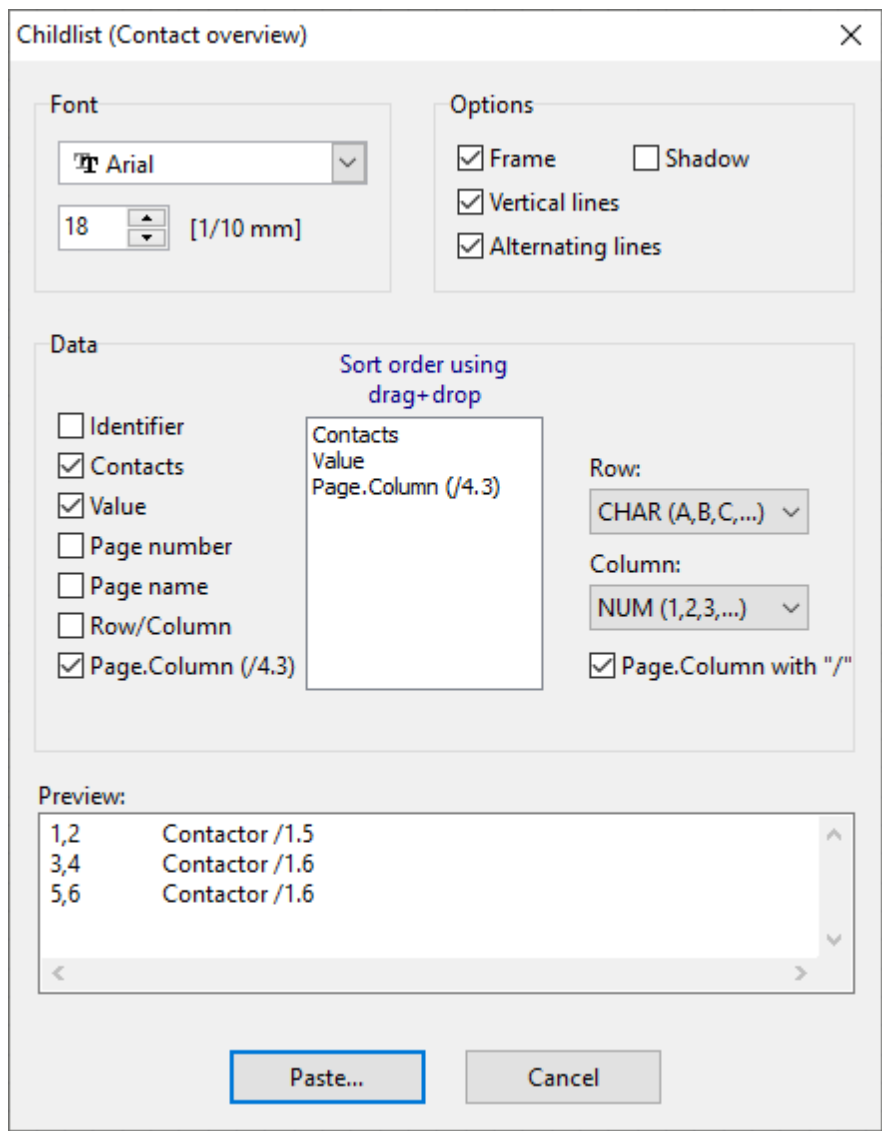
Child-List (Contact overview)

If you select a parent with subordinate children, all children are listed in the component properties.

Here you can also create a child-list (contact overview) for the component:

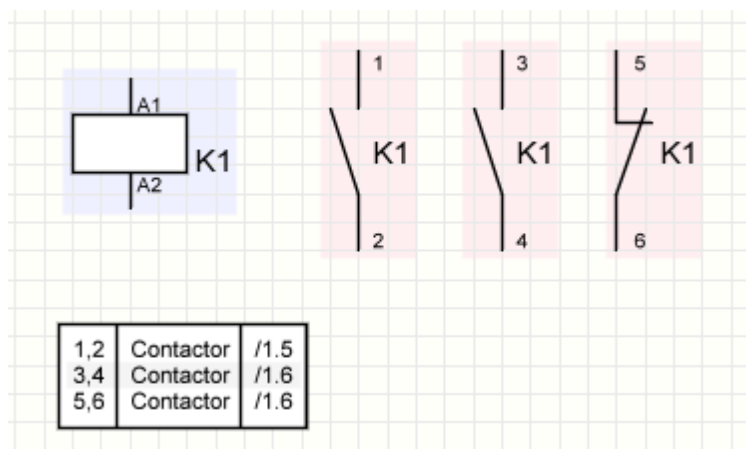


After clicking on Create Childlist (Contact overview), this dialog appears:




Here you can make certain settings for the contact overview.

With a click on Paste... you can then place the contact overview on your plan:



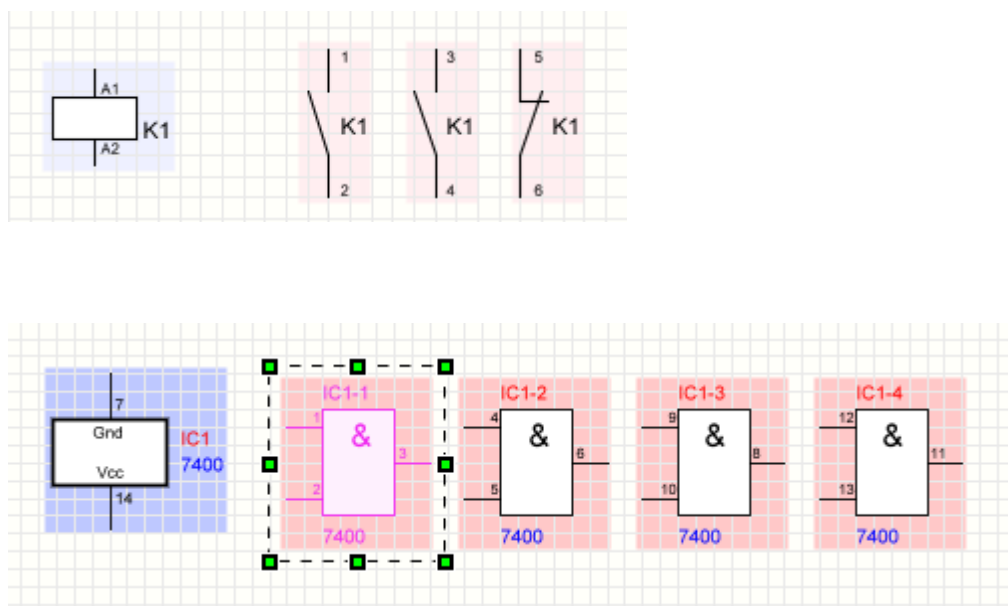
Colorize Parent-Child

sPlan can color-mark all parent-child components on the plan. This enables you to see at a glance which components are defined as parents or children, and which children belong to which parent.

 To switch this function on/off, please use this button in the upper toolbar of sPlan.

All parents are now highlighted in blue, all children in a light reddish background.

If you now select a parent or a child, the parent and all associated children are colored more strongly and you can immediately see how the components belong together:

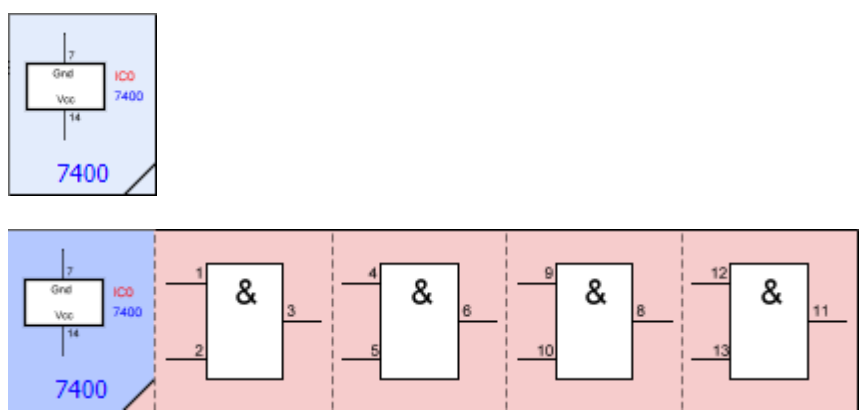


Parent-Child in the Library

If it makes sense, you can include a complete parent-child structure as 1 component in the library.

That would make sense e.g. for the above example of a 7400-IC with its 4 gates.

To do this, simply select the parent component and all of its children. Now click with the right mouse button on the parent and select the entry Copy selected component (with children) to the library. Only the parent component is added to the library, but the children are stored in the parent. You can recognize this by the fact that a small triangle is displayed in the library for this component at the bottom right. If you move the mouse to this triangle, the associated children are displayed to the right of the component:



If you drag such a component onto your plan, the parent component and its children will be created.

Create Parent-Child-List

You can create a hierarchical list with all parents and their children.

This is useful, for example, for an extensive project with contactors and contacts, spread across different pages. The associated children for all parents are displayed in this list. This gives you a complete overview of all the pages in the project.

To create this list, select the entry Create Parent-Child-List... from the Functions menu.

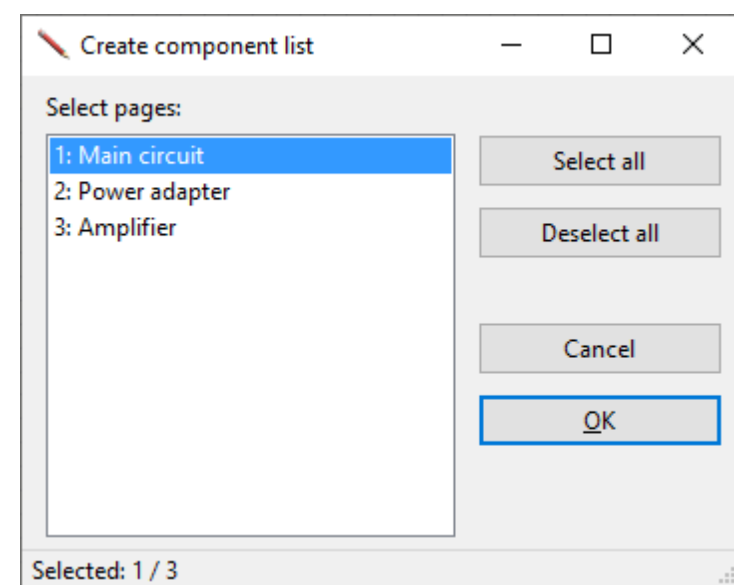
Component List

[Components](#) >>

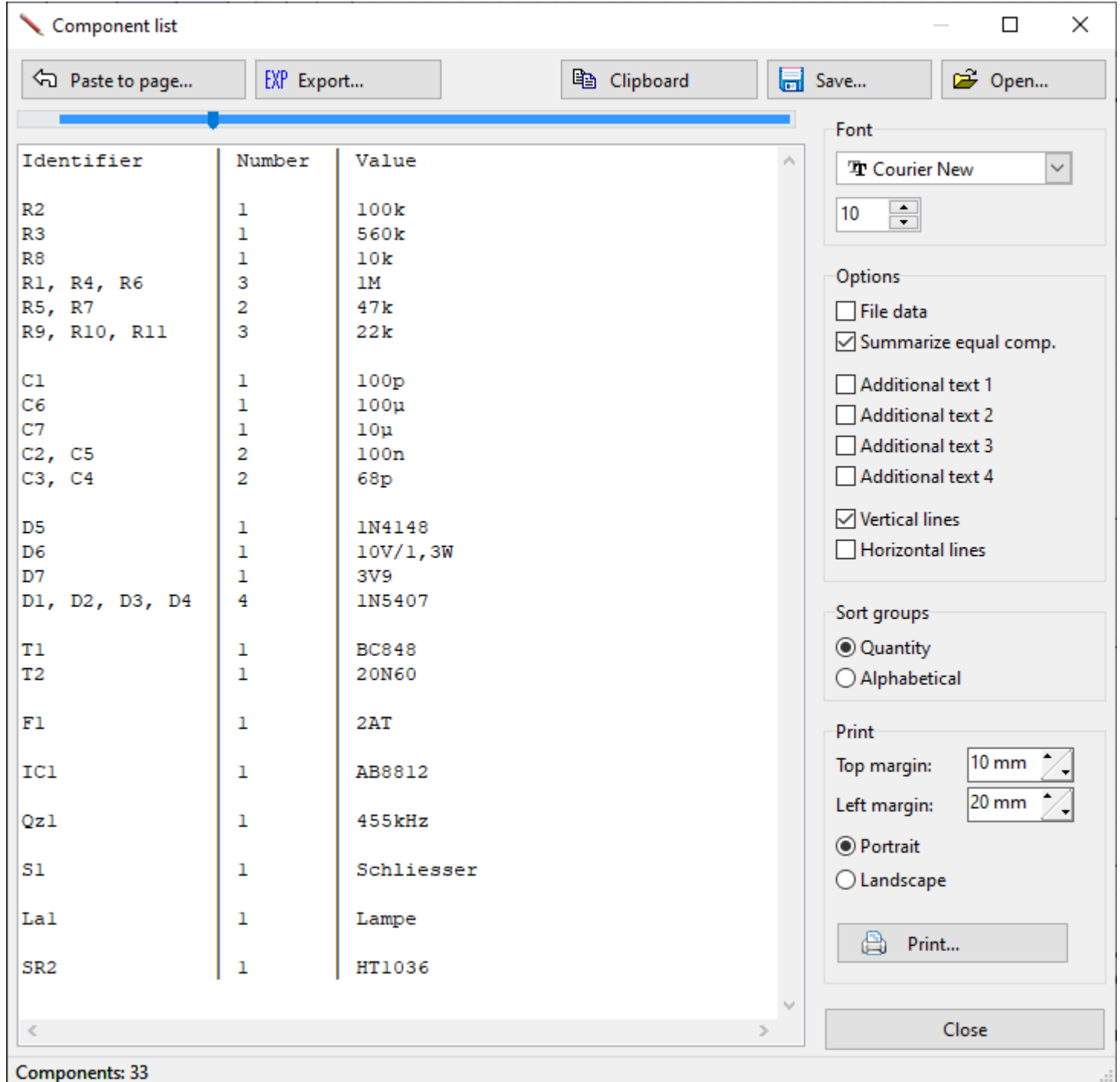


An important function of sPlan is the creation of component lists. If required, these are created fully automatically by the software. You can then edit, print, load or save the component list by hand as you wish.

To create a component list, select the Create component list... command from the Components menu or click on the corresponding button in the toolbar. The following window then appears:



Here you select the pages over which the component list should extend. After you have confirmed this information, the component list appears:



You can edit the component list as you wish within the component list window. You also have the following options here:

Font

Here you can set the font and the font size of the component list.

Options

File data

With this option, various additional information is included in the component list:

Date and Time

Name and folder of the project file

Names of the individual pages for the component list

This data is inserted at the top of the component list.

Please note that if you change this option, the component list will be completely recreated. Any changes that you have made by hand will be lost.

Summarize equal components

If you activate this option, all components with the same value are automatically combined into one entry in the component list.

What looked like this before, for example ...

R1 = 1k

R2 = 1k

R3 = 1k

... then becomes:

R1, R2, R3 = 3 x 1k

This increases the clarity of the component list considerably.

Please note that if you change this option, the component list will be completely recreated. Any changes that you have made by hand will be lost.

Additional text 1..4

Here you can show or hide extra columns for the additional texts (Z1..Z4)

Vertical lines

This allows you to add vertical dividing lines to the component list

Horizontal lines

This allows you to add horizontal dividing lines to the component list

Sort groups

Quantity

If you select this option, the component group that is most frequently represented will appear first in the component list. All other groups then also appear in the order of their frequency.

Alphabetical

If you select this option, the component groups are sorted alphabetically based on their identifiers.

Print

Top margin / Left margin

Before printing, determine the top and left print margins so that the component list is not printed right at the edge.

Portrait / Landscape

Determine the paper size before printing.

Print

Prints the component list

Paste to page...

With the button Paste to page... you can insert the component list directly on the current page:

Export

Field separator

☒ Semicolon
☐ Tabulator

Options

☐ Field names in 1. row
☐ Insert empty lines

Save...

Cancel

Identifier	Number	Value
R2	1	100k
R3	1	560k
R8	1	10k
R1, R4, R6	3	1M
R5, R7	2	47k
R9, R10, R11	3	22k

Clipboard

This function copies the component list to the Windows clipboard. From here it can be inserted by any other program (e.g. Word). You can then continue to edit the component list with the other program as you wish.

Save...

With this button you can save the current component list. The component list is saved in the universal "rich text format" and has the ending ".RTF". Most word processing programs can read this format.

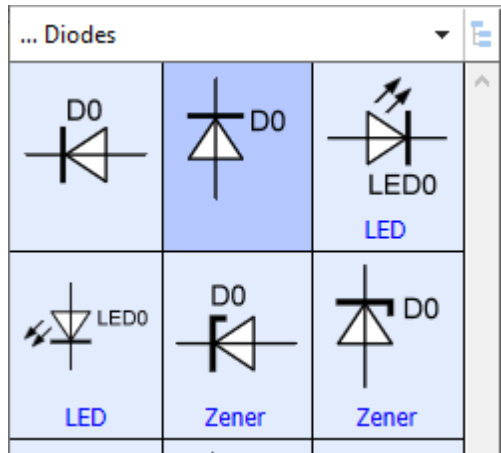
Open...

With this button you can reopen an already saved component list.

The Component Library



The component library is located on the left edge of the sPlan window:



The components are simply dragged from the library onto the drawing (drag & drop). To do this, click with the mouse on the desired component in the library. With the mouse button still held down, drag the component to the desired position on the drawing. There you release the mouse button again.

The current library page is always displayed above the library. Simply click on this field and a menu with all available library pages will open. Here you can simply select a new library page. Since the library has a directory structure, you can also branch into subdirectories here. The top entry "<---" always takes you back one level.



With this button you can display the library completely as a tree structure.

You can rearrange the library as you wish:

- Copy components
- Sort components
- Delete components
- Create new components
- Copy library pages
- Delete library pages

Rename library pages

Create new library pages

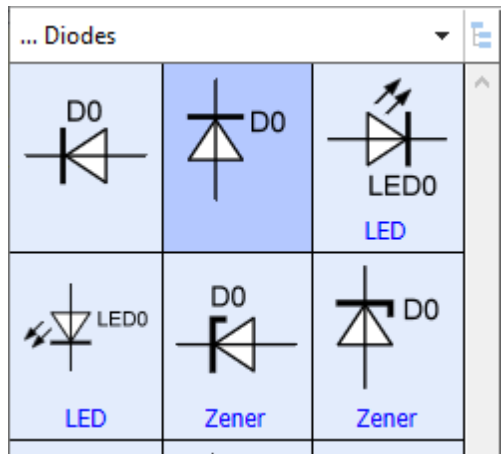
The library has its own local menu (context menu). You can do this by moving the mouse over the library and clicking the right mouse button once. Here you can quickly and easily access all library management commands.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

The Component Library

[The Component Library](#) >>

The component library is located on the left edge of the sPlan window:



The components are simply dragged from the library onto the drawing (drag & drop). To do this, click with the mouse on the desired component in the library. With the mouse button still held down, drag the component to the desired position on the drawing. There you release the mouse button again.

The current library page is always displayed above the library. Simply click on this field and a menu with all available library pages will open. Here you can simply select a new library page. Since the library has a directory structure, you can also branch into subdirectories here. The top entry "<---" always takes you back one level.



With this button you can display the library completely as a tree structure.

You can rearrange the library as you wish:

- Copy components
- Sort components
- Delete components
- Create new components
- Copy library pages
- Delete library pages

Rename library pages

Create new library pages

The library has its own local menu (context menu). You can do this by moving the mouse over the library and clicking the right mouse button once. Here you can quickly and easily access all library management commands.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Representation of the Library

[The Component Library](#) >>


You can influence how the library is displayed on the left-hand side of the screen. You can determine how many columns the library should have and whether the signatures of the components should be shown or hidden. You set these options directly below the library:



The two buttons "+" and "-" increase or decrease the number of columns in the library.

With the button next to it you can show or hide the component signatures.

Use the two arrows up and down to select the previous or next library page.

The magnifying glass icon lets you browse the library

You can determine the width of the component library directly with the mouse. To do this, move the mouse to the right border of the library. The mouse pointer will now turn into a double arrow. You can now determine the width of the library by holding down the mouse button.

Edit a Library page

[The Component Library](#) >>



You can design the component library according to your wishes. You receive all the necessary commands when you right-click on a component in the library:

Properties

This calls up the [component properties](#) of the currently selected component.

Alternatively, you can simply double-click the desired component.

Properties (all)

This calls up the [component properties](#) for all components on the current library page.

Changes you make here are applied to all components on the current library page. So be careful here.

New

When you execute this command, a new component is added directly to the bottom of the library. As a default, sPlan always creates a new component as a square with a side length of 20mm as a placeholder. The properties of the new component are displayed on the right edge of the window. You can now change this here or call up the component editor.

Duplicate

If you want to create a component from another similar component, right-click on the component to be copied in the library and then select the Duplicate command. A copy of the component appears immediately at the bottom as the last component on the library page. The properties of the new component are displayed on the right edge of the window. You can now change this here or call up the component editor.

Delete...

This will delete the selected component from the library page.

Library page ->

New...

This creates a new blank library page.

After executing this command, enter the desired name for the new page. e.g. "resistors". When assigning names, make sure that

the library pages are sorted alphabetically. You can also use the name to determine the position of the page in the list.

Copy...

With this you copy a complete library page.

Here, too, a new library page is created just like with the "Library page -> New" command. The new page is not empty, however, but contains a copy of all components of the currently selected library page.

Rename...

With this you can rename the current library page.

When assigning names, make sure that the individual pages are then sorted alphabetically. You can also use the name to determine the position of the page in the list.

Delete...

This will delete the current library page.

After confirming the security question, the page is then irrevocably removed from the library.

Empty...

This will delete all components from the current library page.

After confirming the security question, the page is not deleted as with "Library page -> delete ...", but it is still available, but empty.

Copy to page ->

This library page only

A new page is created and all components of the current library page are automatically placed on this new page.

All library pages of this level

A new page is created for each library page and all components of the respective library page are automatically placed on this new page.

Extras ->

Create backup ...

This will create a backup of your library as a ZIP file.

Restore backup...

You can use this to restore a previously created backup.

Reload library

With this you force a new reading of the library. This is particularly necessary if you have made changes at the file level in the library directory. This saves you from having to restart sPlan.

Explorer...

This calls up Windows Explorer with the current library directory.

This function is only intended for experienced users to work directly with the library files (* .LIB). Here these files can be copied, removed, renamed or re-sorted at file level.

Sort components within a page

To change the position of a component, simply drag it to the desired position. Simply click on the component, hold down the mouse button, drag it to the desired position within the library and then release the mouse button again. The position at which the dragged component is inserted is shown with a red square while dragging.

Reorder or group library pages

The library in sPlan has a directory structure. Library files can be sorted and grouped in subdirectories. However, this functionality is only directly accessible on the Windows file level. With the function Extras -> Explorer (as described above) you can open a Windows directory window with the current library directory. Here you will find all LIB files sorted in the respective subdirectories. Experienced users can create their own subdirectories here on file level, copy LIB files, etc. in order to adapt the directory structure of the library to your requirements.

Copy components from one library page to another

To copy a component from one library page to another, proceed as follows:

First select the library page that contains the relevant component.

Drag this component onto your schematic.

Now select the library page into which the component is to be copied.

Now click with the right mouse button on the component on the drawing and select the command Copy selected component (s) to the library.

Tip:

With the last command you can not only copy 1 component, but any number of components from the drawing to the library in one fell swoop.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Variables / Text Constants



In sPlan you can also use variables and text constants within text elements instead of fixed texts.

Text constants

Text constants are simply stored texts that are available on demand at any time. If your drawings often contain the same or similar texts, it makes sense to save these as text constants. Every time you have to re-enter this text, you can then access these [text constants](#) and save yourself typing.

Variables

Variables are an expanded form of text constants. They consist of a variable name and its content, the variable text. The variable name is a placeholder for the actual variable text.

There are 2 different types of variables in sPlan. There are predefined fixed variables and there are freely definable user variables. The predefined [fixed variables](#) are available anytime and anywhere. The [user variables](#) can be defined by the user within a project and are only ever valid within this project.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

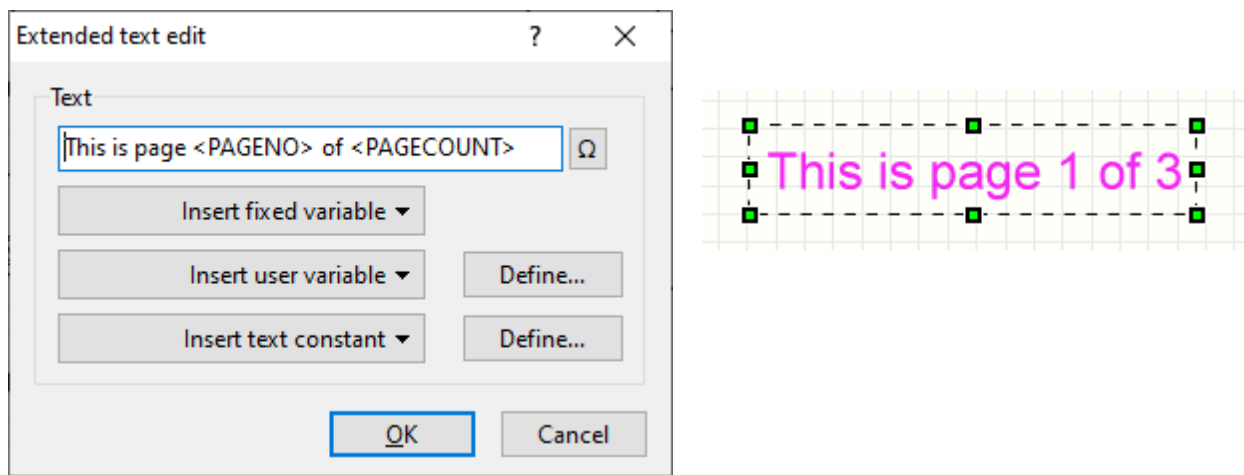
Fixed Variables

[Variables / Text Constants >>](#)


Variables can be used as placeholders within a text.

There are a large number of predefined so-called fixed variables in sPlan. You can use these variables as you wish within a text and thus automate the texts in a way. The variable name must always be embedded in the texts with the characters "<" and ">". A variable looks like this, for example: <PAGENO>

Here is a simple example:



You can reach the Extended text edit dialog simply by double-clicking on the text.

Here the 2 variables <PAGENO> and <PAGECOUNT> have been used in the text. These two variables supply the page number of the current page and the total number of pages in the project. The variables change their content automatically. So if a new page is added to the project, the text automatically changes to "This is page 1 of 3".

Here is a list of all predefined fixed variables:

<PAGENO>

The page number

<PAGECOUNT>

The total number of pages

<PAGENAME>

The name of the page

<FILENAME>

The name of the file (if saved)

<FILENAME_PURE>

The name of the file (if saved) without file extension

<FILEPATH>

The folder in which the file is saved

<FILEDATE>

The date of the file (if saved)

<FILETIME>

The time of the file (if saved)

<VERSION>

The program version (= "sPlan 8.0")

<TIME>

The current time

<DATE>

The current date

Variables that are only available within a component:

<BEZ>

A copy of the part identifier

<VALUE>

A copy of the component value

<Z1>

A copy of the additional text 1

<Z2>

A copy of the additional text 2

<Z3>

A copy of the additional text 3

<Z4>

A copy of the additional text 4

If you have defined a [form](#) in the page properties, the following additional variables are available:

<COLNUM>

The number of the COLUMN in which the component is located

<COLCHAR>

Alphabetical letter for the number of the COLUMN in which the component is located (A, B, C ...)

<ROWNUM>

The number of the ROW in which the component is located

<ROWCHAR>

Alphabetical letter for the number of the ROW in which the component is located (A, B, C ...)

Variables that are only available in connection with the [Parent-Child relationship](#):

Variables for use in Child components:

<PARENT_ID>

Identifier of the parent

<PARENT_VALUE>

Parent's value

<PARENT_PAGENO>

Page number of the parent

<PARENT_Z1>

Additional text 1 of the parent

<PARENT_Z2>

Additional text 2 of the parent

<PARENT_Z3>

Additional text 3 of the parent

<PARENT_Z4>

Additional text 4 of the parent

<PARENT_CONTACT_1>

Contact 1 of the parent (...2, ...3, etc.)

<PARENT_COLNUM>

The number of the COLUMN in which the parent is located

<PARENT_COLCHAR>

Alphabetical letter for the number of the COLUMN in which the parent is located

<PARENT_ROWNUM>

The number of the ROW in which the parent is located

<PARENT_ROWCHAR>

Alphabetical letter for the number of the ROW in which the parent is located

<CHILDNO>

Child number

<CHILDCHAR>

Alphabetical letter corresponding to the number of the child (a, b, c ...)

Variables for use in Parent components:

<CHILD_PAGENO>

Page number(s) of the child

<CHILD_PAGENAME>

Page number of the (first) child

<CHILD_COLNUM>

The number of the COLUMN in which the (first) child is located

<CHILD_COLCHAR>

Alphabetical letter for the number of the COLUMN in which the (first) child is located

<CHILD_ROWNUM>

The number of the ROW in which the (first) child is located

<CHILD_ROWCHAR>

Alphabetical letter for the number of the ROW in which the (first) child is located

These <CHILD_...> variables always return the data of the first child.

If you have linked several child-components with the parent, you can also specify the desired child number directly in the variables. Simply add an underscore followed by the child-number to the end of the variable BEFORE the ">". For example, the variable <CHILD_PAGENO_2> returns the page-number of the 2nd child, or <CHILD_COLNUM_4> returns the column-number of the 4th child.

Variables that are only available within [text links](#):

<LINK_PAGENO>

The page number on which the target text is located

<LINK_PAGENAME>

The name of the page on which the target text is located

<LINK_TEXT>

The text of the target text

<LINK_COLNUM>

The number of the COLUMN in which the target text is located

<LINK_COLCHAR>

Alphabetical letter for the number of the COLUMN in which the target text is located

<LINK_ROWNUM>

The number of the ROW in which the target text is located

<LINK_ROWCHAR>

Alphabetical letter for the number of the ROW in which the target text is located

<LINKFROM_PAGENO>

The page number on which the source text is located

<LINKFROM_PAGENAME>

The name of the page on which the source text is located

<LINKFROM_TEXT>

The text of the source text

<LINKFROM_COLNUM>

The number of the COLUMN in which the (first) source text is located

<LINKFROM_COLCHAR>

Alphabetical letter for the number of the COLUMN in which the (first) source text is located

<LINKFROM_ROWNUM>

The number of the ROW in which the (first) source text is located

<LINKFROM_ROWCHAR>

Alphabetical letter for the number of the ROW in which the (first) source text is located

Copyright © ABACOM GbR, Germany. All Rights Reserved.

User Variables

[Variables / Text Constants](#) >>



User variables can be used as placeholders in a text. They consist of a variable name and its content, the variable text. The variable name is a placeholder for the actual variable text.

The content of the variable then appears on the drawing instead of the variable name.

Here is a small example:

For example, if you define a variable called <PROJECTNAME> and have assigned the text "Measuring amplifier project 45X2-9" to this variable, the text "Measuring amplifier project 45X2-9" is displayed on the drawing for a text element with the content <PROJEKTNAME> .

You could of course also write the text directly without a variable. The advantage of a variable, however, is that if you use this variable several times in your project (possibly even on each page), then you only need to change the content of this variable, and the text will also change everywhere on the schematic where this variable was used.

 User variables

—

□

×

These variables will be saved together with the file.

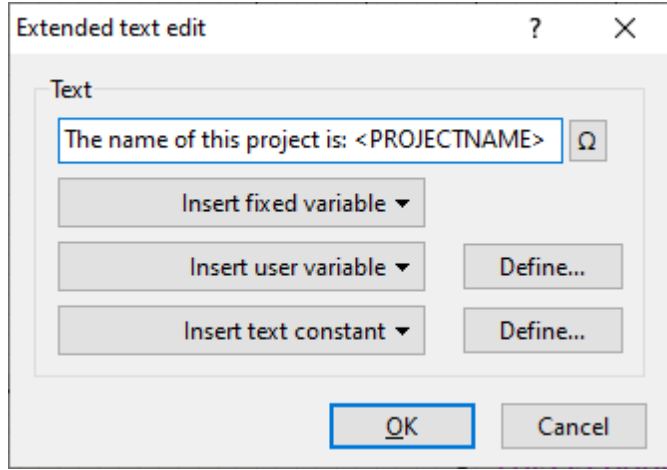
Name	Text
PROJECTNAME	Measuring amplifier project 45X2-9

New entry

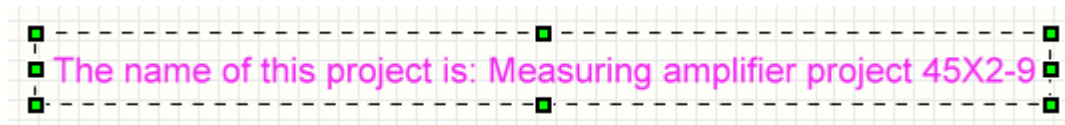
Delete entry

OK

Cancel



You can reach the Extended text edit dialog simply by double-clicking on the text.



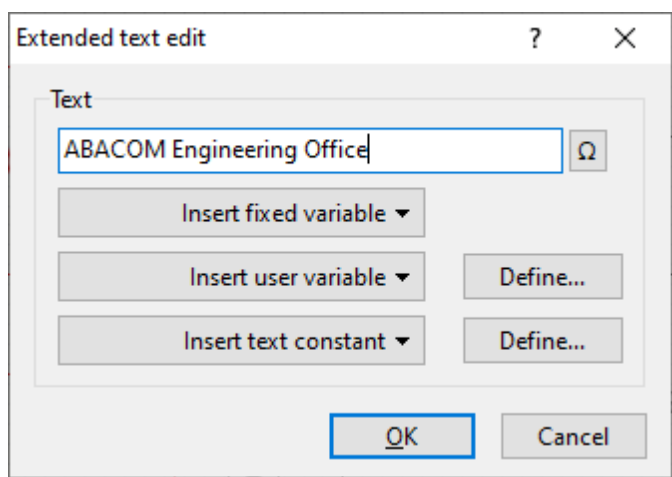
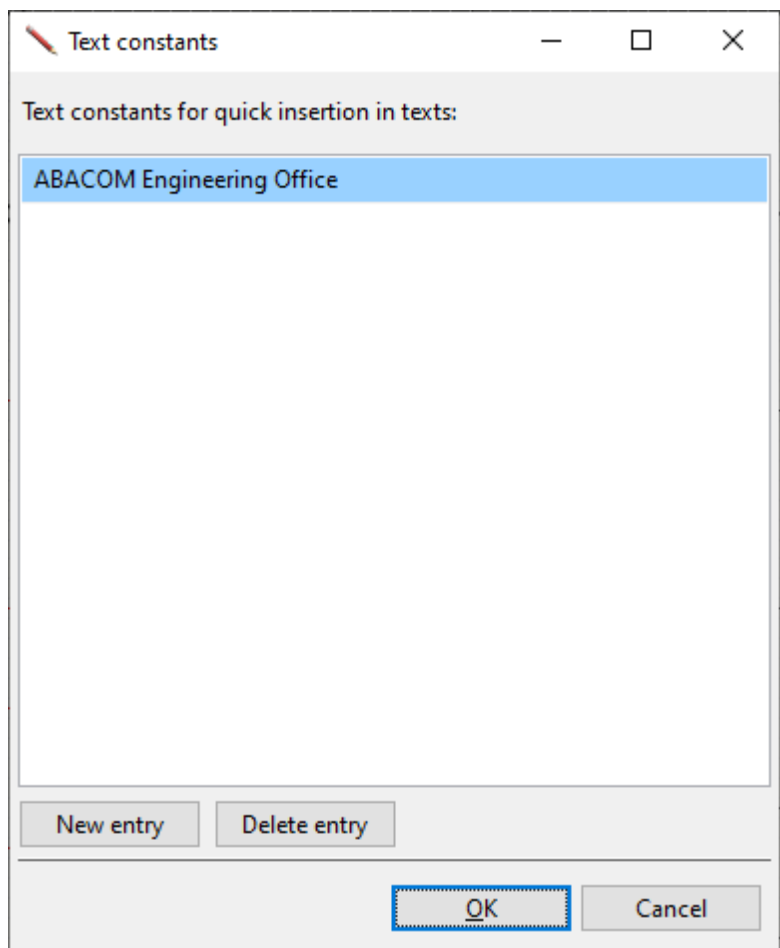
Note that all user variables are always project-related. They are defined and saved within a project (i.e. a file). They are therefore only ever valid within this file.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

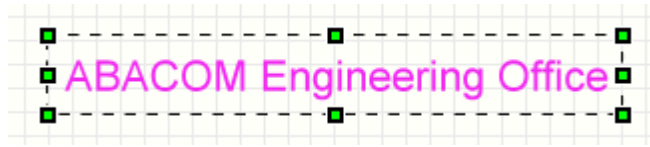
Text Constants

[Variables / Text Constants](#) >>
⏮ ⏪ ⏩ ⏭

Text constants are simply stored texts that are available on demand at any time. If your drawings often contain the same or similar texts, it makes sense to save these as text constants. Every time you have to re-enter this text, you can then access these text constants and save yourself typing.



You can reach the Extended text edit dialog simply by double-clicking on the text.



Copyright © ABACOM GbR, Germany. All Rights Reserved.

Miscellaneous Functions



Copyright © ABACOM GbR, Germany. All Rights Reserved.

Active Text-Links

[Miscellaneous Functions](#) >>


In sPlan you can convert normal texts into active links.

There are 2 types of links: external and internal links:

External links:

You can use this to create a link to a website (e.g. "www.abacom-online.de") or refer directly to a specific file (e.g. a PDF file with a data page).

Internal links:

With sPlan you can create active links from one point in the drawing to another point. It does not matter whether the link is on the same page or a completely different page within your project. You can automatically follow the links with simple clicks of the mouse. You can therefore define jumps to completely different parts of your project and thus create interactive drawings.

The link is created in the properties of a text:

Text-Link - [1]

Target

on page	Text
2: Power adapter	Out 12

Delete target

Select new target...

Publish as target

External link:

File...

☐ Reverse text direction

Internal Link:

With sPlan you can create active links from one point in the drawing to another point. It does not matter whether the link is on

the same page or a completely different page within your project. You can automatically follow the links with simple clicks of the mouse. Here you can define jumps to completely different parts of your project and thus create interactive drawings.

A click on the "Select target ..." button shows a list of all possible jump targets for your project. Select the desired jump destination from this list. If you have not yet approved any other text as a jump target, this list is initially empty.

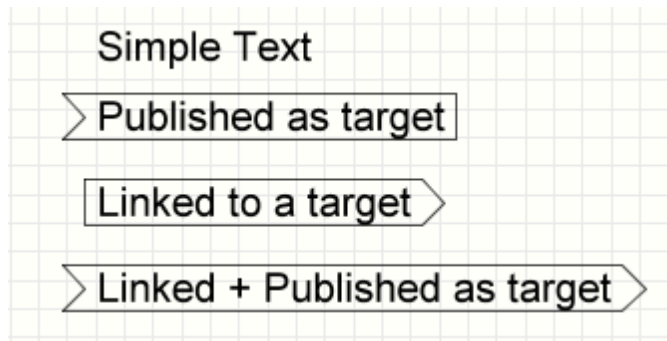
With Delete target you can remove a defined jump target.

With Select new target... you can select a new jump destination, even if another jump destination has already been defined.

With the "Publish as target" option, you determine whether the text should be available as a jump target for other links.

If you want to remove the target option from a text, click again on the "No longer publish as target" button.

Text elements with active links are automatically marked:



All possible jump targets receive an arrow pointing inwards on the left side.

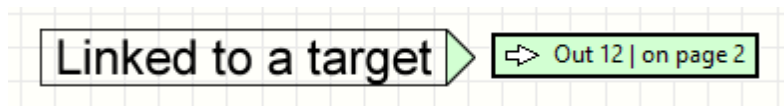
All links with a target are given an arrow pointing outwards on the right-hand side.

This identification makes it possible to immediately identify which text elements are linked.

Follow a link

If you point to an active link with the mouse, the target of this link is displayed.

You can then execute the link with a double click.



If you move the mouse over the right outer arrow, a small window will appear with the link target.

With a double click on the right outer arrow you get to the linked text.



If you move the mouse over the left inner arrow, a small window will appear with the text linked to this text.

With a double click on the left inner arrow you get to the link that linked to this text (source).

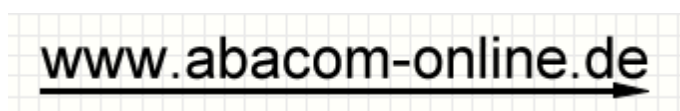
Create a Link-List

In sPlan you can create a link-list. This list contains all the links together with their names and pages. To create the link-list, select the Create link list... entry from the Functions menu.

External Link:

Here you can enter an external link to a website (e.g. "www.abacom-online.de") or refer directly to a specific file (e.g. a PDF file with a data page).

Your text will then appear underlined on the plan, e.g. like this:



If you double-click on the underscore with the mouse, you execute this external link after a security query and you will either be taken to the specified website or the corresponding file will be opened.

Note that the text that appears on the map is not necessarily the same as the text on the external link.

Usually, however, it makes sense to compare the displayed text with the link.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Convert Line <> Polygon

[Miscellaneous Functions](#) >>



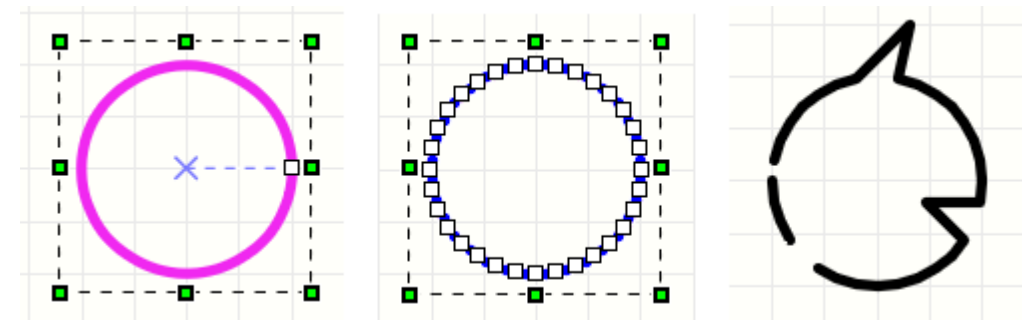
In sPlan you can convert a line into a polygon or convert a polygon into a line.

A rectangle, a circle or a bezier curve can also be converted into a polygon or a line.

That sounds a bit banal at first, but it can be quite helpful in some situations and offers interesting opportunities.

To do this, simply click on the desired element with the right mouse button and select the Convert to Polyline or Convert to Polygon command.

Here are some application examples:



Circle was drawn

Circle was converted to line

(click with the right mouse button on the circle and select the command Convert to Polyline)

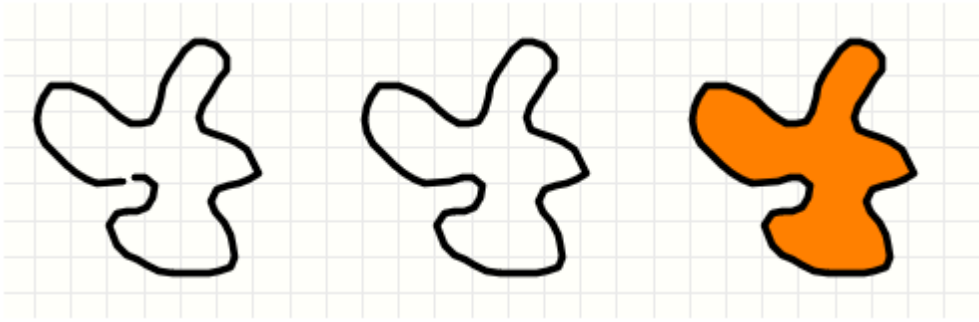
Some nodes were moved

The line was cut in some places

(click with the right mouse button on the node and select the command Split line)

Some nodes have been removed

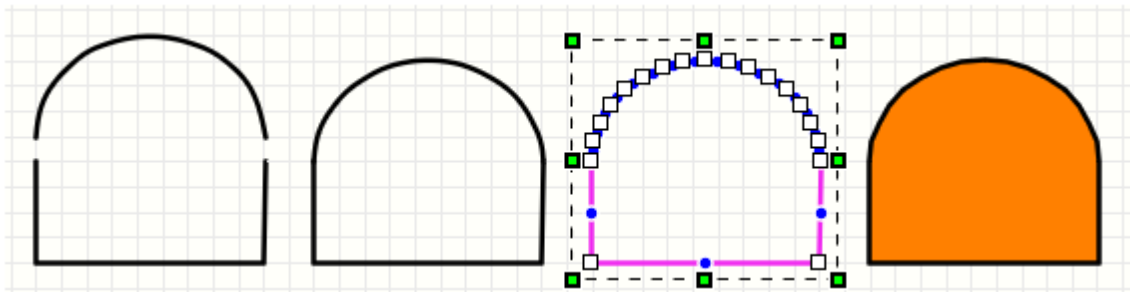
(right-click on the node and select the command Delete node)



Freehand line was drawn

Freehand line converted to a polygon
(right-click on the freehand line and select the Convert to Polygon command)

A fill was assigned to the polygon



A geometric figure was drawn from a line and a semicircle

The two elements were moved together

The semicircle was converted to a line
(click with the right mouse button on the semicircle and select the command Convert to Polyline)

The two lines were joined so that they become one line
(click with the right mouse button on a node where both lines touch and select the command Join lines)

The line was converted to a polygon
(click with the right mouse button on the line and select the command Convert to Polygon)

A fill was assigned to the polygon

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Printing

[Miscellaneous Functions >>](#)

Of course, you also have the option of putting your drawing on paper. The software has a very intuitive and clear print preview. In the print preview you can determine the printer, set the printer, freely select the scaling, position the printout freely on the paper, etc. All settings that you make in the print preview are immediately visible in the preview, so that you do not have any experience unpleasant surprises.

Use the "Print" command in the "File" menu or the corresponding toolbar button to access the print preview. You can also enlarge the window as required to improve the accuracy of the preview.

Print preview

HP Universal Printing PCL 6

Setup...

Pages clipped0

Close

Scale

1:1

User

100%

Offset

X: -12 mm

Y: -12,4 mm

Reset

Banner

X: 1

Y: 1

Overlap [mm]: 0

Functions

Orientation

Automatically

Portrait

Landscape

Printing

Current page

All pages

Selection

Print [1]

no name of this project is: Measuring amplifier project 4562 E

ABACOM Engineering Office

This is page 1 of 3

Sample Desc.

SPUR 100 02 20 V 3

U1 T1 02 20 V 3

SPUR 100 02 20 V 3

Table

Table with 5 columns: No, Keterangan, Unit, Qty, and Remarks. Row 1: Dimmer, 1, 1, 1, 1. Row 2: 1, 1, 1, 1, 1.

At the top of the print preview you can select the printer you want and with Setup... you can call up the setup dialog for the selected printer.

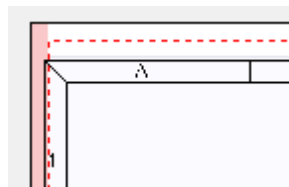
Below the print preview there is a scroll bar in which you can select the page to be printed with one click.

Paper preview and the red frame

The paper is shown as a white page. The red frame indicates the printable area on the paper. The size of this frame depends on your printer or its printer driver.

You can move the drawing freely on the paper. To do this, click with the mouse on the white surface of the paper and, with the mouse button still held down, drag the drawing to the desired position. The displacement of the drawing is displayed as an offset in the left bar. With a click on Reset, this offset can be set directly to 0 again, thus undoing the shift.

As soon as part of the drawing is outside the printable area, the corresponding frame side is colored red:



In addition, the Print button changes from green to red to show that the printout would be cut off with the current settings.

In the lower page bar, all pages that are cut off are automatically marked with a [!].

Scale

Here you can decide whether you want the expression to be exactly "1:1" or whether you prefer to scale the expression freely. If you have selected "User", you can freely set the scaling with the slider. You can follow the scaling online in the print preview.

Offset

You can move the drawing freely on the paper. To do this, click with the mouse on the white surface of the paper and, with the mouse button still held down, drag the drawing to the desired position. The displacement of the drawing is displayed as an offset in the left bar. With a click on Reset, this offset can be set directly to 0 again, thus undoing the shift.

Banner

Here you have the option of printing a drawing on several pages.

Enter how many pages you want to use in the X or Y direction. In the Overlap field, you can provide an overlap of the individual pages.

mm. You can now scale and move the printout so that it fits on all pages.

Functions

Adjust (size and position)

Scales and centers the drawing so that it fits on the page as best as possible

Center

Centers the drawing on the page

All pages -> ...

Automatically applies the function to all existing pages

Apply to all pages -> ...

All other pages take on the corresponding value of the current page

All pages -> Reset print settings

Resets all print settings for all pages

Orientation

With Automatically, the page orientation is always set as it was defined in sPlan (portrait or landscape).

You can also force the page orientation into the appropriate format with portrait or landscape format.

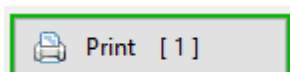
Printing

With the option Current page only the page currently displayed is printed out.

With the option All pages all pages of your project are printed out.

With the option Selection you can determine yourself which pages should be printed out. To do this, enter the page numbers in the field separated by commas, or click on "..." to select the individual pages via a selection dialog.

With the button Print you start the printout:



The number in square brackets indicates the number of pages to be printed.



If the button is red, this indicates that at least one page will be cut off during the printout.

See the section Paper preview and the red frame above.

Close

This closes the print preview and returns you to your drawing.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Export

[Miscellaneous Functions](#) >>



You can export your drawing in various file formats. This allows you to integrate your drawings into other programs or publish them on the network. You can choose between 5 different file formats:

JPG

PNG

BMP

EMF

SVG

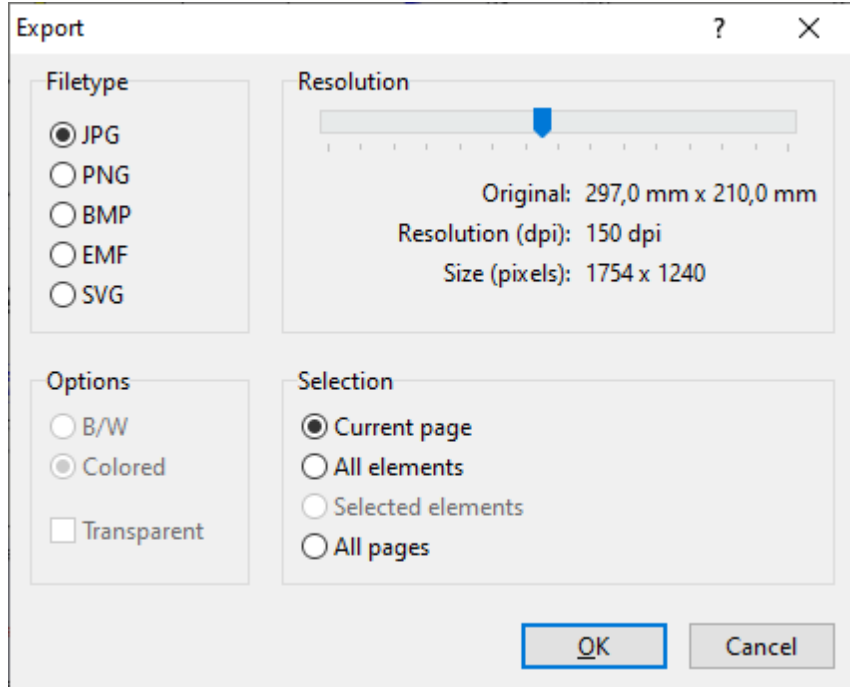
JPG, PNG and BMP files are pixel format files. The conversion of a vector graphic (sPlan itself works with vector graphics) into a pixel format is always lossy. With the setting of a high resolution, however, you can minimize these losses here. However, this is then at the expense of the file size.

As a rule, the JPG format is preferable to the BMP format, as it creates much smaller files with the same resolution.

EMF files are vector format files. Since sPlan also works in vector format, a loss-free export is theoretically possible. The EMF format is a Windows-specific vector format that can be read by many other graphics programs. Unfortunately, there are often compatibility problems here, so that the result is not always satisfactory. We recommend avoiding this format. It is actually only available in sPlan for reasons of compatibility.

The SVG format is also a vector format. SVG stands for Scaleable Vector Graphics. SVG files can now be used directly by many browsers (Firefox, etc.) and other programs, and should definitely be preferred to the EMF format.

To export the drawing, select the "Export..." function from the "File" menu. The following dialog then appears:



Filetype

Here you enter the desired file type of the target file.

Resolution (only for JPG and BMP)

When exporting to a JPG or BMP file, it is necessary to specify the resolution of the pixel graphic to be created.

Use the slider to set the desired resolution. Please note that the file size of the target file increases with increasing resolution.

Color (only for BMP)

This information is only necessary for the BMP export. The other file formats are generally colored.

If possible, please select the "S / W" option here. Only if you absolutely need a colored drawing, please select the "Color" option. The reason for this is simple: a colored bitmap requires approximately 24 times as much memory as a monochrome bitmap. The same bitmap occupies, for example, 500 Kbytes of memory in b / w, as a colored bitmap, a full 12 Mbytes of memory. Even if you have drawn the drawing in color, you can create a b / w bitmap if it is enough for you. The individual colors are reproduced using a grid.

You can see the required memory value for the bitmap online in the window.

Transparent (only for PNG)

Creates a graphic with a transparent background

Selection

Here you specify what you want to export:

Current page

The entire page is exported in full size with all elements.

All elements

Everything is also exported, the page size is determined by a comprehensive rectangle over all elements.

Selected elements

Only the selected elements are exported.

All pages

All pages of the project will be exported. Each page is exported to a separate file.

The files are each appended with the page number, so several files are automatically created, such as:

- filename_1.jpg
- filename_2.jpg
- filename_3.jpg
- filename_4.jpg

After you have made and confirmed the desired settings, a "Save As" dialog appears in which you can specify the target folder and the file name. The file is then generated and saved.

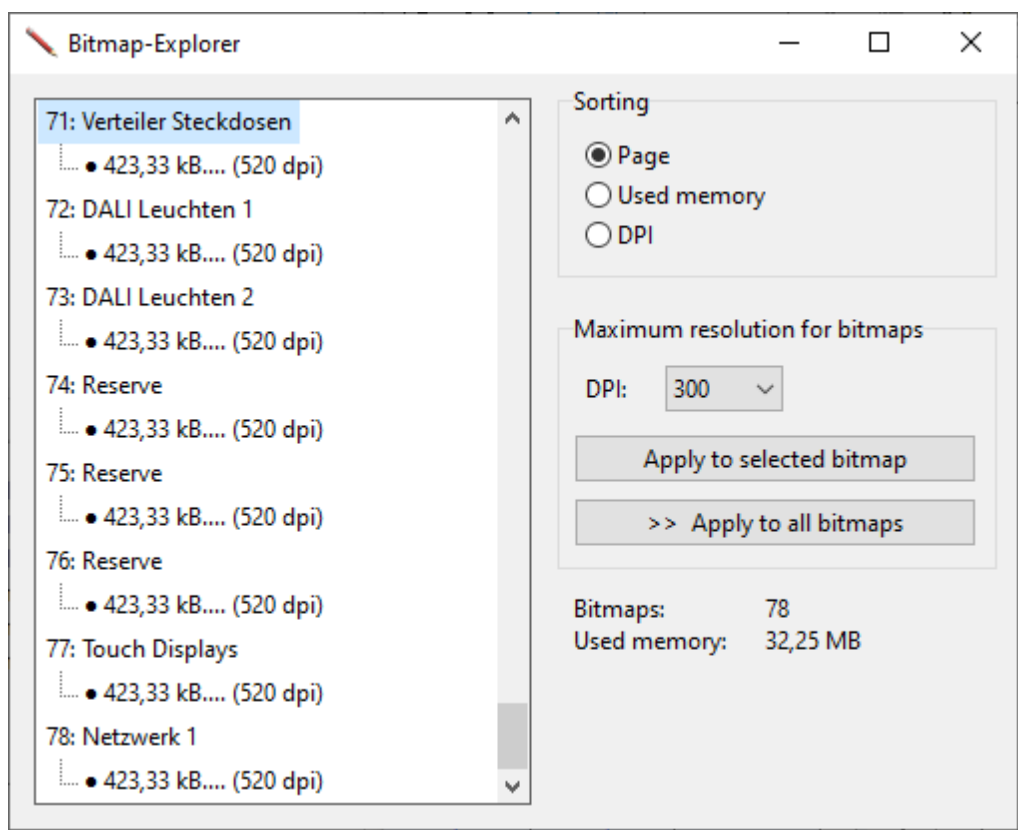
Copyright © ABACOM GbR, Germany. All Rights Reserved.

Bitmap-Explorer

[Miscellaneous Functions](#) >>


All bitmaps used in your project are listed in the Bitmap-Explorer. In this way you can quickly get an overview of all bitmaps, their resolution and their memory requirements.

You can open the Bitmap-Explorer at any time with the Bitmap-Explorer... command from the Functions menu. If you have selected a bitmap, you can also open the Bitmap-Explorer directly in the properties panel on the right-hand edge of the window:



Here you can directly specify a maximum value for the resolution (DPI) of the bitmaps and automatically adjust all bitmaps with a higher resolution to this maximum value. Depending on the bitmaps used, this can save a lot of memory and thus make your sPlan project more handy and faster.

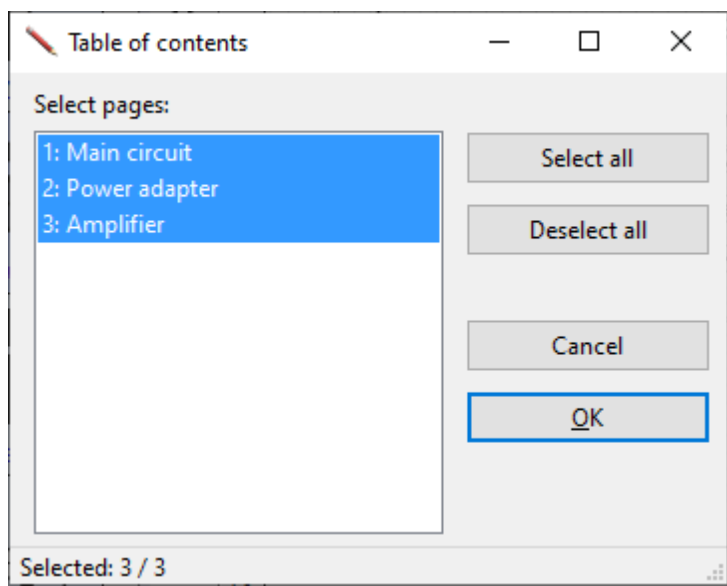
Paste Table of Contents

[Miscellaneous Functions](#) >>


If desired, a table of contents can be inserted directly into your drawing as a kind of table.

The table of contents is like a list with all the pages in your project.

To create a table of contents, select the Paste Table of Contents... command from the Functions menu. The following window then appears:



Here you can select all pages that should be included in the table of contents.

The following dialog then appears:

Table of contents

Font

Courier New

16 [1/10 mm]

Options

☒ Frame ☒ Shadow

☒ Vertical lines

☐ Alternating lines

Height of the table (3 Rows)

☒ Maximum (adjusted to page height) (= max. 180 Rows)

☐ Break manuell after 48 Rows

No break

Paste Cancel

Here you can still make certain settings.

After clicking on Paste you can then place the table of contents on your drawing:

Page	Name
1	Main circuit
2	Power adapter
3	Amplifier

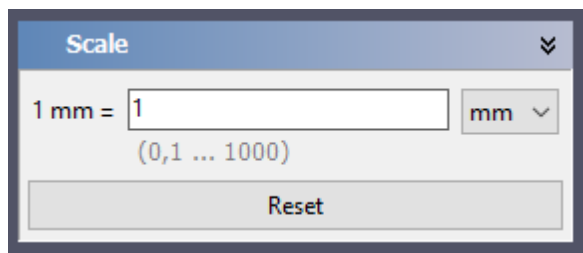
Scale

[Miscellaneous Functions](#) >>


You can give each page in sPlan its own scale.

sPlan then automatically takes this scale into account for all coordinates, rulers, dimensions, etc.

The properties of a page are displayed in the properties panel on the right edge of the window if no other element is currently selected on the plan. You can also find the Scale section here:



Search / Replace

[Miscellaneous Functions](#) >>



sPlan offers a special search function to search for components or texts and, if necessary, to replace them automatically. Particularly in the case of somewhat larger projects with a large number of pages, it can be difficult to identify a specific component or text on the drawing.

To activate the component search, select the Search/Replace... command from the Edit menu or click directly on the corresponding button (right-aligned in the toolbar):



The search panel is then displayed on the right-hand side of the window.

You can hide the search panel again at any time with another click.

At the top you can choose whether you want to search for components or for texts:

Search

Components

Text

Identifier:

R

Replace:

All pages

Value:

Replace:

All pages

☐ Only component list elements
 ☐ Case sensitive
 ☒ Resolve <Variables>
 ☒ All pages

Update

All pages

1: Main circuit

R1 / 1M

R2 / 100k

R3 / 560k

R4 / 1M

R5 / 47k

R6 / 1M

R7 / 47k

R8 / 10k

R9 / 22k

R10 / 22k

R11 / 22k

SR2 / HT1036

Search

Components

Text

Search:

R

Replace:

All pages

☐ Case sensitive
 ☒ Resolve <Variables>
 ☒ All pages

Update

All pages

1: Main circuit

pr.:

Dimmer

Published as target

Linked to a target

Linked + Published as target

The name of this project is: Mea

ABACOM Engineering Office

2: Power adapter

3: Amplifier

Search only

Enter your search text for the component search in the Identifier and/or Value fields, or for the text search in the Search text field.

The search results are automatically sorted by pages and displayed in a tree structure.

If you click on a search result, the element is automatically displayed on the drawing. If the element is on a different page, the page is also switched automatically. So you can easily control and examine all search results.

Search and Reaplace

First, proceed exactly as described for Search only above so that the search results are displayed.

Replace a single element

If you now select a result you can use the button Element -> to replace the text in the corresponding element with the text specified in the Replace field. This only replaces the text in the one selected element.

Replace all elements of a page

You can also replace all elements of a page at once. To do this, select the superordinate page in the search result instead of a single element. The Element -> button now becomes a Full page -> button. With this you can now replace all elements of this page at once.

Replace all elements on all pages

You can also replace all elements of all pages at once. To do this, select the top entry All pages in the search result instead of an individual element or a page. The Element -> button now becomes an All pages -> button. With this you can now replace all elements on all pages (i.e. the complete result list) at once.

Search options

When searching, you still have a few options that influence the search result:

Only component list elements

This option is only available when searching for a component. If you activate this field, only those components will be taken into account that have activated the "Add to component list" field. This is likely the case for most components anyway.

Case sensitive

The search is case-sensitive or not.

Resolve <Variables>

If there are variables in the text, either the original text with the variable names ("This is page <PAGENO>") or the text with the resolved variables ("This is page 1") is used for the search. For example, if you want to search for variable names directly, you would have to deactivate this option.

All Pages

If this option is activated, the search automatically extends over all pages in the project. Otherwise, the search is only ever made

on the current page.

The "Update" button

This allows you to restart the search at any time and update the results.

The list of results is automatically updated with every change in the search panel. However, if a search has been made and elements are then deleted or changed on the plan, a manual update may be necessary.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Keyboard Control

[Miscellaneous Functions](#) >>


You can also use the keyboard to access many of the sPlan functions if required.

Functions that are often required such as copy, paste, duplicate, delete, etc. can be accessed using special key combinations. These key combinations are displayed next to the menu entry in the sPlan main menu.

Other functions can be controlled via the keyboard:

Drawing modes

There is a hotkey for each drawing mode in the vertical toolbar. So you can simply select the corresponding drawing mode with the keyboard instead of the mouse

The following hotkeys are preset as standard:

Edit	= ESC
Line	= L
Connection	= P
Rectangle	= R
Circle	= C
Polygon	= O
Bezier	= U
Freehand line	= H
Special forms	= S
Dimensions	= D
Text	= T
TextRect	= E
Bitmap	= F
Zoom	= Z
Measure	= M

In the [General Settings](#) you can define the hotkeys yourself if necessary.

Move elements with the arrow keys

Selected objects on the drawing can also be moved with the arrow keys on the keyboard. The elements are shifted by one grid point in the desired direction. If you also press the CTRL key, you can switch the grid to fine (0.1 mm) and fine-tune it using the keyboard.

Select elements with the TAB key

With the TAB key you can select the individual elements of the drawing one after the other. Each press of the TAB key selects the next element. If you also press the SHIFT key, the previous element is selected instead of the next element.

Call the Component-Editor with Alt-Return

When a component is selected, you can call the [Component-Editor](#) with Alt-Return.

Zoom using the keyboard

F5 zooms in on the whole page

F6 zooms in on all elements of the drawing

F7 zooms in on all selected elements of the drawing

+ zooms in

- zooms out

Follow a text link with F8

With the function key F8 you can follow an [active text-link](#). To do this, the text element with the link must be selected.

Translation into other languages

[Miscellaneous Functions](#) >>


Most of the texts used in sPlan are saved in a file called "Translations.txt".

This file is located in the folder "C:\ProgramData\sPlan80\".

The name of this folder may vary depending on the version of Windows.

This file contains all texts in the following format:

```
1100;Bearbeiten;Edit;Editer
1101;Rückgängig;Undo;Retour en arrière
1102;Wiederherstellen;Redo;Retour en avant
...
```

Each line defines exactly one text

Each line begins with an identification number that sPlan uses to identify the text

This is followed by 3 texts, separated with ";"

The 1st text is used for the language selection GERMAN

The 2nd text is used for the language selection ENGLISH

The 3rd text is used for the language selection FRENCH

If you want to enter a new text, you can simply add the desired text as the 4th text:

```
1100;Bearbeiten;Edit;Editer;My own text
```

Note that your own text must be separated with a ";" from the last text

As soon as a fourth text is available, sPlan will always use it, regardless of the selected language.

In this way, you could completely translate the sPlan into a new language.

Please note that the changes in this file only take effect after restarting sPlan.

You can also simply delete the "Translations.txt" file.

The next time sPlan is started, this file is automatically created again with the 3 standard languages (German, English, French).

Only the CHANGES you want are saved in this file.

So you can simply delete this file at any time in order to reactivate the default sPlan translation.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Software-Registration

[Miscellaneous Functions](#) >>



This option has nothing to do with the actual function of the software. You can use it to register your software directly with ABACOM, the manufacturer of this software. We will then automatically notify you of updates and new versions.

You can register in the main menu under Help -> Software registration...

We recommend registering by email.

Note:

If you obtained the software directly from ABACOM, this separate registration is not necessary.

Copyright © ABACOM GbR, Germany. All Rights Reserved.

Tips and Tricks



With <ESC> or with the right mouse button you can automatically return to the standard mode from any mode.

With the CTRL key you can deactivate the active grid snap (or the angle snap when rotating) at any time. In this way you can also reach points outside the grid without completely deactivating the grid snap.

With the SHIFT key you can deactivate the set angle snap at any time when drawing a line.

If you click on a selected element again, the surrounding [sizers](#) change and you can now rotate and shear the element with the mouse.

When you have selected a line or a polygon, you can move individual line segments within the line or the polygon by holding down the <ALT> key at the same time.

You can also position the texts of a component such as identifier, value or contacts directly on the drawing. Simply click on the text element (the component must not be selected beforehand) and move it to the desired position. You do not need to call up the component editor for this. This function can be deactivated in the [General Settings](#) if necessary.

With a double click on an empty space you can move the visible image section with the mouse. Simply double-click on an empty space, hold the mouse down and move the image section with the mouse.

You can easily edit the arrangement of the components within the library using drag & drop. Click on a component in the library and simply drag it to the new location within the library.

You can easily change the width of the component library with the mouse.

If you want to add [forms](#) to your drawings, sPlan offers special functions for this.

If you would like to present your drawings on the Internet, first export the drawings in a high resolution, e.g. in JPG format. Alternatively, you can offer your plans as *.SPL files for download and refer to the license-free [viewer](#).

The "sPlan-Viewer"



The sPlan-Viewer is a free and independent program with which sPlan files can be opened, viewed and printed out. This gives every user access to sPlan files.

The sPlan-viewer is freeware. You are free to pass on the viewer and thus also give other users the opportunity to view or print out your drawings without sPlan having to be installed on the relevant computer.

The sPlan-Viewer is freely available here on our homepage:

<https://www.electronic-software-shop.com/lng/en/support/free-viewer-software/>

Copyright © ABACOM GbR, Germany. All Rights Reserved.