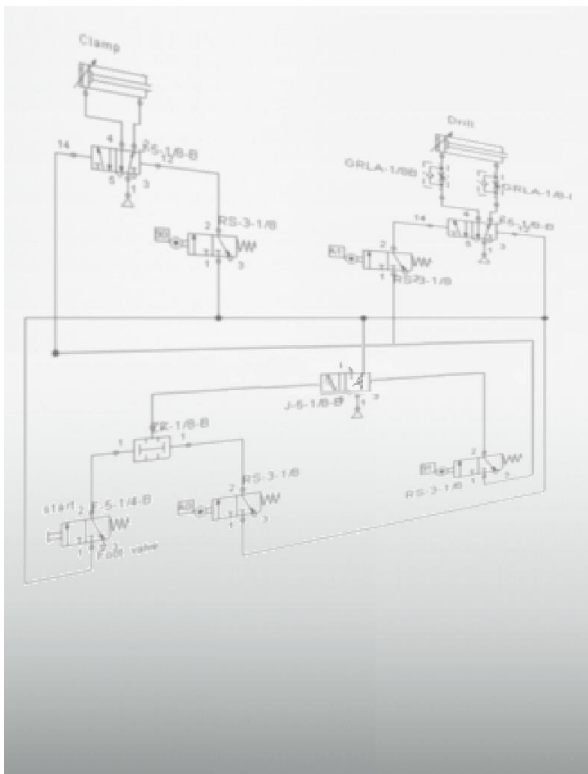




FluidDraw® 5

User's Guide



FluidDraw is an application for creating pneumatic circuit diagrams. Not only the standard circuit symbols are available but also all of the components contained in the Festo product catalog along with their part numbers and technical details. If you have also installed the Festo product catalog then the current catalog data is always available in FluidDraw. In addition to its own proprietary file format, FluidDraw also supports DXF files and circuit diagrams created with older FluidDraw versions and the FluidSIM pneumatic simulator.

Contents

1.	First steps	7
1.1	Create a new circuit diagram	7
1.2	Organize symbols, libraries and circuit diagrams	10
1.3	Insert symbol from menu	10
1.4	Symbol libraries	13
1.4.1	Create user library	14
1.5	Circuit files	15
2.	Library and project window	16
2.1	Change window position	16
2.2	Automatic show and hide	16
3.	Edit circuit	18
3.1	Insert and arrange symbols	18
3.2	Using the Festo product catalog with FluidDraw	18
3.3	Connect connectors / ports	22
3.4	Insert T-connector	23
3.5	Lay lines	24
3.6	Define the characteristics of lines	25
3.7	Delete line	26
3.8	Set the properties of the connectors	27
3.9	Define component connector	28
3.10	Delete component connector	28
3.11	Configure a way valve	28
3.12	Configure cylinder	30
3.13	Group symbols	32
3.14	Create macro objects	32
3.15	Delete symbol groups and macro objects	33
3.16	Align symbols	33
3.17	Mirror symbols	33
3.18	Rotate symbols	34
3.19	Scale symbols	35

4.	Drawing frame	37
4.1	Editable labels	37
4.2	Use a drawing frame	38
5.	Additional tools for creating drawings	43
5.1	Drawing tools	43
5.1.1	Grid	43
5.1.2	Alignment lines	43
5.2	Drawing levels	44
5.3	Cross-references	46
5.4	Drawing functions and graphical elements	47
5.4.1	Pneumatic line	48
5.4.2	Line	48
5.4.3	Rectangle	50
5.4.4	Circle	51
5.4.5	Ellipse	52
5.4.6	Text	54
5.4.7	Image	54
5.5	Check drawing	56
6.	Component attributes	57
6.1	Component attributes in the Properties dialog window	57
6.2	Attributes of the text components	62
6.3	Link text components with attributes	64
6.4	Text components with specified links	66
6.5	Change the properties of multiple objects simultaneously	67
7.	Parts list management	69
7.1	Display parts list	69
7.2	Find components from the parts on the circuit diagram	69
7.3	Set parts list properties	71
7.4	Export parts list	74

8.	Manage projects	76
8.1	Create new project	76
8.2	Project nodes	76
8.2.1	Project attributes	77
8.2.2	Project archiving	78
8.3	Circuit and parts list nodes	80
8.4	Attributes of the nodes of a project	82
9.	Circuit input and output	84
9.1	Print circuit diagram and parts list	84
9.2	Import DXF file	86
9.3	Export circuit	86
10.	Options	88
10.1	General	88
10.2	Save	90
10.3	Folder Locations	91
10.4	Language	92
11.	Menu overview	93
11.1	File	93
11.2	Edit	94
11.3	View	97
11.4	Circuit	99
11.5	Library	99
11.6	Draw	100
11.7	Insert	101
11.8	Project	102
11.9	Tools	103
11.10	Window	104
11.11	Help	105

12.	Functional diagram	106
12.1	Edit Mode	107
12.1.1	Set diagram properties	107
12.1.2	Table textboxes	108
12.1.3	Adjust the presentation of the diagrams	110
12.2	Draw diagram curve	112
12.3	Insert signal elements	112
12.4	Insert text boxes	113
12.5	Draw signal lines and insert signal connections	115
12.5.1	Free-draw signal lines	115
12.5.2	Drawing signal lines from signals	117
12.5.3	Draw signal lines from diagram support points	117
12.6	Insert additional signal lines	118
12.7	Insert row	118
12.8	Delete row	118
12.9	Additional editing functions	119
12.9.1	Zoom	119
12.9.2	Undo editing steps	119
	Index	120

First steps

Chapter 1

1.1 Create a new circuit diagram

.....> Select the **New** entry from the **File** menu.

An empty circuit diagram window opens in which you can insert symbols and connect them with lines. You should set the diagram size in advance.

.....> Select the **Paper Size...** entry from the **Circuit** menu.

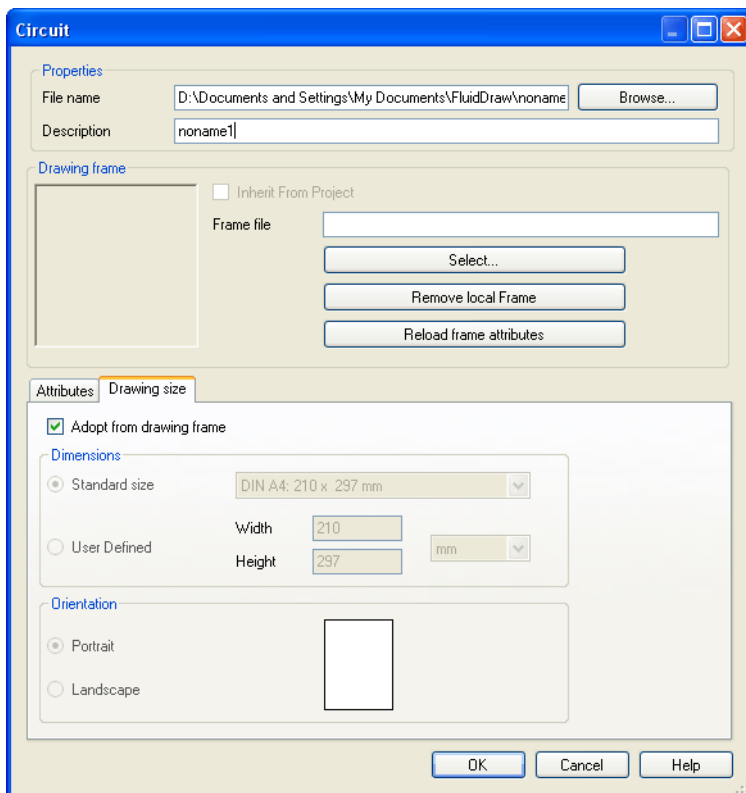


Figure 1/1: Circuit dialog window, Drawing Size tab: Setting the diagram size

If you utilize a drawing frame then FluidDraw can automatically adjust the paper size. If you wish to manually define the paper size then deactivate the “Adopt from drawing frame” option and select the drawing’s desired dimensions and orientation. If the diagram’s dimensions are larger than your printer’s printing area then you can distribute your drawing across several pages ([tiles](#)). To retain a better overview you can assign attributes to every circuit diagram.

→ Click the “Attributes” tab.

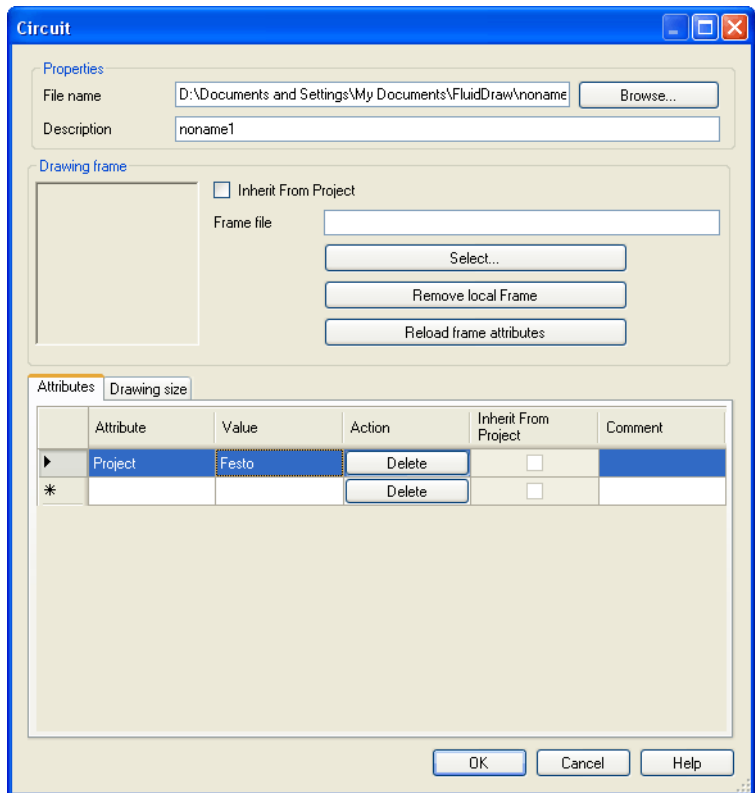


Figure 1/2: Circuit dialog window, Attributes tab: Creating attributes

The attribute table allows you to save any desired data in the form of attribute value pairs. The associated placeholders (attributes with the same name) in the drawing frame are replaced by the values entered.

You can also access this dialog window directly by using the **Circuit** menu and the **Properties...** menu entry.

1.2 Organize symbols, libraries and circuit diagrams

In order to support the organization of the different document types in FluidDraw all of the circuit files are assigned to one of three groups.

Symbols

Symbols are formal, abstract models providing a graphical representation of the function of a component or component group. This can consist of simple circuit symbols or entire circuits. Symbols can be inserted into their own circuits and linked between the connecting points. The insert can be performed either via the **Insert** menu or via drag and drop from a **library window**. Symbols can be compiled into library files with the **lib** file suffix.

Libraries

Libraries are hierarchically organized collections of symbols. In addition to the standard library, which users cannot modify, users can compile any desired number of their own libraries. You can find functions for organizing the libraries on the **Library** menu as well as on the context menu of the currently active library. To switch libraries use the tab in the upper margin of the library window. Library files have the **lib** file suffix.

Circuits

The default location for circuits is in the **FluidDraw** folder contained in the My Documents folder defined by the operating system. These have the **circ** file suffix. Note: It is worthwhile to create a new sub-folder in the **FluidDraw** directory for every customer or for every folder **project**.

1.3 Insert symbol from menu

To find a specific symbol you can either enter characteristic key words in the Find Symbol dialog windows or navigate through the hierarchical structure.

→ Where necessary open a new window and select the **Find** **Symbol Description...** entry on the **Insert** menu.

The Find Symbol dialog box then opens. You can enter your search terms in the “Search” input line. Separate the individual search terms using commas or spaces. Neither the order of the search terms nor upper or lower case letters have any influence on the search results.

→ For example, enter **pressure, valve**.

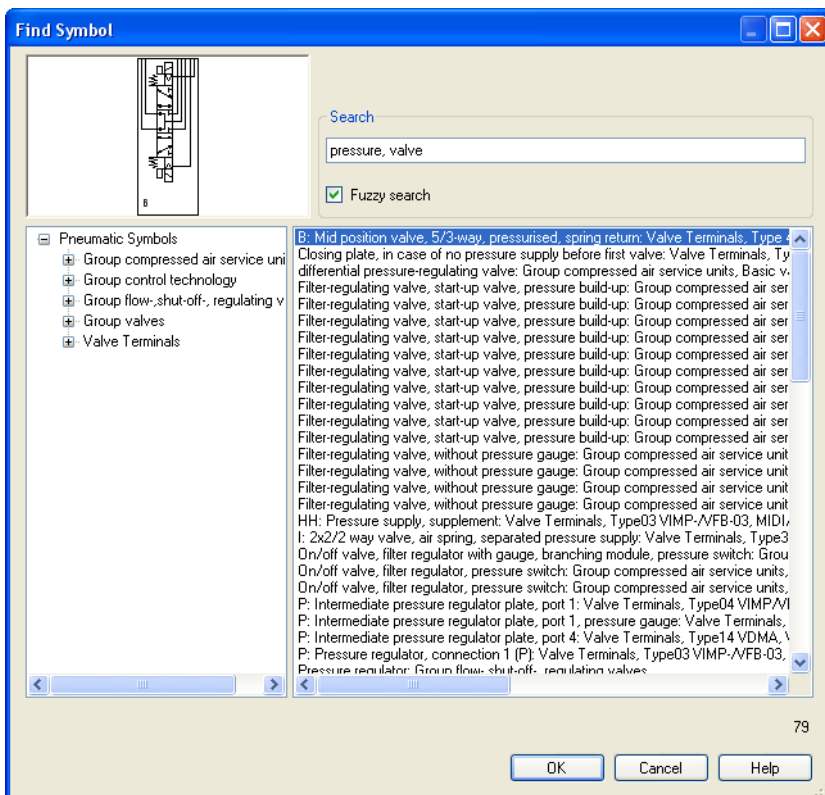



Figure 1/3: Find Symbol dialog window

You can see the symbols found in the two results lists. The library hierarchy is displayed on the left-hand side although only those branches appear which contain matching symbols. An alphabetical list of the search hits is displayed on the right-hand side. The symbol of the highlighted entry is shown in the preview. Once you have found the symbol that you were looking for, you can select it using the **OK** button or by double-clicking on the appropriate row in the results list. The symbol then “hangs” on the mouse pointer and is placed by left-clicking on the drawing area.

With the Fuzzy Search option you can activate a tolerance in order to achieve results despite minor typing errors or spelling variants.

In addition to the option of inserting a specific symbol using the Find Symbol dialog window you can also insert a real component including the part number and technical details from the Festo product catalog. You can find details regarding the use of the Festo product catalog with FluidDraw under [Using the Festo product catalog with FluidDraw](#).

1.4 Symbol libraries

FluidDraw is capable of managing multiple libraries, each of which is displayed in a tab in the library window. Libraries which cannot be modified in FluidDraw are labeled with a lock symbol  in the tab. This applies to standard library as well as to symbol folders which FluidDraw does not manage itself or for which the registered user does not possess write authorization.

Every library is displayed hierarchically. Every hierarchy level can be shown or hidden by clicking on the group name. Right-clicking in a library opens a context menu offering the following menu items for editing the library:

View

Defines the size of the symbols displayed. **Small**, **Normal** and **Large** are available.

Expand All

Opens all of the hierarchy levels.

Collapse All

Closes all of the hierarchy levels.

There are three types of library:

The standard library

This library is delivered with FluidDraw and cannot be modified.

Symbol folders

Circuit and symbol files saved on the disk can be used as libraries in FluidDraw. The files from the selected folder are added as a library via the **Library** and the **Add Existing Symbol Folder...**

menu item. The library hierarchy corresponds exactly to the folder hierarchy. These libraries cannot be modified in FluidDraw. The files must be modified directly.


User libraries

You can create and subsequently modify new libraries via the **Library** menu and the `MenuItemAddNewLibrary` menu item (see section [Create user library](#)). Using drag and drop you can move the symbols and groups within the library as desired.

1.4.1 Create user library

In order to be able to access frequently used symbols (or circuits) more quickly multiple symbols can be compiled into libraries. Libraries are saved as files with the `.lib` file suffix. You can create new libraries using the **Library** and the **Add New Library...** menu item. Right-clicking in the new library opens a context menu with which you can modify the library.

The following options are available:

Copy	Copies the selected symbols to the clipboard
Paste	Inserts the symbols into the library from the clipboard. These symbols can also be partial circuits.
Delete	Removes the selected symbols from the library.
Rename...	Edits the text which libraries display below the symbol
Add Existing Symbols...	Opens a dialog window for selecting symbol files to be copied into the library as new symbols.
Copy to Other Library	Copies the selected symbols into another library. The available libraries are listed in a submenu. Only the currently open libraries are displayed here (appearing as tabs in the library window) which are not write-protected (recognizable as no lock symbol  is displayed).

New Sub-Folder...	Creates a new hierarchy level below the active group. The active group is the one belonging to the area below the mouse point and can be recognized by the dark blue coloring.
Remove Sub-Folder	Removes the hierarchy level which the mouse pointer is located in.
Rename Sub-Folder...	Enables you to rename the hierarchy level which the mouse pointer is located in.

1.5 Circuit files

FluidDraw circuit files have the **circ** files suffix and are saved as compressed XML files. An option on the **Tools** menu under the **Options...** menu item enables the deactivation of the compression so that the circuit files can be viewed as plain text. This can be useful for version management software, for example.



However, please note that editing a **circ** file outside FluidDraw can cause the file to become unreadable or corrupted upon re-import.

Library and project window

Chapter 2

2.1 Change window position

The library is fixed on the left-hand side by default and the project window (when a project is open) is on the right-hand side. To separate the window from the anchoring: Move the mouse pointer to the upper margin. Click and hold the left mouse button. Move the window a small distance towards the centre of the screen. Now release the left mouse button. Now you need to move the library window to the lower right and the project window to the lower left. Once a window has been separated from the anchor it can be moved freely.

How to re-anchor the window: Move the mouse pointer to the upper margin. Click and hold the left mouse button. Move the window as far as possible to the right or the left. Now release the left mouse button. The window snaps into place. This enables you to anchor the library window to the right and the project window to the left, for example. You can also anchor both windows on the same side. In this case you can move the desired window into the foreground by clicking on the associated tab.



2.2 Automatic show and hide

The tabs offer an additional, practical function: Automatically showing and hiding the library or project window. Click on the associated vertical tab “Library” or “Project” at the edge of the window. This causes the window to be hidden providing a larger area for the drawing. In order to make the window visible again move the mouse pointer over the tab; the window opens once again. As soon as you have performed your operations in this window and move the mouse pointer to a circuit window the library window and / or the project window are automatically hidden. To

deactivate the function click the associated tab again (this then appears depressed).

Chapter 3

3.1 Insert and arrange symbols

You can insert symbols into the symbol window being edited using the [dialog window Find Symbol](#) and by means of the [libraries](#). You can also transfer objects from every other window by selecting them and dragging them to the desired window. Alternatively you can also utilize the clipboard by selecting the items, choosing the **Copy** menu item on the **Edit** menu, bringing the target window into the foreground and selecting the **Paste** menu item on the **Edit** menu. Using the mouse pointer to drag objects from one window to another copies them. Dragging the objects from one position to another within a window moves them. In order to copy within a window you need to hold down the **Shift** key while moving the mouse pointer. You can recognize operation by the shape of the mouse pointer. During a move-operation a cross with arrows  appears, when copying a plus symbol  also appears in the lower right-hand corner of the cross.

3.2 Using the Festo product catalog with FluidDraw

The standard FluidDraw installation already contains all of the products from Festo. That is why an extract from the Festo product catalog is installed in multiple languages. Note: In order to remain up to date, we recommend always installing the latest product catalog. FluidDraw recognizes more current versions of the catalog and utilizes these for the product search.

There two primary options for inserting symbols with part numbers and technical details.

Via symbol search ([Insert](#) menu and [Find Symbol Description...](#) menu entry) or from the FluidDraw library window

If you wish to insert a symbol from the FluidDraw symbol library, simply select a graphical image which represents the function of a component or component group. For the majority of the symbols there is a large number of different products with different part numbers and technical details (attributes and parameters). To select a product right-click on the [dialog window Properties](#) symbol and activate the [Find...](#) button.

About the Festo product catalog

The product catalog represents a complete data base of the available Festo products. If you wish to insert the associated symbols with part numbers and technical details for certain products then select the [From Festo Catalog...](#) menu item on the [Insert](#) menu. The Insert from catalog dialog window appears. Enter your key words into the "Search" input line. The upper table contains the search result. Selecting a row there displays the associated symbol in the preview. Select a product by double-clicking on a table row or via the [Add to Selection](#) button. Selected products are collected in the lower table. When leaving the dialog window the symbols of the selected products are inserted into the circuit window. These symbols contain the part numbers and technical details of the previously selected products in the form of attribute value pairs.

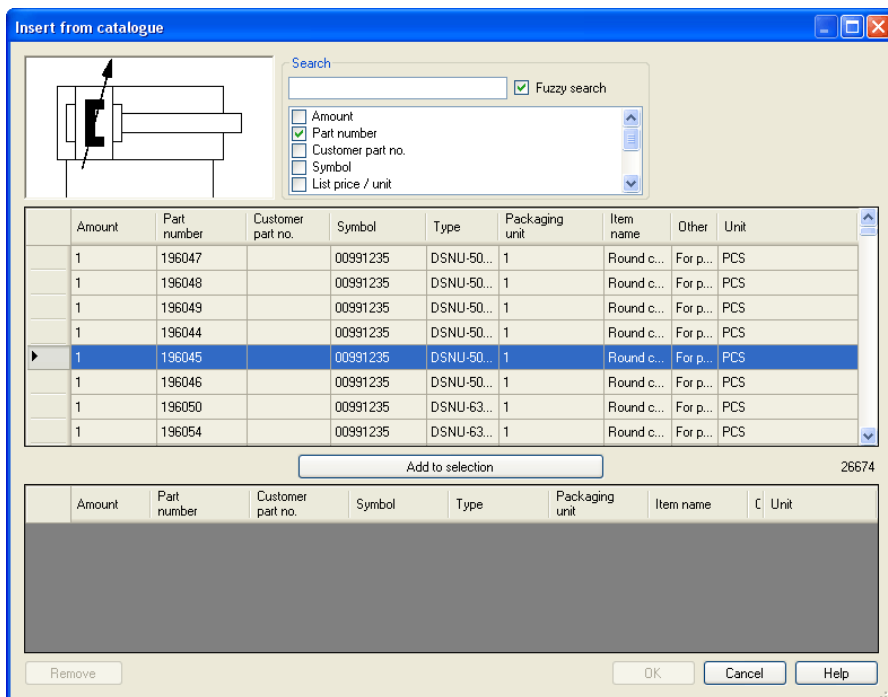


Figure 3/1: Insert from catalog dialog window

If you wish to insert one or more symbols from an existing shopping cart select the **From Festo Shopping Basket...** menu item from the **Insert** menu.

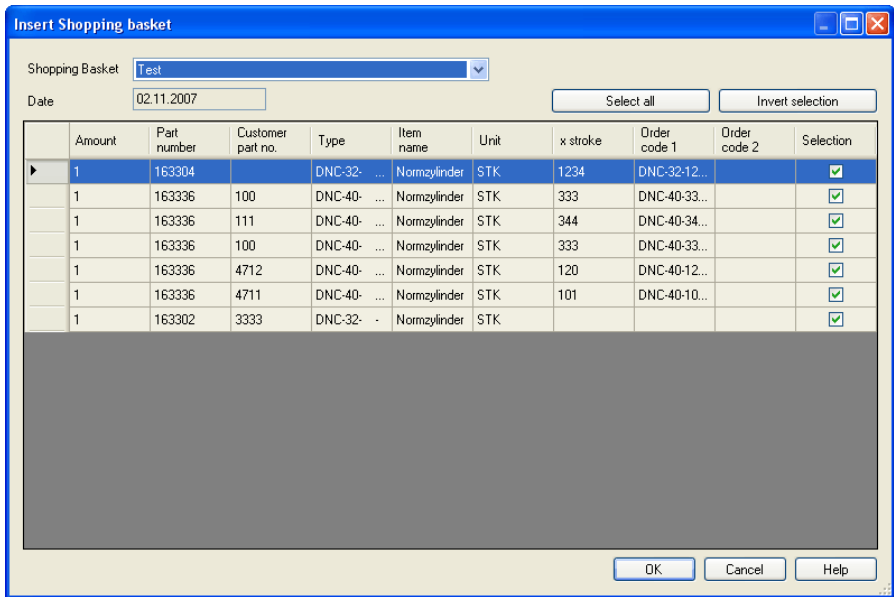



Figure 3/2: Insert Shopping Basket dialog window

If you have selected a product which does not possess a graphical illustration then a text appears in the circuit diagram which takes on the component attributes instead of the symbol. Both this text and the identifiers for the symbols appear in the [parts list](#).

Using the [Tools](#) menu and the [Options...](#) menu item you can modify the settings that influence the generation of symbol designations and accessories. Where desired FluidDraw automatically generates text elements acquired from the type attribute of the associated symbol

3.3 Connect connectors / ports

In order to connect two component connectors with a line move the mouse pointer to the component connector. You can recognize a connector by the small circle at the end of a symbol's connection line. As soon as you have touched the line the mouse pointer transforms into a crosshair .

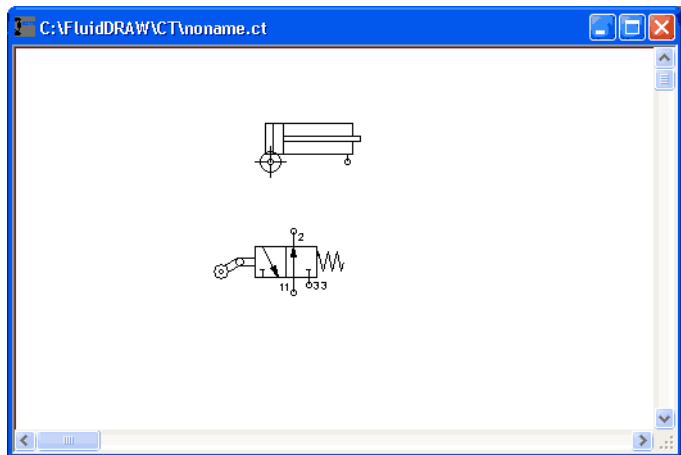




Figure 3/3: Mouse pointer as a crosshair over a component connector.

→ Now press the left mouse button and move the mouse pointer to the connector which you wish to connect with the first.

You can recognize when you are over the connector by the shape of the mouse pointer . If the mouse pointer is over a connector which is already connected to a line then the prohibited sign  appears and no line can be connected.

→ Release the mouse button once you have touched the second connector.

FluidDraw automatically lays a line between the two connectors.

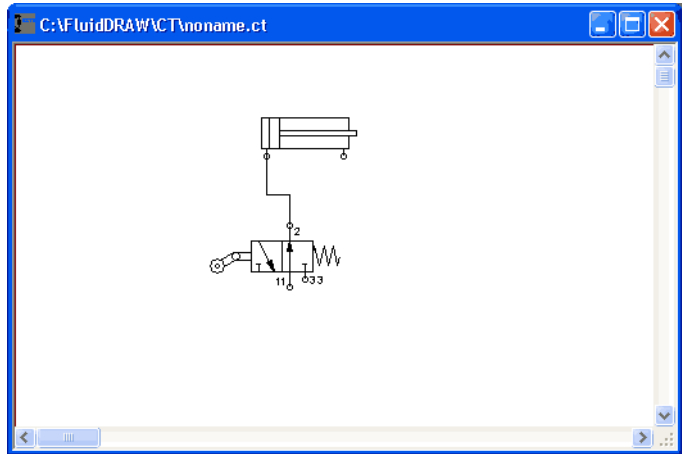


Figure 3/4: Line between two connectors




Support positions can be defined when laying lines. Simply release the mouse button while laying the line and click the desired position. The line is completed once you click the second connector or click the same point twice. You can cancel the action by pressing the **Esc** key.

3.4 Insert T-connector

You do not need to utilize a special symbol in order to insert a T-connector. FluidDraw automatically inserts a T-connector when you lay a component connection on a line or lay a line segment to a connection. If you wish to connect two lines then you can also draw one line segment to another; FluidDraw then inserts two T-connectors and connects these with a new line.

→ Move the mouse points to a connector and press the left mouse button.

As soon as you touch the line the mouse pointer transforms into a crosshair .

→ Release the mouse button once you have touched the desired point on the line.

FluidDraw inserts a T-connector and automatically lays a line.

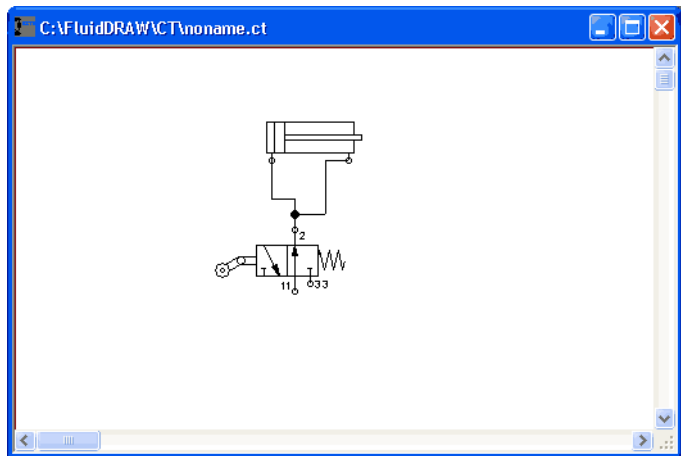



Figure 3/5: Line connection with inserted T-connector

Every T-connector can connect up to 4 lines with each other.

3.5 Lay lines

Once you have joined two connectors you can then adjust the length of the lines. You can move the line segments in parallel by moving the respective line segment. You can see when you have touched the line on the basis of the line-selection shape  of the mouse pointer.

- Press the left mouse button and move the line segment orthogonally to the desired position.

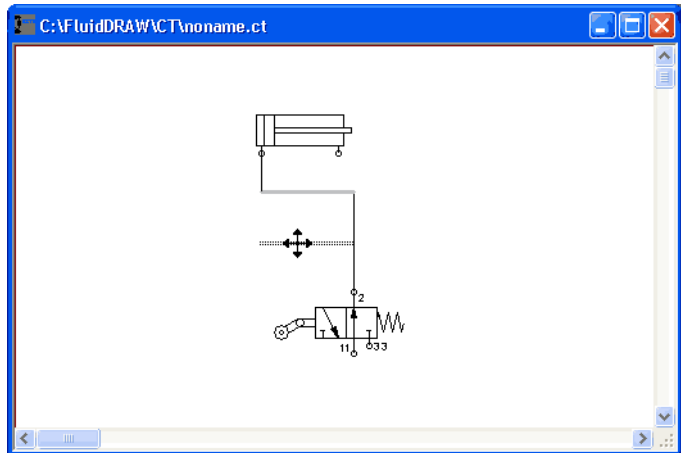


Figure 3/6: Moving a line segment

- Release the left mouse button; FluidDraw adjusts the adjoining line segments in such a way that they remain connected.

If you move a line segment with is directly connected to a component connector then FluidDraw may insert additional line segments in order to prevent gaps.

3.6 Define the characteristics of lines

You can set the style, the color and the [drawing layer](#) of the lines by double-clicking on a line segment or selecting the line segment and choosing the [Properties...](#) menu item from the [Edit](#) menu. The Line Properties dialog box then opens. The settings are applied to the entire line section up to the next connector or T-connector.

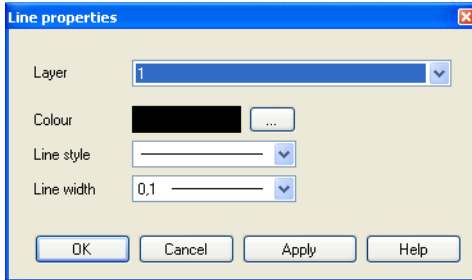


Figure 3/7: Line Properties dialog window: Set the properties of a pneumatic line

Layer	Sets the drawing layer of the line.
Color	Sets the color of the line.
Line Style	Sets the line style of the line.
Line Width	Sets the line thickness of the line.

Note: Working lines are typically displayed as solid lines, pilot lines as dashed lines.

3.7 Delete line

To delete a [line](#) you can either select an associated line segment and press the **Del** or select the **Delete** menu item on the **Edit** menu or select a component connector and press the **Del** key. In both cases not the connector itself is deleted but rather the line.

If you delete a T-connector which three or four lines are attached to then all of the lines are deleted. If, on the other hand, only two lines are connected then only the T-connector is deleted and the two lines combined into one.

3.8 Set the properties of the connectors

You can assign a component connector an identification and a blanking plug or silencer by double-clicking on the connector or by selecting the connector and choosing the **Properties...** menu item on the **Edit** menu. The Connector dialog box then opens.

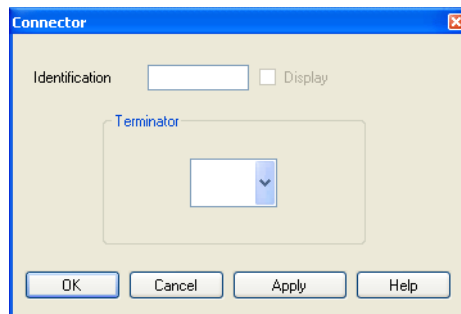


Figure 3/8: Connector dialog window: Set the properties of a connector

Identification

You can enter a text identifying this connector into the input line. If the Display option is activated then the identification is displayed in the circuit diagram.


Terminator

Open the symbol list containing connector terminations by clicking on the button using the arrow. Select a suitable silencer or a blanking plug. Note: Please note that this symbol list is only available if a **line** is not connected to the connector. If you wish to connect a line to a sealed connector you first have to remove the blanking plugs or the silencer. Select the empty field on the symbol list with the connector terminations.

3.9 Define component connector

To define a new component connector select the **Define Connector** menu item on the **Edit** menu. FluidDraw switches to a special mode in which the next mouse-click defines the new connector. If you wish to cancel the action press the **Esc** key.

Connectors in symbols can be set at any desired position.

Note: To set the connector precisely at the desired position we recommend zooming the display window in closely to the relevant symbol. As soon as the mouse pointer is over a symbol the mouse pointer converts into a cross-hair . Left-clicking sets the new connector.

3.10 Delete component connector

To delete a component connector select this and choose the **Delete Connector** menu entry on the **Edit** menu.

Note: Please note that the **Delete** menu entry or the **Del** do not delete the connector but rather only one possibly connected [line](#).

3.11 Configure a way valve

If you require a certain valve which you cannot find in the standard FluidDraw library you can create your own valve symbol with the help of the valve editor.

→ Insert a 5/n way valve into a circuit window from the “Standard symbols / Pneumatic Symbols / Symbols” library to create new symbols / configurable way valves.

To set the valve body and the actuation types of the way valves double-click on the valve. The [dialog window Properties](#) then opens. Click the “Configure Valve” tab to access the valve editor.

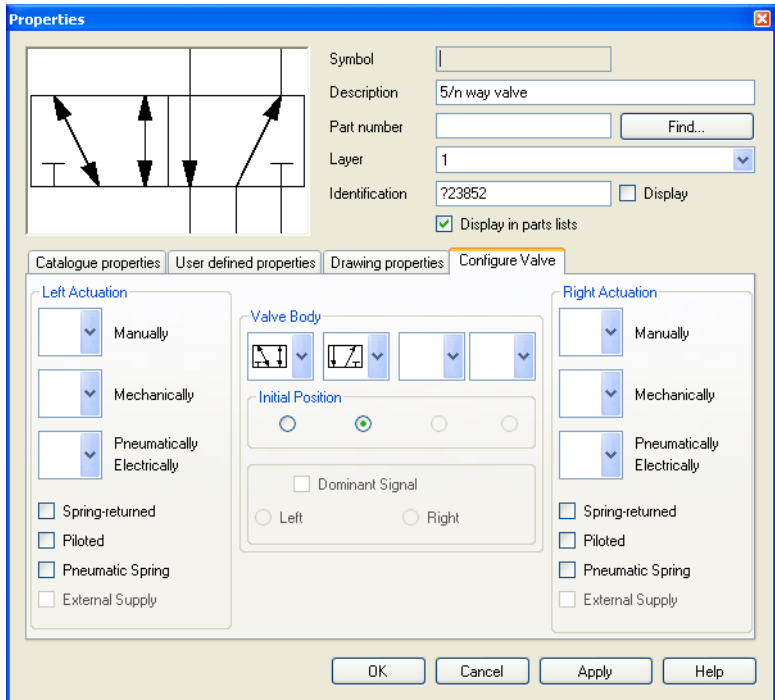


Figure 3/9: Properties dialog window: Configure Valve tab

Left Actuation – Right Actuation

You can select the actuation types of the valve from the “Manually”, “Mechanically” and “Pneumatically/Electrically” categories for both sides of the valve. Click the button with the mouse pointer and select a symbol element. A valve can possess multiple actuation types at the same time. If you do not want an actuation type from a category then select the empty field on the list. In addition you can also define whether each side should possess a spring return, pilot control, pneumatic spring or external supply.

Valve Body	A configurable valve can possess a maximum of four switching positions. One valve body can be selected for each switching position. Click the associate button with the mouse points to open the symbol elements list. Select a symbol element for each switching position.. If you want to use less than four switching positions then select the empty field on the list for the positions not required.
Initial Position	Here you set which switching position the valve should use in the normal position. Note: When determining this setting ensure that it does not conflict with a possible spring return.
Dominant Signal	This provides a graphical identification of whether the right or left signal should dominate in the event of equally strong signals.
External Supply	This creates an additional connector for connecting the external supply for the controller.

3.12 Configure cylinder

If you require a certain cylinder which you cannot find in the standard FluidDraw library you can create your own cylinder symbol with the help of the cylinder editor.

→ Insert a double-acting cylinder into a circuit window from the “Standard symbols / Pneumatic Symbols / Symbols” library to create new symbols / configurable cylinders.

To configure the cylinder double-click on the cylinder. The [dialog window Properties](#) then opens. Click the “Configure Cylinder” tab. This brings you to the cylinder editor.

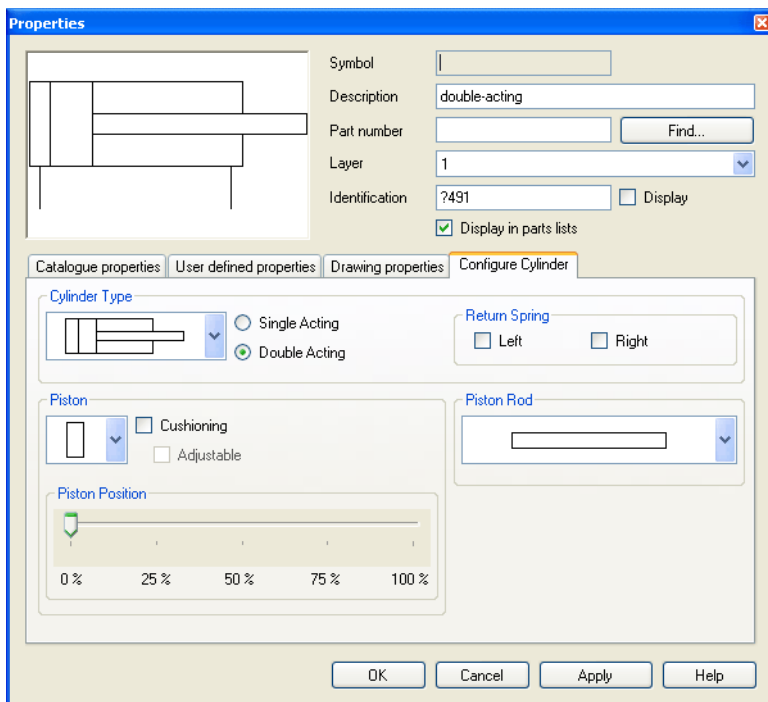


Figure 3/10: Properties dialog window: Configure Cylinder tab

Cylinder Type

Click the button with the mouse pointer to open the symbol elements list. Select a cylinder type. Set whether the cylinder should be single-acting or double-acting.

Return Spring

Specifies whether a spring for the return should be inserted into the right or left cylinder chamber.

Piston

Click the button with the mouse pointer and select a symbol element for the piston. Set whether the cylinder should possess end position cushioning and whether this should be adjustable.

Piston Rod

Click the button with the mouse pointer and select a symbol element for the piston rod.

Piston Position

Using the slider you can specify the relative piston position in 25 % steps. 0 % stands for the completely retracted and 100 % extended piston.

3.13 Group symbols

If you wish to combine multiple symbols into a group select these and choose the **Group** menu item on the **Edit** menu. Groups can also be nested, i.e. already grouped objects can be grouped again.

A group primarily serves as a drawing aid and does not represent any new components. Every element in a group is included in a **parts list** in exactly the same way as without being grouped. Double-clicking on a group element opens the **dialog window Properties** of group element clicked.

If you wish to combine multiple symbols into a new component with its own attributes create a **macro object**.


3.14 Create macro objects

If you wish to combine multiple symbols into a new component with its own attributes then select these and choose the **Create Macro Object** menu item on the **Edit** menu. This creates a new macro object. Macro objects are included in the **parts lists** as independent components. The original symbols are removed from the parts lists. It is no longer possible to modify their component attributes.


3.15 Delete symbol groups and macro objects

To delete a group or a macro object select the group or the macro object and select the **Ungroup/Break Off** menu item from the **Edit** menu. This always deletes the outermost group. In order to delete nested groups you must repeat the operation.

3.16 Align symbols


To align objects with each other, select these and choose the desired orientation from the **Align** menu item on the **Edit** menu or click the appropriate button  on the toolbar.


3.17 Mirror symbols

The symbols can be both horizontally and vertically mirrored. Select the desired mirror axis from the **Mirror** menu item on the **Edit** menu or click the appropriate button  on the toolbar. If you select multiple objects at the same time then every object will be mirrored separately. If you wish the operation to use a common mirror axis then group the objects before the operation.

The geometrical properties can be entered directly via the **“Drawing Properties”** tab. Enter a minus sign in front of the corresponding scaling factor in order to mirror the symbol.

3.18 Rotate symbols

The symbols can be rotated in 90 degree steps or with the help of the mouse pointer. To rotate objects in 90 degree steps, select the desired angle from the **Rotate** menu item on the **Edit** menu or click the appropriate button  on the toolbar. If you select multiple objects at the same time then every object will be rotated separately. If you wish the operation to use a common axis of rotation then group the objects before the operation.

You can also rotate symbols with the help of the mouse pointer by clicking and dragging the edge of the symbol. In order to do this FluidDraw must be in **Enable Rotate** mode. This mode can be activated or deactivated using the **Enable Rotate** menu item on the **Edit** menu or by clicking on the corresponding button  on the toolbar. Note: Activating the **Enable Scale** mode deactivates the **Enable Rotate** mode and vice versa.

→ In **Enable Rotate** mode click the edge of the symbol and hold down the mouse button.

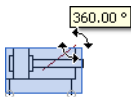


Figure 3/11: Rotate symbol

The current angle of rotation and guides are displayed.

→ Hold down the mouse button and move the mouse pointer to the desired position. The angle changes in 15 degree steps.

Note: If you also press the **Shift** key then you can rotate the symbol steplessly.

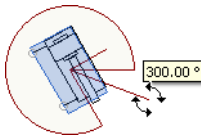



Figure 3/12: Rotate symbol

The angle of rotation can be entered directly via the “Drawing Properties” tab.

3.19 Scale symbols

The component symbols can be scaled with the help of the mouse pointer. In order to do this FluidDraw must be in **Enable Scale** mode. This mode can be activated or deactivated using the **Enable Scale** menu item on the **Edit** menu or by clicking on the corresponding button  on the toolbar. Note: Activating the **Enable Scale** mode deactivates the **Enable Rotate** mode and vice versa.

→ In **Enable Scale** mode click the edge or corner of the symbol and hold down the mouse button.

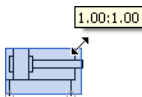


Figure 3/13: Scale symbol

The current scaling relationship to the original sizes is displayed.

→ Hold down the mouse button and move the mouse pointer until the desired size is reached. The scaling relationship is changed in 0.25 steps. Note: If you also press the **Shift** key then you can scale the symbol steplessly.

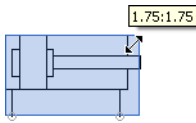


Figure 3/14: Scale symbol

You can simultaneously mirror the symbol by moving the mouse pointer over the middle of the symbol to the opposite side.

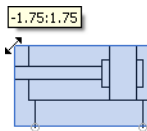


Figure 3/15: Mirror symbol

The scaling factor can be entered directly via the “[Drawing Properties](#)” tab.

Drawing frame

Chapter 4

In FluidDraw drawing frames are “circuit diagrams” which consists of a title block and the frame with the field divisions. They can also be displayed in other circuit diagrams. Existing CAD drawing frames can be imported using the **DXF import** menu item on the **File** menu. No order to make a drawing frame usable in different projects and circuit diagrams a number of texts in the title block must be editable. These texts are Author, Creation date, Project name, Page name and Page number, for example. In FluidDraw these are **text components** with Attribute Link.

4.1 Editable labels

Texts in the title block of a drawing frame are **text components**. You can use the imported texts or insert new text components at the desired positions. Editable texts are **text components** with Attribute Link. These texts are replaced by the corresponding attribute values of the project and the circuit.

→ Double-click on a text component to open the Properties dialog window.

→ Enter the name of the attribute to be linked into the text field, e.g. “creator” and activate the “Attribute Link” option. Note: You can also utilize and imported text as an attribute name.

The attribute name is utilized as a placeholder. These attributes are displayed in angle brackets in the “circuit diagram” with the drawing frame. These attributes are replaced by the corresponding attribute values when using the drawing frame in a project or circuit diagrams. Edit the attribute values in the Properties dialog window of the project or the circuit diagram. Clicking on a text component of the drawing frame opens the Properties dialog window of the project or the circuit which contains the attribute which the text component refers to.

4.2 Use a drawing frame

Drawing frames can be copied into a project and/or a circuit diagram. Insert the drawing frame via the Properties dialog window of a [project](#) or a [circuit](#).



All of the objects belonging to a drawing frame are inserted into the circuit diagram as a copy. Changing the file with the drawing frame at a later date does not have any effect on the circuit diagram which the drawing frame was inserted into.

When copying the objects belonging to the drawing frame all of the attributes are generated with the [drawing frame's text components](#) refer to and which do not exist in the project or circuit.

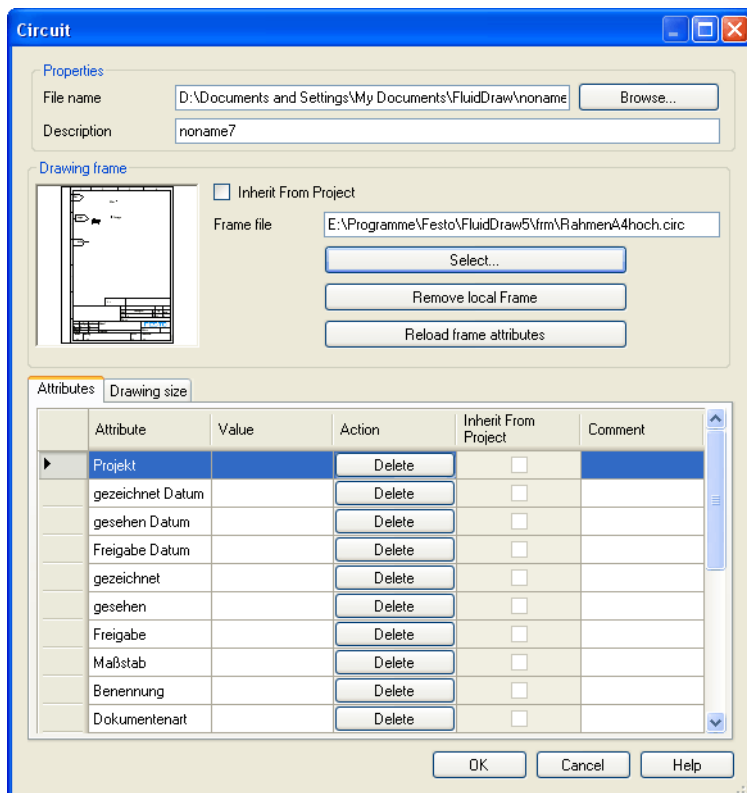


Figure 4/1: Circuit dialog window: Insert drawing frame

Inherit From Project

If this option is activated then the drawing frame specified in the project is copied into the circuit. The path and the file used are displayed in the “Frame File” line.


Pick...

This button opens a dialog in which an included frame can be selected. These frame files are located in the **frm** directory and are compiled in the **project file frames.prj**.

Remove Local Frame

Removes the drawing frame from the circuit. The drawing frame’s attributes remain attributes of the project or circuit.

Reload Frame Attributes

When inserting a drawing frame the attributes [of the text components of the drawing frame](#) are listed. These attributes are saved with the project or the circuit drawing and can be edited and deleted. The  button reloads all of the attributes from the drawing frame, thus updating the attribute list of the project or the circuit drawing.

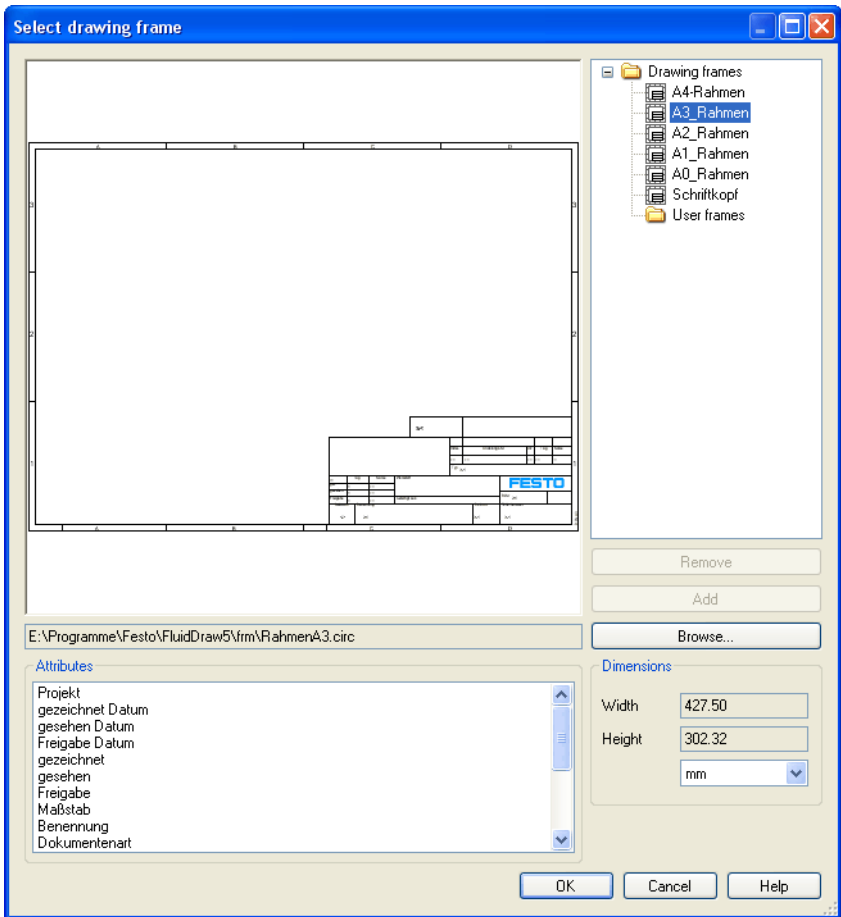


Figure 4/2: Select Drawing Frame dialog window

All of the available key files are displayed as a tree on the right-hand side. This tree enables the selection of the desired drawing frame. The drawing frame is displayed in the preview. In addition the “Attributes” list displays all of the attributes of the texts belonging to the drawing frame which are set as links.

Browse...

Opens a dialog window to select a circuit file to be inserted as a frame.

Additional tools for creating drawings

Chapter 5

5.1 Drawing tools

5.1.1 Grid

For arranging the symbols and laying the [lines](#) it is often practical to show a point or line grid. You can activate or deactivate the displays of the grid using the [Show Grid](#) menu item on the [View](#) menu. You can also set [grid settings](#) using the [Options...](#) menu item and the [Tools](#) menu.

In order to simplify handling the lines snap into place when they are close to a grid line. This makes it easier to find the precise position when moving the lines. Note: However the grid snap function may not be desired as it prevents free positioning. In this case you can hold down the [Ctrl](#) during the moving operation; This temporarily deactivates the grid snap function.

5.1.2 Alignment lines

Connectors between symbols should be exactly horizontal or vertical wherever possible. Then they can be connected with a straight line.

FluidDraw supports the exact positions firstly via the [grid snap](#) and secondly via automatically displaying red alignment lines while moving the selected objects.

→ Open a circuit file which contains multiple objects. Select one of these and slowly move it back and forth above and next to other objects.

Observe the dashed red lines which appear when two or more connectors lie above each other.

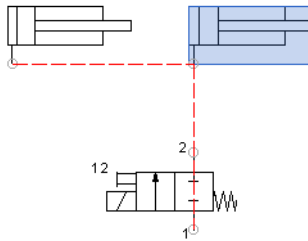


Figure 5/1: Automatically display the alignment lines

In order to simplify handling the connectors snap into place when they are close to an alignment line. This makes it easier to find the precise position when moving. Note: However the grid snap function may not be desired as it prevents free positioning. In this case you can hold down the **Ctrl** during the moving operation; This temporarily deactivates the alignment lines and the grid snap function.

5.2 Drawing levels

FluidDraw supports 256 drawing levels which can be shown / hidden and locked / unlocked individually. In addition you can also define the color and line thickness for every drawing level. Using the **Drawing Layers...** menu item on the **View** menu you can set the properties of the individual layers and also insert a description.

Note: The **drawing frame** is located on layer “0” by default.

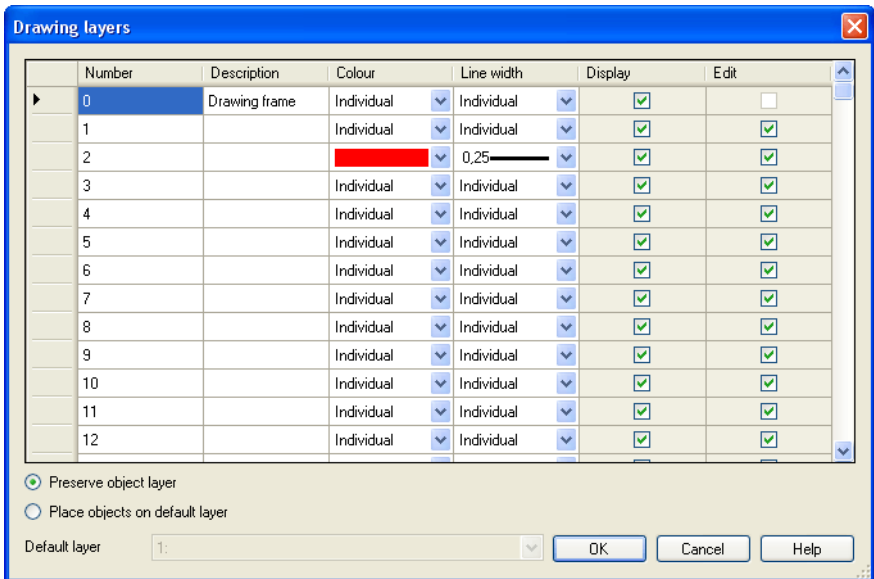


Figure 5/2: Drawing Layers dialog window

Under “Default Layer” you can select the drawing layer which new objects should be inserted into. If you do not want to change the layer of the symbol when it is inserted then select the option “Preserve Object Layer”.

Objects located on a drawing layer for which the “Edit” option is deactivated are visible however they cannot be selected and thus neither moved nor deleted. This enables you to lock a drawing frame, for example, if this was inserted manually and not via the [dialog window Properties for the circuit or project](#). In order to still be able to edit objects on layers of this type you must activate the “Edit” option for the corresponding layer.

Drawing layers for which the “Display” option is deactivated are neither visible nor can they be edited.

5.3 Cross-references

Cross-references serve to link parts of a circuit drawing associated with each other when the entire drawing is distributed across multiple pages. Thus lines can be interrupted, for example, and continued on another page. FluidDraw not only offers suitable symbols for cross-references but also the possibility of jumping directly from an existing cross-reference to the corresponding target as long as both are identified with an identical label. The circuit drawings involved must belong to the same [project](#).

Using the [Properties...](#) menu item on the [Edit](#) open the Cross-reference dialog window. Alternatively you can open this dialog window by double-clicking on the cross-reference symbol of via the [Properties...](#) context menu.

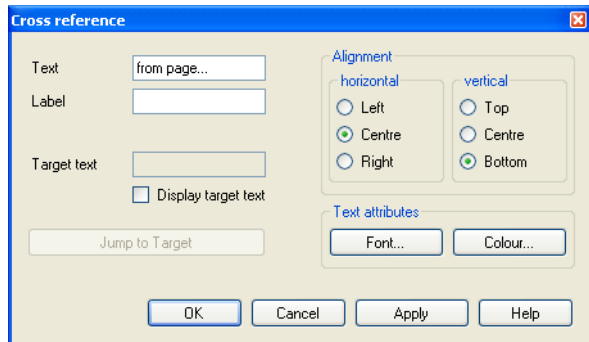


Figure 5/3: Cross-reference dialog window

Text	This text is displayed in the cross-reference
Label	Defines the labels via which the linked cross-references are identified.
Target Text	The text displayed here is stored as text for the corresponding cross-reference.

Display Target Text

If this option is activated the text of the corresponding cross-reference is displayed in the symbol below the text for this cross-reference.







Jump to Target








Activating this button opens the circuit window containing the corresponding cross-reference. The associated symbol is indicated by an animation.

5.4 Drawing functions and graphical elements

Graphical elements can be inserted into a circuit using the **Draw** menu or by activating the drawing function on the toolbar. In order to prevent you from unintentionally moving other symbols when drawing you enter a special mode for the drawing functions in which you can only perform the selected drawing functions. After every drawing operation FluidDraw returns to the normal editing mode. To insert another drawing element you must select the associated menu item or the corresponding drawing function on the toolbar. Note: If you wish to draw several elements in a row without leaving the special drawing mode then you can select the menu item or the corresponding drawing function on the toolbar while holding down the **Shift** key. The drawing mode then remains active until the menu item or the drawing function is deactivated by selecting it again or another drawing function is selected without the **Shift** key being pressed.

The drawing functions toolbar contains the following functions:

-  Switches to the normal editing mode.
-  Switches to the mode for drawing a **line**.
-  Switches to the mode for drawing a **rectangle**.
-  Switches to the mode for drawing a **circle**.
-  Switches to the mode for drawing a **ellipse**.
-  Switches to the mode for drawing a **text**.

-  Switches to the mode for drawing an [image](#).
-  Switches to the mode for drawing a [pneumatic line](#).
-  The graphical elements are drawn in the color shown.
-  The graphical elements are drawn in the line style shown.
-  The graphical elements are drawn in the line thickness shown.
-  The line beginnings are drawn with the symbol shown.
-  The line ends are drawn with the symbol shown.

5.4.1 Pneumatic line

A pneumatic line is drawn by defining two end points. A pneumatic line consists of two connectors with a line between. Both connectors can be used as the initial situation for additional connections. The line segments of pneumatic lines are always perpendicular to each other.

The line attributes of the line can be defined in the [dialog window Properties for the line](#).

5.4.2 Line

A line is drawn by defining two points. In contrast to a pneumatic line this line is purely a drawing element. As such it can be drawn at any desired angle however does not allow connections with pneumatic connectors.

The line-specific properties can be defined in the Drawing Properties tab of the Properties dialog window for the line:

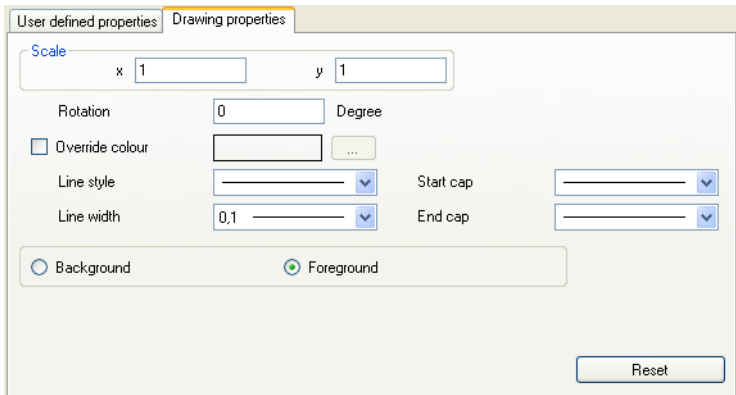


Figure 5/4: Properties dialog window for a line: Drawing Properties tab

Scale	Defines the scaling in the x or y direction.
Rotation	Defines the angle of rotation in degrees.
Override Color	Defines the color of the drawing element.
Line Style	Defines the line style.
Line Width	Defines the line thickness.
Start Cap	Defines how the beginning of the line is displayed.
End Cap	Defines how the end of the line is displayed.
Background	Sets the location of the drawing element to the background. This means that all of the circuit symbols are on top of this. For example symbols are not covered by filled drawing elements.
Foreground	Sets the location of the drawing element to the foreground. This means that all of the circuit symbols are underneath this. For example symbols are covered by filled drawing elements.

5.4.3 Rectangle

A rectangle is drawn by defining two diagonally opposite corner points.

The rectangle-specific properties can be defined in the Drawing Properties tab of the Properties dialog window for the rectangle:

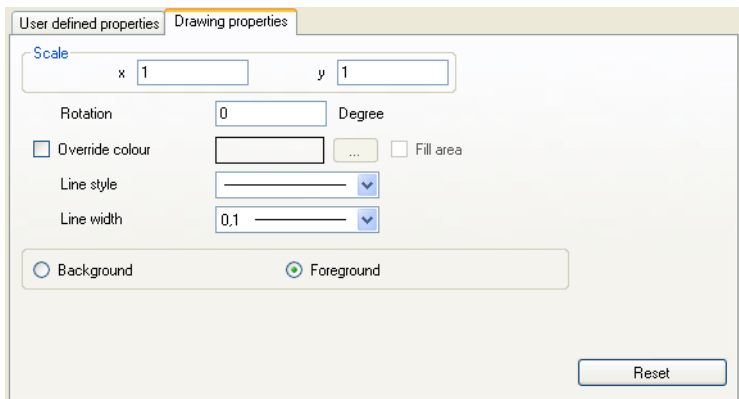


Figure 5/5: Properties dialog window for a rectangle: Drawing Properties tab

Scale	Defines the scaling in the x or y direction.
Rotation	Defines the angle of rotation in degrees.
Override Color	Defines the color of the drawing element.
Fill Area	Fills the rectangle with the color listed.
Line Style	Defines the line style.
Line Width	Defines the line thickness.

Background

Sets the location of the drawing element to the background. This means that all of the circuit symbols are on top of this. For example symbols are not covered by filled drawing elements.

Foreground

Sets the location of the drawing element to the foreground. This means that all of the circuit symbols are underneath this. For example symbols are covered by filled drawing elements.

5.4.4 Circle

A circle is drawn by defining a middle point and a radius. You can draw a circular arc by subsequently entering the beginning and final angles into the Drawing Properties tab in the Properties dialog window of the circle.

The circle-specific properties can be defined in the Drawing Properties tab of the Properties dialog window for the circle:

The screenshot shows the 'Drawing properties' tab of a properties dialog window for a circle. The window has two tabs: 'User defined properties' and 'Drawing properties'. The 'Drawing properties' tab is active and contains the following controls:

- Scale:** Two input fields for 'x' and 'y', both set to '1'.
- Rotation:** A numeric input field set to '90' and a unit dropdown menu set to 'Degree'.
- Override colour:** A checkbox that is unchecked, followed by a color selection box and a 'Fill area' checkbox that is also unchecked.
- Line style:** A dropdown menu showing a solid line.
- Line width:** A numeric input field set to '0,1'.
- Arc:** Two numeric input fields for 'Starting angle' and 'Ending angle', both set to '0'.
- Start cap:** A dropdown menu showing a rounded end.
- End cap:** A dropdown menu showing a rounded end.
- Background/Foreground:** Two radio buttons. 'Background' is selected (indicated by a blue dot), and 'Foreground' is unselected (indicated by a green dot).
- Reset:** A button located at the bottom right of the dialog.

Figure 5/6: Properties dialog window for a circle: Drawing Properties tab

Scale

Defines the scaling in the x or y direction.

Arc	Defines the starting and final angles of a circular arc.
Rotation	Defines the angle of rotation in degrees.
Override Color	Defines the color of the drawing element.
Fill Area	Fills the circle with the color listed.
Line Style	Defines the line style.
Line Width	Defines the line thickness.
Start Cap	Defines how the beginning of the line is displayed.
End Cap	Defines how the end of the line is displayed.
Background	Sets the location of the drawing element to the background. This means that all of the circuit symbols are on top of this. For example symbols are not covered by filled drawing elements.
Foreground	Sets the location of the drawing element to the foreground. This means that all of the circuit symbols are underneath this. For example symbols are covered by filled drawing elements.

5.4.5 Ellipse

An ellipse is drawn by defining a middle point and two radii with parallel axes. You can draw an ellipse by subsequently entering the beginning and final angles into the Drawing Properties tab in the Properties dialog window of the ellipse.

The ellipse-specific properties can be defined in the Drawing Properties tab of the Properties dialog window for the ellipse:

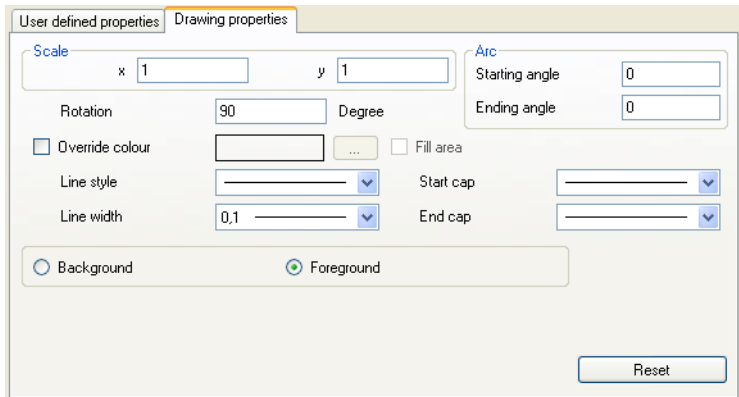


Figure 5/7: Properties dialog window for an ellipse: Drawing Properties tab

Scale	Defines the scaling in the x or y direction.
Arc	Defines the starting and final angles of an ellipse.
Rotation	Defines the angle of rotation in degrees.
Override Color	Defines the color of the drawing element.
Fill Area	Fills the circle with the color listed.
Line Style	Defines the line style.
Line Width	Defines the line thickness.
Start Cap	Defines how the beginning of the line is displayed.
End Cap	Defines how the end of the line is displayed.
Background	Sets the location of the drawing element to the background. This means that all of the circuit symbols are on top of this. For example symbols are not covered by filled drawing elements.

Foreground

Sets the location of the drawing element to the foreground. This means that all of the circuit symbols are underneath this. For example symbols are covered by filled drawing elements.

5.4.6 Text

A text is inserted at the position of the mouse pointer by clicking. The [dialog window Properties](#) for entering the text and setting the attributes then opens.

5.4.7 Image

An image is inserted at the position of the mouse pointer by clicking. The dialog window for selecting an existing image file then opens.



Large background images can massively decrease the processing speed as the screen has to be re-drawn whenever symbols are moved or edited.

The image-specific properties can be defined in the Drawing Properties tab of the Properties dialog window for the image:

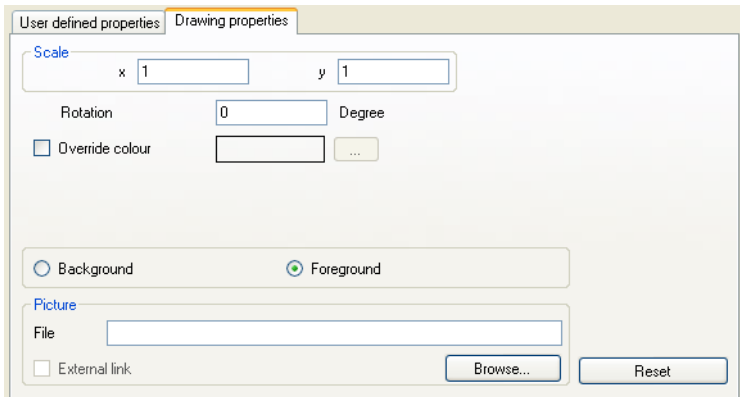


Figure 5/8: Properties dialog window for an image: Drawing Properties tab

Scale	Defines the scaling in the x or y direction.
Rotation	Defines the angle of rotation in degrees.
Override Color	Defines the color of the drawing element.
Background	Sets the location of the drawing element to the background. This means that all of the circuit symbols are on top of this. For example symbols are not covered by filled drawing elements.
Foreground	Sets the location of the drawing element to the foreground. This means that all of the circuit symbols are underneath this. For example symbols are covered by filled drawing elements.
File	Defines the image file for the image.
External Link	When this option is active FluidDraw only saves a link to the image file. However if this option is deactivated then the image is embedded in the circuit diagram. This is the preferable variant if you wish to pass on or archive the circuit diagram.

5.5 Check drawing

Using the **Check Drawing** menu item on the **Circuit** menu you can cause FluidDraw to check your drawing for possible drawing errors. FluidDraw displays the following messages where necessary:

- Duplicated identification labels
- Unresolved link targets
- Superimposed objects
- Lines through connectors
- Open connectors

If messages are displayed then the corresponding objects are selected after confirmation in the dialog window.

Component attributes

Chapter 6

6.1 Component attributes in the Properties dialog window

The FluidDraw circuit symbols largely correspond to the components in the [Festo product catalog](#). Even if you have not installed the [product catalog](#) FluidDraw read most of the component attributes from the database delivered with the program. If you have installed a newer version of the [product catalog](#) then FluidDraw automatically accesses the current data. Open the [dialog window Properties of the components](#) by double-clicking on a symbol or via the [Properties...](#) menu item on the [Edit](#) menu.

Properties

Symbol: 00991235

Description: Adjustable cushioning, sensing

Part number: 163309 Find...

Layer: 1

Identification: 1A1 ☐ Display

☒ Display in parts lists

Catalogue properties | User defined properties | Drawing properties

Attribute	Value	Display
Part number	163309	<input type="checkbox"/>
Symbol	00991235	<input type="checkbox"/>
Type	DNC-32-100-PPV-A	<input checked="" type="checkbox"/>
Packaging unit	1	<input type="checkbox"/>
Item name	Standard cylinder	<input type="checkbox"/>
Other	As per ISO 15552, with profile cylinder barrel, for proximity sensing, with adjustable cushioning at both end positions.	<input type="checkbox"/>
Unit	PCS	<input type="checkbox"/>

OK Cancel Apply Help

Figure 6/1: Properties dialog window for a component:

The properties of a component are saved in attribute value pairs. The attributes are divided into different groups. The first group contains the general properties:

Symbol	00991235
Description	Adjustable cushioning, sensing
Part number	163309 Find...
Layer	1
Identification	1A1 <input type="checkbox"/> Display
<input checked="" type="checkbox"/> Display in parts lists	

Figure 6/2: Extract from the Properties dialog window of a component: General properties

Symbol Name	Displays the name of the circuit symbol. The symbol name is mapped to appropriate products in the Festo product catalog . Users cannot edit the symbol name.
Description	Contains a more detailed description of the complete name of the circuit symbol where applicable. Users cannot edit this,
Part number	The part number uniquely identifies a product. You can enter the part number manually or search via the Find... button in the product catalog . The component attributes saved in the catalog are only automatically taken over if the selection is made via the product catalog. You can find these attributes in the Catalog Properties tab of the Properties dialog window for the component.
Layer	In this selection list you define the drawing layer for the symbol. Depending on the setting for the drawing layer it may not be possible to display or edit the symbol. In order to make the symbol visible or change the settings you have to temporarily activate the drawing layer on the Drawing Layers... menu item from the View menu.
Identification	Here you can assign a label which uniquely identifies the components in the circuit diagram. If the “Display” option is selected the label is shown in the circuit diagram as text. Note: When inserting or copying circuit diagram symbol FluidDraw automatically assigns a unique label. An automatically assigned label text begins with a question mark and may be edited by the user. FluidDraw issues a warning message if you assign a label which already exists.
Display in Parts Lists	Deactivate this option if you do not want the symbol to appear in the parts list.

Component attributes from the product catalog as well as attributes assigned by the user are displayed in tabular form in different tabs. The table entries from the catalog cannot be edited. The value of an attribute is listed in the “Value” column. This value can be displayed in the circuit diagram next to the symbol as text if the “Display” option is activated.

The component attributes taken over from the product catalog are displayed in the “Catalog Properties” tab.

Catalogue properties		User defined properties	Drawing properties
	Attribute	Value	Display
▶	Part number	163309	<input type="checkbox"/>
	Symbol	00991235	<input type="checkbox"/>
	Type	DNC-32-100-PPV-A	<input type="checkbox"/>
	Packaging unit	1	<input type="checkbox"/>
	Item name	Standard cylinder	<input type="checkbox"/>
	Other	As per ISO 15552, with profile cylinder barrel, for proximity sensing, with adjustable cushioning at both end positions.	<input type="checkbox"/>
	Unit	PCS	<input type="checkbox"/>

Figure 6/3: Extract from the Properties dialog window of a component: Catalog Properties tab

In the “User Defined Properties” tab you can enter your own component attributes.

Catalogue properties		User defined properties		Drawing properties
	Attribute	Value	Display	Comment
▶	Supplier	Festo	<input type="checkbox"/>	
*			<input type="checkbox"/>	

Figure 6/4: Extract from the Properties dialog window of a component: User Defined Properties tab

In order to edit an entry the corresponding row must be selected by clicking on it. Clicking on the table cell to be modified again then enables you to edit the entry in the cell.

A cell can be deleted by first clicking on the cell to select it. Then the selected cell can be deleted using the **Del** key.

You can add any desired attributes by filling out the empty cells on the last row.

In the “Drawing Properties” tab you can enter a number of geometrical properties which influence how the symbols are displayed in the circuit diagram.

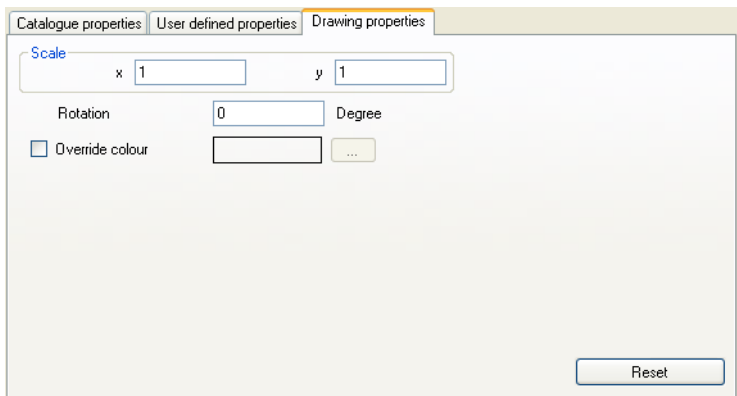


Figure 6/5: Extract from the Properties dialog window of a component: Drawing Properties tab

Scale	Defines the scaling in the x or y direction. The scaling factor can also be defined with the help of the mouse pointer. This is described in the “ Scale symbols ” section.
Rotation	Defines the angle of rotation in degrees. The angle of rotation can also be defined with the help of the mouse pointer. This is described in the “ Rotate symbols ” section.
Override Color	If this option is activated then another color can be selected for the symbol.
Reset	Resets the geometrical settings to the default values: Scaling to 1, rotation to 0 and “Override Color” deactivated.

6.2 Attributes of the text components

In FluidDraw text components serve, firstly, to insert comments and labels and, secondly, can be defined by the labels and accessories without displaying a symbol. Open the dialog window Properties of the text component by double-clicking on a text or via the **Properties...** menu item on the **Edit** menu.

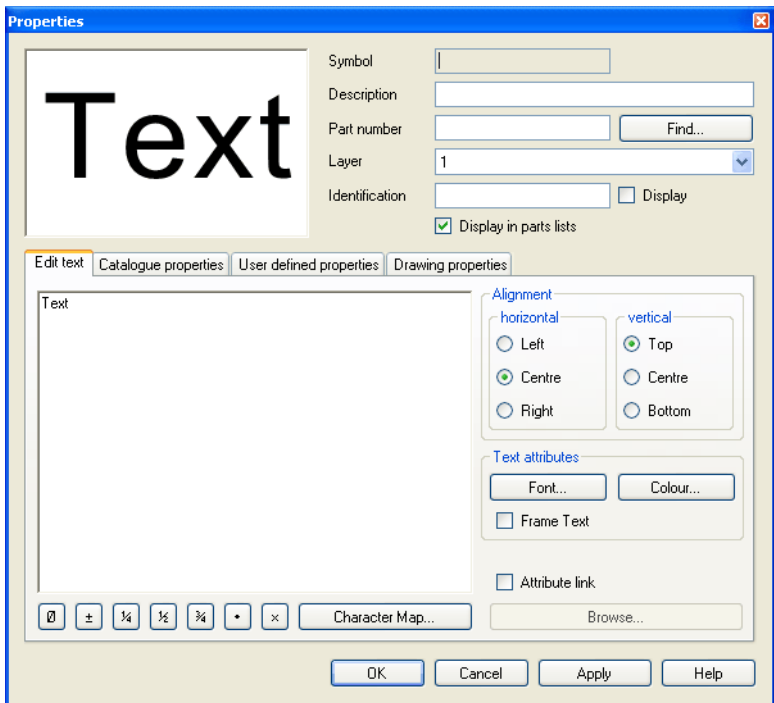


Figure 6/6: Properties dialog window for a text component:

A text component possesses all of the attributes of a **standard component**. The text properties are located on the “Edit Text” tab.

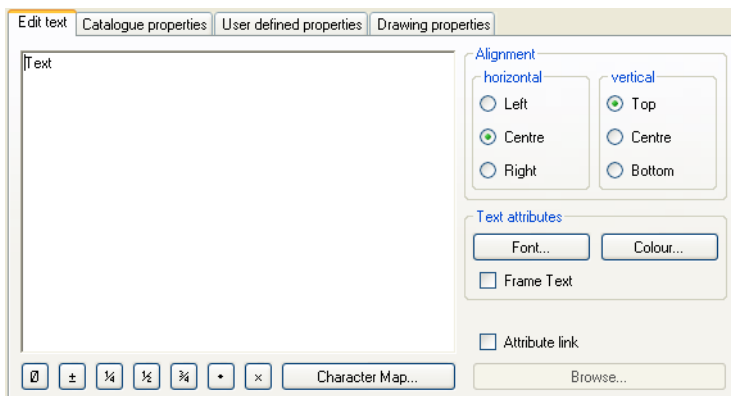


Figure 6/7: Extract from the Properties dialog window of a text component: Edit Text tab

Edit Text	Enter your text into the input field on the left-hand side. You can also enter text with multiple lines. Press the Enter key to create a line break.
Alignment	Defines the horizontal or the vertical alignment of the text within the text field.
Font...	Defines the text font.
Color...	Defines the text color.
Frame Text	Draws a frame around the text field.
Attribute Link	If this option is activated then the text entered is not shown but rather the text identifies a link to an attribute. The value of the selected attribute is shown in the circuit diagram. You can find a detailed description of this function under Link text components with attributes .

6.3 Link text components with attributes

Text components can also display the attributes of other components, the attributes of a circuit diagram or the project. The text components must be linked to the corresponding attribute. The linked attribute is defined in the entry field of the “Edit Text” tab in the [dialog window Properties of the text component](#). The activated Attribute Link option determines that the text is not displayed in the text field but rather should be interpreted as a link.

Example:

Assuming that your project is called “Project1” and possesses the “Supplier” attribute with the value “Festo”. You project contains the “Circuit1” circuit and you would like to display the value of the “Supplier” attribute, in this example namely the value “Festo”, in this circuit diagram.

→ Insert a text component into the circuit diagram by selecting the text element on the drawing elements toolbar and then left-click on the circuit diagram. The Properties dialog box for the text component then opens. Note: You can open the dialog window of an existing text component by double-clicking on it or via the [Properties...](#) menu item on the [Edit](#) menu.

→ Activate the Attribute Link option and then click the [Browse...](#) button.

A window opens displaying the hierarchy of all of the available attributes. Please note that only those objects are listed with possess a label assigned by the user. Labels beginning with a question mark “?” are not listed. These are labels which have been automatically assigned by FluidDraw. The desired attribute is located under “Attribute Tree” - “Project1” - “Supplier”.

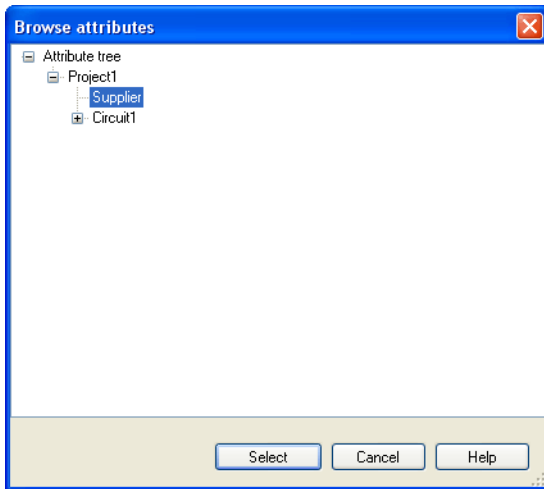


Figure 6/8: Browse Attributes dialog window

→ Select “Supplier” and then click the **Select** button.

The “Project1.Supplier” value is displayed in the input field and “Festo” in the preview. The complete and unique name of the attribute contains all of the hierarchy levels beginning at the project. The individual layers in the name are separated by a full stop. It is also possible to enter only the attribute name into the input field. In this example: “Supplier”. The hierarchy is then searched from the bottom up for the attribute, beginning with the text components. If the attribute searched for is not found in that component then the circuit and then the project are searched for the attribute.

If the attribute is not found then the attribute name is displayed in the circuit diagram in angle brackets. The missing attribute can be created at a later point in time. The link is then created automatically.

The concept of linking text components with attributes is also utilized for labeling [drawing frames](#).

6.4 Text components with specified links

Text components are automatically generated for displayable component attributes and connector labels which are linked to the corresponding attributes. Open the Display Attribute dialog window with which you can adjust how the text is displayed by double-clicking on the text component or via the **Properties...** menu item on the **Edit** menu.

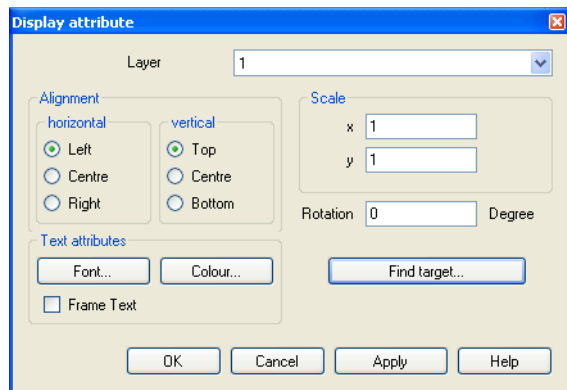


Figure 6/9: Display Attribute dialog window

Alignment	Defines the horizontal or the vertical alignment of the text within the text field.
Scale	Defines the scaling factor of the text in the x or y direction. Note: You can also change the text size by selecting another font size in the font type dialog window. You can open this dialog window using the Font... button under Text Attributes.
Rotation	Defines text's angle of rotation in degrees.
Font...	Defines the text font.

Color...	Defines the text color.
Frame Text	Draws a frame around the text field.
Find Target...	The target of a linked text is the attribute which the text refers to. This button opens the Properties dialog window of the object which contains this attribute.

6.5 Change the properties of multiple objects simultaneously

If multiple objects are selected, choosing the **Properties...** menu item on the **Edit** menu opens a dialog window with a limited selection of properties. It only contains those properties which meaningfully relate to all of the selected objects. This enables you to change the font for multiple texts in a single operation, for example. The more different object types (symbols, connectors, drawing elements, texts, etc.) that are selected, the less common properties are available.

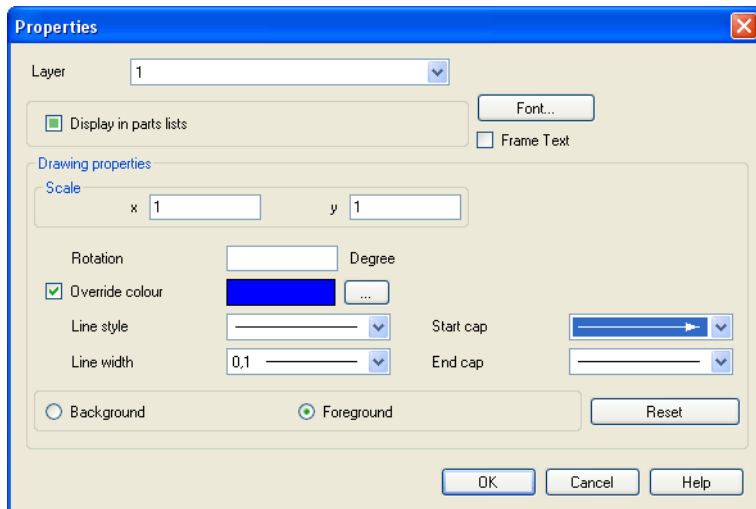


Figure 6/10: Properties dialog window for multiple selected objects

Parts list management

Chapter 7

FluidDraw creates parts lists which are automatically updated in the background while editing the circuit diagram.

7.1 Display parts list

We recommend setting up one file specifically for the parts list in a project. If you wish to create a circuit diagram without creating a project you can switch the view for this circuit diagram over to the parts list view. Selecting the [Parts List](#) menu item on the [View](#) menu enables you switch from the circuit diagram view to the parts list view. Conversely, selecting the [Circuit Diagram](#) menu item on the [View](#) menu enables you switch from the parts list view to the circuit diagram.

A parts list is displayed as a table in the window. You can edit the following options for editing the table:

- Attributes which the user can edit can be entered directly into the corresponding table field.
- Clicking on the column header sorts the rows on the basis of the column clicked. Clicking on the head of the same column again reverses the sorting order.
- You can adjust the column width by dragging the edges of a column header.
- You can change the order of the columns by dragging and dropping a column header in another position.

7.2 Find components from the parts on the circuit diagram

FluidDraw enables you to easily find components on the parts list in the circuit diagram and vice versa.

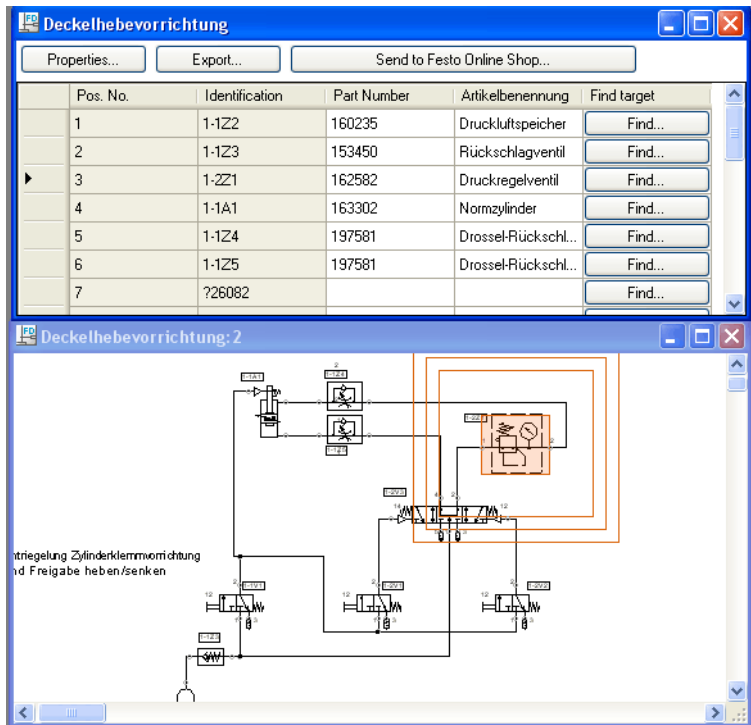



Figure 7/1: Finding components from the parts list

You have the following options:

- Clicking on the **Find...** button on the "Find Target" parts list column switches to the circuit diagram display and an animation identifies the corresponding components. If the same circuit diagram is already open in another window than the parts list view remains in the active window. The already opened window containing the circuit diagram is moved into the foreground and an animation identifies the corresponding circuit diagram symbol.

- When you click on a cell in the parts list table an animation identifies the corresponding component in all of the opened windows displaying the circuit diagram.
- If you click on a column header in the parts list table then both the entire column of the parts list as well as the corresponding components in the associated circuit diagram windows are selected.
- If you click on a component in a circuit diagram then both this component and the corresponding row in the open parts lists are selected.

7.3 Set parts list properties

In the [parts list view](#) clicking on the  button opens the Parts List Properties dialog window.

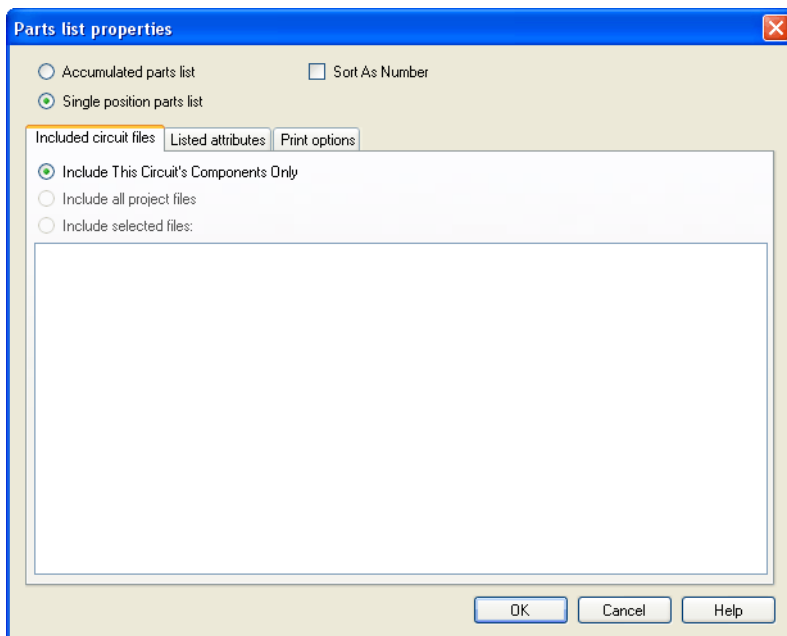


Figure 7/2: Parts List Properties dialog window: Included Circuit Files tab

Accumulated Parts List	Compiles the components with the same attributes.
Single Position Parts List	Lists all of the components individually
Sort As Number	Interprets the column contents as numbers, so that “10” appears after “2”, for example.
Include This Circuit’s Components Only	Only lists the components of the corresponding circuit diagram .
Include All Project Files	Lists all of the components of the active project.
Include Selected Files:	Lists all of the components of the selected circuit diagrams belonging to the active project.

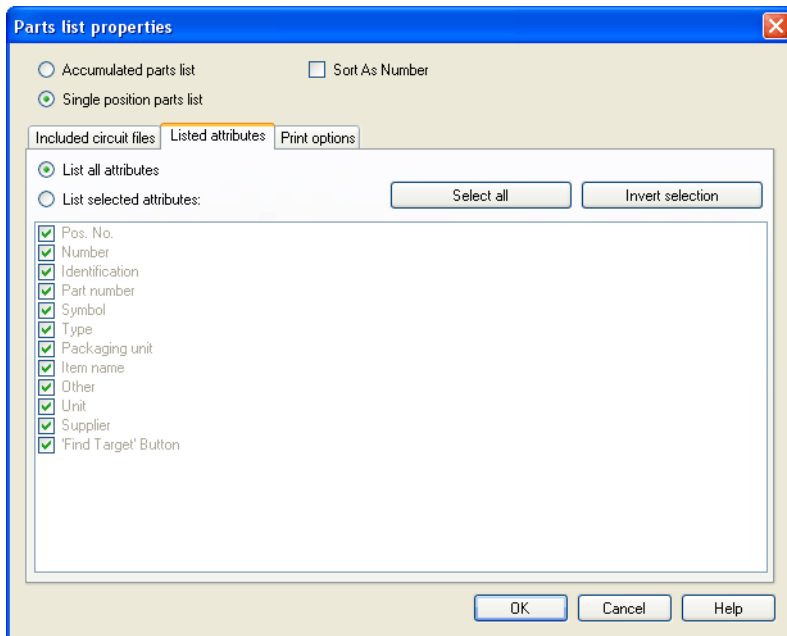


Figure 7/3: Parts List Properties dialog window: Listed Attributes tab

List All Attributes

Lists all of the component attributes within a parts list row.

List Selected Attributes:

Only lists selected component attributes within a parts list row.

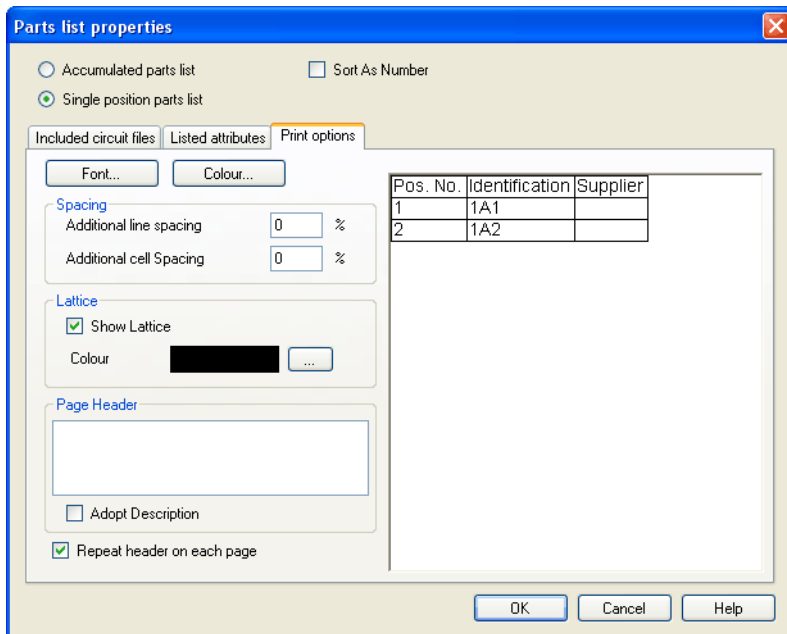


Figure 7/4: Parts List Properties dialog window: Print Options tab

In this tab you can adjust the appearance of the parts list printout. In the preview on the right-hand side you can immediately see the effects of the adjustments. You can find information regarding printing a parts list in the [“Print circuit diagram and parts list”](#) section.

7.4 Export parts list

You can export the parts list as a text file.,

→ Click the **Export...** button in the [parts list view](#).

A dialog window for selecting a file or entering a new file name appears. After you have entered a file and left the dialog field you can define the following format settings for the export in the Parts List Export dialog window:

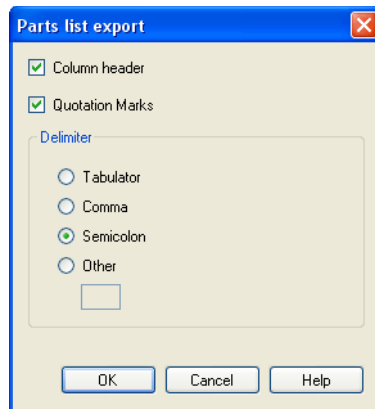


Figure 7/5: Parts List Export dialog window

Column Header

If this option is selected then the names of the attributes appear in the first row of the text file.

Quotation Marks

Select this option to set the field elements in quotation marks.

Delimiter

The selected separator is used to separate the columns.

Manage projects

Chapter 8

FluidDraw supports the management of projects by compiling different files under one name in a single project file. This has the following benefits: When opening a project all of the associated data is loaded. It is possible to rapidly access files belonging to a project via the project window. [Parts lists](#) whose elements are located in different circuit diagram files can be maintained in one project.

8.1 Create new project

→ Select the [New...](#) menu item on the [Project](#) menu and enter a file name for the new project.

Project files possess the `.prj` file suffix. By default they only contain links to files contained in the project. If you wish to pass on a project then you must either pass on all of the associated files along with the project file or activate the [“Save Projects As Single File”](#) option.

In the project window a project is represented as a hierarchical list. The uppermost element is the project node. Below this are the associated circuit nodes and parts list nodes which can be differentiated from each other via the symbol in front of the name.

8.2 Project nodes

Every project possesses one project node as the uppermost element. All of the project-specific properties are saved with the project node. Right-clicking on a project node opens a context menu. You can also find the menu items on the [Project](#) menu. Here you can find the functions for adding and removing files, among other things.

8.2.1 Project attributes

Under the **Properties...** menu item on the **Project** you can set the properties for the project.

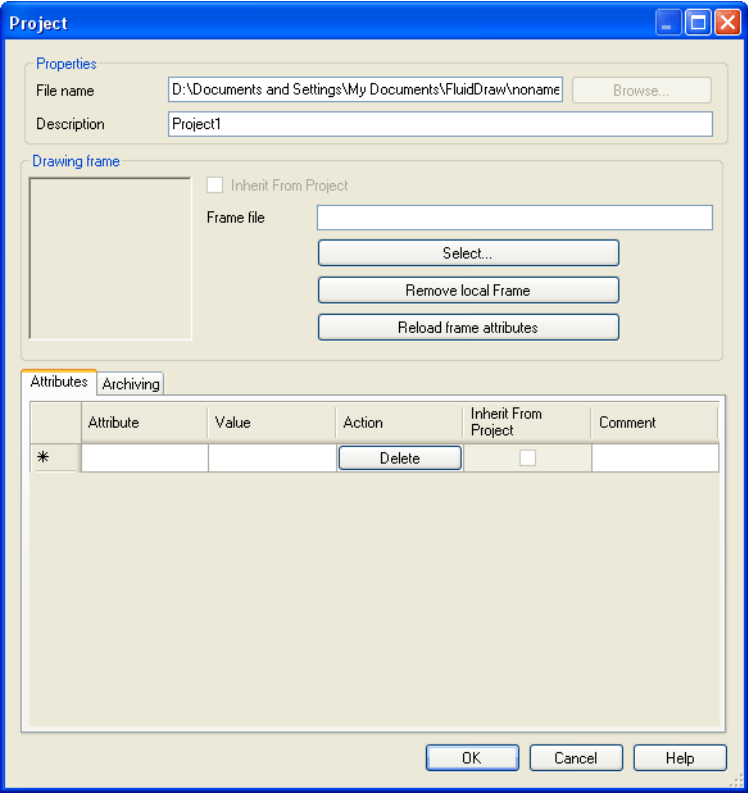


Figure 8/1: Project dialog window: Properties of a project

Properties The file name field displays the name of the project file along with the complete path. You can enter a name for the project in the

description field. This is displayed in the upper margin of the project window and next to the project node.

Drawing Frame

Here you can define the properties of the drawing frame. This is described in the [Drawing frame](#) section.

Attributes

On the Attributes tab in the Project dialog window you can create and edit project-specific attributes such as “Creator” or “Creation date”. The use of attributes is described in the [Attributes of a project node](#) section.

8.2.2 Project archiving

Project files generally consist of links to the files contained therein. However, where desired FluidDraw can also save the project along with all of the associated files in one single file. This serves to simplify passing on the project, among other things. On the Attributes tab activate the Save Projects As Single File option.

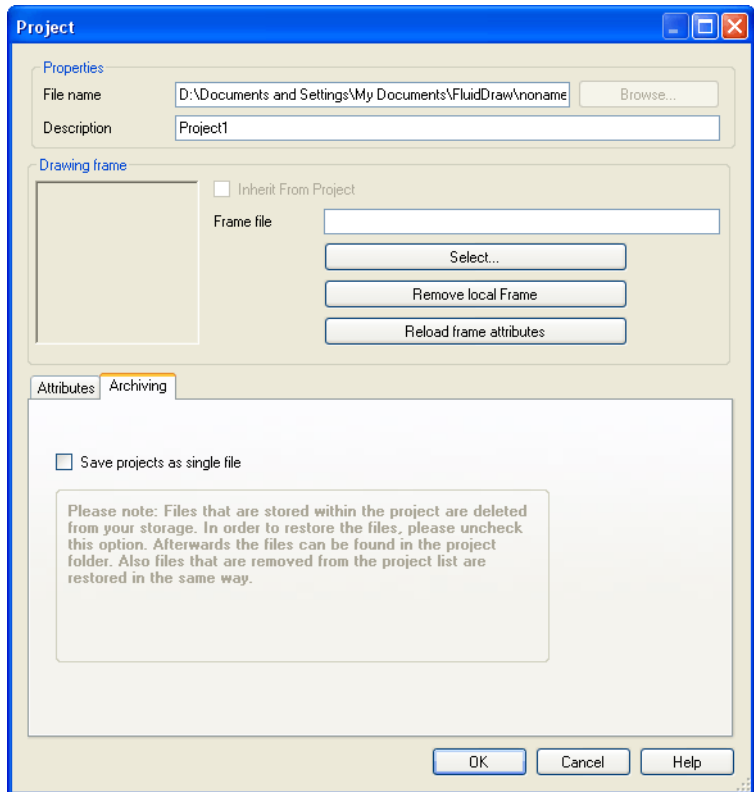


Figure 8/2: Project dialog window: Archiving tab

Save Projects As Single File

If this option is activated all of the files belonging to the project are saved in a single file.



Please note the advice displayed under this option.

8.3 Circuit and parts list nodes

A circuit node is created below the project node for every circuit file which belongs to the project. This also applies to [parts lists](#) as parts lists are saved as circuits. These are only displayed differently in the window. All of the circuit-specific or parts list-specific properties are saved with the circuit node. Right-clicking on a circuit node opens a context menu containing the following menu items:

Open...	Opens the selected circuit or the selected parts list in a window. Double-clicking on the node in the project tree also opens the window.
Close Windows	Closes the window of the selected circuit or the selected parts list.
Remove From List	Removes the selected circuit or the selected parts list from the project.
Rename...	Changes the description of the selected circuit or the selected parts list.
Properties...	In this dialog window you can enter data for the circuit or the parts list. Note: Please note that the additional parts list-specific properties can be set using the window view of the parts list

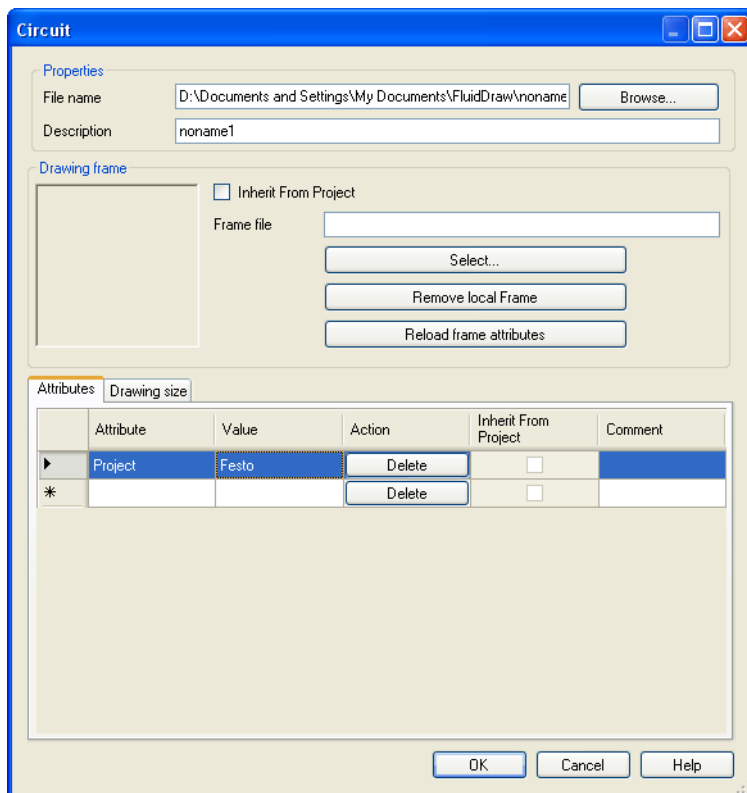


Figure 8/3: Circuit dialog window: Properties of a circuit

Properties

The file name field displays the file name of the circuit along with the complete path. The file name is carried over to the “Description” field and can be edited there. This entry is displayed in the upper margin of the project window, next to the project node.

Drawing Frame

Here you can define the properties of the drawing frame. This is described in the [Drawing frame](#) section.

Attributes

Here you can create and edit circuit-specific attributes such as “Page number” or “Designation”. The use of attributes is described in the [Attributes of a project node](#) section.

Drawing Size

Here you can define the dimensions and alignment of the circuit which are relevant for printing.

8.4 Attributes of the nodes of a project

Any desired number of attributes can be created for every node of a project. The attributes are listed in a table on the “Attributes” tab in the Project or the Circuit dialog window. New attributes can be entered into the empty cells at the end of the table.

The attributes of the project node are automatically transferred to all of the circuit and parts list noted (inherited) and are thus available to all of the circuit diagrams. This concept is particularly useful for the [drawing frames](#) if project attributes should be displayed in the circuit diagram, for example. You can find additional information in the [Link text components with attributes](#) section.

Circuit attributes carried over from a project cannot be edited initially. The corresponding row in the Attributes tab of the Circuit dialog window is greyed out and the “Inherit From Project” option in the column is activated.

However it is possible to over-write a carried-over attribute in a circuit. In order to do so the “Inherit From Project” option in the column must be deactivated. The value of the attribute can then be edited. This can be used to assign individual page numbers to the [drawing frame](#), for example. If the “Inherit From Project” option is reactivated then the attribute values are carried over from the project again.

Attributes		Drawing size			
	Attribute	Value	Action	Inherit From	Comment
	Anzahl Änderungen		Find target...	<input checked="" type="checkbox"/>	
	Änderung Datum		Find target...	<input checked="" type="checkbox"/>	
	Änderung Name		Find target...	<input checked="" type="checkbox"/>	
	Abteilung		Find target...	<input checked="" type="checkbox"/>	
*			Delete	<input type="checkbox"/>	

Figure 8/4: Extract from the Project or the Circuit dialog window – Properties of a node in the project: Attributes tab

Attribute	This column contains the name of the attribute.
Value	This column contains the value of the attribute.
Action	The buttons in this column allow the following actions: If the attribute was created in the super-ordinate project node then the “Find Target...” action is available. Clicking this opens the Project or the Circuit dialog window of the project node as the target which contains the corresponding attribute. If the attribute has been created in the same node then the “Delete” action is available, with which the attribute can be deleted.
Inherit From Project	If the attribute has been created in the same node then this option is deactivated and greyed out. This applies to all of the attributes of the project node as this does not possess a super-ordinate node. For subordinate nodes the value of the attribute from the project node is carried over when this option is activated. After deactivating this option it is possible to locally overwrite the attribute value.
Comment	Here you can enter a comment on an attribute.

Chapter 9

9.1 Print circuit diagram and parts list

FluidDraw circuits and parts lists can be printed by opening the Print dialog window via the [Print...](#) menu item on the [File](#) menu.

In FluidDraw a parts list is the representation of a circuit as a table. If you wish to print an individual parts list then the corresponding circuit must be shown in the [parts list display](#).

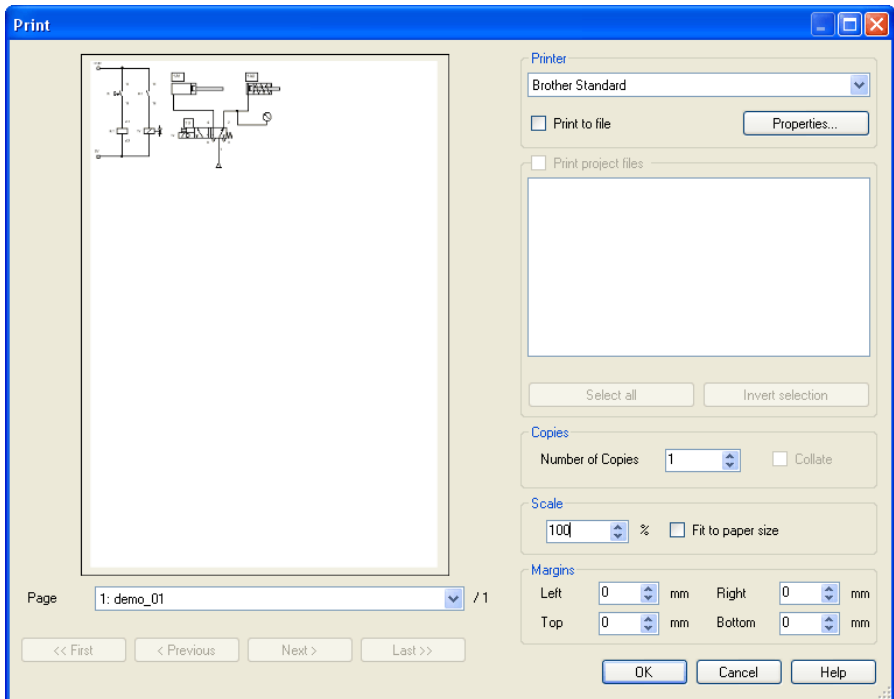


Figure 9/1: Print dialog window:

Printer	Select the desired output device from the list of the available printers.
Properties...	This button enables you to open the dialog for setting the printer options.
Print to File	Activate this option if you wish to write the print data to a file.
Print Project Files	When you work in a project you can select which circuits and parts lists are to be printed.
Copies	

Select the number of copies. If the printout consists of multiple pages you can cause FluidDraw to print a correspondingly sorted sequence.

Scale

You can increase or decrease the output size by entering a scaling factor. If the dimensions of the area to be printed are larger than the printable area on the paper then the printout is spread across multiple pages (tiled). You can see the page divisions in the print preview. Note: Please note that it may not always be possible to precisely comply with the page margins. Therefore you should plan a certain amount of leeway for the page margin.

Page

You can select which pages the print preview displays.

9.2 Import DXF file

Files saved in the DXF format can be imported and retain most of their element attributes. If several special points were taken into account when creating the drawing using a CAD program then FluidDraw can easily expand the symbol library.

9.3 Export circuit

The FluidDraw- circuits can be saved as a file with a variety of different formats. The BMP, JPG, GIF, WMF, PNG, DXF and TIF formats are available. Once you have selected the **Export...** menu item on the **File** menu a dialog window opens in which you can define the different settings for the export. The settings depend on the format selected.

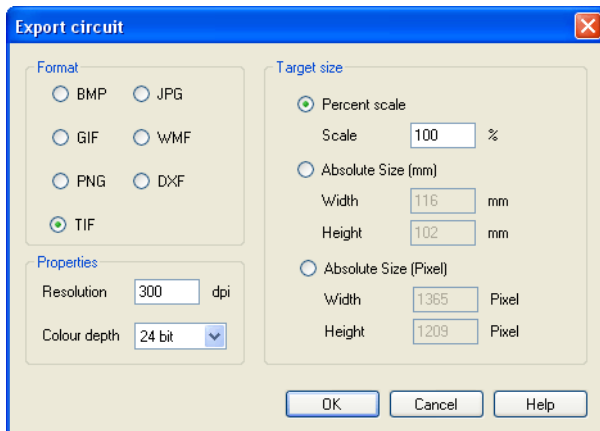


Figure 9/2: Export Circuit dialog window

You are then prompted to select a file name or enter a new one.

Select a suitable resolution for the image file.



Please note that a high resolution generates extremely large files which mean the export may take longer. However you can cancel the image export at any time.

Options

Chapter 10

You can access the various program settings in the Options dialog window via the [Options...](#) menu item on the [Tools](#) menu.

10.1 General

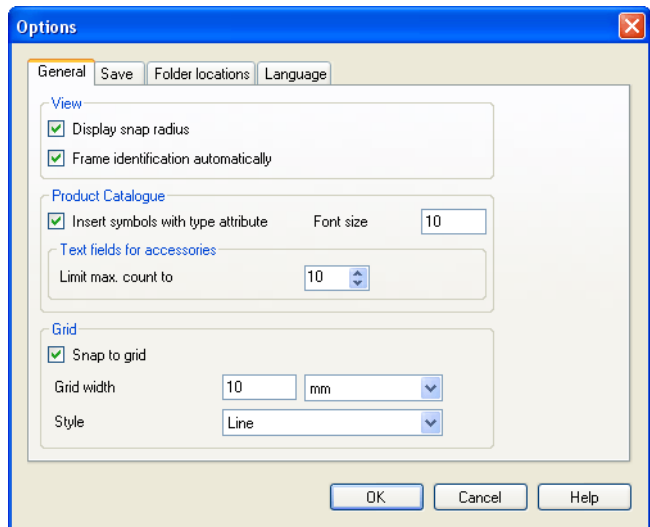


Figure 10/1: Options dialog window: General tab

View

In order to simplify connection of connectors FluidDraw draws a small circle around the connectors on the circuit symbols. Deactivate the Display Snap Radius option in order to disable the display of the snap-in circles.

Activate the Frame Identification Automatically option if labels should automatically be framed. This simplifies the depiction of circuits in accordance with the standards.

Product Catalog	Activating the “Insert Symbols With Type Attribute” option causes a text field to be added with the circuit symbol which contains a link to the attribute with the type designation of the product when components are inserted from the Festo product catalog .
Font size	Sets the font size of the texts generated by the “Insert Symbols With Type Attribute” option.
Text fields for accessories	When inserting accessories without symbols texts with the type designation of the products are displayed. Here you can limit the number of text fields inserted.
Grid	Here you can set the grid width and the style. To display the grid select the Show Grid menu item on the View menu.

10.2 Save

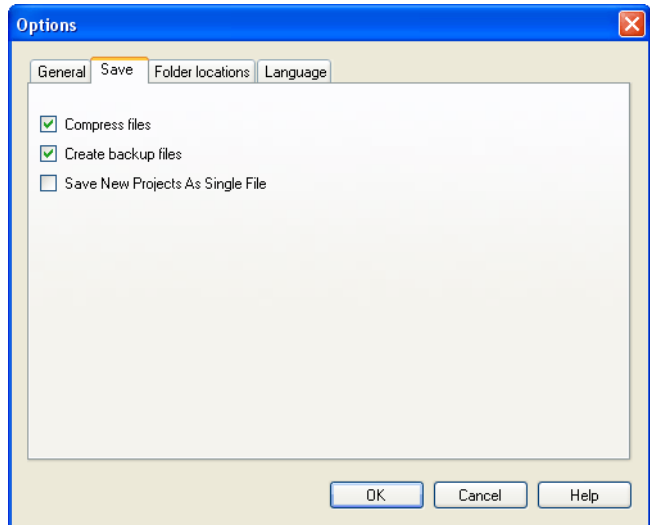


Figure 10/2: Options dialog window: Save tab

Compress Files

FluidDraw normally compresses the saved files in order to save space on the disk. If you deactivate the option the files are saved as XML files without compression. You can find additional information in the [Circuit files](#) section.

Create Backup Files

The “Create Backup Files” option causes the previous versions of saved files are created with the **bak** file suffix. This enables you to restore previous versions.

Save New Projects As Single File

[Project files](#) generally consist of links to the files contained therein. However, where desired FluidDraw can also save the project along with all of the associated circuit files in one single file. This serves to simplify passing on or archiving the project, among other things. You can set whether a project saves links or all of the associated files using the [Project dialog window – Properties of the project](#). This option defines the default setting for new projects.

10.3 Folder Locations

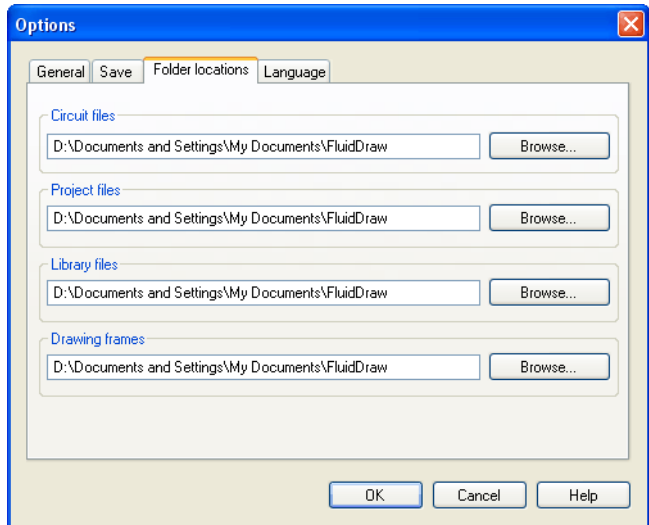


Figure 10/3: Options dialog window: Folder Locations tab

Circuit Files

This is the default path for opening and saving circuits.

Project Files

This is the default path for opening and saving projects.

Library Files

This is the default path for opening and saving libraries.

10.4 Language

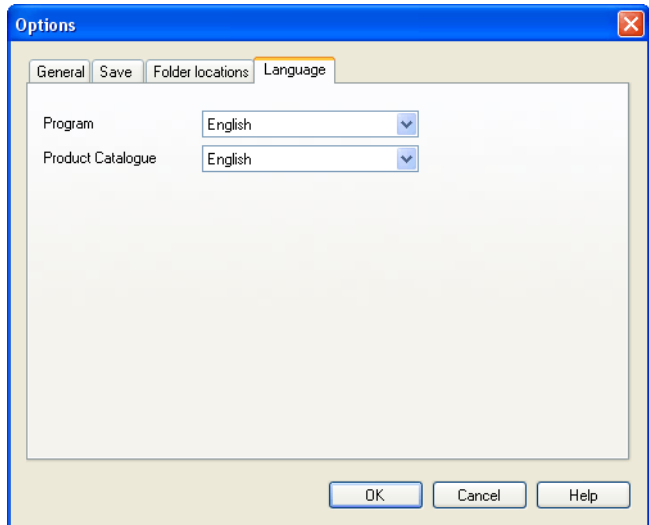


Figure 10/4: Options dialog window: Language tab

Program

Defines the languages for the program interface including dialog windows and messages.

Product Catalog

Defines the language for the [Festo product catalog](#).

Menu overview

Chapter 11

11.1 File

New

Opens an empty window for creating a new [circuit diagram](#).

Open...

Opens the dialog window for selecting a saved file. Files in [DXF format](#) are automatically converted into FluidDraw circuit files.

Close

Closes the active window.

Save

Saves the circuit diagram in the active window to disk.

Save As...

Saves the circuit diagram in the active window to disk under a new name.

DXF Import...

Opens the dialog window for selecting a saved [DXF file](#). The selected file is subsequently converted into a FluidDraw circuit diagram. The original file remains unchanged.

Export...

Exports the circuit diagram in the active window to the data storage medium as a BMP, JPG, GIF, WMF, PNG, DXF or TIF file.

Page Setup...

Sets up the printer properties for a page to be printed.

Print Preview

Opens a print preview as a page view.

Print...

Opens the FluidDraw Print dialog window to set the [print options](#) for the active circuit.

Recent Files

A file can be selected for opening from a list of the recently opened files.

Exit

Ends FluidDraw.

11.2 Edit

Undo

Undoes the last editing action for the active circuit.

Redo

Takes back the last undo-action for the active circuit.

Cut

Deletes the selected objects and adds them to the clipboard.

Copy

Copies the selected objects to the clipboard

Paste

Pastes the objects from the clipboard into the active circuit.

Delete

Deletes the selected objects from the active circuit.

Select All

Selects all of the objects from the active circuit.

Group

Groups the selected objects.

Create Macro Object

Creates a **macro object** out of the selected objects.

Ungroup/Break Off

Ungroups the selected **groups** or **macro objects**.

Enable Scale

Activates or deactivates the “enable scaling” mode which **enables the scaling of symbols** with the help of the mouse pointer.

Enable Rotate

Activates or deactivates the “enable rotate” mode which **enables the rotation of symbols** with the help of the mouse pointer.

Align

Aligns the selected objects with each other.

Rotate

Rotates the selected objects by 90, 180 or 270 degrees anti-clockwise. **Grouped objects** are rotated around the centre point of the group rectangle.

Mirror

Mirrors the selected objects horizontally or vertically. **Grouped objects** are mirrored along the axis of the group rectangle.

Define Connector

Creates a new **component connector** in a circuit symbol with the next left-click.

Delete Connector

Deletes the selected **component connector**.

Properties...

Opens the Properties dialog window for the selected circuit symbols in order to enter the **component properties**. This also enables you to carry over attributes from the **Festo product catalog**.

The Connector dialog window is opened for the selected component connector enabling you to enter the **connector properties**.

When a line segment is selected a dialog window opens enabling you to enter the [line segment properties](#).

If multiple objects are selected then a [dialog window with a limited selection of properties](#) appears. It only contains those properties which affect all of the selected objects.

11.3 View

Circuit Diagram

Switches from the [parts list view](#) to the circuit diagram view.

Parts List

Switches to the [parts list view](#).

Library

Shows or hides the libraries window.

Project

Shows or hides the project file management window.

Standard Size

Displays the circuit diagram in its original size.

Previous View

Displays the previous view of the circuit diagram. Repeatedly activating this function switches back and forth between the views previously used.

Fit To Window

Set the zoom level to fit the entire circuit diagram into the window.

View Detail

Defines the new view by drawing a rectangle while holding down the left mouse button.

Zoom In

Zooms the view of the circuit diagram in by one step. Three steps correspond approximately to doubling the zoom factor.

Zoom Out

Zooms the view of the circuit diagram out by one step. Three steps correspond approximately to halving the zoom factor.

Show Page Borders

Shows or hides the page borders in the form of a red rectangle. This rectangle displays the borders of the paper format defined on the [Paper Size...](#) menu. Using the [print preview](#) you can decide whether and how the drawing is distributed among multiple sheets when printing.

Show Grid

Shows a [background grid](#) in the active circuit window. You can adjust the grid settings on the [Tools](#) menu under the [Options...](#) menu item.

Drawing Layers...

Opens a Drawing Layers dialog window enabling you to set the properties of the [drawing layers](#).

11.4 Circuit

Check Drawing

Checks the active circuit diagram for [drawing errors](#).

Paper Size...

Opens a dialog window enabling you to set the [paper size](#).

Properties...

Opens an dialog window enabling you to set the [circuit properties](#).

11.5 Library

Add New Library...

Creates a new [library](#).

Add Existing Library...

Opens the dialog window enabling you to select a saved library file with the **.lib** file suffix. The library saved in the file is added to the library window.

Add Existing Symbol Folder...

Opens the dialog window for selecting an existing folder. The entire contents of the selected folder, including all of the circuit symbols and sub-folders contained therein are displayed as a library.

Close Current Library

Removes the active library from the list in the library window. The library file is not deleted and can be opened again using the **Library** menu and the **Add Existing Library...** menu item.

Rename Active Library...

Opens a dialog window for entering the name of the library. This function is only available to libraries which the user has created. For write-protected libraries the folder name is shown in the tab.

Sort Current Library Alphabetically

Sorts the contents of the active library alphabetically. This function is only available to libraries which the user has created. Write-protected libraries are automatically sorted and cannot be resorted by the user.

11.6 Draw

This menu contains functions for freely drawing **graphical elements** such as lines, rectangles, circles, etc as well as for inserting texts and images.

Line

Draws a **line** by defining two defined end points.

Rectangle

Draws a **rectangle** by defining two diagonally opposite corner points.

Circle

Draws a [circle](#) by defining a middle point and a radius.

Ellipse

Draws an [ellipse](#) by defining a middle point and a radii on parallel axes.

Text

Inserts a [text](#) at the mouse position.

Picture

Inserts an [image](#) at the mouse position.

Pneumatic Line

Draws a [pneumatic line](#) by defining two end points.

11.7 Insert

Find Symbol Description...

Opens the [search dialog](#) for selecting a symbol via its appearance or its description.

From Festo Catalog...

Opens the [Festo product catalog](#) to select a component via the product features.

From Festo Shopping Basket...

Opens the [Festo product catalog](#) to transfer components from an existing shopping cars.

From File...

Opens the dialog window for selecting an existing text file in CSV format which was created by an export from the Festo product catalog.

11.8 Project

New...

Opens the dialog window for creating a new [project](#).

Open...

Opens the dialog window for selecting an existing [project](#).

Close

Closes the active [project](#).

Save

Saves the active project.

Save as Component Library...

Saves the active project as a library. As a result a new library symbol is generated for every circuit. The new library bears the name of the project with the **lib** file suffix.

Add New Circuit File...

Creates a new circuit file and adds this to the active project.

Add New Parts List...

Creates a new parts list and adds this to the active project.

Add Existing Files...

Opens the dialog window for selecting existing files to be added to the active project.

Add Current Window

Adds the active window to the file list of the open [project](#).

Remove Current Window

Removes the active window from the file list of the open [project](#).

Properties...

Opens the dialog window for entering the [project properties](#).

11.9 Tools

Options...

Opens the dialog window with [program settings](#), [file paths](#) and [language options](#).

Restore Defaults...

Resets the program settings to the default values: This enables you to undo unintentional changes made to the program. Note: Use this

function if you have the impression that FluidDraw is behaving unexpectedly or files and windows have suddenly disappeared.

11.10 Window

New Window

Opens a window with an additional view of the active window.

Navigation Pane

Opens an overview window with a scaled-down view of the entire circuit diagram. The currently visible area of the active window appears as a white area. The non-visible area of the drawing is shaded grey. By drawing a rectangle in the overview window with the mouse pointer you can define the section of the circuit diagram displayed in the active window. Simply left-clicking on the overview window moves the visible area while retaining the zoom level.

Cascade

Arranges the windows so that they overlap.

Tile Horizontally

Arranges the windows into horizontal tiles.

Tile Vertically

Arranges the windows next to each other.

Close All

Closes all open windows.

11.11 Help

Contents

Displays the contents of the FluidDraw help pages.

Index

Displays the index of the FluidDraw help pages.

Search

Displays the search dialog for the FluidDraw help pages.

About...

Displays the FluidDraw program information.

Functional diagram

Chapter 12

You can find the function diagram in the standard FluidDraw library. You can apply the functions from the [Edit](#) menu to the function diagram. Double-click on the function diagram or open the Functional diagram using the [Properties...](#) menu item on the [Edit](#) menu.

The buttons in the upper toolbar of the editor are for editing a function diagram. The following six buttons determine the editing mode:



[Edit Mode](#)



[Draw diagram curves](#)



[Insert signal elements](#)




[Insert text boxes](#)



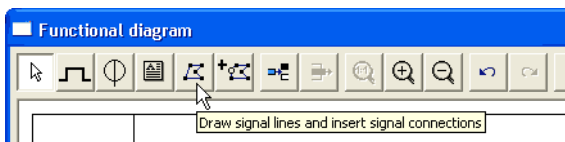
[Draw signal lines and insert signal connections](#)




[Insert additional signal lines](#)


The selected mode is highlighted in white or displayed as a pressed button.  shows that signal lines can be drawn in the diagram area by clicking, for example.

If you hover the mouse pointer over a button for a short period of time then a short description is displayed.



12.1 Edit Mode

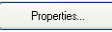
 This mode serves to adjust the objects in the function diagram. Elements in the diagram can be moved. Changing the size of text boxes is only possible in this mode.

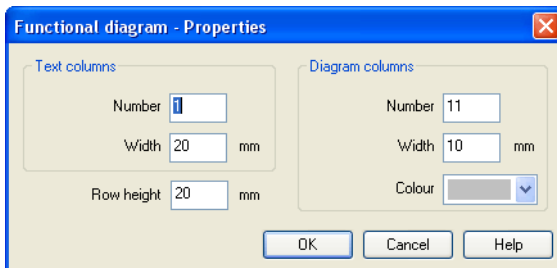
Movement and dragging operations can be cancelled using the  **Esc** key.

If the mouse pointer is moved outside the window area while the left mouse button is held down the view automatically scrolls.

Double-clicking on a diagram element (diagram row, text box, signaling element, etc.) opens a dialog window in which you can make the desired adjustments.

12.1.1 Set diagram properties

Clicking on the  button opens a dialog window in which you can set the diagram properties.



Text columns – Number

If the number of text columns is changed than all of the table text boxes are distributed horizontally and evenly.

Text columns – Width

If the width of the text columns is changed than all of the table text boxes are distributed horizontally and evenly.

Diagram columns – Number

The diagram columns are located on the right-hand side of the function diagram. The diagram curves can be drawn in this area. The number of diagram columns can also be changed by dragging the mouse pointer to the right-hand edge of the diagram.

Diagram columns – Width

Color

The color which the grid lines are drawn in the diagram area.


Row height

Defines the row height of all of the rows.

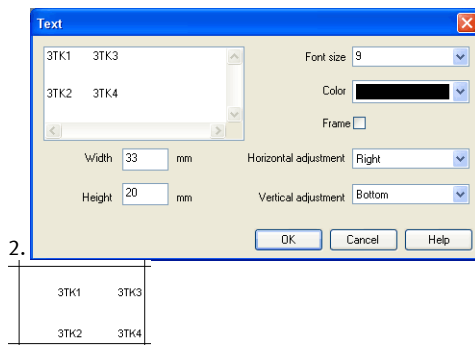
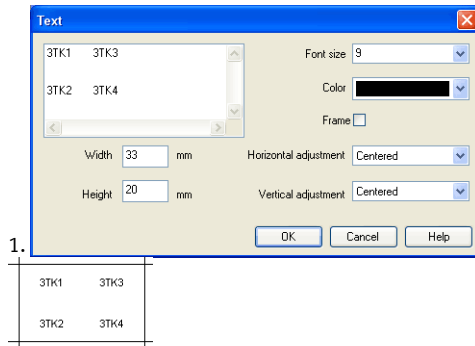
12.1.2 Table textboxes

The table textboxes are located on the left-hand side of the function diagram.

Description	Designation	Function	State	Sc
				3
				2
				1

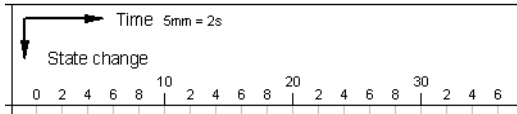
- Font size The font size of the text to be shown.
- Color A selection from sixteen standard colors for the text to be displayed.
- Width The width of the selected table column can be changed by dragging the mouse pointer.
- Height The height of the selected table column can be changed by dragging the mouse pointer.
- Horizontal adjustment The following alignment options are available: “Left” “Center” and “Right”.
- Vertical adjustment The following alignment options are available: “Top”, “Center” and “Bottom”
- Alignment within a table cell If a text within a textbox should resemble a table then set a tab stop between the text parts. The text is displayed in the textbox in accordance with the number of tab stops and the specified horizontal and vertical alignment. To enter tab stops hold down the  key.

Examples:



12.1.3 Adjust the presentation of the diagrams

The area in which curves can be drawn is located on the right-hand side of a diagram row.

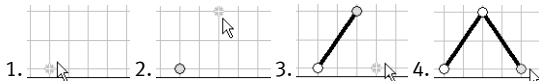


States – Number	The entry defines the number of states and thus the number of horizontal lines in the diagram row.
States – Base state	Horizontal lines are drawn through the base state with a thin pencil.
Numbering – Start column	The start column defines which column the numbering should begin with.
Numbering – Start number	The start number defines which number the numbering should begin with.
Numbering – Number	The number specified who many steps need to be numbered.
Numbering – Step width	Defines the step width between two numbers.
Numbering – Loop	If this field is selected then an equals sign and the start number are always displayed after the last number.
Representation – Display arrows	If this field is selected then two arrows are displayed.
Representation – Display grid	If this field is selected then the background grid is displayed.
Representation – Display text 1	If this field is selected then a textbox is shown which can serve as a label. This textbox belongs to the selected row and cannot be moved to another row.
Representation – Display text 2	If this field is selected then an additional textbox is shown which can serve as a label. This textbox belongs to the selected row and cannot be moved to another row.
Representation – Line color	Sets the color of the diagram lines.

12.2 Draw diagram curve



In this mode diagram curves can be drawn. Supporting points can only be inserted on the grid. A supporting point is drawn at every left-click.



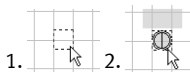
If you hold down the left mouse button then supporting points can be moved as in the selection mode.

Selected supporting points are shown in grey. Use the **Del** key to delete a selected symbol.

12.3 Insert signal elements



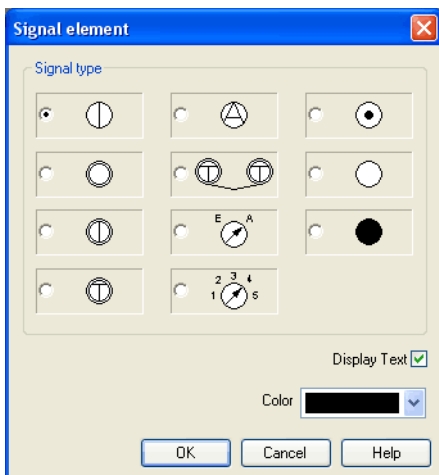
In this mode signaling elements can be inserted by left-clicking.



If you hold down the left mouse button then signaling elements can be moved as in the selection mode.

Selected signaling elements are shown in grey. Use the **Del** key to delete a selected signaling element.

Double-clicking on a signaling element in the selection mode opens a dialog window in which you can modify how the signaling element is shown.



Signal type


Display description

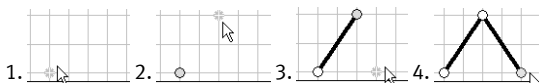
If this field is selected then a text is shown which can serve as a label for the signaling element.

Color

The signaling element is displayed in this color.

12.4 Insert text boxes

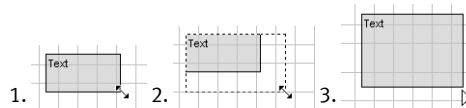
 In this mode textboxes can be inserted by left-clicking.



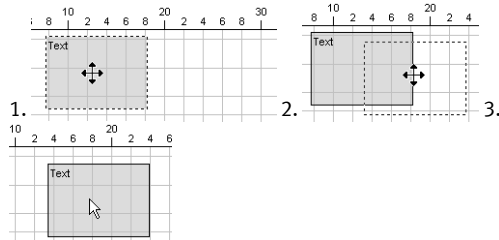
Selected textboxes are shown in grey. Use the **Del** key to delete a selected textbox.

In “Edit Mode” the size and position of a textbox can be changed using the mouse pointer.

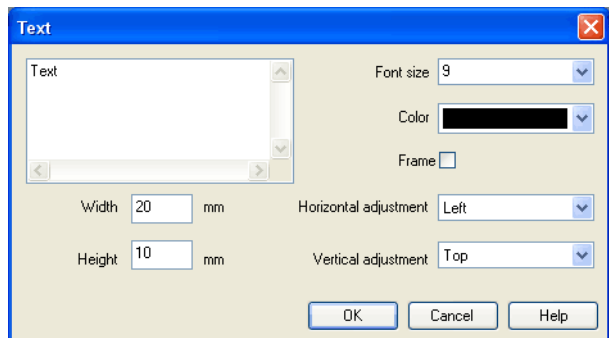
Adjust size:



Move textbox:



Double-clicking on a textbox in the selection mode opens a dialog window in which you can modify how the textbox is shown.




Font size




The font size of the text to be shown.


Color	A selection from sixteen standard colors for the text to be displayed.
Frame	If this field is selected then a frame is drawn around the textbox.
Width	The width of the textbox.
Height	The height of the textbox.
Horizontal adjustment	The following alignment options are available: “Left”, “Center” and “Right”.
Vertical adjustment	The following alignment options are available: “Top”, “Center” and “Bottom”

12.5 Draw signal lines and insert signal connections

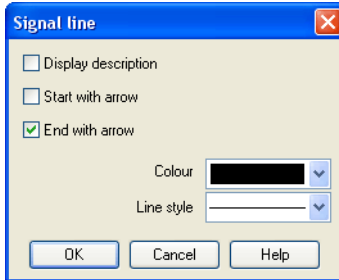
 In this mode signal lines can be drawn freely or automatically laid from signaling elements.

12.5.1 Free-draw signal lines

An additional supporting point is set with every left-click. The process ends when you click the current line while holding down the  key, press the  key or switch the editing mode. Use the  key to delete a selected signal line. If only a supporting point is selected then only this is removed from the line.

In “Edit Mode” the supporting points belonging to the signal lines can be moved. If the  is held down while setting or moving supporting points then the supporting point is vertically or horizontally aligned.

In the selection mode double-clicking can change the way the line is displayed.



Display description

A framed label on the line and a label next to the framed label is shown. The framed label can be moved freely along the line. The additional label can be moved freely.

Start with arrow


An arrow is displayed at the beginning of the line. The arrow can be moved freely along the line.

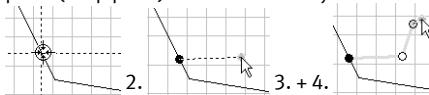
End with arrow

An arrow is displayed at the end of the line. The arrow can be moved freely along the line.

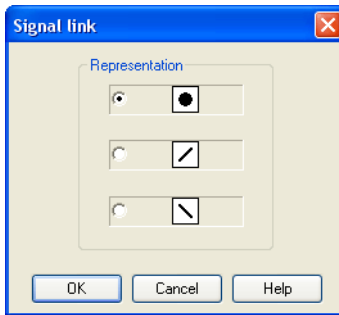
Color

Line color


If you click on a finished signal line to activate the “Freely move signal lines” mode  then a signal logic is inserted. This logic point (snap point) can then be freely moved along the line 1.

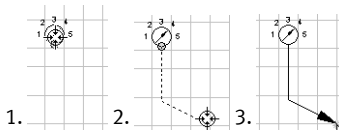


In the selection mode you can change how the signal logic is displayed by double-clicking on it.




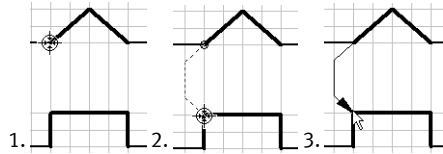
12.5.2 Drawing signal lines from signals

In the “Freely draw signal lines”  signal lines can be drawn from signals. Left-click on a signal and hold down the mouse button. Drag the mouse pointer to a position which you have selected as the end point of a signal line. The signal line will be drawn once you release the mouse button.




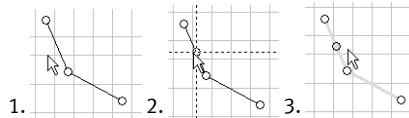
12.5.3 Draw signal lines from diagram support points

In the “Freely draw signal lines” mode  signal lines can be drawn from curve support points. Left-click on a supporting point and hold down the mouse button. Drag the mouse pointer to a second supporting point. The signal line will be drawn once you have the mouse pointer over the second supporting point and release the mouse button.




12.6 Insert additional signal lines

In the “Insert additional signal lines” mode  additional supporting points can be inserted into existing signal lines.



12.7 Insert row


Clicking on the “Insert row” button  inserts a new diagram row above the current selection. If no cell is marked then a new row is attached to the end of the diagram.


12.8 Delete row


Clicking on the “Delete row” button  deletes a selected diagram row. If nothing is selected then the function is not available.

12.9 Additional editing functions


12.9.1 Zoom


Using the  button the view is reset to the standard zoom level.

Using the  button increases the zoom level.

Using the  button decreases the zoom level.

12.9.2 Undo editing steps

Using the  button you can undo the last 50 editing steps.

Using the  button you can redo the previously undone editing steps.

Index

?

? Menu	105
--------------	-----

A

Accessories	89
Alignment	33
Alignment lines	43
Archiving	
project	79
Attribute link	64

B

Backup copies	91
Blanking plugs	27
BMP	
export	86

C

Check	
drawing	56
Circle	51
Circuit	15
check	56
input	84
nodes	80
open	80
output	84
print	84
properties	80
XML file	15
Circuit menu	99
Component attributes	57
Compression	
files	91
Configurable cylinder	30
Configurable way valves	28
connector	
define	28
Connector	
connect	22

delete	28
identification	27
properties	27
seal	27
Create drawing frame	37
Cylinder	
configure	31
Cylinder editor	31

D

Delete	
line	26
Dialog	
circle	51
Circuit	7, 81
component	58
connector	27
diagram properties	108
diagram size	7
drawing areas	111
drawing frame	39, 41
drawing layers	45
ellipse	53
export parts list	75
Functional diagram	107
image	55
line	26, 49
line options	116
pneumatic line	26
print	85
project	77
rectangle	50
signal logic	117
signaling elements	113
text	62
text options	109
Drawing element	47
Drawing frame	37, 38
import	37
insert	38
label	37

Drawing level	44
Drawing menu	100
DXF	
export	86
DXF file	
import	86
E	
Edit	
circuit	18
Edit menu	94
Ellipse	52
Export	86
F	
Festo product catalog	18
File menu	93
Find target	69
G	
GIF	
export	86
Graphical element	47
Grid	43
Grid dimension	43
Grid options	90
Group	32
create	32
delete	33
Guides	43
H	
Hide	
automatic	16
Hide window	16
I	
Image	54
Image file	54
Insert	
Symbol	10

Insert menu	101
Insert T-connector	23

J

JPG	
export	86

L

Layer	44
Libraries	10
Library	13, 16
Library menu	99
Library window	16
Line	48
delete	26
lay	24
move	25
pneumatic	48
Line segment	25
Line type	
set	25
Link	64

M

Macro object	
create	32
delete	33
Menu	
circuit	99
drawing	100
edit	94
file	93
insert	101
library	99
project	102
tools	103
view	97
window	104
Menu. ?	105
Mirror	33, 35

O

Options	
Folder Locations	91

General	88
Grid	90
Language	92
Save	90
text fields	89

P

Parts list	69
display	69
edit	69
export	74
find components	69
find target	69
nodes	80
open	80
print	84
properties	71
sort	69
Pilot line	25
Pneumatic line	48
PNG	
export	86
Port	
connect	22
Print	84
Product catalog	18
Project	16, 76
add	76
nodes	76
print	84
properties	76
remove	76
Project archiving	79
Project menu	102
Project window	16
Properties dialog	7
circle	51
circuit	81
component	58
connector	27
cylinder	31
drawing layers	45
ellipse	53
image	55

line	26, 49
pneumatic line	26
project	77
rectangle	50
text	62
way valve	29
R	
Rectangle	50
Rotate	34
S	
Scale	35
Search	
Symbol	12
Show	
automatic	16
Show window	16
Silencers	27
Storage locations	91
Symbol	
align	33
group	32
insert	18
mirror	33, 35
move	18
rotate	34
scale	35
Symbol library	13
Symbols	10
T	
Text	54
Text attribute	64
Text components	62
Text link	64
TIF	
export	86
Tiles	
Printout	7
Toolbar	
drawing	47

Tools menu _____ 103

V

Valve editor _____ 29

View menu _____ 97

W

Way valve
 configure _____ 29

Window menu _____ 104

WMF
 export _____ 86

Working line _____ 25

X

XML file _____ 15