Making sense of your networks
Making business sense

AND
Advanced Network Design

LibEdit
UserManual

www.and-solution.com

Advanced Network Design

User Manual

AND LibEdit

Version: IV/C

Author: various

Document name: and_libedit_en.pdf

This manual contains 199 pages

Date of publication: 2018-03-26

AND Solution GmbH
Karl-Schmid-Strasse 14, 81829 Munich, Germany
Tel. +49 (89) 743533-0
Fax. +49 (89) 743533-999

Contents

1 General information

The AND Editor is an add-on for the AND Planning Program.
It is used both for editing and modifying the components and for creating new libraries.
It enables you to modify technical data and the shape of component symbols as desired,
to create and maintain cables, connectors, costing objects, amplifier objects.
Components can be moved or copied by drag and drop.
Symbols can be displayed in multiple forms and sizes (frames).
This data is automatically converted on importing from libraries of the DOS version.
The component editor is offered in two versions:

- as a pure component editor
- as an extended component editor with integrated calculation functionality for amplifiers (ARD - Amplifier Raster Design)

Using the ARD program, you can calculate the entire CSO/CTB spectrum of coaxial CATV amplifiers for any frequency raster because of the fewer CSO/CTB measured values.
It is immaterial which rasters the measured values are available for.
As an alternative to CSO/CTB measured values, a series of KMA/IMA data items,
e.g. from a DIN measurement station, can be used instead.

It is possible to simulate or calculate the change in the CSO/CTB spectrum of the amplifiers for preemphasis or gain ripple (input and output level alignment).
The measurement station no longer has to be calibrated to a linear amplifier output level to be able to determine standard parameters.

With ARD, you can perform raster-independent comparisons of different CSO/CTB measurement series. The software also helps you define the frequencies to be measured for the CSO/CTB spectrum in the case of multi-channel measurement.
Of course, you can also document the calculated data and store it in the library.

1.1 Target Group of this Manual / Required Skills

The target group of this manual is user of the software LibEdit.
It is assumed that this target group has knowledge of the use of personal computers and Windows operating system. Knowledge of using CAD-software is not required.

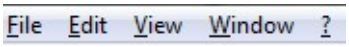
1.2 Starting the AND component editor

When you start the AND Component Editor an application window opens.
You can vary the size of the window.
Using the pull-down menu, you can make several basic settings, create a new library and/or open an existing library.

1.2.1 The Toolbar

Toolbar file operations		Shortcut	Function
New		Ctrl+N	Creates a new library; any open libraries remain open.
Open		Ctrl+O	Loads and opens a library from a data medium.
Save		Ctrl+S	Saves the current library. If you have not yet assigned a name to the library you are now prompted to do so ("Save As" function).
Cut		Ctrl+X	Cuts the selected object to the clipboard.
Copy		Ctrl+C	Copies the selected object to the clipboard.
Paste block		Ctrl+V	Inserts an object from the clipboard.
Print		Ctrl+P	Prints out all created objects.
Print characteristic curve/spectrum			Prints out the characteristic curve/spectrum of any amplifier instances that have also been created.
About the Component Editor			Displays program information, version number and copyright.

1.2.2 The menu bar



1.2.2.1 File

Menu Item	Shortcut	Function
New	Ctrl+N	Creates a new library; any open libraries remain open.
Open	Ctrl+O	Loads and opens a library from a data medium.
Close		Closes the currently active library.
Save as		Saves the currently active library in the set directory path.
Saving options		Allows you to select save options. For example, this is where you can activate write and/or read protection.
Excel-Export		The objects selected in the dialog box are exported to an Excel list.
XML-Export		The library is exported to an XML file.
Print	Ctrl+P	A library can be printed out according to a selected layout.
Page view		The library is displayed the print preview in the selected lay-out.
Set printers		This is where you set a local printer or select a network printer.
Change directory		Sets the storage path.
Exit		Closes the component editor.

1.2.2.2 Edit

Menu Item	Shortcut	Function
Cut	Ctrl+X	Removes the marked object and copies it to the clipboard.
Copy	Ctrl+C	Copies the marked object and transfers it to the clipboard.
Insert	Ctrl+V	Inserts the content of the clipboard.
Search	Ctrl+F	Search for an object in the library.
Edit basic properties		Edit the basic properties of the basic data.
Edit objekt-attributes		Edit the object attributes.
Supplier		Edit the supplier data.
Mounting groups		Create and edit mounting groups.
Lokal references		Edit local references.
Color codes		Edit color codes.
Eldanorm-comparison		Price information is adjusted by comparison with an Eldanorm file.
Basic language		Change basic language of the basic data.
Raster		Edit raster list.
Set version		Set version number for all objects of the library.

1.2.2.3 View

Menu Item	Shortcut	Function
Symbols Status View		Show and hide toolbars.
Change language		Change menu language.
Global references		Displays all cables, connectors and optical references.

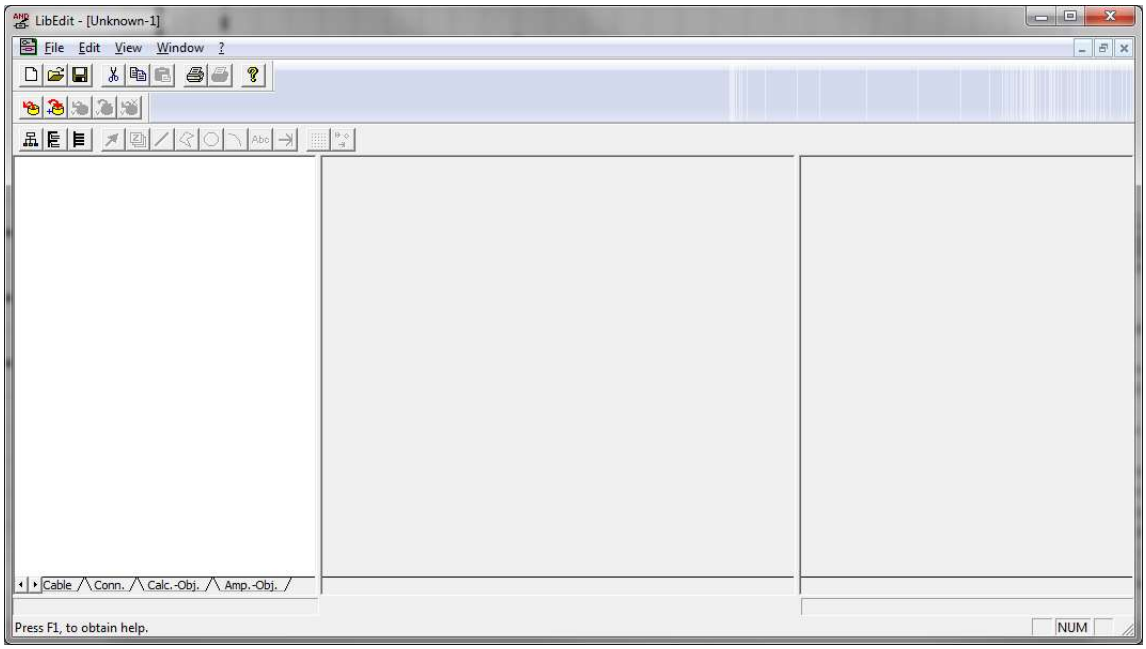
1.2.2.4 Window

Menu Item	Shortcut	Function
Cascades Tiles		Cascades or tiles open windows. All open libraries are listed in the lower screen section. The active library is indicated by a check mark.

In addition, the last 4 libraries to have been opened are listed under menu item “File”.

1.3 Screen layout

When you load an existing library or create a new library, a window with three sections opens. You can set the width of each of the sections individually by moving the dividing bars with the mouse.



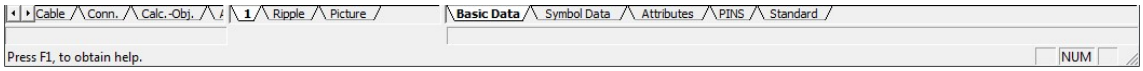
You can open multiple libraries at once, for example, to copy components from one library to another.
A separate window with its own toolbar opens for each library

.

With this toolbar, you can activate additional functions that apply to a particular library. This toolbar is not to be confused with that containing the standard functions (New, Open, Save).

Within each section you can activate context menus with the right mouse button, for example, to create new objects, frames or data.
For further information, please turn to the descriptions of the individual sections.

To save space, several windows are placed on top of the other in each section.
Each window has an activation button, which has the appearance of a index card tab.
A window can be brought to the foreground by clicking on the tab with the mouse.



1.4 Left-hand section

The objects contained in the library are displayed in the left-hand section.
Objects can be combined into groups much like files can be grouped into directories.
However, unlike files and directories, it is not possible to combine several groups to form a higher-level group.

You can expand these groups with this button  .

To collapse expanded groups, click this button  .

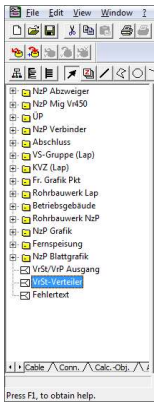
Objects can be displayed in two different ways:

- According to object type
- According to user group

According to object type

We distinguish between five different object types:

- Symbols
- Cables
- Connectors
- Costing objects
- Amplifier objects (not to be confused with the amplifier symbols)

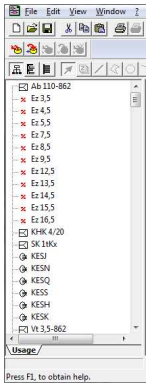


According to user group

Objects of different types that are, however, used in a related context
(for example, all amplifiers, cables and connectors used for network level 4) can be combined to form user groups. In this case, the "tab" is superfluous but is kept to maintain the familiar display.

Note: The assignment of an element to a group is not altered when you switch user group.

The object type is indicated by the icon used to display them.



You can switch between two types by clicking on the relevant button in the toolbar.

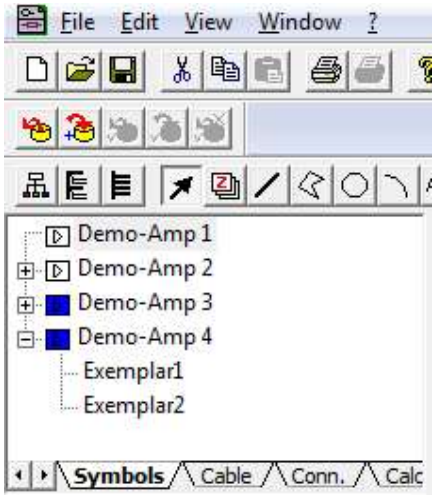
If you click the  button,

the display is switched over and you can now see all library objects at once.
You can reverse to the previous display by clicking on the button again.
You can create or delete objects individually or within a particular group by means of the context menu that you activate with the right mouse button.

ARD:

In ARD, the amplifier symbols have a special function.
If measured data exist for an amplifier, a "+" box appears to the left of the amplifier.
When you start the program, no measured data is loaded and the amplifier symbol appears white. When you double-click the amplifier or single-click the "+" box,
the measured data is loaded (search for the relevant data, if necessary).

The amplifier symbol is now blue.
If you expand the amplifier, all instances of the amplifier are shown as its "children".
The following figure shows all possible states of an amplifier:



Amplifier without measured data

Measured data exist but are not loaded

Amplifier with measurement data loaded but not expanded

Amplifier with loaded measured and expanded measured data

The measured data is stored in files with the extension *.amp. A file exists for each amplifier.
The name of the file usually follows the convention *amplifiername.amp*.

The working directory of the program indicates where the measured data is stored and where to search for the measured data to load it.
If you do not find the measured data while trying to load it, you can search for the measured data file in a dialog box.
You can set the working directory path with menu item “File / Change directory”.
You can import and export the measured data of an amplifier with the context menu (right mouse button).
You can also use the import function to add measured data to the existing measured data.
You also create a new instance using the context menu.

1.4.1 Copying, moving, and deleting components

Entire components can be duplicated either by drag and drop or by copy and paste.
Drag and drop means grabbing the component (in the left section of the screen) with the mouse and moving it to another part of the display ("dragging" it) and then releasing it with the mouse button ("dropping" it). Either a copy is made at the new position and the original is kept or the original is deleted and generated at the new position, i.e. it is moved.

The standard drag and drop operation moves an object within a library and copies it outside a library.

You can influence the drag and drop operation by additionally pressing the

Alt or the Strg key.

The Alt key forces the components to be moved, the Strg key,

on the other hand, forces copying.

If it is not possible to "drop" an object onto the current position of the mouse, this is indicated by the mouse pointer.

Besides mouse-based copying, it is also possible to move components in and out of the clipboard using the copy and paste function.
This is done by selecting cut, copy and paste from the menu, by clicking the relevant buttons in the toolbar or using shortcuts

Strg + X for cutting,
Strg + C for copying, and
Strg + V for inserting or "pasting"

Note: Neither of these procedures allows you to place components in windows not associated with their type, for example, creating symbols in a window for cables.

ARD:

Drag and drop cannot be used for instances of amplifiers.

However, examples can be copied to the clipboard and the copy then inserted at another position, using the Edit menu or using shortcuts with

Strg + C or Strg + V.

Components and instances are deleted as follows:

Choose an object in the left section and either select the command

"Delete" from the context menu or press the Del key directly.

You can also delete an entire group.

1.4.2 Recovering data

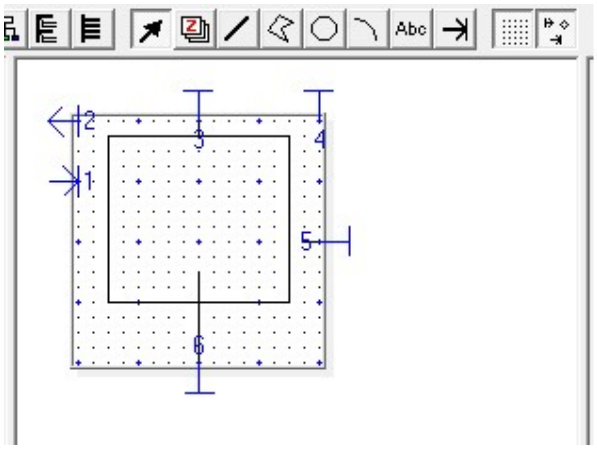
If you mistakenly delete an object, you can recover the data by exiting the library without saving. The library then returns to the state it was in when you last saved it.
You can restore deleted measured data and rasters by the same method.
Back-up copies with the extension *.bak are created when you save libraries, measured data and rasters.
You can also restore the status prior to the last time you saved by loading the relevant *.bak library.
To return measured data to the status prior to the last time you saved, import the *.bak file of the amplifier in question from the working directory (measured data are imported by means of the context menu item "Import Measured Data").

The old imported data are appended to the existing data.
All instances are now duplicated.
Delete the version you do not require from each duplicated instance.
Any changes to the rasters are ignored if you exit the library or the program without saving.
A full list of the rasters is stored in the file *Raster.rst*.
The file *Raster.bak* contains the status of the raster list before your last changes.
In order to restore the old raster status you must overwrite the file *Raster.rst* with *Raster.bak*.

1.5 Center section

This section is activated whenever an existing symbol or cable is selected in the left section or when a new symbol or cable is created.

Symbol drawing



This is where the symbols are drawn.

1.5.1 Drawing symbols

You can draw lines, semicircles, circles and polygons using this toolbar.

As soon as you have completed a drawing action you can move, modify or delete the element you have just drawn.

The frame defines the size of the symbol and can be set individually for each drawing.

The frame cannot be made smaller than the symbol itself.

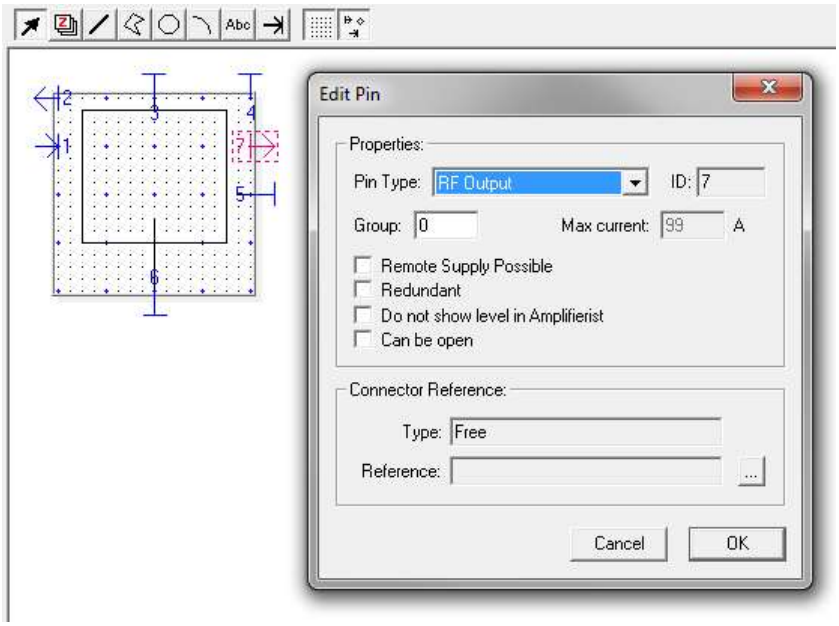
Using this  button you can define connections (pins).

Connections can only be placed on the highlighted grid points.

You can select a connection type by clicking the pin.

The connection is marked red. It is currently not possible to select more than one pin.

Double-clicking on a connection opens the following dialog box with data about this connection:



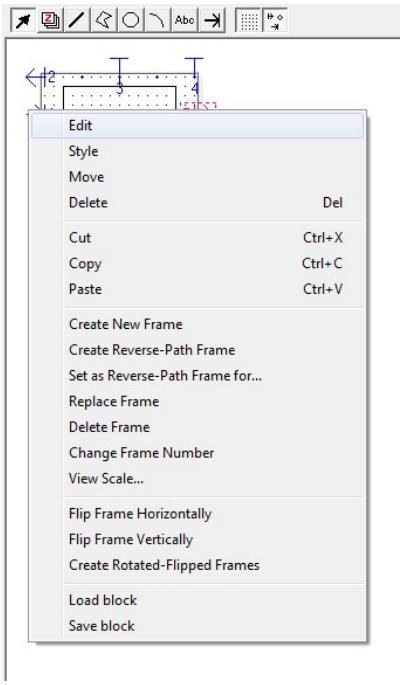
It is possible to activate and deactivate these pins via the button  .
This function does not affect calculation and display in AND.

Using the  button, you can assign text to the symbol, independently of the grid.

If surfaces have been drawn, you can define the sequence of the levels with the  button.

Clicking the  button hides the grid

1.5.2 Context menu



A context menu opens when you press the right mouse button (mouse pointer must be positioned on the center section).

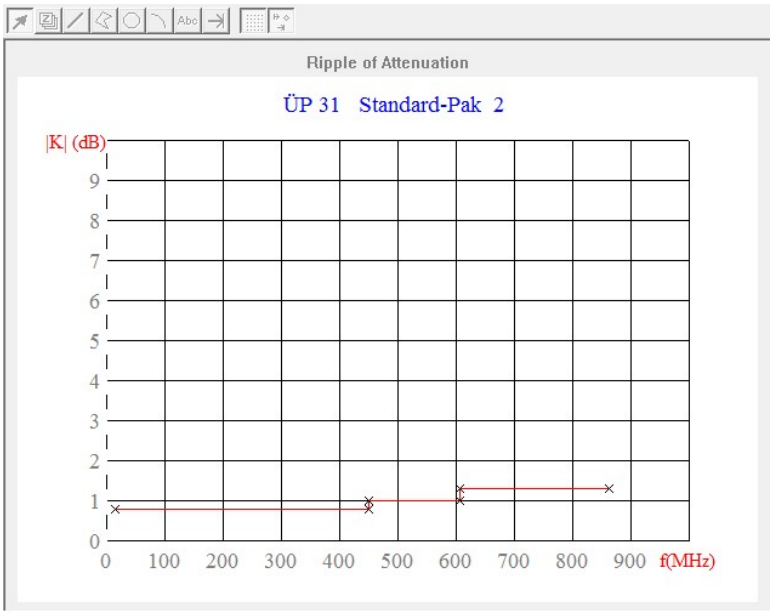
You can now:

- Edit pins
- Change line styles
- Move selected drawing objects
- Execute the functions cut, copy, paste
- Create new frames, delete frames, generate a reverse-path frame or set a reverse-path frame.
- As it is possible to create several drawings for each symbol, you can implement different forms of representation for one and the same symbol, for example, for different drawing hierarchies.
  However, only the first drawing can ever be written to the AND-DOS-compatible part.
- Scale the view
- Load a block
- Save a block.

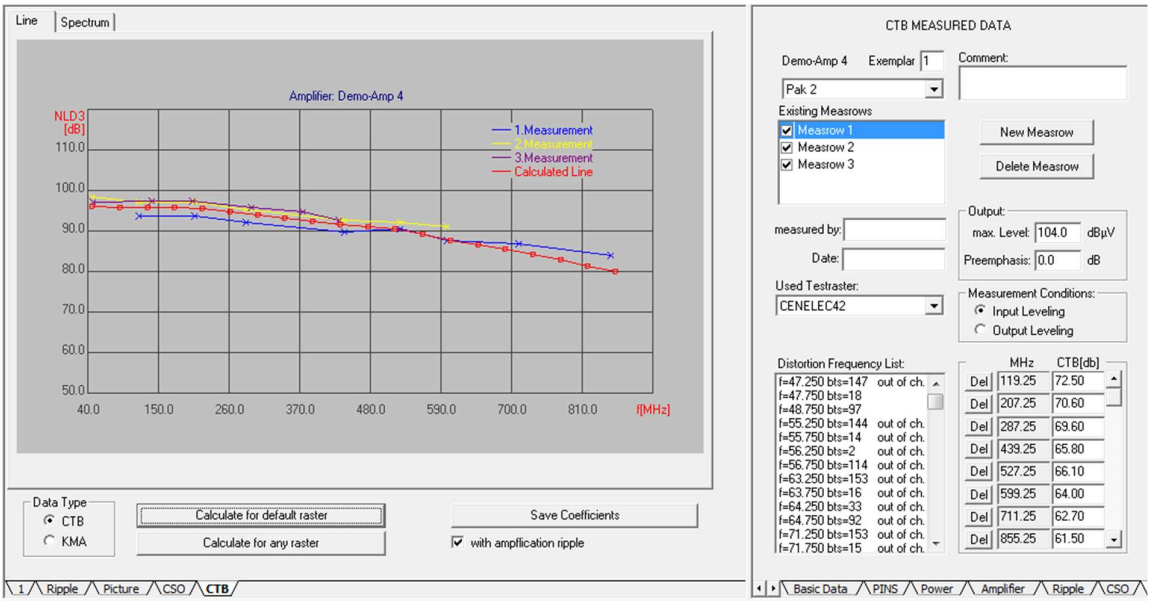
You will find more detailed information in the description of drawing functions.

1.5.3 Frequency slope display window

This tab is used to graphically display the data for the frequency slope of the attenuation and gain from the right-hand section.
It appears for cables and symbols that could have frequency slope data.
For symbols, the interpolation between the measuring points is linear, for cables, the interpolation algorithm uses theoretical knowledge about the attenuation curve of a coaxial cable.

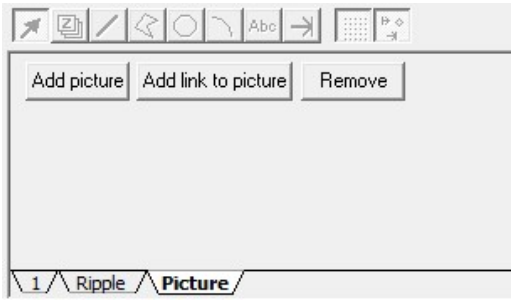


1.5.4 CSO, CTB calculation window



These windows only occur in the ARD version.
For symbols representing amplifiers with loaded measured data or an additional instance of the amplifier, two additional tabs labeled "CSO" and "CTB" appear for the calculation of the characteristic curve and the noise spectrum from the measured data in the right-hand section. Please turn to the relevant chapters for more detailed information.

1.5.5 Inserting an image



With this function you can assign an image, assign a link or delete an image already stored.

1.5.6 Overview of ARD

With the program "*Amplifier Raster Design*" (ARD) you can calculate the entire CSO/CTB spectrum of coaxial CATV amplifiers for any frequency raster in cases where you have fewer CSO/CTB measured values.
It is irrelevant to which raster the measured values apply.
Instead of CSO/CTB measured values, you can also use a series of KMA/IMA data items, for example, from a DIN measurement station.

It is possible to simulate or calculate the change in the CSO/CTB spectrum of the amplifiers during preemphasis or gain ripple (input or output level alignment), i.e., the measurement station no longer has to be calibrated to a linear amplifier output level to be able to determine standard parameters.

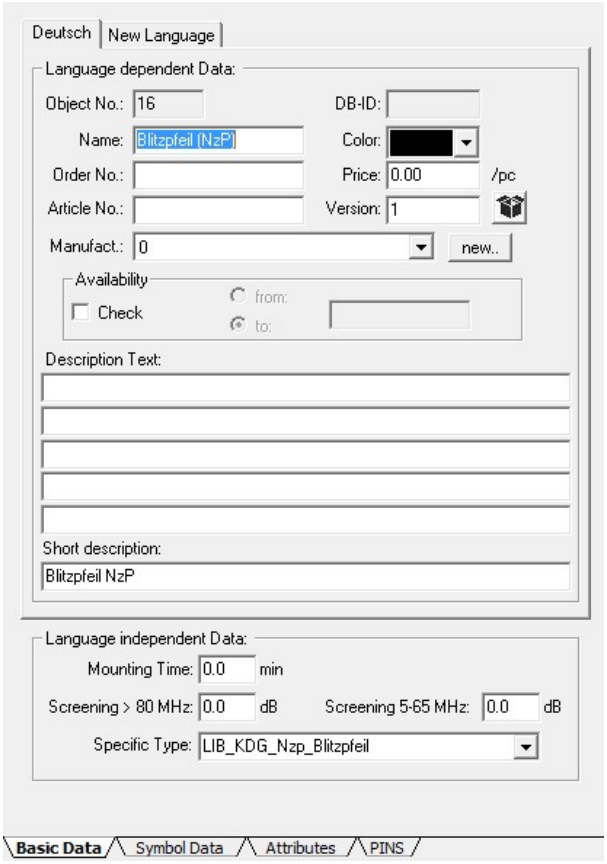
With "Amplifier Raster Design" you can compare different CSO/CTB measurement series of a coaxial CATV amplifier.
The software also helps you to define frequencies to be measured for the CSO/CTB spectrum in multiple channel measurement.
Of course, you can also document the calculated data and store it in the library.

Who is "Amplifier Raster Design" intended for?
CATV developers and operators of a multiple-channel measurement station whose efficiency will increase manifold through the use of ARD.
DIN 45004 measurement station users can calculate the CTB and CSO spectrum for CENELEC and any other rasters from a KMA/IMA data series.

1.5.7 Overview of NRD

Another program, the "*Network Raster Design*" (NRD) , is available to CATV developers, network planners and operators.
This program permits the cascade calculation of CNR, the CSO/CTB spectrum of CATV networks with respect to the output level ripple of the CATV amplifier and its preemphasis.
The NRD can also calculate the CNR, CSO/CTB spectrum for laser-optical CATV systems and their cascading with coaxial CATV networks.

1.6 Right-hand section



The data of the components are edited in the right-hand window of the AND component editor.

In ARD, the right-hand section is also used to enter and select measured data.
The right-hand section consists of a selection of windows, only one of which is ever visible at any one time.
To switch from one window to another, select the tab on the lower edge of the display which places the associated window in the foreground.
Besides the basic data, the symbol data, the object attributes and the pins (component connections), you will also find the "data packets" which exist for all symbol types.

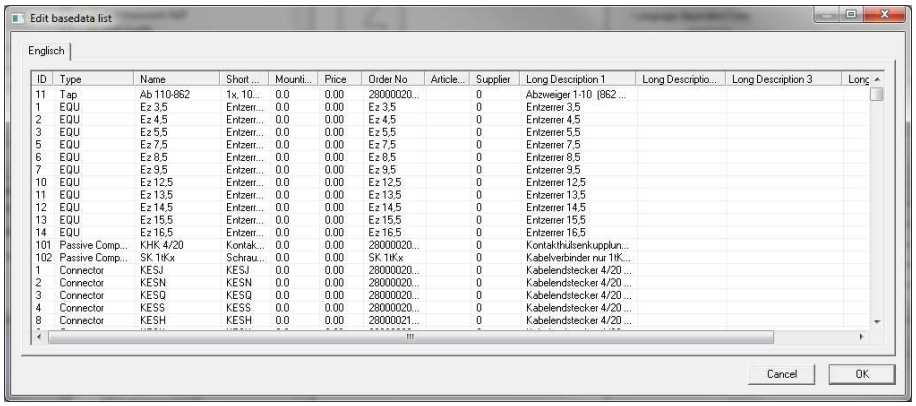
1.6.1 Basic data

The basic data are identical for all elements and can be created in several languages.
The basic data consist of two areas:
a language-dependent area, which is located inside the frame of language-dependent data, and a language-independent section.
Components can contain language-dependent basic data in several languages.
The language-dependent data are stored separately for each language.

For example, if you change the supplier in French, the supplier does not change in another language.
The language-independent data, however, apply globally to all languages.

- **Language-dependent data:**
  Name, order No., item No., supplier, price, available packet sizes, availability, tender text and brief description
- **Language-independent data:**
  Mounting time, display color, shielding values, version and special type designations.

Menu item "Edit/Edit basedata list" helps you to update the basic data information for all objects in the library in question.



The list contains all objects of the library.
The basic data can be generated in multiple languages.
There is a separate tab card for each language that is created.
This is where you change all basic data contained in the library.
Edit the data by clicking on the line you wish to change.
If no basic data is entered for an object in one of the other languages, three dashes are displayed in that line:

1.6.2 Symbol data

This is where the symbol properties are edited.

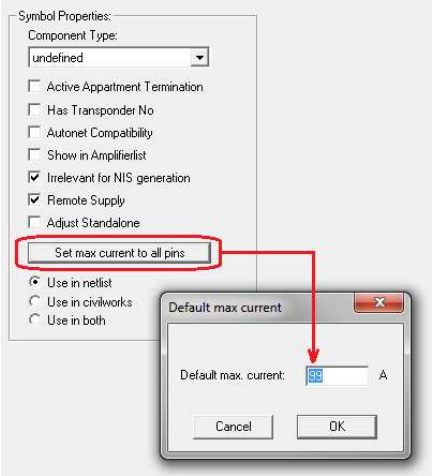
This tab only appears if “Symbols” is selected in the left-hand window.

The component type of the symbol must be defined before the associated data packets can be generated.
A data packet specifies the RF, electrical or optical data between the different connections.
If only one data packet is defined, it applies to all outputs opposite the input.
You can only create new data packets in the window with the basic data by means of the context window.
Which packet types can be created depends on the component type that has been defined.
The following data packets are possible:

- Standard
- Antenna
- DWDM
- Terminal
- Equalizer
- Filter
- Power supply unit
- Opt. standard
- Opt. attenuator
- Opt. receiver
- Opt. filter
- Opt. transmitter
- Opt. amplifier
- Parabolic antenna
- Reverse-path receiver
- Bridge Point (NTU)
- Optical Bridge Point
- LNC
- Splice
- Wall outlet
- Trace
- Converter
- Amplifier

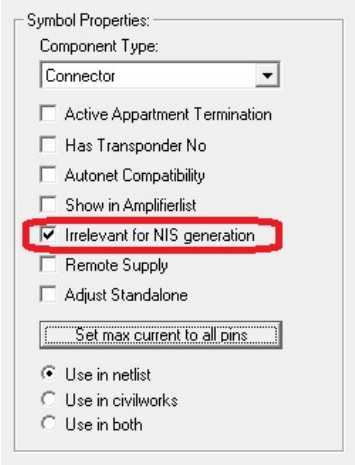
For component type “tap-off”, you can enter attenuation values of the tap outlet not only via the standard packet but also directly at the pin (attenuation identical across the entire frequency spectrum).

1.6.3 Symbol properties



This is where additional symbol properties are defined:

- Automatic capability
- Entry of this symbol in an amplifier list
- All pins are power passing



NIS graphically represents the hierarchical structure of the network.
By selecting the “Irrelevant for NIS generation” checkbox,
you define that the relevant component will be ignored during generation of the NIS structure.

1.6.4 Attributes

Attributes can be assigned to all objects.

An object attribute consists of a name and its associated value.

The name of the attribute must be unique.

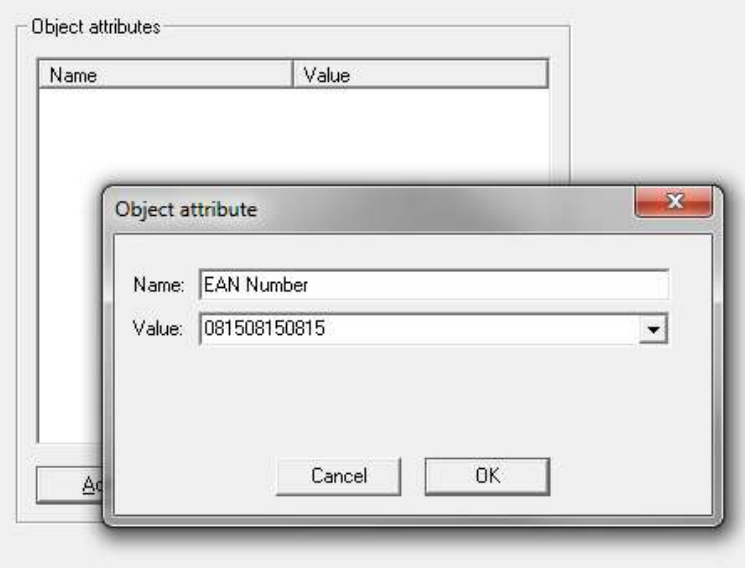
If you wish to assign additional object attributes to an object, click the Add. button.

Enter the name of the object attribute and the value.

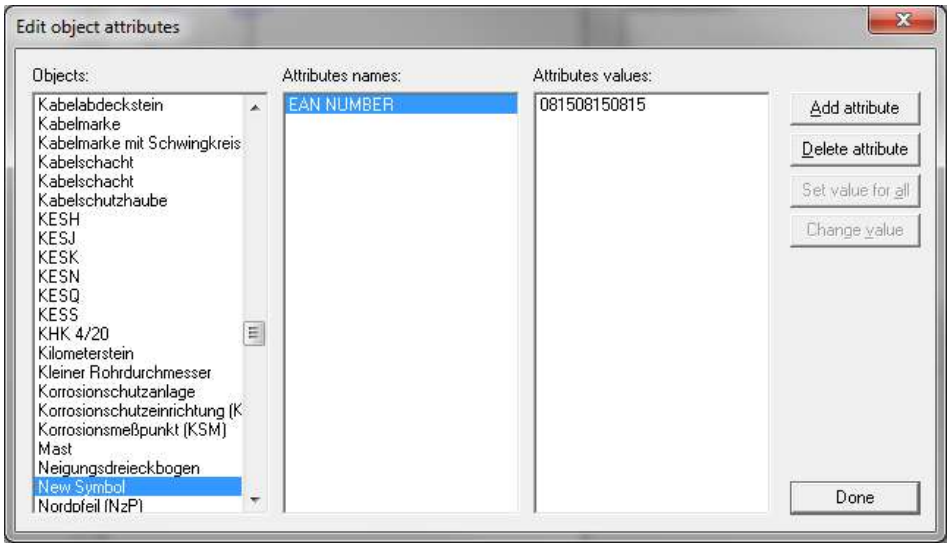
Then confirm this dialog box with OK.

If the name already exists, this name will not be created.

You can edit existing object attributes with Edit .



Help is provided under the “Edit/Edit object attributes” menu item for quickly editing multiple objects in a single step.



All objects of the library are displayed on the left-hand side.

You can select one or more objects in the list to change their attributes.

(You can select multiple objects at once by pressing the Strg key + clicking on the objects.)

All subsequent actions are applied to the selected objects.

If you want to insert a new object attribute, click Add attribute.

Now enter a name that already exists to change the existing object attributes.

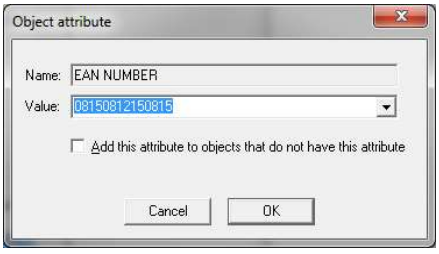
The center list contains the designations of the object attributes that exist in the selected objects. If the same object attribute is found in multiple objects, it only appears once in the list.

You can only select one entry from the list.

To delete object attributes, click Delete attribute.

All the values for the objects pertaining to the selected object attributes are displayed in the right-hand list. To change these values, click Change value.

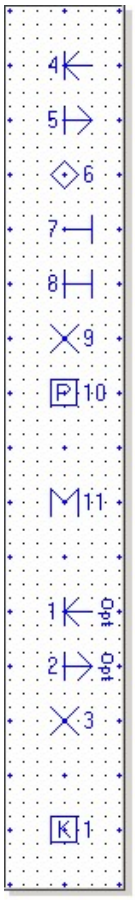
Only those object properties showing the selected value are changed.
If you want to change value that applies to all object attributes that you selected in the left-hand list, click Set value for all and then enter the new value in the "Object attribute" window:



Here you can additionally define that the new attribute is to be applied to all objects that do not yet feature this attribute.

Check the check box accordingly.

1.6.5 Component connections (pins)



The following tab of the right-hand section shows the connection list for symbols. A connection is displayed red in the center section when it is selected. Depending on the component type you have selected, the following connection types are possible:

- Electrical connections
  - RF input
  - RF output
  - RF user output
  - RF tap
  - Loop-through input/output
  - RF isolation point
  - Power supply
- Mechanical connections
  - Mounting
- Optical connections
  - Opt. input
  - Opt. output
  - Opt. separator
  - Mech. cable connection (splice box)

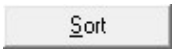
1.6.6 Adaptation

Clicking the 

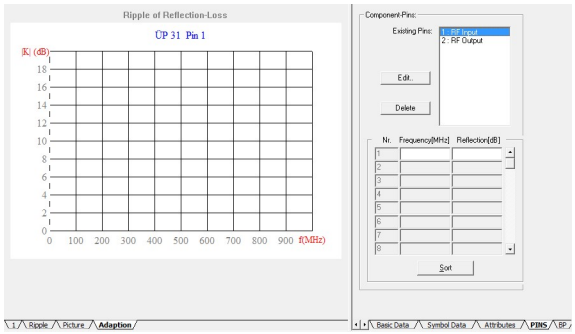
button allows you to enter existing value pairs in the table in any sequence.

This button is only active when at least one connection is defined.

You can start a new line by pressing the  key.

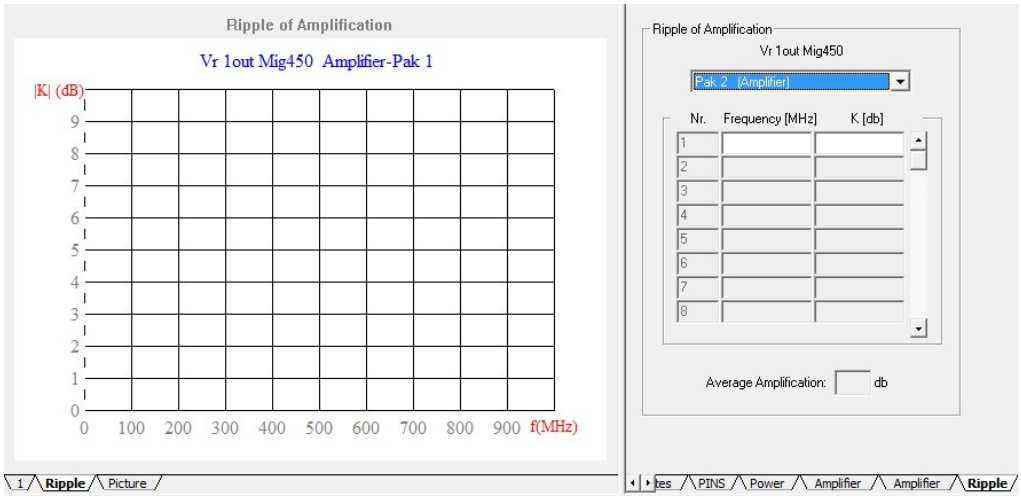
To sort your entries in ascending order and have them displayed as a diagram in the center window, click the 

button.



1.6.7 Frequency slope measured data window

This window only opens if a symbol, cable or amplifier object is selected, if the symbol is an amplifier, a filter or an equalizer, or if the symbol contains a standard packet or is a cable. It is used to edit measured data for the frequency slope of a data packet (i.e. of an input/output pair). The entered data is displayed in the center section.



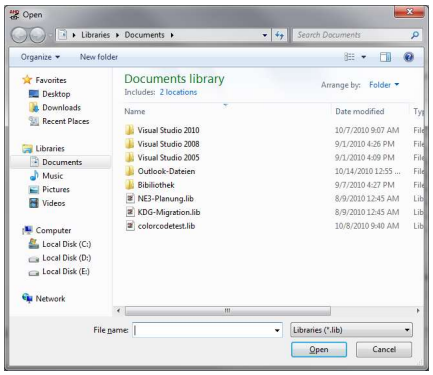
2 Editing Libraries Graphically

2.1 Loading / opening libraries

You can load an unlimited number of component libraries, each containing an unlimited number of objects for each heading simultaneously.

You can call up existing libraries using the menu item “File/Open” , using the toolbar  or with the shortcut Strg + O.

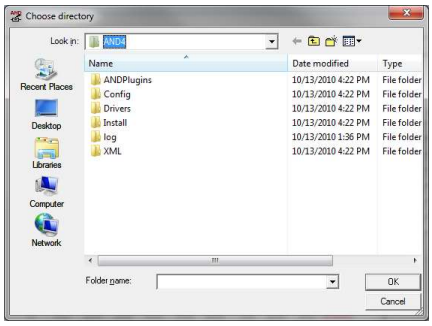
A file selection dialog box opens, in which you can select a library and load it with Open or by double-clicking with the left mouse button.



You can abort the action without making a selection using the Cancel button or Esc key.

2.2 Changing the working directory

You can change the working directory with menu item “Choose directory”.
The following dialog box opens.



If the library you wish to edit is not located on your computer, you can set the path in the network, if you have a network, with the Network button.

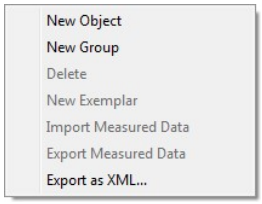
2.3 Creating and editing libraries

If you want to edit an existing library, load it as described above.
If you want to create a new library, select menu item "File/New",

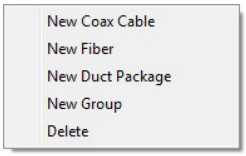
click  or press the shortcut  +  .

An empty worksheet opens.
In the left-hand section you can select a category (Symbol, Cable, etc.) using the tab at the bottom of the screen and with the right mouse button you can then open the pop-up menu

for symbols



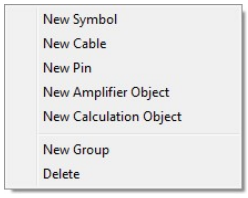
for cables



for connectors

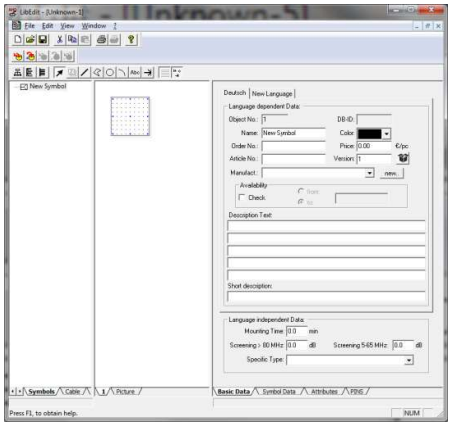
for amplifier objects and costing objects.

If display according to user groups is selected, the following menu opens:



Select the menu item for the component you wish to create.
In this pop-up menu you can, of course, also delete components again or create a new group.
Multiple components are combined into one group.

If you want to draw a new symbol by activating the "New symbol" field,
the following screen will open:



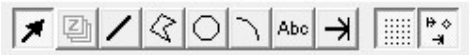
If you want to increase or decrease the size of the symbol frame place the mouse pointer on the lower right-hand edge of the symbol frame.
The mouse pointer changes shape. Now, keeping the left mouse button pressed, drag the symbol frame to the required size.

You can also increase or decrease the length and width of individual symbol frames.
You can also change the frame of the symbol even after you have begun to draw the component.

However, the frame can never be smaller than the symbol itself.

2.4 The system drawing function

The symbol is drawn graphically in the center section.
You can activate different drawing modes on this



toolbar.

2.4.1 Selection mode

You switch to selection mode with the  button.

This is where you change the size of the frame, define and draw blocks, move text, edit pins, select drawing elements, and move or modify them.

2.4.2 Block funktion

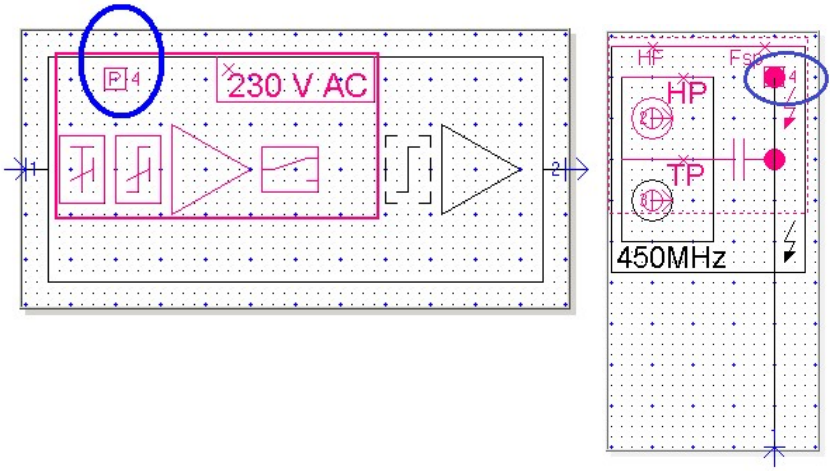
If you want to delete or move an existing line group, for example, drag a border around the objects in question using the left mouse button. They are then displayed in red.

You can also select objects with the Shift key  .

Keeping the Shift key pressed, click one after the other on the objects that you want to select.

If you click an object that is already selected, it is removed from the selection again.
If you want to add more objects to your selection, keep the Shift key pressed and insert other objects using the selection square.
It does not matter whether these objects were already selected or not.

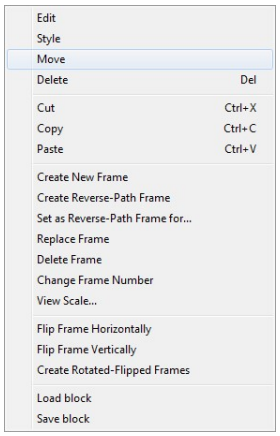
Example: Creating a block



The pin selected in the block (blue circle) is included in all actions. See also "Showing and hiding connections"

2.4.3 Moving objects

After selecting objects, press the right mouse button to open a drop-down menu.



Now you can move the selected objects.
The easiest way of moving objects is to grab them with the mouse and then drag them to another position.

2.4.4 Cutting, copying and pasting objects

These actions can be performed with the help of the context menu.
Other options are also available for performing these actions:

with the shortcuts familiar to you or by selecting one of the buttons shown here.

To cut Strg + X or  or Entf
(copy to clipboard)

To copy Strg + C or 

To paste Strg + V or 

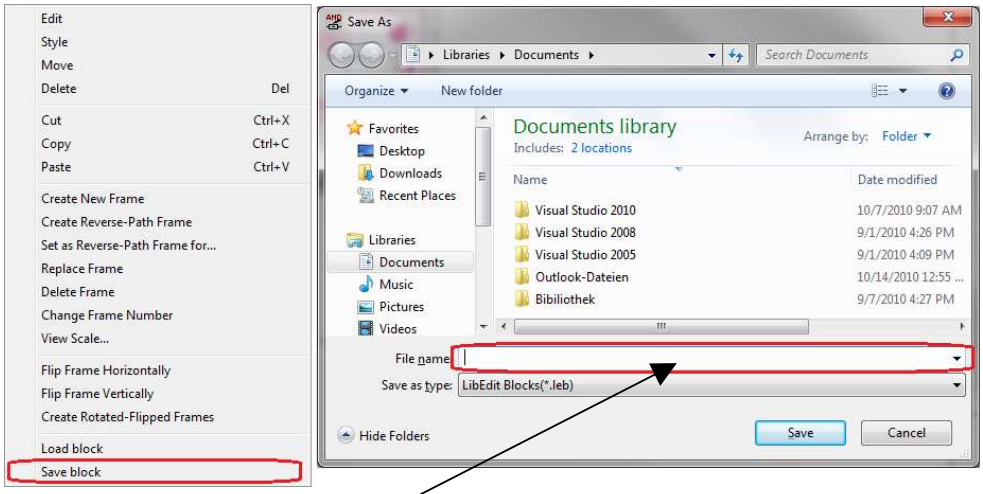
If you keep the Ctrl key pressed while you move an object you will create a copy of the object.
Once you have copied an object or a group of objects to the clipboard,
you can select another symbol or draw a new symbol and then insert the contents
of the clipboard into it.

The clipboard is overwritten with the commands Strg + X or  or Entf.

2.4.5 Saving and loading objects

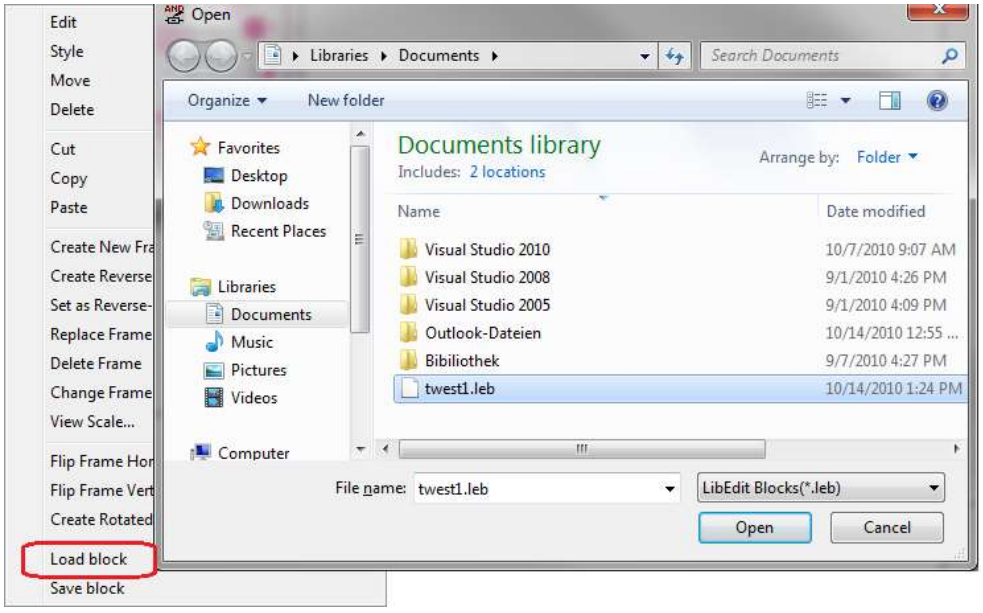
These actions can only be performed using the context menu.

You can save an object or a group of objects to a file to use them again later.
Select the objects and right-click one of the selected objects.
Then select the function **Save Block** from the context menu**.**



Enter a valid file name here.

If you want to use an object you have saved again, proceed as follows.
Select the **Load Block** function via the context menu .



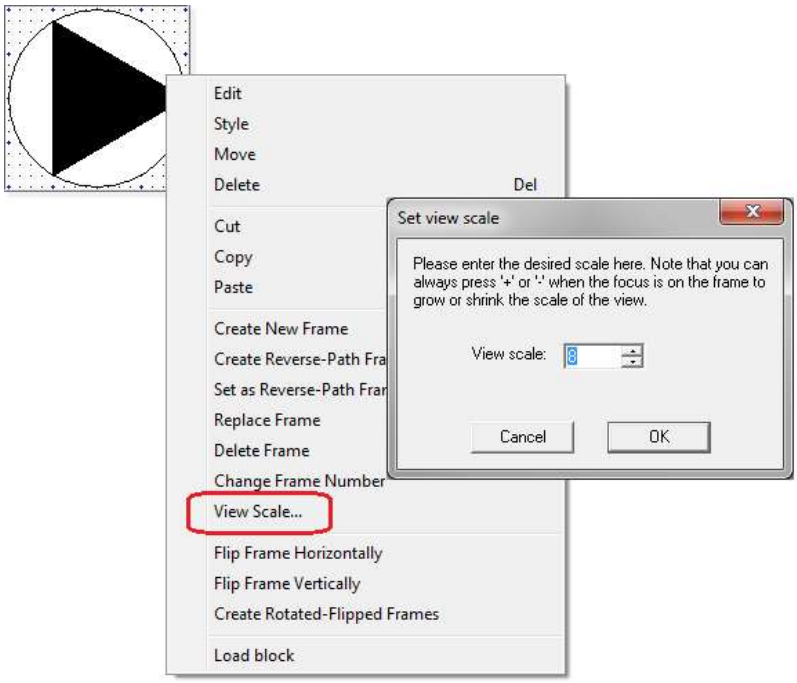
Make your selection here.
The object is now "attached" to the cursor.
Click the left mouse button to deposit the object inside the grid.

Complete the action by clicking the left mouse button again or pressing Esc.

2.4.6 Scaling a view

This function allows you to change the size of the object in order to edit it.
The scaling factor can be anywhere between 1 and 50.

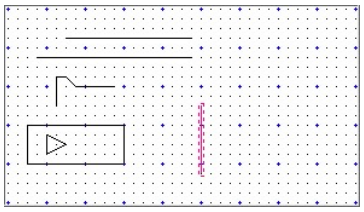
Alternatively, you can also scale the object step by step with the  or  keys.



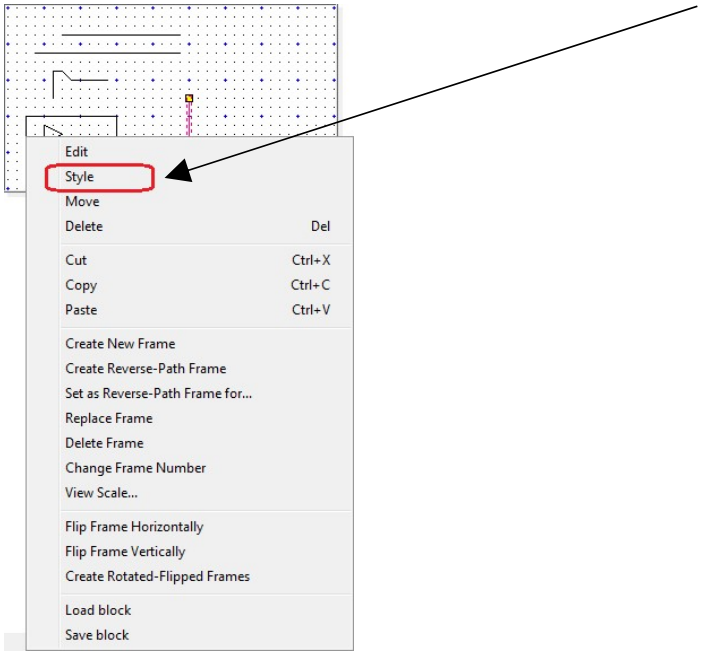
This function has no effect on the AND planning program.

2.4.7 Drawing a line

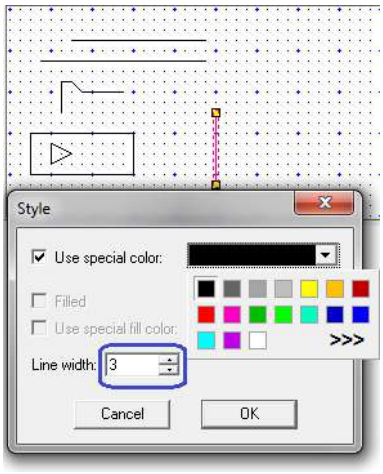
If you want to draw a line, press this button.
Click the starting point of your line and keeping the mouse button pressed drag a line to the end point.



Besides familiar functions such as Copy, Delete, etc., the context menu shown below (right mouse click) also includes functions for influencing the style of the lines.



Here you can modify the line color and line thickness of individual lines or sections of lines.



The altered line thickness is only visible when you call it up in AND.
Before you can delete, edit or move a drawn object,
you must mark it. You can mark a line by clicking it once with the left mouse button when the mouse pointer is positioned directly over the line.
If multiple objects are adjacent to each other, the object nearest the click point is selected.

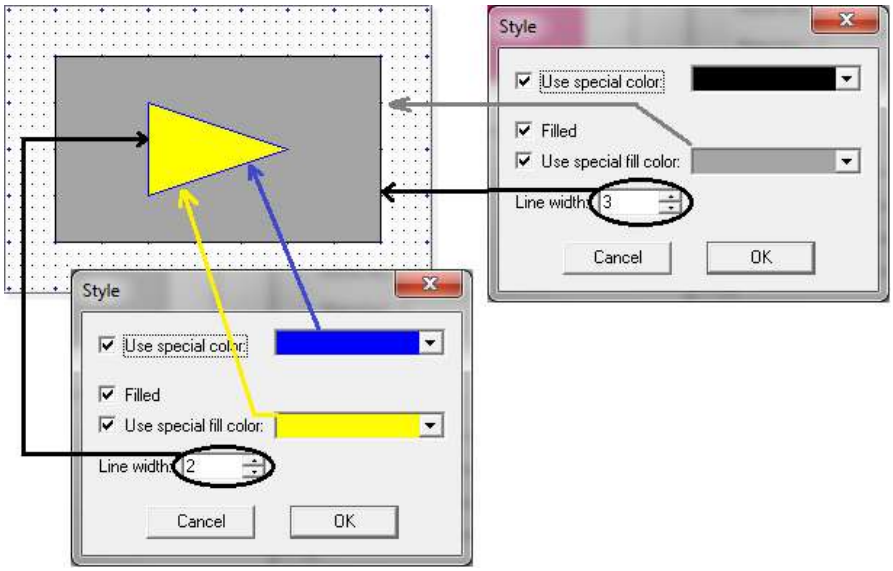
However, if you want to select a different object, press the tab key.
Selection is only possible from among the objects that are near the click point.
If you cannot select the object you wish to edit, click again at a different position.
The element to be edited is then displayed in red and you can execute the relevant function.

You can undo this action by clicking the Cancel button or pressing the Esc key.

2.4.8 Drawing Polygons

If you want to draw polygons, select the  button.
Make a click for each corner of the polygon.

To join up the polygon, click the right mouse button or press the Esc key.



You can subsequently change the shape of the polygon.
Grab one of the corner points with the mouse pointer by moving the mouse pointer over the object while keeping the left mouse button pressed.
Now you can release the corner points anywhere within the grid.
You cannot generate additional corner points or delete them.
You can edit individual polygons using the menu.

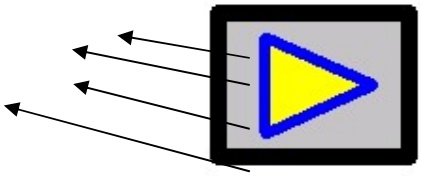
Please remember that older versions of AND may not support this function.

Here are some examples showing how they are represented in AND.

The line widths defined here are only visible in AND.

The defined filler color – yellow – is not visible until the polygon is deselected.

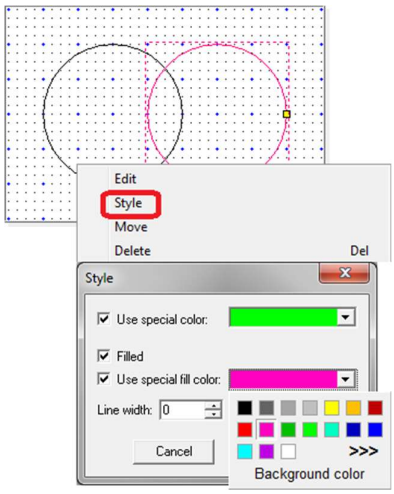
Here is the result in AND.



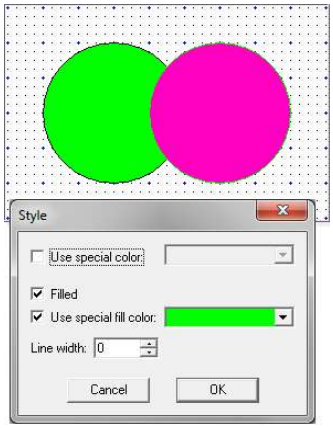
Line width: 3, Color: black
Polygon filled, Color: gray
Line width: 2, Color: blue
Polygon filled, Color: yellow

2.4.9 Drawing circles

If you want to draw a circle, click this button.
First click a point that is to be the center point of the circle.
Then pull the circle to the required size while keeping the left mouse button pressed.
When you release the mouse button the circle is fixed on the grid.
Circles are edited in the same way as polygons.



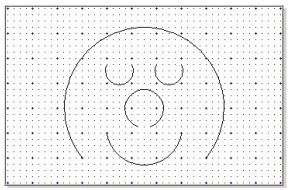
2nd step



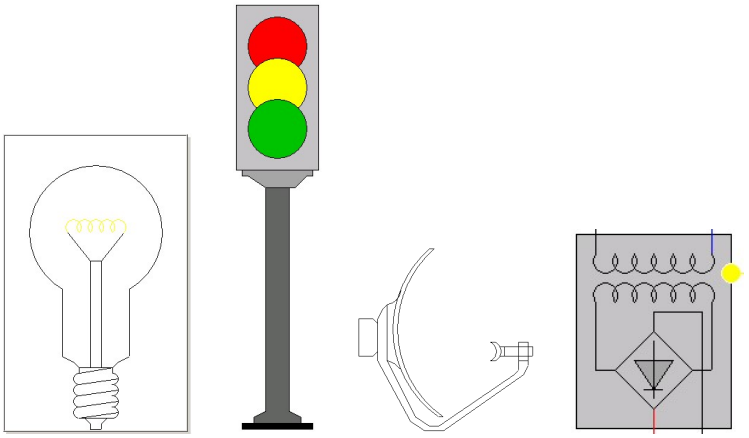
Please remember that older versions of AND may not support this function.

2.4.10 Drawing open circles, arcs

If you want to draw **arcs**, click this button.
You define the opening of the arc with the first and second click you make in the drawing grid.
A red line appears.
The subsequent dragging motion with the mouse defines the size of the arc.
You complete this action with the third and final click.


Arcs, too, can be drawn in different ways.

Examples:



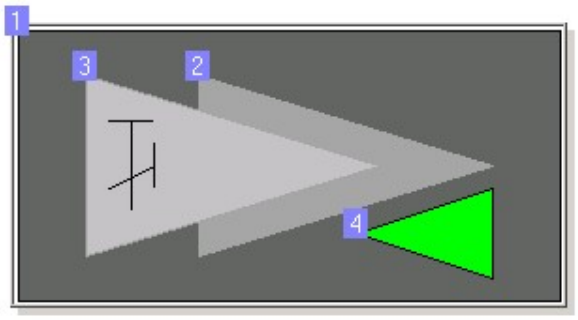
This variety of possibilities also allows you to draw graphical symbols used in other technologies, so that you can prepare sophisticated documentation for other systems, too.

Please remember that older versions of AND may not support this function.

2.4.11 Order of the drawing levels

Polygons and circles can be shown filled.
It is therefore necessary to define the order in which objects that overlap are to be displayed.
Click this button to enter Z-order edit mode.
Numbers are now assigned to each object, indicating on which drawing level each object is located.
The individual objects are numbered starting from the lowest number.
The object assigned the lowest number is located at the bottom level (background).
The subsequent numbers indicate in which higher level the remaining levels are located (foreground).

Lines are always located in the foreground.



Now click the objects in the sequence they are to overlap each other.
Start from the background.
The last object to be clicked can be recognized by the number in the blue box.
If you do not wish to start at the backmost object (number 1) but at a different position,

keep the 

pressed while clicking the object from which you want to start numbering.

Release the 

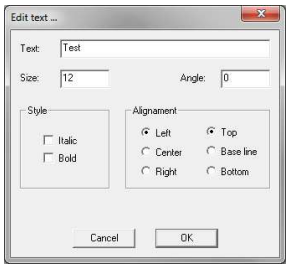
button and continue numbering as described above.

You can restart numbering by clicking the  button again.

Free texts are also subject to Z-order.

2.4.12 Inserting free texts,

If you want to label objects, select this button.
Now, left-click on the position where the text is to appear in the drawing.
The "Edit Text" dialog box opens. Enter a text here and adapt the formatting.



The text is positioned when you confirm your entries.
If you want to edit your text, go to selection mode.

Double-click to return to the "Edit Text" dialog box.
You can access the change color option from the context menu opened with a right mouse click.

Remember that older versions of AND may not support this function.

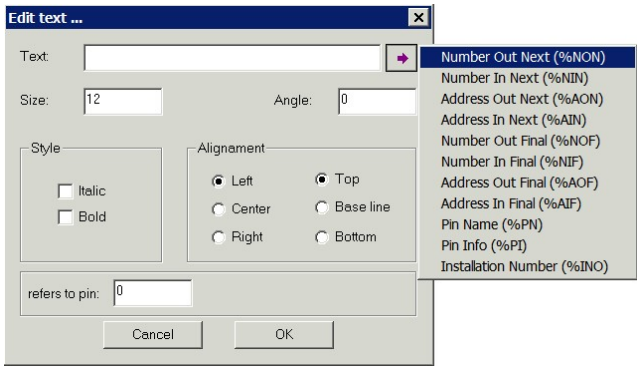
2.4.13 Dynamic Labels

See also the description of dynamic labels in the AND Client documentation.

2.4.13.1 Create Dynamic Label

For creating a dynamic label for a pin, press this button: 

Draw the rectangle for the text. The following dialog will appear:



By clicking on the arrow, you can choose a placeholder variable for the dynamic label and enter additional text manually. Multiple placeholders can be entered, too.

The dynamic label has to be assigned to a pin to which it refers. This pin is used to calculate the value of the dynamic label in the connected network. To refer a label to a pin, enter the internal pin Id in the field "refers to pin".

Moreover some formatting options exist.

2.4.13.2 List of variables for dynamic labels

The list in the dialog contains only the most important variables. More combinations of 3 letters are available (for dynamic labels and splice/patch reports):

<**What**:1character><**Where**:2 characters>

Examples:

NON **N**ame**O**ut**N**ext

AIF **A**dress**I**n**F**inal

The following components of the variables are available:

What	Description
N	**N**umbers (Path of numbers, possibly relativized, see dynamic Labels)
A	**A**ddress (inherited from hierarchy if empty for object)
C	„**C**ontext", hiearchy context: hierarchy without lowest level (splice/pin)
T	**T**ype (of the lowest path, not inherited)
I	**I**nfo (type dependent)
O	**O**bject-Id (Installation No., Pin name, ..) of lowest hierarchy level, (without replacement by type or replacement string)
P	**P**ath (whole hierarchy path, not relativized)
J	Ob**J**ect-No. of object containing the pin (incl. replacement)
M	**M**ounting (Object-No.+Rack-Pos.) of object containing the pin

	(incl. replacement)
R	**R**ack-Position of object containing the pin (incl. replacement)
Y	Cit**Y** of object
Z	**Z**ip code of object
L	Innermost **L**ocation rectangle with non empty installation number
1 .. 9	Path part of the object (1* = O*)

Where	**Stands for**	**Description**
TH	**Th**is	Start/reference point (list in the report/label start)
ON	**O**ut **N**ext	Next connection point, following connection away from symbol
OF	**O**ut **F**inal	Last connection point, following connection away from symbol
IN	**I**n **N**ext	Next connection point, following connection through symbol
IF	**I**n **F**inal	Last connection point, following connection through symbol
PP	**P**artner **P**in	Partner pin of starting point (used for mounting pins)
TP	**T**raverse	Partner pin of traversed symbol (for future implementation of dynamic labels for links)

There are (13+9)*7 = 154 combinations, e.g. "NON". "1TH" .

Optionally, bundle and fibre information can be appended to each of these values to do so, append e.g. to %NON for an ONT having more than one fibre:

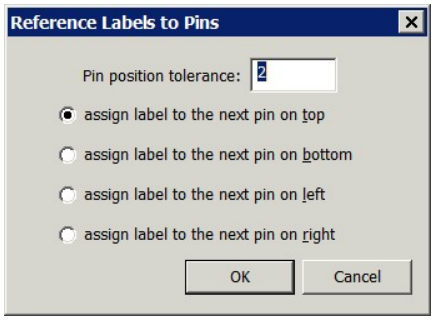
- .B<m> (m=1,..): route for bundle m (up tp first bundle split)
- .B<m>.F<n> (m=1,.., n=1,..): route using bundle m, fibre n
- .F<n> (n=1, …): route using bundle 1, fibre n
- If m=0 and n=0, routing proceeds without wire information.

Examples:%NON.B1.F2 , %AON.F1

2.4.13.3 Automatic pin assignment

Dynamic labels may also be created without assigning them to a pin manually and can later be assigned to a pin according to their position to the nearest pin. To activate this function, select multiple dynamic labels and from the context menu select the command „Reference Labels to Pins".

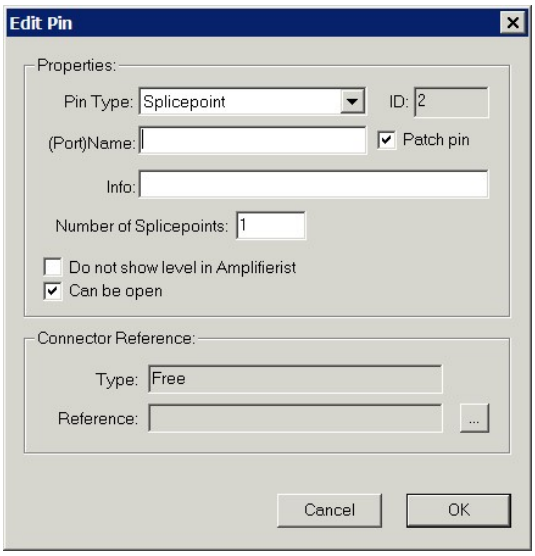
The following dialog will appear:



You may enter the maximum tolerance of the distance of the dynamic label to the pin as well as the direction of assignment. If the assignment cannot be done for all labels, a message appears.

2.4.13.4 Pin Data for display in dynamic labels and reports

For displaying the pin in a dynamic label or in a splice/patch report, at least the **(Port)Name** should be provided:



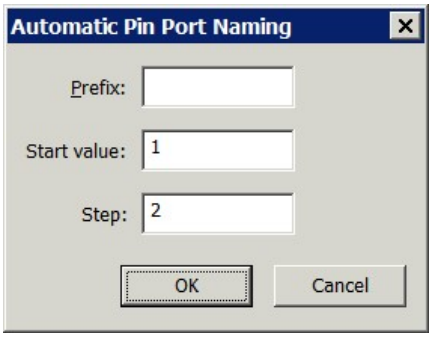
Dynamic labels allow to display values from the next/last connection points in the connected network. Since not all pins should be considered as connection points (e.g. only the patch ports of the front of a patch field should be considered but not the pin of the back of the patch field with the whole bundle connected), pins have to be activated as being a **Patch pin**.

Patch pins define (together with splice points, terminating devices and path dividers) the **connection points** to be considered and displayed in dynamic labels and splice/patch reports. Moreover, patch pins may be displayed in splice/patch reports (like for the dynamic labels, only the front patch ports of the patch field should be listed, but not the back pin for the bundle).

2.4.13.5 Automatic naming of Pin/Port Names

Pin/port names may automatically be assigned. Chose "Autogenerate Pin Port Names" from the context menu of a selection of pins.

The following dialog is displayed:



You may enter

- a prefix
  (e.g. for selected pins 1,…,9 you may choose prefix „P0" and for selected pins 10,11,12, you may choose „P", such that the alphabetic sorting oft he port names will work as expected)

- the start value
  (e.g. 10 (with prefix "P"))
- the step (e.g. in combination with start value 1 resp. 2 if there are two rows of pins with the even and odd numbers in the patch field).

Please take care for names which are alphabetically sortable, such that hierarchy sorting works in splice/patch reports, i.e.

not: P1, P10, P11, P12, P2, P3, P4, P5, P6, P7, P8, P9

but: P01, P02, P03, P04, P05, P06, P07, P08, P09, P10, P11, P12 .

2.4.13.6 Patch fields

Patch fields have the same data as splice boxes, only some behavior is different:

- for patch fields, pins are by default activated as patch pins,
- automatic splice assignment by default is „Row assignment",
- Patch pins by default have a female connector reference, spliceboxes have none.

Row assignment of splices automatically assigns the pin with the maximum number of splice points the lowest with the lowest number splice points. The assignment order is determined from left to right and then downwards, i.e. first the uppermost row of pins is assigned left to right, then the next downward row, …

2.4.13.7 Parent hierarchy for Symbols

Mounting symbols can be given a parent hierarchy, by marking the child symbol in the tab **Symbol Data** as being an **Assembly Unit** and by giving it an **Assembly Unit Level** lower than the assembly unit level of the parent.

Example: assembly unit level 0 for a patch field symbol and assembly unit level 1 for a rack symbol. If in AND the patch field is then mounted into the rack, the rack can be recognized as being a parent by considering the assembly unit level. E.g. for the rack with installation number R0117 and the patch field with installation number PF04 and an pin with name P01 the hierarchy path of the pin can be computed to be R0117-PF04-P01 if the assembly unit data has been entered as explained above.

See also the notes on the two documentation styles.

2.5 Connections (pins)

The connection selection is dependent on the component type.

The component type should be defined first.

The connection types (see below, Section 2.5.1) can be structured in three groups:

- RF connections:
  - RF input
  - RF output
  - RF user output (only useful for outlets)
  - RF tap (only useful for taps. Use with other component types is strongly discouraged.)
  - RF isolation point (internal intermediate point)
- Optical connections:
  - Optical input
  - Optical output
  - Opt. separator (optical intermediate point)
  - Optical connection (non-directional optical connection)
- Other connections:
  - Mounting (for placing multiple components on or next to each other)
  - Power supply (power passing connection)
  - Mechanical cable connection (mechanical contact point for splice boxes)

RF connections are possible for RF components; optical connections, for optical components. The types "Mounting" and "Power supply" are possible for all components.

2.5.1 Connection types

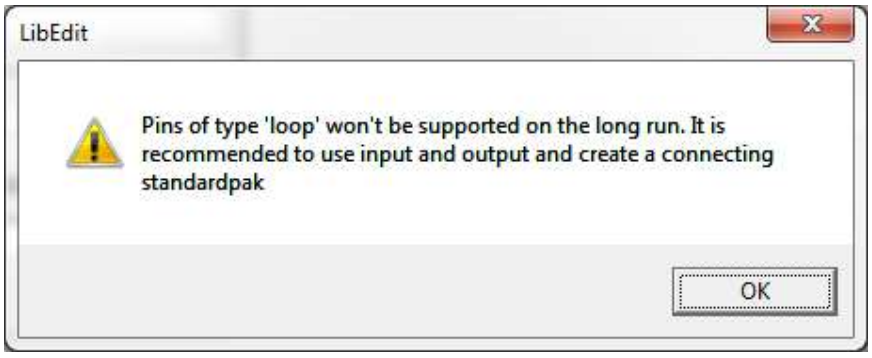
A list of possible connection types is given in this overview.

RF input	• a signal entry point of an RF data packet
RF output	• a signal exit point of an RF data packet
RF user output	• a signal exit point of an RF data packet • can alternatively be assigned a tap loss in the "Attenuation" field, this side loss with respect to the input is identical across the entire frequency spectrum • used as a subscriber connection for antenna wall outlets and for test sockets on amplifiers • a measuring point but no cable can be connected here
RF tap	• a signal entry point of an RF data packet • can alternatively be assigned a tap loss in the "Attenuation" field, which is, however, identical across the entire frequency spectrum • used for tap outputs of multi-taps and taps
RF isolation point	• internal signal entry or exit point between two RF data packets, also RF separator • used, for example, to define different gain levels of an RF amplifier • not used as a component input or output because even though it is possible to select the connector reference here, this is not taken into account in AND
Opt. separator	• internal signal entry or exit point between two optical data packets • not used as a component input or output because even though it is possible to select the connector reference here, this is not taken into account in AND
Mounting	• used to implement mechanical connections or semi-automatic/automatic combination of individual symbol parts to form one whole object
Optical input	• a signal entry point of an optical data packet
Optical output	• a signal exit point of an optical data packet
Optical connection	• an optical signal entry or exit point of a standard optical data packet. Only possible for component types: "Terminal" and "Opt. adapter"
Mech. cable con-nection	• only possible for splice boxes and used to implement mechanical connections between a fiber-optic cable and a splice box, the number of contact points describes the max. number of fiber strands in a fiber-optic cable for this pin in AND
Power supply	• entry or exit point of the power supply

2.5.1.1 Pins of type “loop” no longer supported

Loop pins are remnants from AND 2. In AND 2, loop pins were necessary to define converters which combine signals. They were also used to define neutral pins for 1:1 connectors, for instance, i.e. input or output pins, depending on the context. There has been no need to create loop pins in LibEdit since AND 3, but for compatibility reasons they have been kept in the program. Loop pins cannot be created any more in LibEdit 4.9 and later versions. AND 4.9 can still calculate objects with loop pins. It converts loop pins to inputs/outputs, depending on the context.

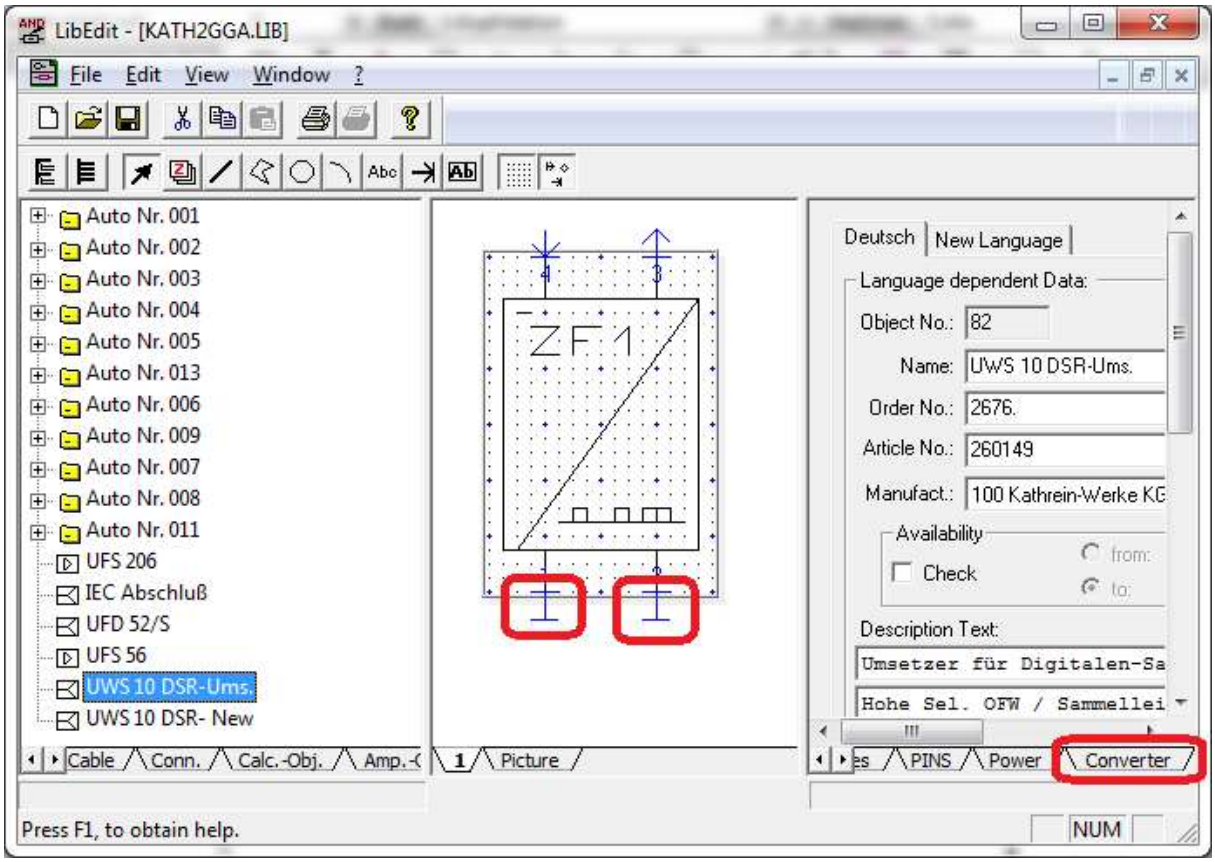
You cannot create loop pins in LibEdit 4.9 and later versions. If an old library is loaded and a symbol with loop pins is selected, the following message appears:



There is no urgent need to revise old components containing loop pins. Those components are still calculable in AND.

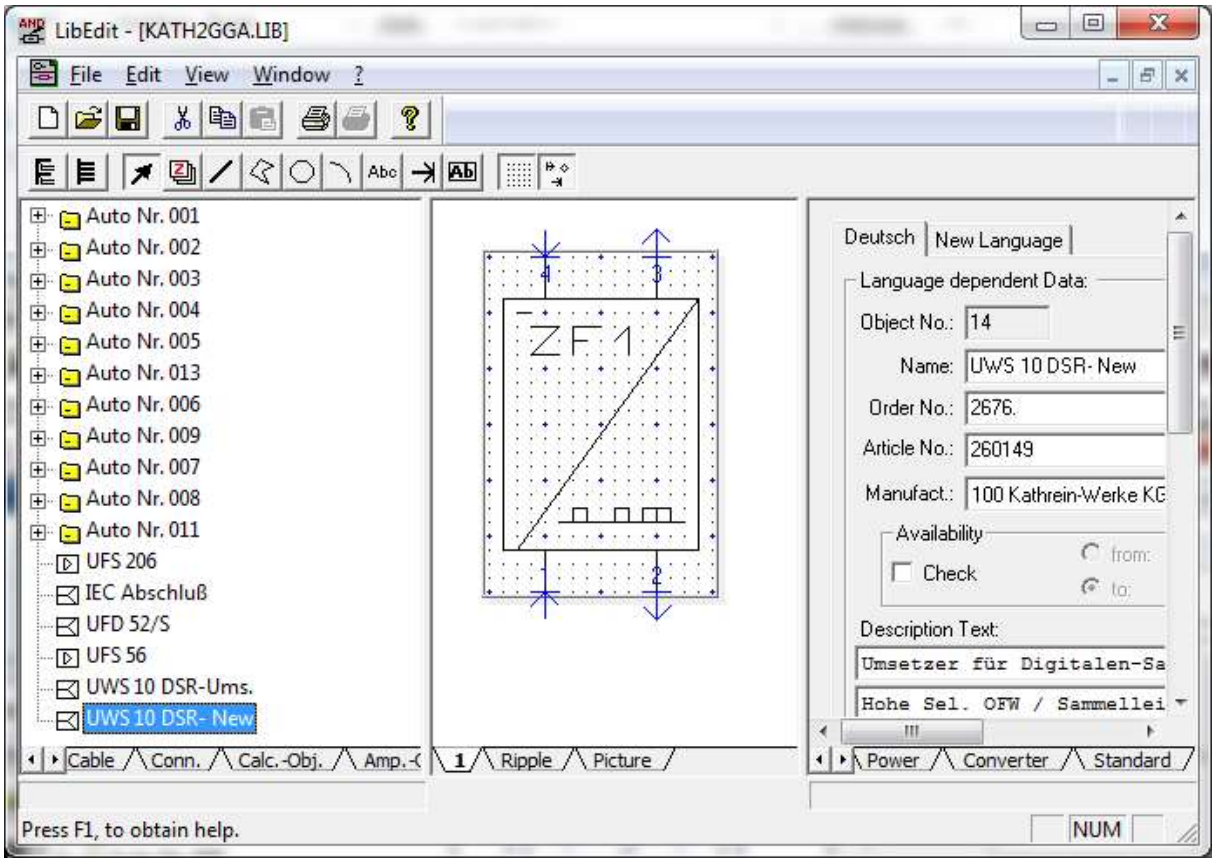
Example of how to redesign such an old component:

Old design:



In the old design, there is only one converter pack from pin3 to pin4; pin1 and pin2 are of type "loop".

New design:



In the new design, the loop pins are converted to one input (pin1) and one output (pin2).

In the base data context menu, a standard pack from pin1 to pin2 is created, and the converter pack is changed to go from input pin4 to output pin3 and pin2.

Since a converter is a component with a fixed direction, in the new design pin2 is a fixed output, and pin1 is a fixed input. If the pins are to remain flexible, you can create a second frame and swap the positions of pin1 and pin2.

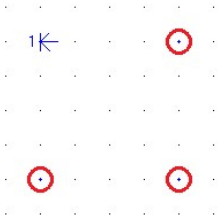
1:1 connectors are objects without direction, therefore there is no need to create a second frame.

2.5.2 Drawing connections

Create a connection for a symbol by clicking this button.
Then click on the insert position for this connection.

You cannot create two connections on the same grid position.

Connections can only be positioned on the highlighted grid points.



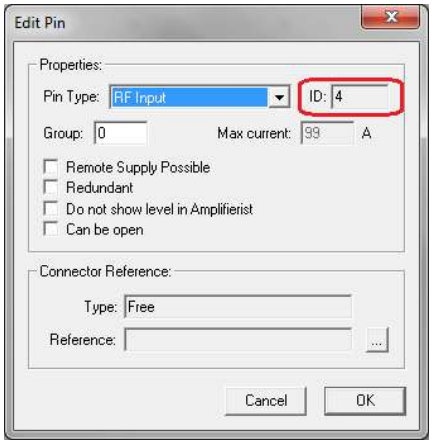
- You can show or hide the drawing grid with the  button.
- You can show or hide the connection grid with the  button.

2.5.3 Optical connections, RF connections, power supply

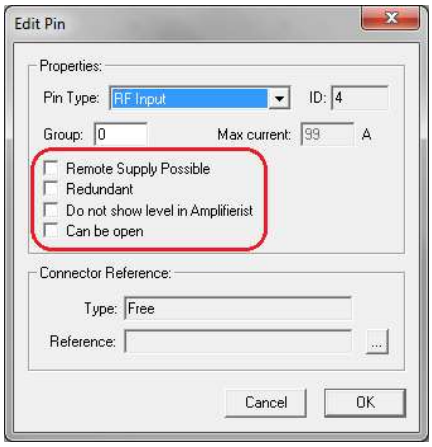
The connection that can be selected depends on the component type.

The component type should be defined first.

In the "Edit Pin" dialog box you can now define the attributes for this connection.



Each new connection defined is automatically assigned a consecutive connection number which cannot be edited.



When you select a check box in the dialog box, the connection in question will be power passing, with the exception of optical connections, now you can enter the max. current carrying capacity of the pin in the enabled "max. current" entry field.

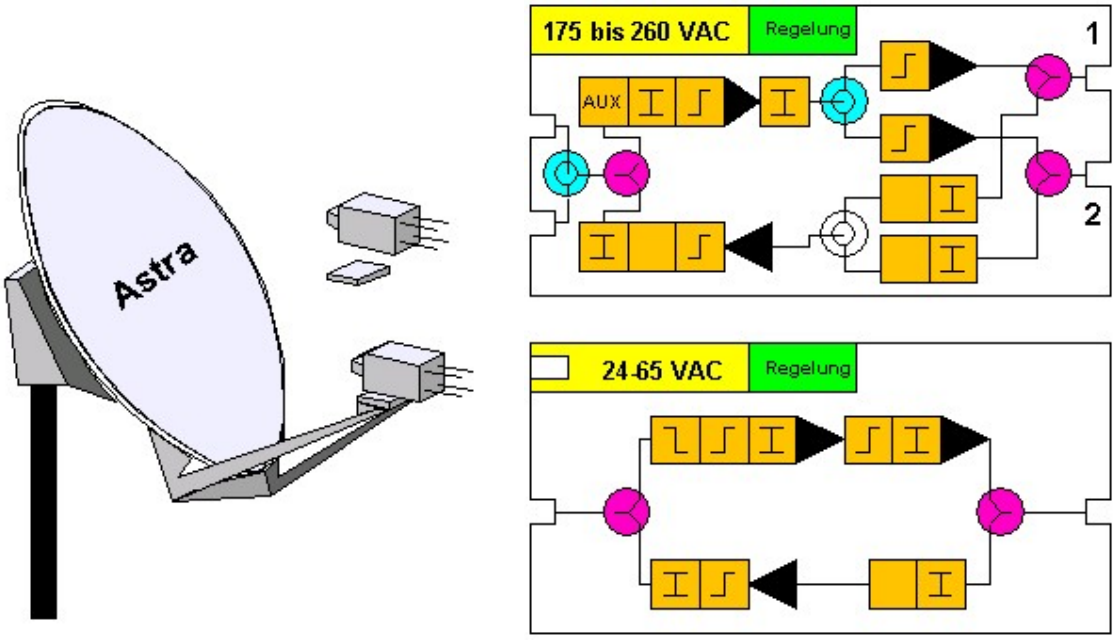
- will be treated redundantly in AND
- will not be included in the amplifier list
- will be accepted by AND as an open connection.

2.5.4 Mounting points

Mounting points have no electrical function.
They permit the semi-automatic and automatic combination of multiple symbols (Macro).
When you use a macro with semi-automatic mounting in AND, after selecting a basic object a dialog box opens prompting you to select one of the objects offered.
In automatic mounting, no dialog box opens when you select a basic object.
The macro is automatically assembled from the available objects in the background and then appears ready for use at the cursor.

A mounting point is inserted in each of the grouped symbols, "automatic mounting" selected where necessary and assigned to the same mounting group.
The mounting points are drawn to match up in the graphical display if the individual symbols are correctly combined.
A macro can contain an unlimited number of components.
Two objects of the same component type cannot be mounted together.
All mounting points of a macro should be located on one basic object.
The objects to be mounted do not have to belong to the same group.

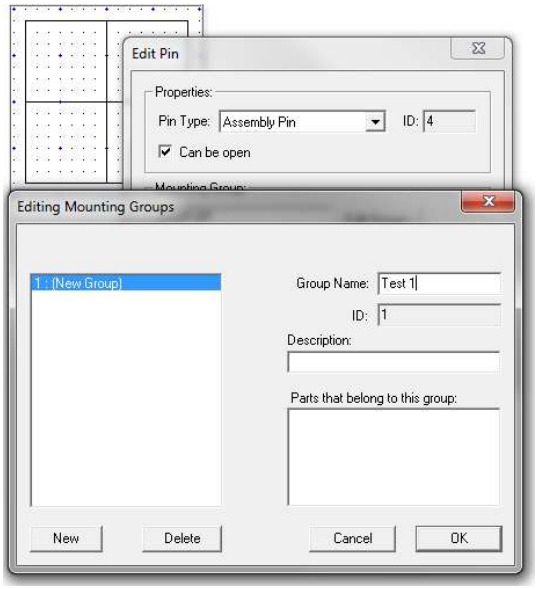
Some examples of macros



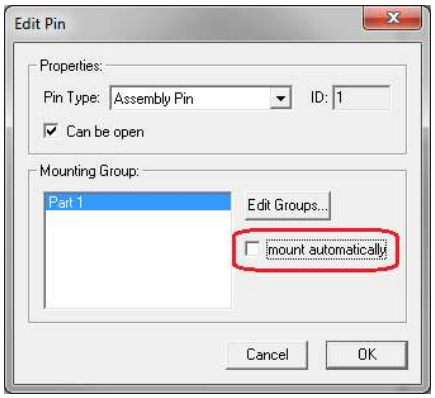
2.5.4.1 Organizing mounting points

The newly defined mounting point is entered in the connection list with an automatically assigned "ID number".
Mounting points are organized in mounting groups in the "Edit Pin" dialog box.
The connection list applies across all symbols for all frames.

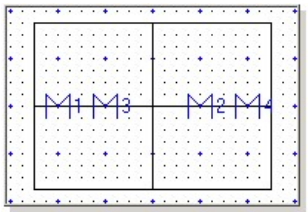
After clicking the Edit Groups..., New buttons in the relevant dialog boxes



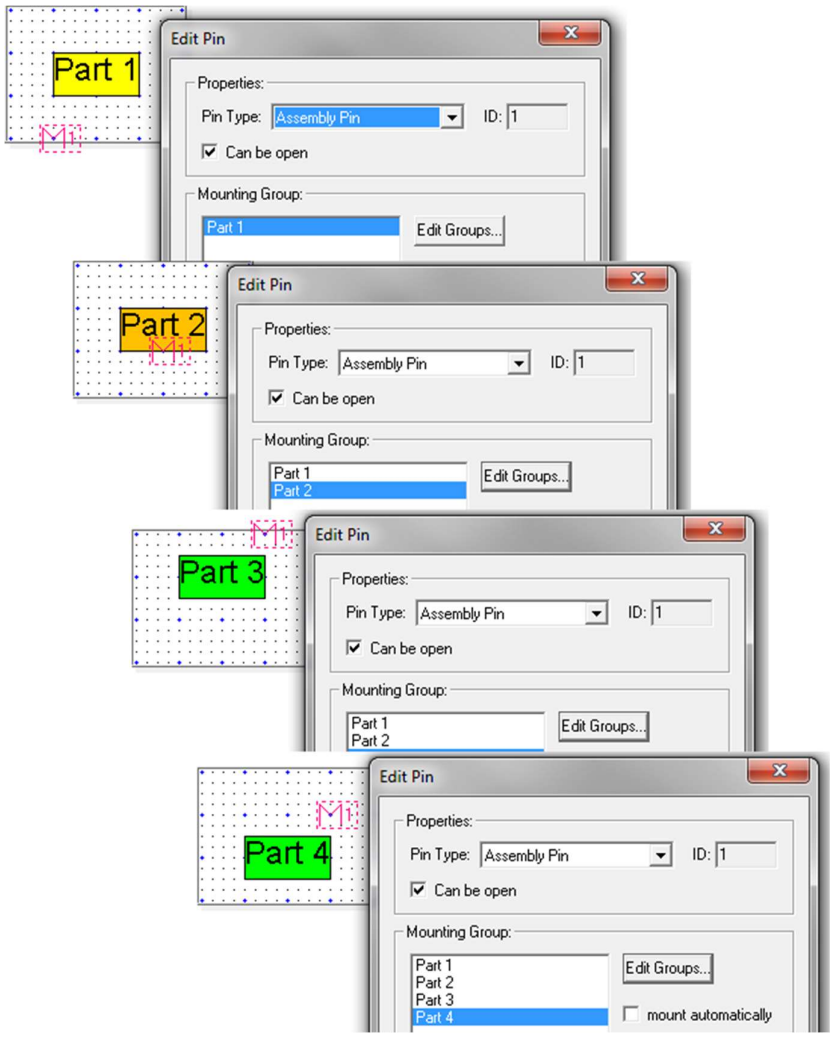
and then labeling the group, you can specify whether the mounting point is to be mounted automatically with another object.



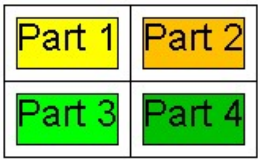
When you have confirmed the dialog boxes with OK the mounting point is aligned on the grid.



Now create the remaining mounting objects and reference the mounting points accordingly.



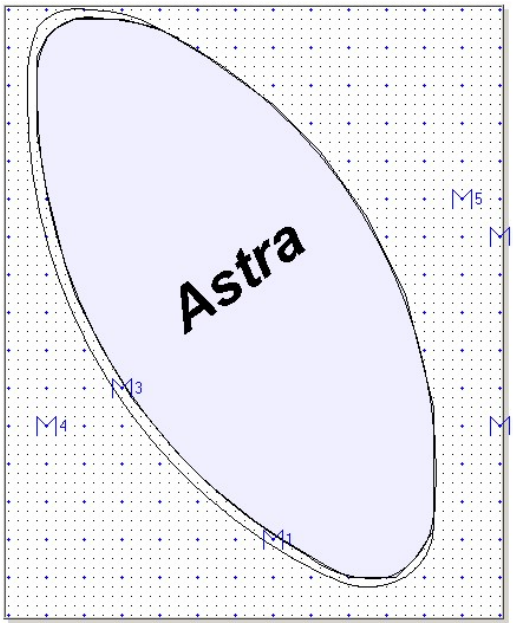
The completed mounting in AND:



2.5.5 Creating macros

The basic object contains ALL mounting points positioned precisely where the corresponding mounting objects are to be inserted.
To generate a macro proceed as follows.
Create all 7 individual components.

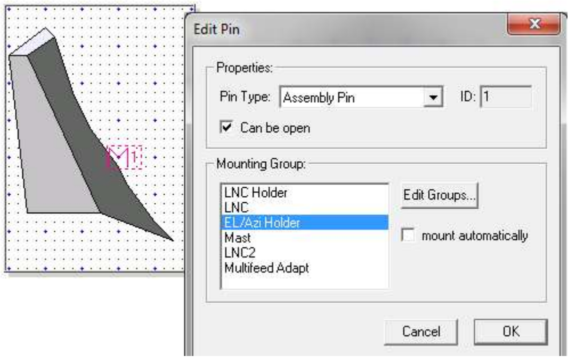
2.5.5.1 The basic object

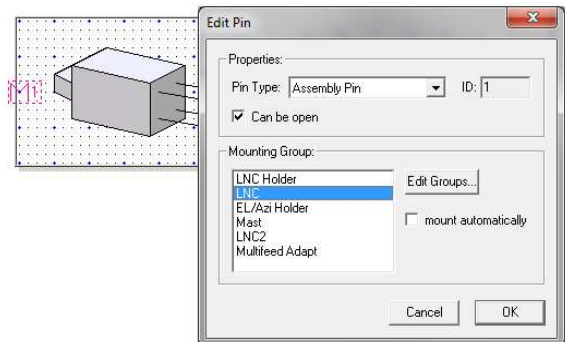


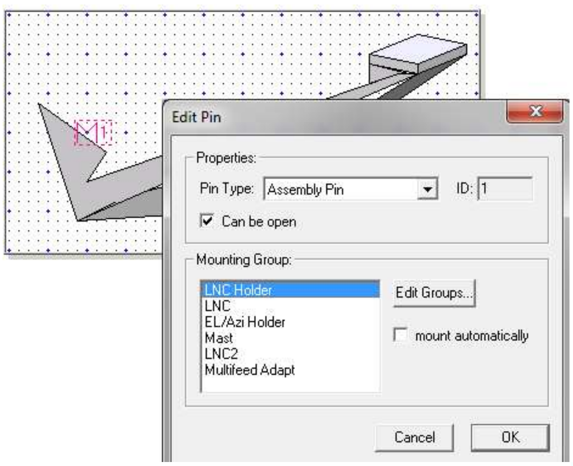
Draw the symbol for the basic object.
Generate and organize the required mounting points.
Create the required mounting groups.
Label them and give them shape, if necessary.

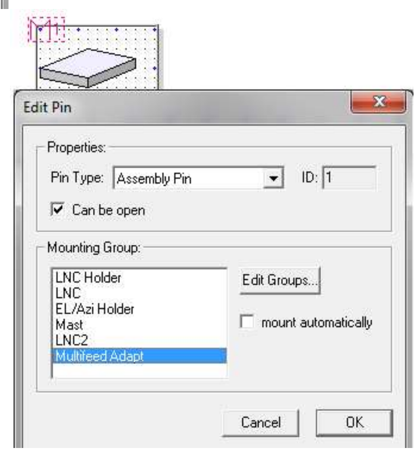
2.5.5.2 The mounting objects

Finally, create all the necessary mounting objects one after the other together with their mounting points and organize them as described above.



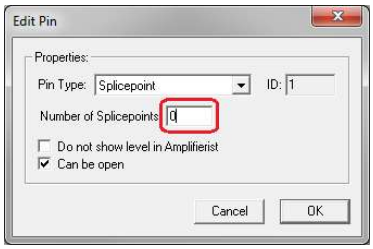




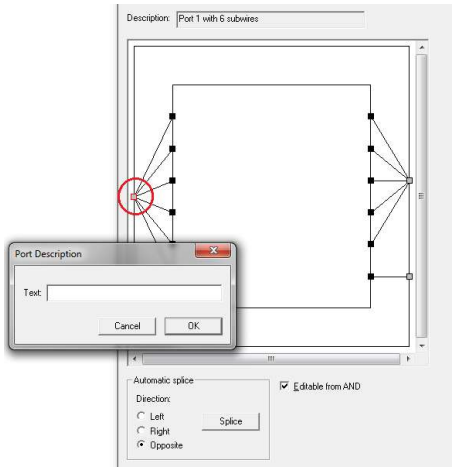


2.5.6 Mechanical cable connection (splice box)

Mechanical cable connections can only be selected for splice boxes.
Enter the number of splice points of this particular connection in the "Number of Splicepoints" entry field.



These splice points are called "pins".
In the cabling area, these connections are drawn in gray in the outer rectangle.
The pins are drawn in black in the inside rectangle.



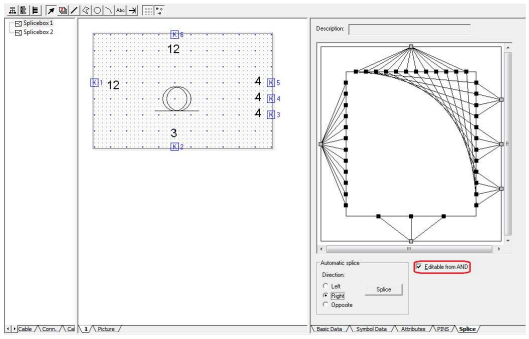
You can enter a name for each connection.

To do this, click the connection, enter a name and click OK.

When you move the mouse over the port, its name will appear in the display field.
You can also have information about the pins displayed when you move the mouse over the pin. However, you cannot enter any data for pins here.

In AND you can plan standard cabling for splice boxes.
Depending on the setting of "Editable in AND" option (see figure above), you can define whether or not the standard values can be edited in AND.

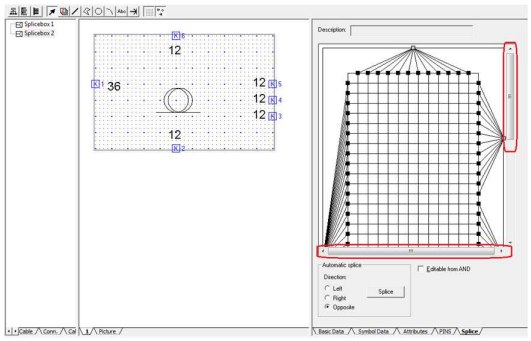
If this option is deactivated, the splice box cabling can only be edited in the component editor. This information is backed up in the splice packet.
A splice box can only contain one splice packet, which is automatically generated by the component editor.



You can also add links to a splice box manually.
A connection can be linked to several pins, but a pin can only have one link.
To add a new link, click on a pin and draw a link to another pin.
This link is displayed in black.
If you do not finalize the link at a pin, that link will be removed again.
It is not possible to make a link to a pin that is already assigned.
You can remove existing links by clicking on an assigned pin while keeping the mouse button pressed.

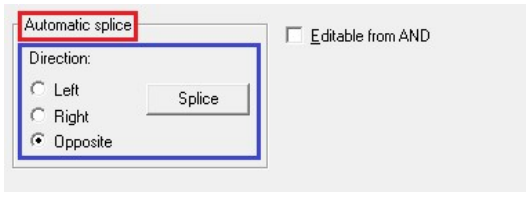
The connecting line turns blue and the pin turns red.
When you drag the line into a vacant space and release the mouse button, the connecting line is removed. However, if you place the connecting line on a new pin, a new link is created.

If more connections and pins are defined than can be displayed, a scrollbar automatically appears



If you attempt to draw a link to a pin outside the visible area, the image automatically scrolls.
The "Automatic s*plice*" function will help you create a link automatically.

Select one of the directions "*Left",* "*Right*" or "*Opposite*" and then click the  button.



If you selected the setting "Left" or "Right", the pins on the right and left sides are connected to the upper and lower pins.
If you selected the setting "Opposite", the left pins are connected to the right pins and the upper pins are connected to the lower pins.
It is not yet possible to automatically remove links that were created automatically.

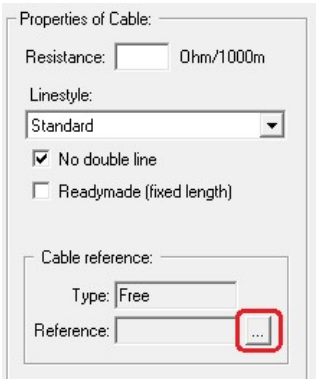
2.5.7 Cable and connector references

References support automatic connector search in AND.
References to associated reference numbers apply across libraries and cannot be changed by the user.
Cable and connector references are necessary for explicitly defining which connection options are available for cables, connectors/adapters in AND.

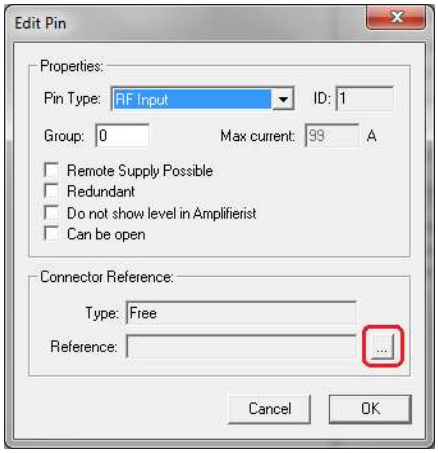
To uniquely identify cables and connectors, one unique reference from a list can be assigned to each type.
This reference can subsequently be replaced by another.
Whenever a new connection, connector or cable is generated, it initially has no reference.

After you have created a new connection, connector or cable, dialog boxes are provided for selecting a reference from a large selection of references.
Click this button to make your selection in the product-specific dialog boxes.

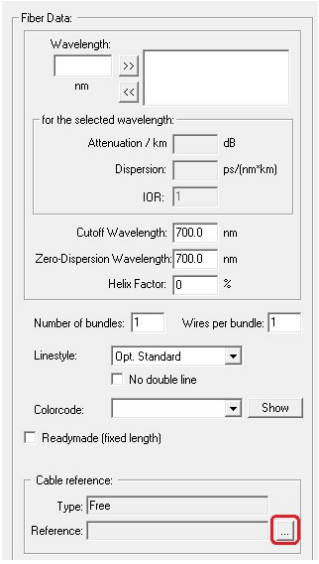
for RF-cables



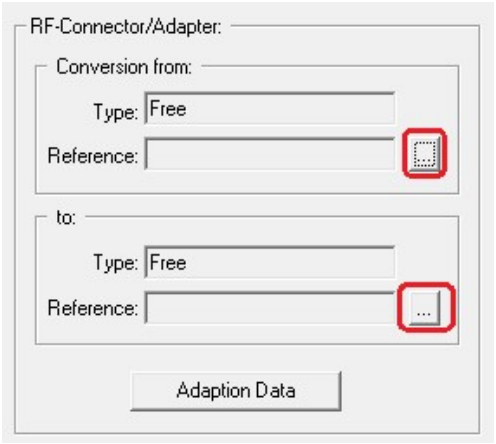
for symbols



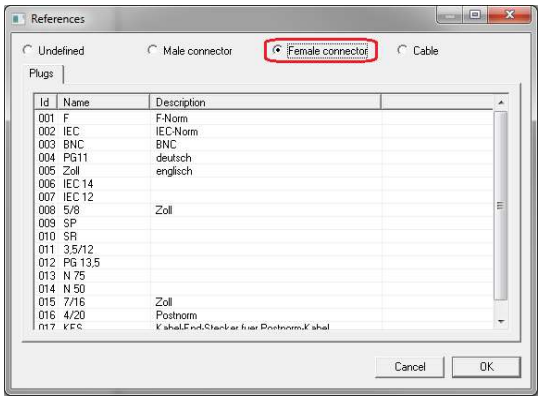
for fiber-optic-cables

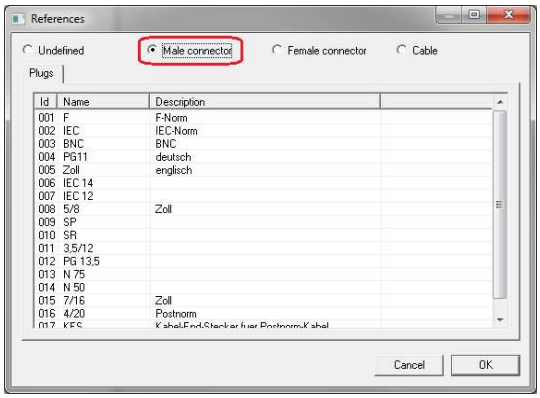


for connectors/adapters

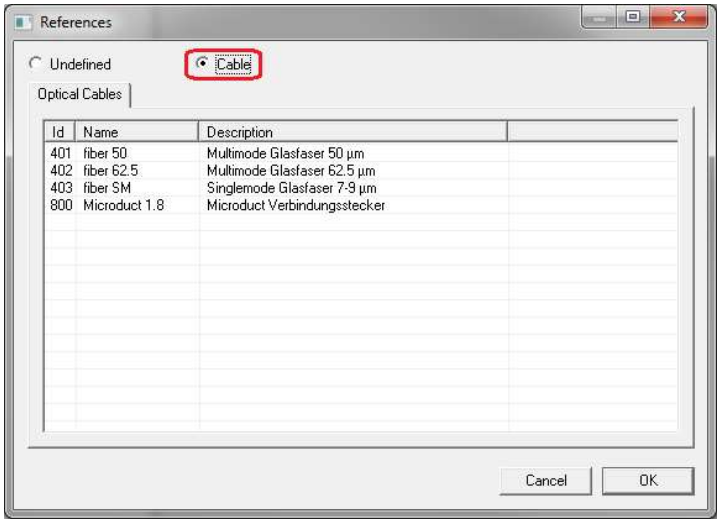


The following selections are available to you for the types RF male/female connector:

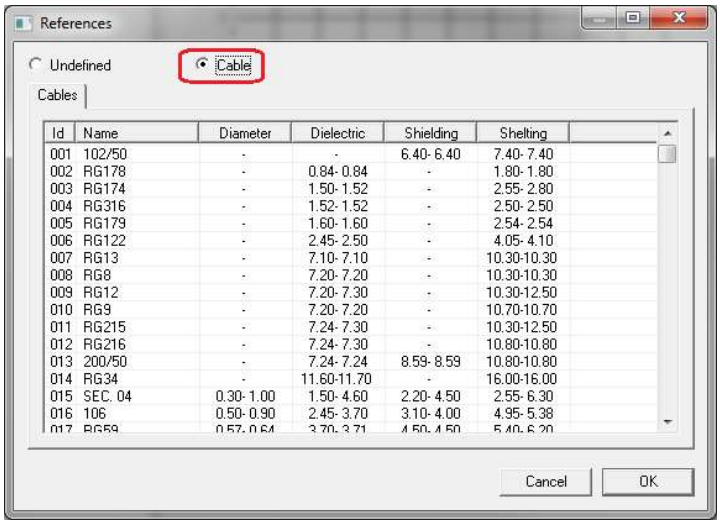




Make your selection from this reference table for optical connector components.

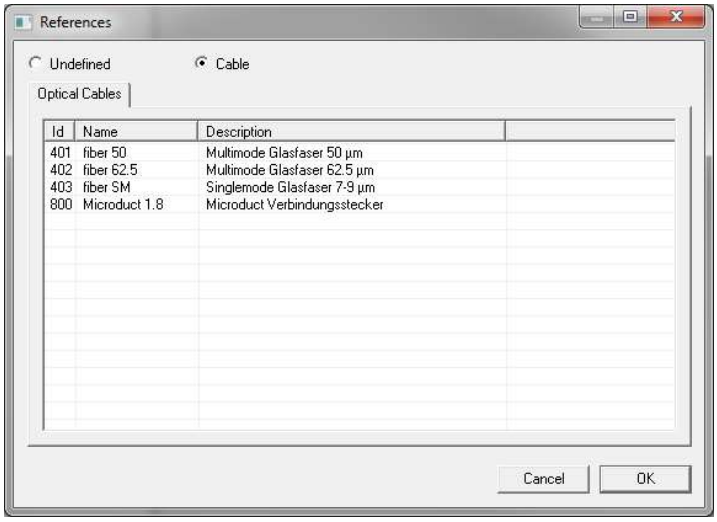


RF cables are identified by their different dimensions.



The assignments of the various cable dimensions are given in the reference table.
The internal reference numbers are listed in the "ID" column.

AND makes the necessary connection between cable and matching connector/adapter based on these numbers.
Select a reference from this table for fiber-optic cable.
This reference describes the type and dimension of an individual fiber.



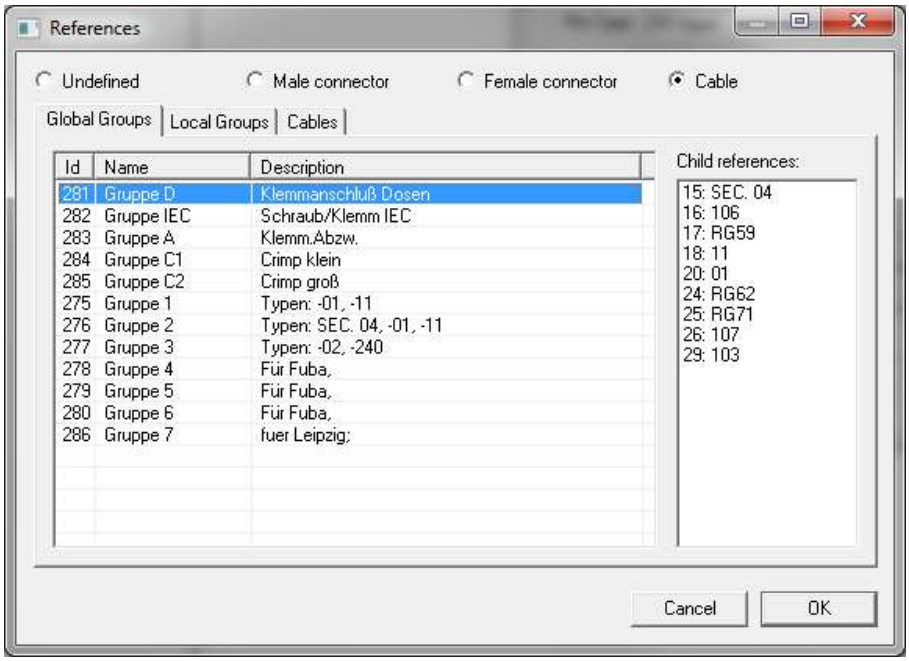
Note: If you do not define any references (undefined), AND may define one of these information types if these products are used.

2.5.8 Reference groups

Reference groups can be generated for each library.

We distinguish between two groups:

The global group



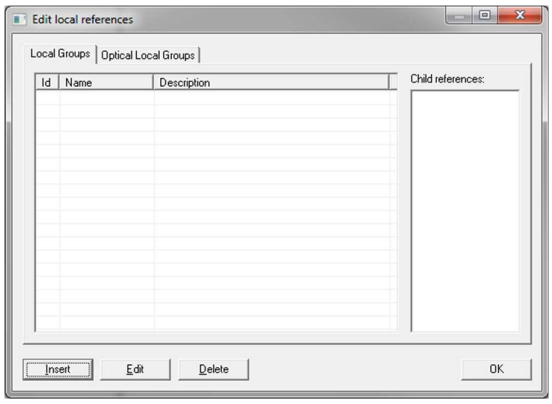
This is where references are logically combined.
That is, references combined in this group represent cables of similar dimensions,
which due to their mechanical properties can be connected to a connection with the same ID.
This is displayed accordingly in the right-hand column “Child references”.
These groups apply across libraries and cannot be changed by the user.

The local group

This group can be freely edited by the user.

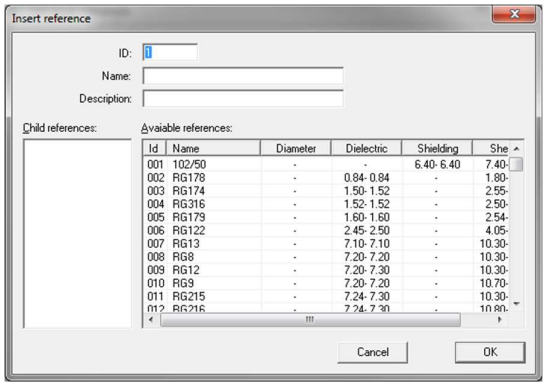
2.5.8.1 Creating reference groups

You can open this dialog box with menu item “Edit local references”

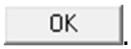


You can open another dialog box with the 

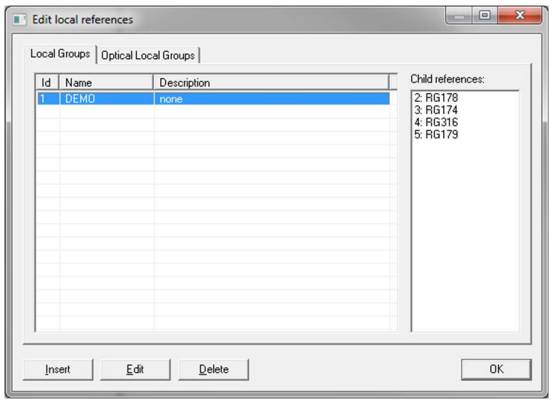
button,
which allows you to create a new local group.



A new ID is automatically assigned to this group.
When you have made your entries under “Name” and “Description”,
the necessary references to this group are added with a double-click.
You can move down the list with the scrollbar.
If you want to remove a cable, delete it by double-clicking it in the list.

When you have completed your entries, confirm with 

.
Now you can either create a new group, continue editing the group or close the dialog box.



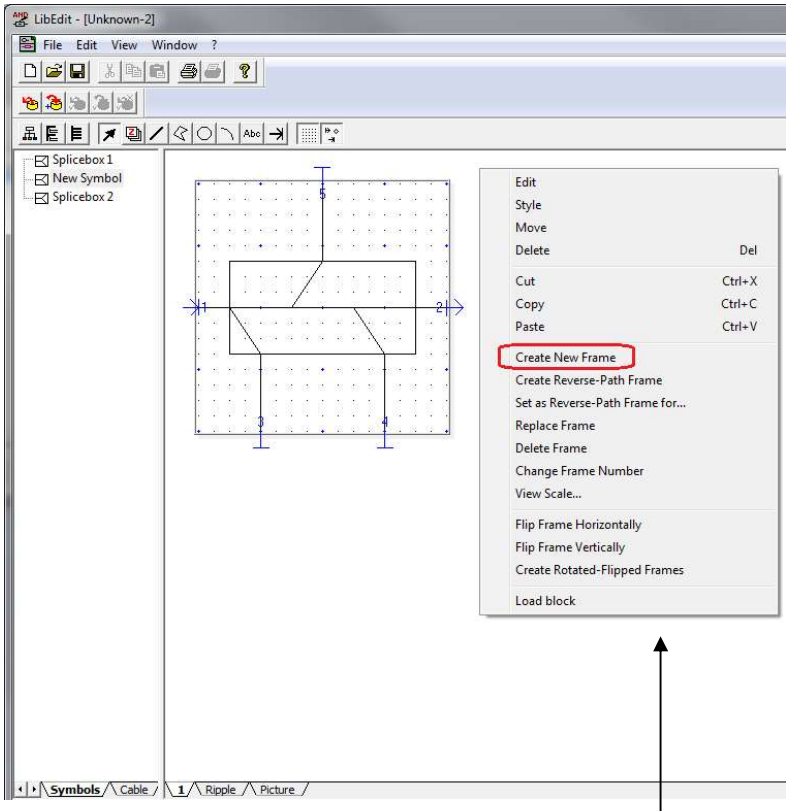
Local groups can also be created for optical cables.

2.6 Frames

Frames allow you to display a symbol in different shapes and sizes.
By displaying a different frame, for example, a tap, you can also prevent cable crossing in AND.
It is possible to generate several frames for each component.
The frames of one particular symbol do not all have to be the same size or have the same number of pins.
Try to avoid altering frames, especially the number and position of the connections.
All changes are executed in AND and may generate error messages.

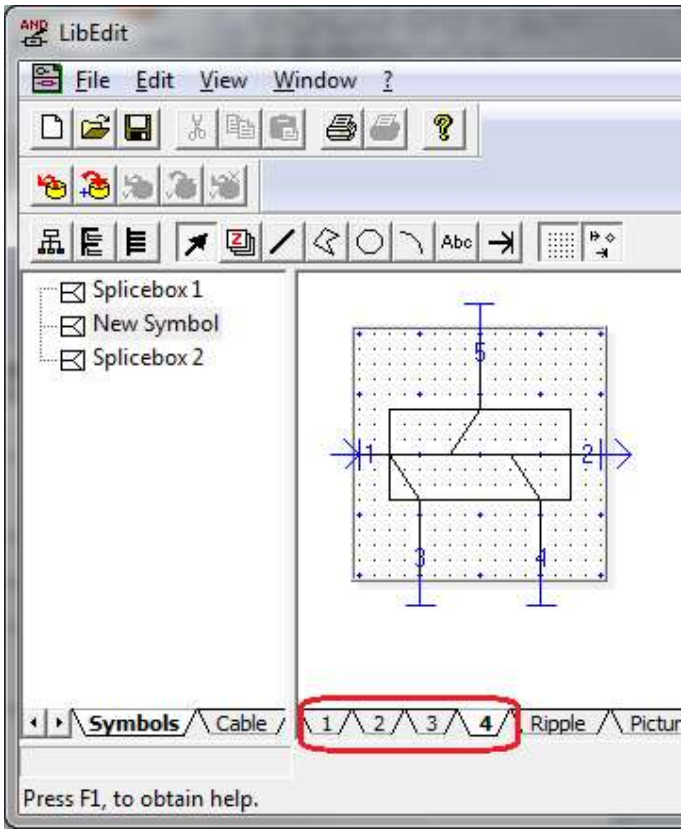
2.6.1 Creating a frame

You can generate a new frame in the left-hand section by selecting an object for which you want to generate a new frame.
In the drawing (center section), open the context menu with a right mouse click and select the "Create New Frame" menu item.



Context menu for symbols.

When you have confirmed the context menu item, the drawing of the current frame is applied and the next frame is generated.
You can create any number of frames in this way.



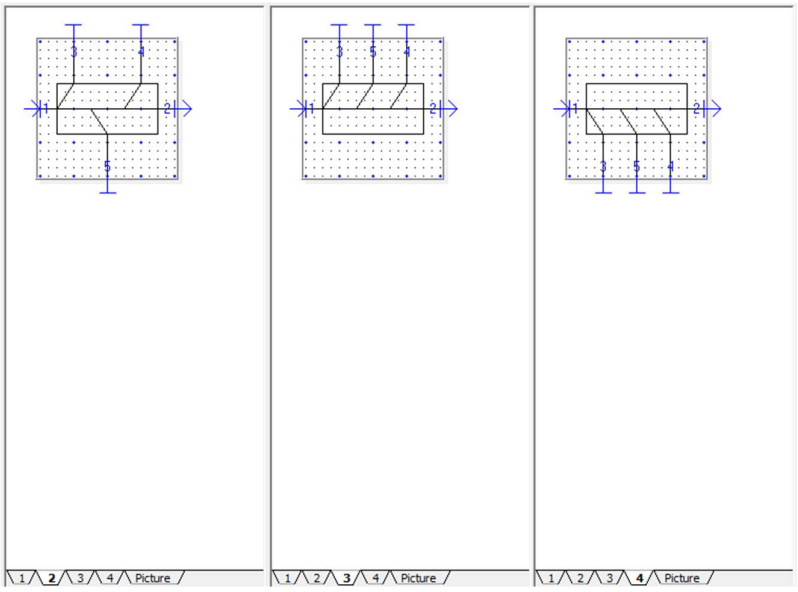
3 additional frames have been created

You can now start editing the newly created frames.

For example, you can change the shape and color or merely the direction of the inputs and outputs.

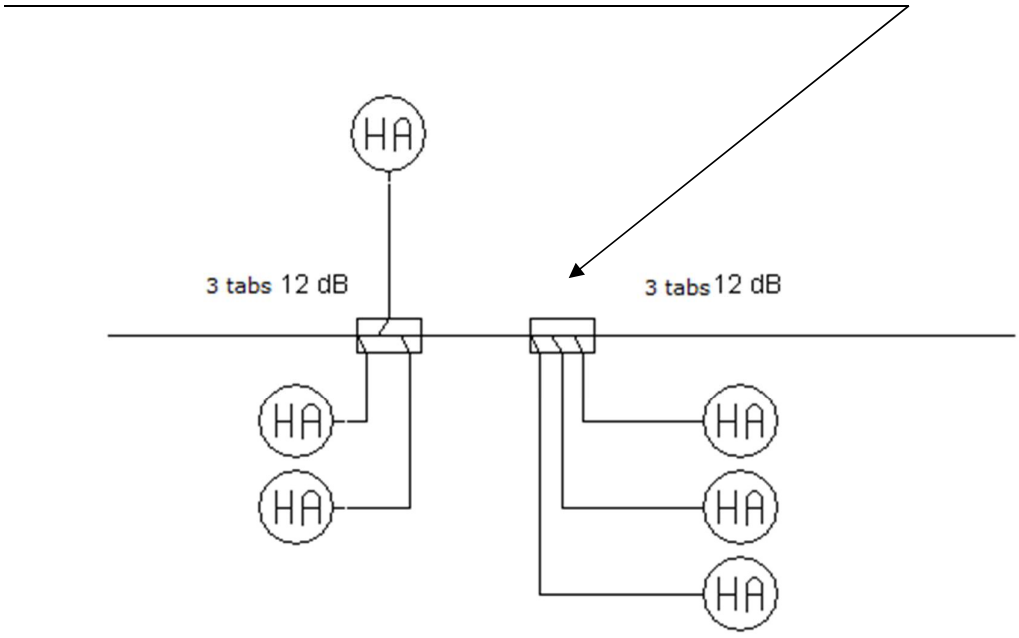
All connection information from the 1st frame is transferred 1:1 and

remains unchanged in all frames.

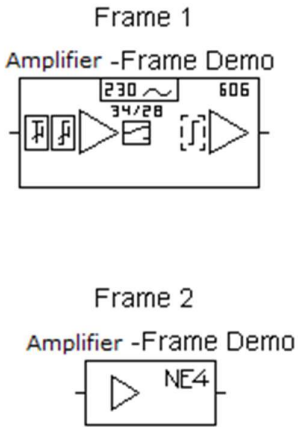


Examples in AND

Here, Frame 4 has been selected to avoid cable crossings.



Frame 2 shows the smaller form of representation of an amplifier.



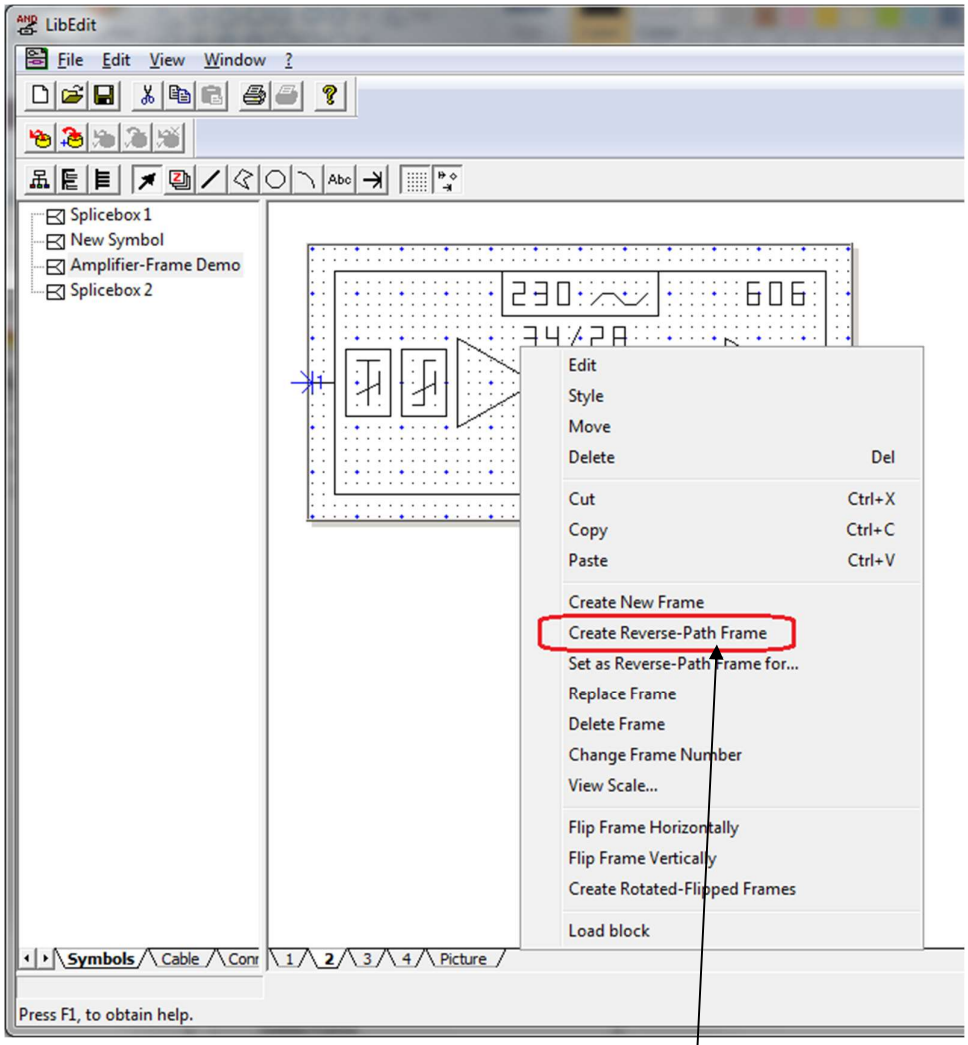
Be careful with Frame 2.

The connections have different positions.

2.6.2 Creating reverse-path frames

Amplifier symbols can be extended with reverse-path frames.
These reverse-path frames are informal in character and have no effect on the electrical properties of the amplifier.
These are defined using the data packet.
Where amplifiers have a socket for the reverse-path amplifier, AND automatically uses this reverse-path frame instead of the normal frame as soon as a reverse-path amplifier module is used.
You can define one reverse-path frame for each amplifier frame.

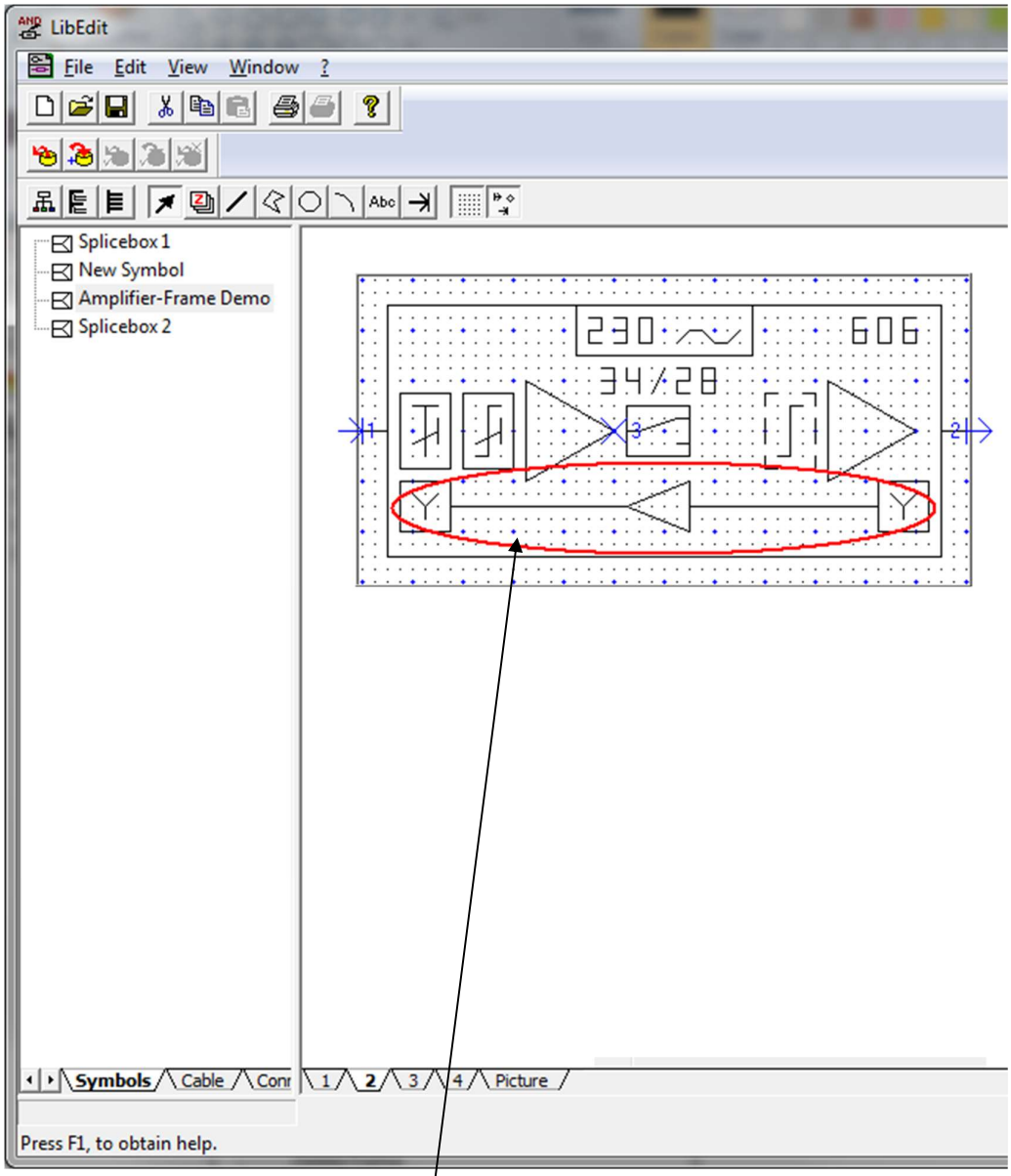
To create a new reverse-path frame, select the amplifier from the toolbar (see left-hand section), then, if necessary, switch to the required frame and select menu item “Create Reverse-Path Frame” from the context menu of the drawing area (center section):



Create Reverse-Path Frame.

A new frame with the same drawing and the same connections is generated exactly as it is when a normal frame is created.
You can now modify this frame by graphically drawing the reverse-path in it.

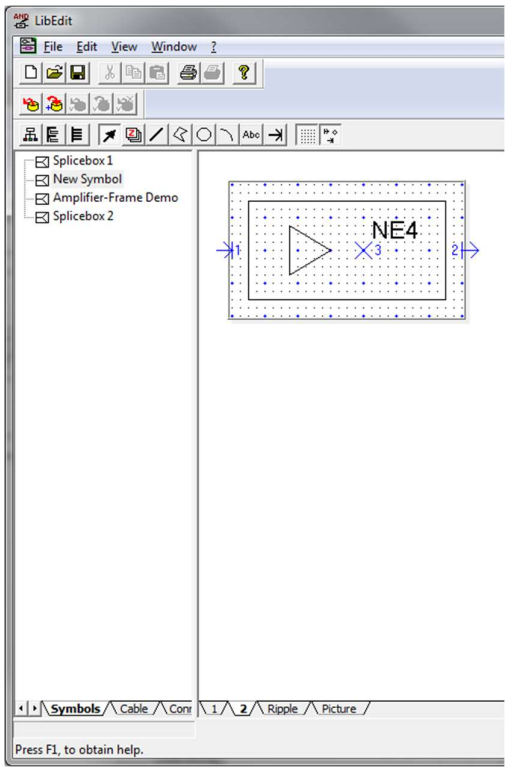
Example:



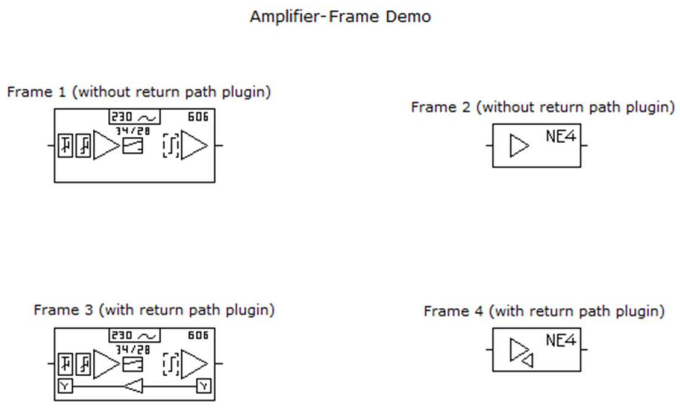
Defined as a reverse-path frame

You cannot generate, move or delete connections in a reverse-path frame, as they have to be identical to the connections of the first frame to which this reverse-path frame belongs.
As soon as you change the connections in the first frame,
these changes are automatically applied to the reverse-path frame.

Here, a second frame (smaller) and a matching reverse-path frame has been generated:



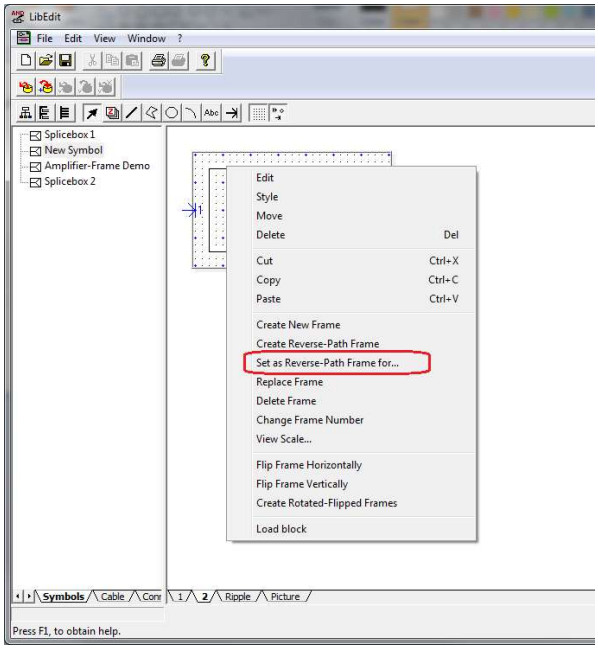
If you use this amplifier in AND, AND either shows the reverse-path frame or the normal frame, depending on whether a reverse-path amplifier module has been inserted or not.
At least one reverse-path module must be defined for this amplifier.



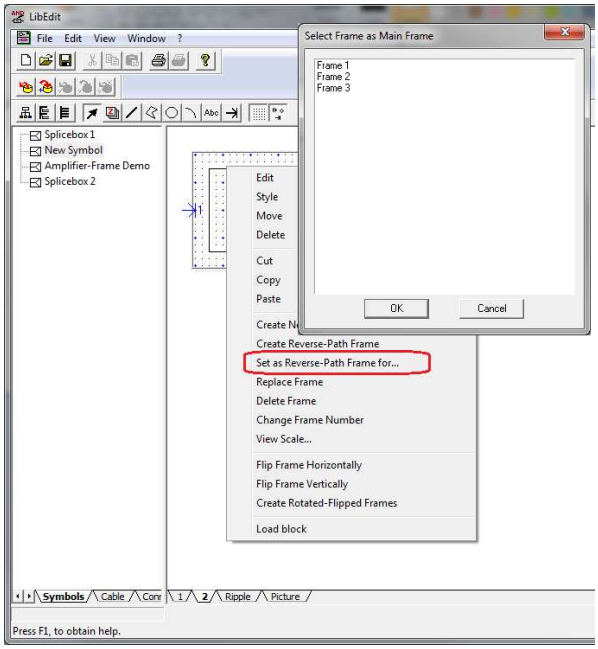
You can now select 4 types of representation for an amplifier,
where Frame 2 and Frame 4 are automatically generated if you select a reverse-path module.

2.6.3 Set as Reverse-Path-Frame for…

As earlier versions of the component editor did not include a reverse-path frame, many users solved the problem by drawing a second frame with a reverse path which they replaced manually in AND whenever they required a reverse path. These frames can now be converted to reverse-path frames in older libraries. To do this, choose menu item "Set as Reverse-Path Frame for..." from the context menu



Frame 2 (drawing 2) is converted to a reverse-path frame when you confirm the action. Now update the symbol graphically. If there is more than one frame without a reverse-path frame, you will be prompted to select the corresponding frame for this reverse-path frame



After confirmation, Frame 3 will become the corresponding frame for Frame 1.

2.6.4 Inserting a frame

This function is used, for example, to replace the current symbol drawing with another.

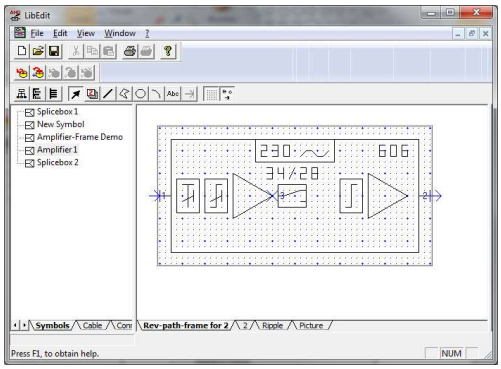
Use the copy-and-paste function to copy a product to the clipboard.

The "Insert frame" function only copies the symbol and all its frames from the clipboard.

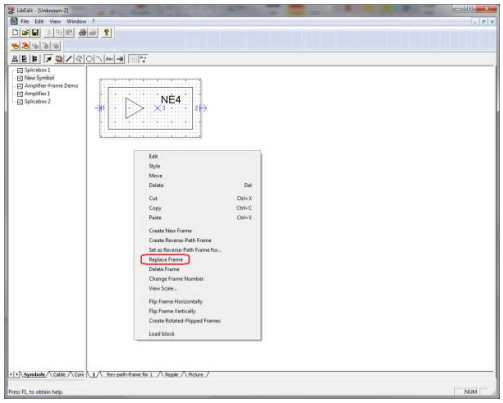
This resets reverse-path frames.

The symbol is copied with all its connections. All other data remain unaffected by this action.

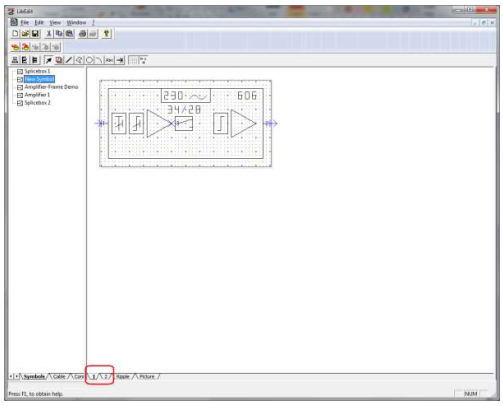
- Copy a symbol to the clipboard.



- Call up the product whose symbol drawing you wish to replace.
- Press the right mouse button in the center section and choose the appropriate menu item.



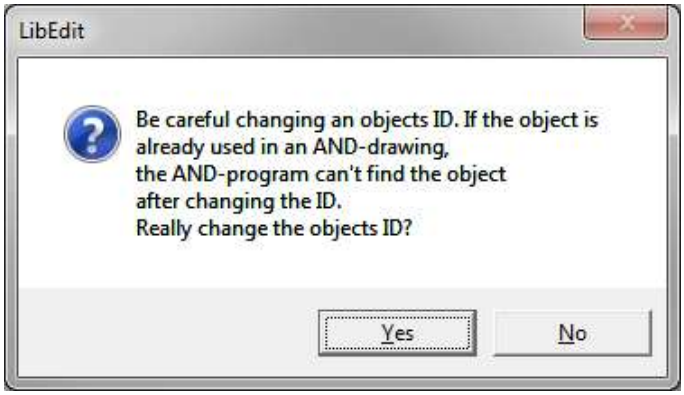
The frame is replaced.



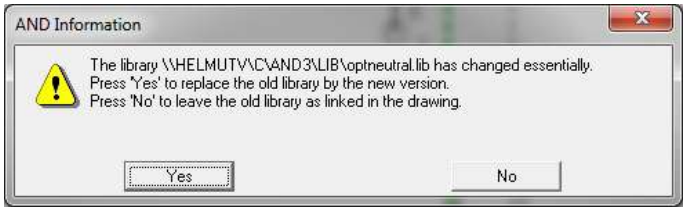
The reverse-path frames of the source symbol have been reset.

Warning:
You must never graphically change components that you have already used in AND,
delete them from their symbol location or assign another object ID number to the object in the editor!

Warning in the AND Component Editor:



Warning in AND after making changes in a library:



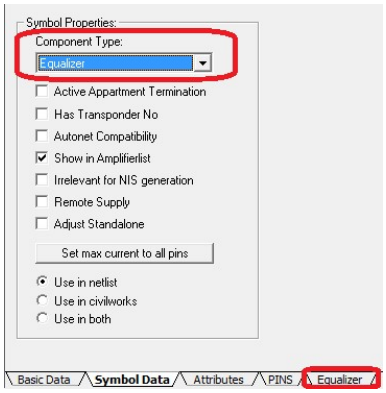
If you were now to click the Yes button in AND,
serious errors would occur in the drawings that you have already created.

However, if you confirm the AND message with No the drawing will be correctly loaded together with its originally linked libraries.

3 Entering / Modifying the Symbol Data

3.1 Creating data packets

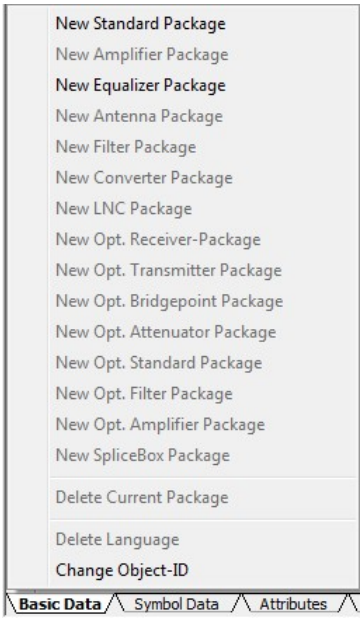
After definition of the component type, the corresponding data packet is generated immediately.



A data packet specifies the RF data between different connections.

If only one data packet is defined, it applies to all outputs opposite the input.

To create additional data packets, press the right mouse button in the Basic data view to open the following context menu:



The active fields depend on the defined component type.

Depending on which component type you have selected under “Symbol Properties”,

the various fields of the menu shown above are active.

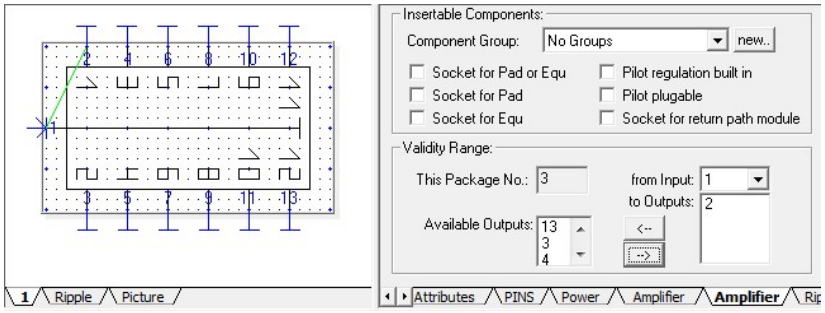
The user, tap, and loop-through connections are exceptions.
These are usually specified by entries in the "Edit pin" dialog box.
The attenuation entered applies to the entire frequency spectrum.

However, additional standard packets can also be created for these connection types.
The corresponding function is selected in the "Edit pin" dialog box.

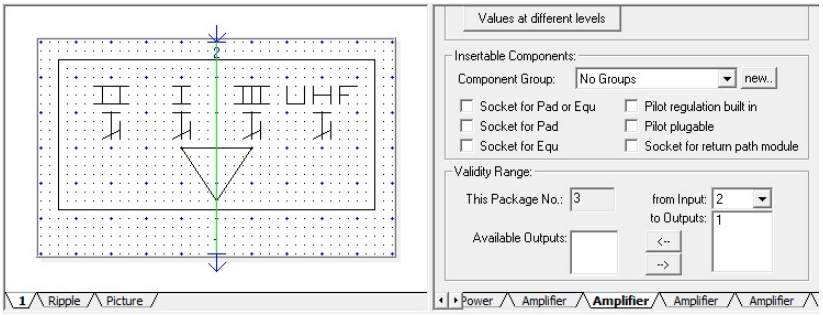
3.1.1 Assignment of data packets

If you have specified additional data packets, they always refer to specific inputs and their associated outputs.
This assignment is indicated by a marking line between the input and the output in the symbol drawing.
There are several possibilities:

- A data packet refers explicitly to one output and one input.

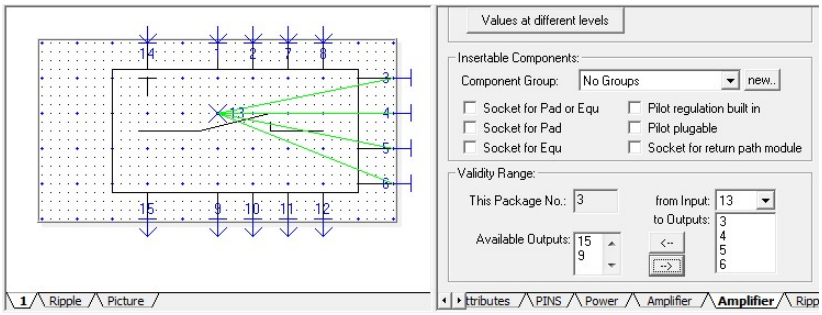


- Packets are created in such a way that all packets refer to the same connection pairs, as is required for split-band amplifiers, for example.



In this example, 4 amplifier packets have been defined for 4 different frequency ranges.

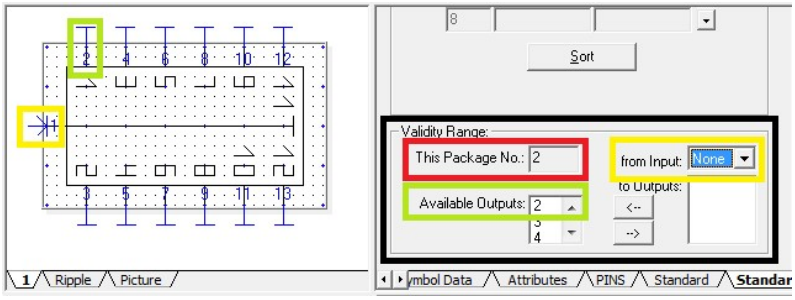
- You define a packet for several outputs with respect to the same input.
  This usually only makes sense if different groups of the same outputs are required.
  In this case, for example, for the subscriber outputs for multi-switches.



You can set "RF isolation points" for the internal component structure and use them both as a packet input and output.

3.1.2 Validity range

The "Validity range" appears at the bottom of the right-hand section of any data packet that can occur more than once or in combination with other packets.
This is where you define the association of the packet to the connections.
This is only possible if more than one packet has been created.
If only one packet exists, it applies to all outputs opposite the input.



To define the validity range of the data packets you have to proceed as follows:

Packet No. 1 is to be associated with connections input "1" and tap outlet "2".

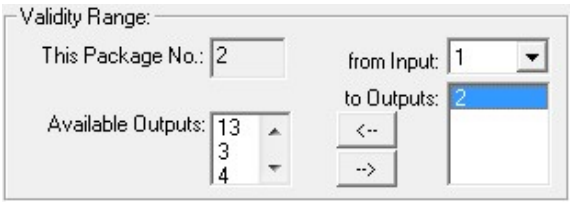
Please do not confuse with the tape labeling.

In the "Input" field, click the button.

A list of available inputs opens 

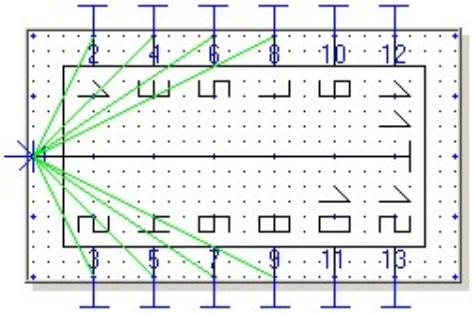
.

Once you have selected the input, you must define the output of the validity range.
To do this, select the relevant output in the *"Available Outputs"* field.
If the output you require is not available, scroll up or down until it appears in the selection window.
Clicking the --> button moves the selected output into "to Output".



At the same time, this validity range is made visible by a marking line, as already described.
If you wish to expand the valid range to include more outputs,
select the relevant outputs from the *"Available Outputs"* field and continue as described above.

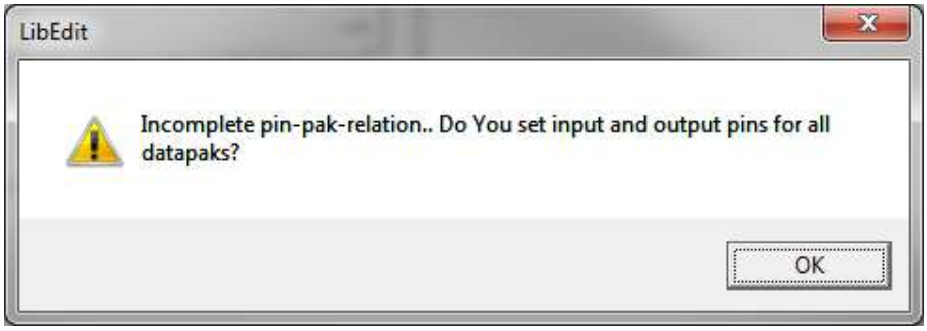
The selected outputs are marked accordingly.



If you want to remove individual connections from the validity range,

select a connection and click the <-- button.

The connection is now displayed again in the "Available Outputs" window.
Now select all the data packets one after the other and define their validity ranges.
If one of the data packets is not assigned to a validity range, the following message appears.



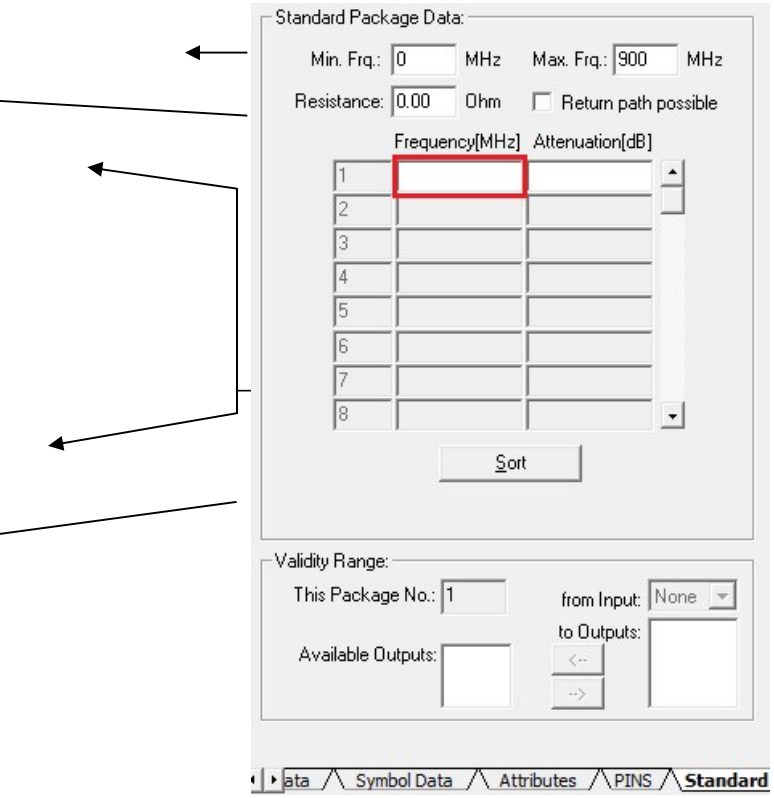
3.2 Creating a data packet RF standard

Data packets of type Standard are *RF data packets* and can be used with the following component types:

Termination/tap box	Equalizer	Multi-tap
Optical receiver	Optical transmitter	Optical amplifier
Passive component	Wall outlet	Tap-off
Converter	Connecting element	Amplifier
Splitter	Filter	

If you click this packet, the "Standard Package Data" dialog box opens

Input options:



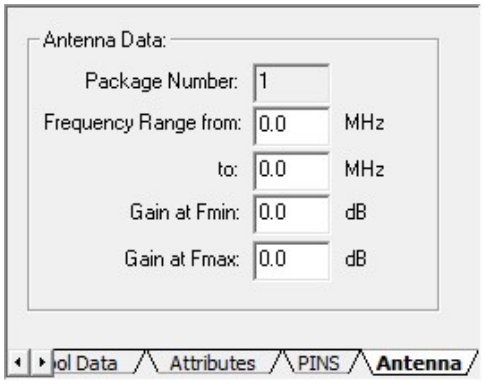
„min .Frq.:" and *„max. Frq.:"* of the forward path

„Resistance"

Attenuation values in the forward path (value pairs)
Click into the first frequency field (marked red here), enter the frequency and switch to the next column with the key to enter the appropriate attenuation value.
This automatically releases the next row.
The Sort button sorts the value pairs depending on the frequency.

„Validity Range"

3.3 Creating a data packet for an antenna



Component type "Antenna" has been selected.
You can now make the following entries in the input form shown in the figure above:

Antenna data:

"Package Number:"
This field is not active and cannot be edited.

"Frequency Range from: ..... MHz"
Here you define the beginning of the receive range of the antenna.

"Frequency Range to: ..... MHz"
Here you define the end of the receive range of the antenna.

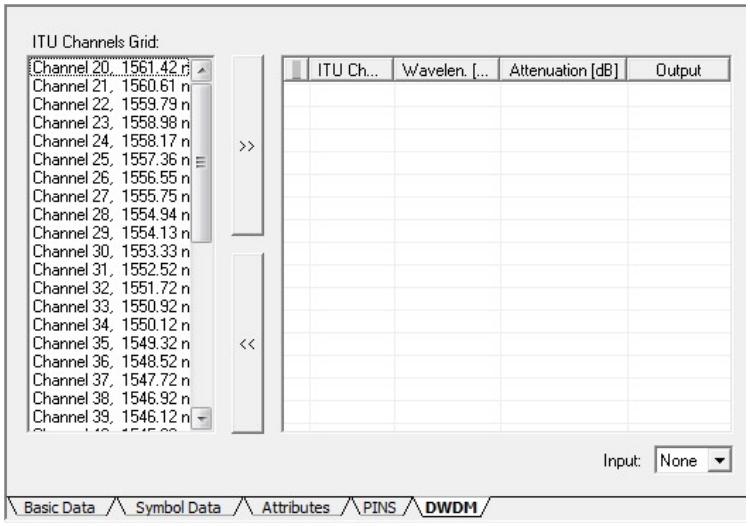
"Gain at Fmin.: ..... dB"
Gain of the antenna at the lower receive frequency.

"Gain at Fmax.: ..... dB"
Gain of the antenna at the upper receive frequency.

You can only create one data packet of type "Antenna".
Please note that no supplementary data packets, such as standard packets, are possible.

3.4 Creating a data packet for DWDM

One method for boosting the capacities of fiber-optic transmission systems is wavelength division multiplexing.
Here you can create a **D**ense **W**avelength **D**ivision **M**ultiplexer (DWDM) / Demultiplexer (DWDDM).



Component type "DWDM" has been selected.

Before you can define a DWDM data packet, the symbol drawing must have at least one optical input and one optical output.

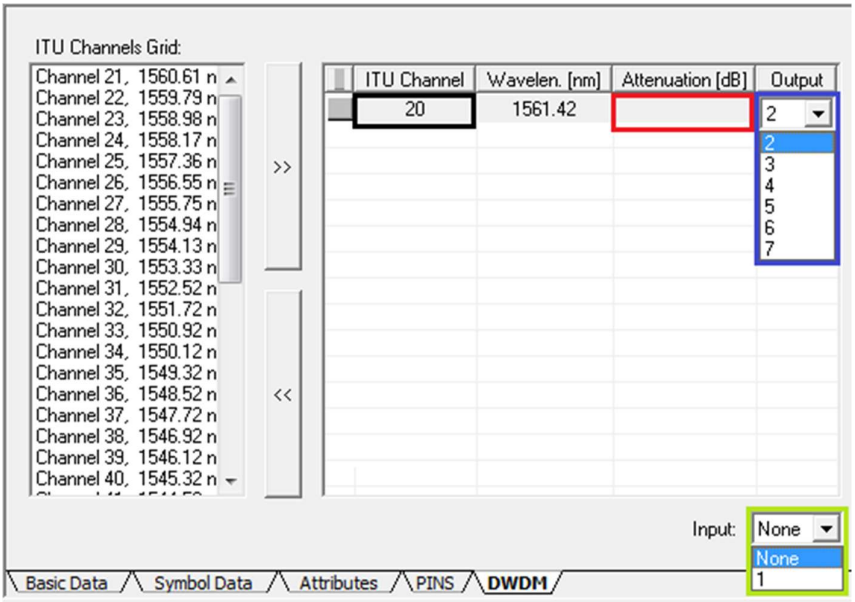
You now have the following input options in the input form.

"ITU-Channels Grid:"

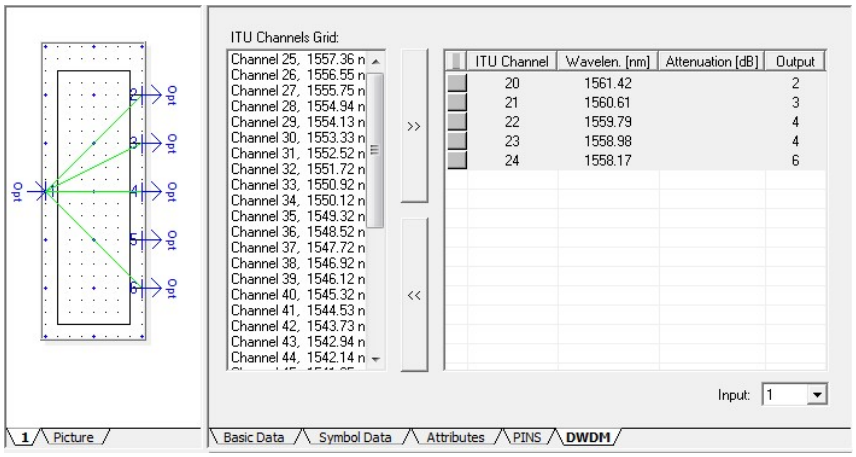
This list contains the defined ITU channel raster from which you can move individual channels to the channel table.

Move buttons

With these buttons you can move ITU channels to the right-hand table or remove them again. You can speed up this action by double-clicking the ITU channel you want to move.



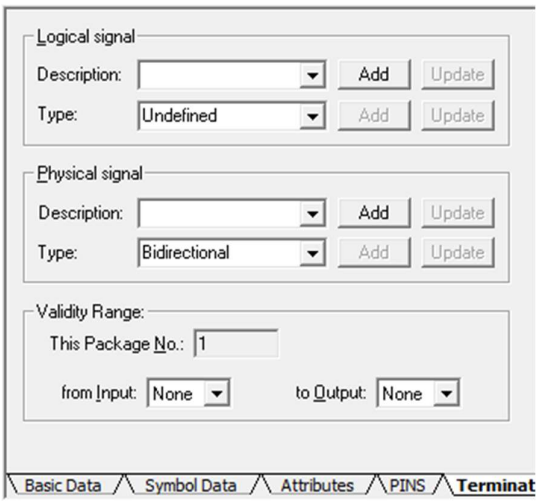
This is where you define the *attenuation* of the selected *ITU channel* between an *input* and the previously defined *outputs*.
All outputs lead to an input!



A DWDM (multiplexer) is also used as a DWDDM (demultiplexer).
Optional standard packets are possible as supplementary data packets here.

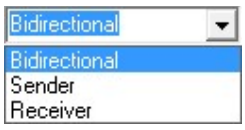
3.5 Creating a data packet for a terminal

In Version 3.3 and higher, AND and LibEdit support symbols for terminals.
These symbols represent optical devices of an unknown type or devices that do not process an analog signal.
Terminals are used to terminate the end of an optical cable.



Component type "Terminal" has been selected.
You can enter a description and type for the logical and physical signal in the terminal data packet.

If you use some of values frequently, you can insert them in the selection list



by clicking 

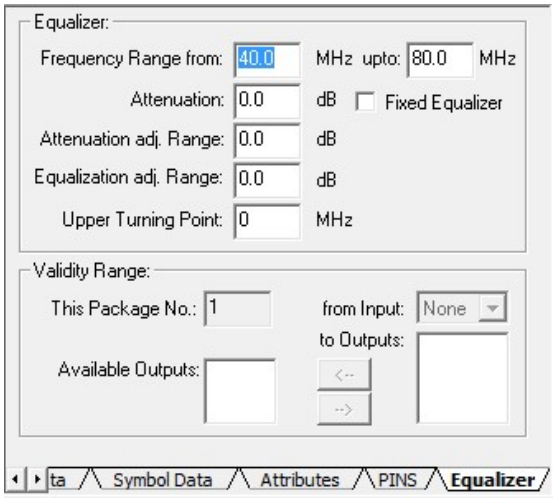
With the exception of the standard settings that LibEdit automatically makes available, your entries are stored locally in a file so that you can access them again later.

Optical transmitters (lasers) and optical receivers also have these settings.

As these objects represent physical elements, only the logical signal has to be described in the terminal data packet.
If you want AND to integrate this terminal in the optical path search you must additionally define an optical standard packet, specifying at least one wavelength.

„Validity Range"

3.6 Creating a data packet for a equalizer



This data packet is used to define an equalizer or an attenuator.

Data:

"Frequency Range from: ..... MHz upto: ..... MHz"
Here you define the beginning and end of the transmission range.

"Attenuation: ..... dB"
Here you enter the through loss (fixed).

"Attenuation adj. Range: ..... dB"
Here you enter the maximum attenuation value of the control range.

"Equalization adj. Range: ..... dB"
Here you enter the maximum equalizer value of the control range.
This value refers to the lowest transmission frequency considering the base.

If the value entered is negative, "negative equalization" with reference to the base is assumed (for example, for cable simulation modules).

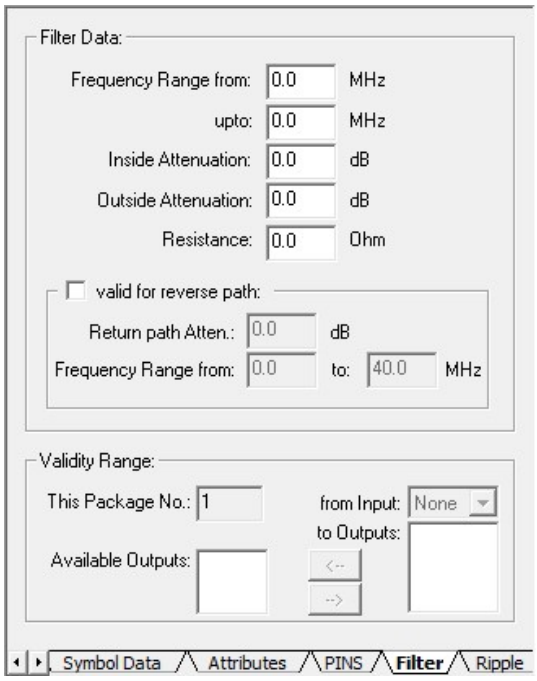
"Fixed Equalizer"

By clicking the Fixed Equalizer field, you define that it is a fixed equalizer.

"Upper Turning Point"
Here you define which frequency will be used as the base of the equalizer.

„Validity Range"
Equalizer and standard packets are possible as supplementary data packets here.

3.7 Creating a data packet for a filter



Data:

"Frequency Range from: ..... MHz upto ..... MHz"
Here you define the beginning and end of the transmission range.

"Inside Attenuation: ..... dB" (pass band)
Here you enter the attenuation within the specified frequency range (through loss).

"Outside Attenuation: ..... dB" (stop band)
Attenuation outside the specified frequency range.

"Resistance: ..... Ohm"
Remote feed impedance for this packet (if the component is power passing).
After you have selected the valid for reverse path: field, the object is suitable for the reverse path.

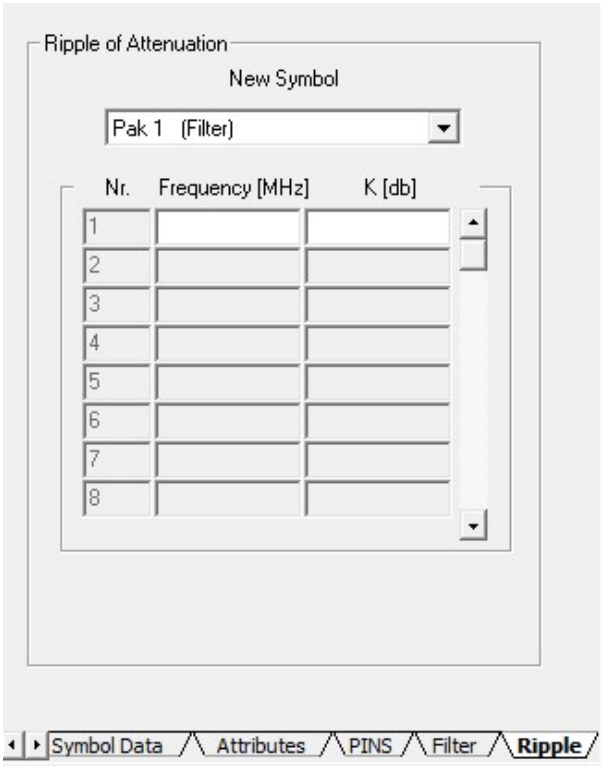
This also activates the following entry fields:
"Return path Atten.: ..... dB"
Here you can enter the attenuation of the reverse-path frequency range.

This is defined under:
"Frequency Range from: .....to..... MHz" (reverse path)

Entry of the reverse-path frequency range

„Validity Range"

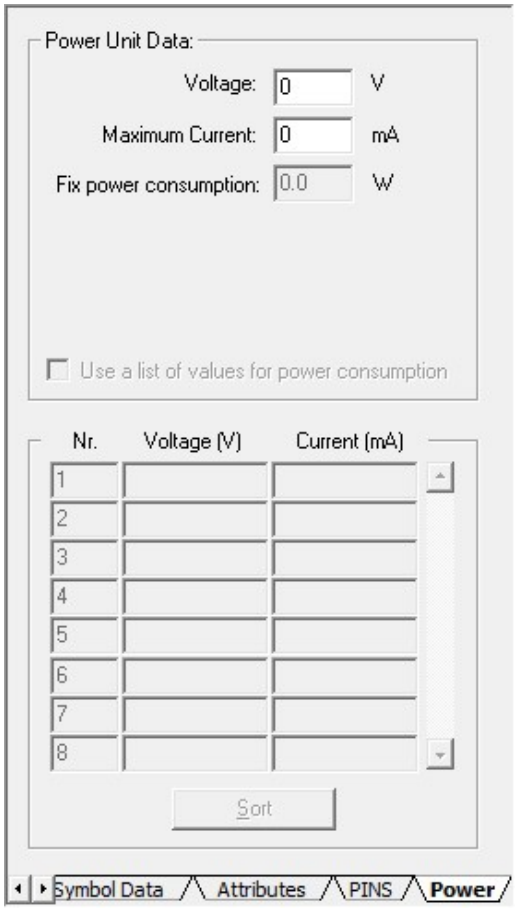
To specify filters better, you can create a frequency slope list with value pairs.



If data is entered in this frequency slope table,
it will only be used for calculations within the frequency window defined in “Filter Data”.

All data outside the frequency window is ignored.
Filter and standard packets are possible as supplementary data packets here.

3.8 Creating a data packet for a power supply unit



Component type "Power supply unit" has been selected.
This power supply unit is a *power source/transformer.*

Notes:

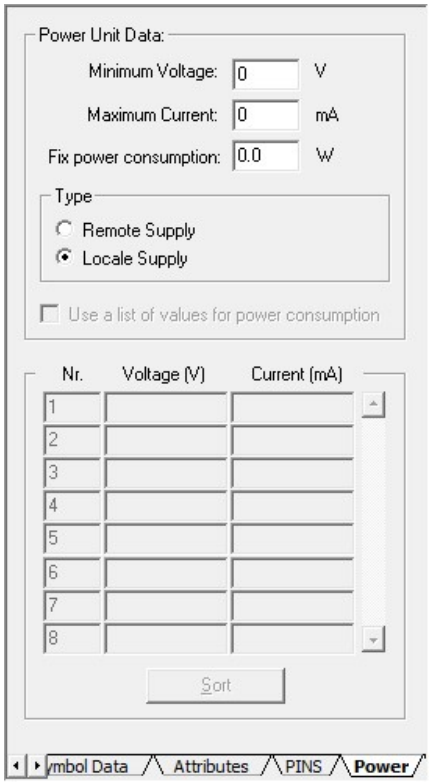
The list for the power consumption cannot be activated here.
Please note that no supplementary data packets, such as standard packets, are possible here.

Data:

"Voltage: ..... V"
This voltage is available to power power passing loads.

"Maximum Vurrent: ..... mA"
Here you enter the maximum current carrying capacity of the power supply unit.

For the following component types, a power supply unit data packet is automatically generated in addition:
amplifier, converter, optical amplifier, optical transmitter, optical receiver.
The power supply units of these types are only for powering ***loads*** and
are integrated into these components.
You can activate the list for the power consumption here.



„Minimum Voltage: ..... V"
Here you enter the minimum voltage value that is required to operate the device.

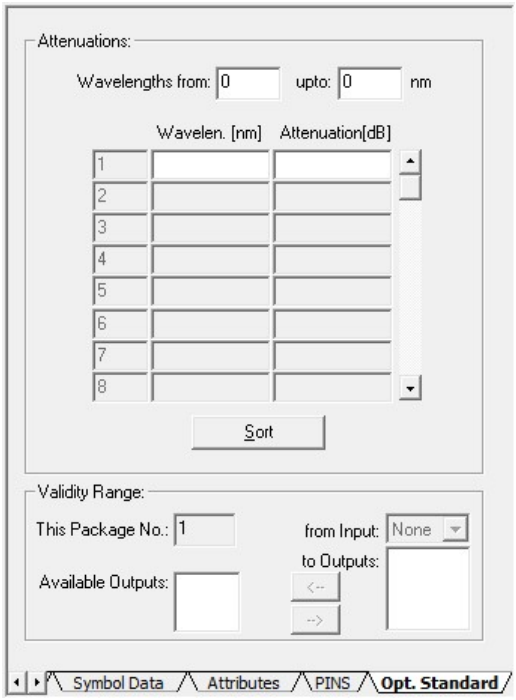
"Maximum Current: ..... mA"
Here you enter the maximum current carrying capacity of the power supply unit.

"Fix power consumption: ..... W"
Here you enter the power consumption of the device.

If you select this button (•) Remote Supply and select this check box
☑ Use a list of values for power consumption,

you can then enter the value pairs in the table that has now been released.
The previously entered values "Maximum Current" and "Fix power consumption" will now be ignored.

3.9 Creating a data packet for an optical adapter



You can set up additional data packets of type opt. Standard.

Data:

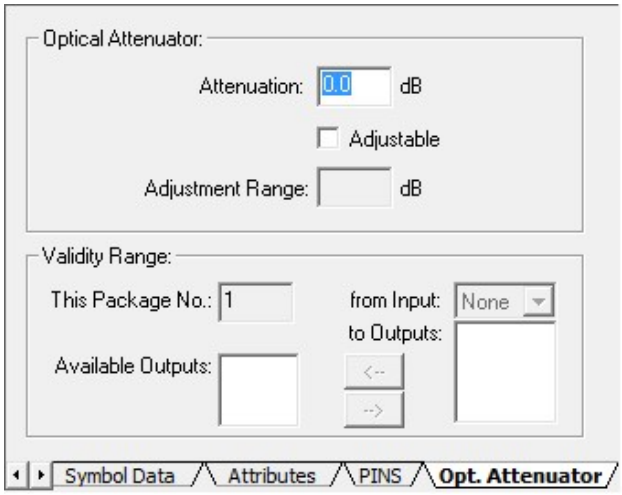
"Wavelengths from: ... upto... nm"
Here you define the beginning and end of the transmission range.

"Wavelen. (nm) Attenuation(dB)"
Here you enter value pairs that exhibit the corresponding through loss depending on the wavelength.

The Sort button sorts the value pairs

„Validity Range"

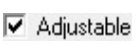
3.10 Creating a data packet for an optical attenuator



You can set up additional data packets of type opt. Standard.

Data:

"Attenuation:... dB"
Here you define the through loss. This value is static.

Selecting this check box 

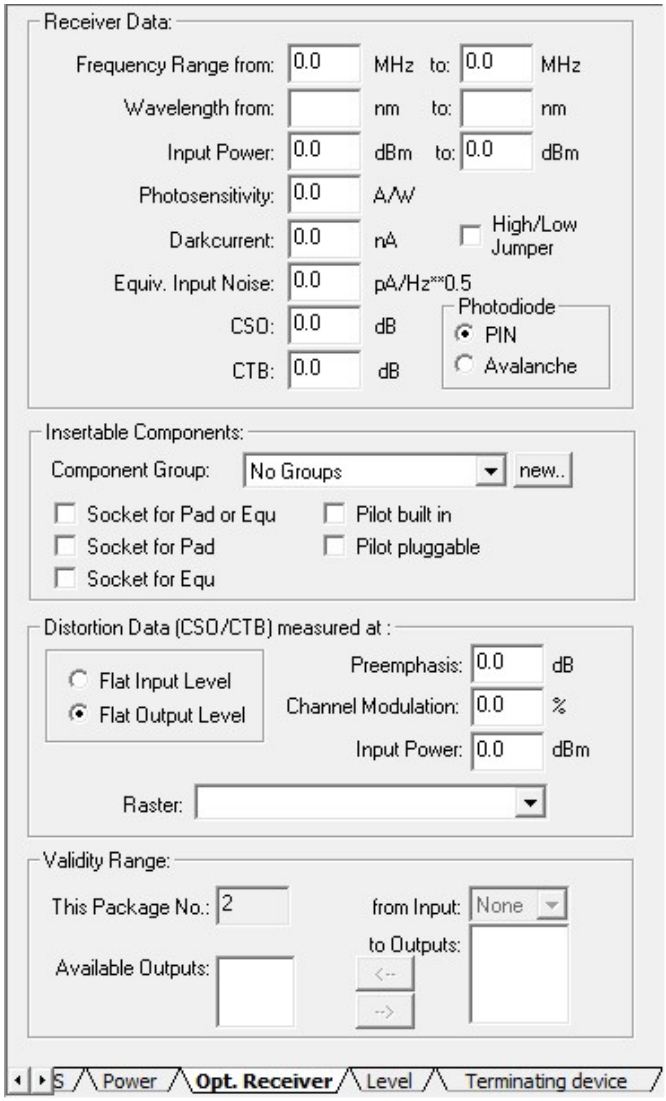
activates entry field 

Here you can define the control range.

This value is added to the through loss in AND.

„Validity Range"

3.11 Creating a data packet for an optical receiver



You can create data packets of type opt. Standard, opt. Receiver and RF Standard.

Data:

"Frequency Range from: ..... MHz to ..... MHz"
Here you define the beginning and end of the transmission range.

"Wavelength from: ..... nm to: ..... nm"
Here you define the receive range of the receiver.

"input Power: ..... dBm to: ..... dBm"
Here you define the working range of the receiver.

"Photosensitivity: ..... A/W"
Here you enter the photosensitivity of the photodiode.

"Darkcurrent: ..... nA"
Here you enter the dark current of the diode.

*"Equiv. Input Noise: ..... pA/Hz**0,5"*
Here you enter the equivalent background noise of the receiver.

Here you define the type of diode:

PIN- Diode
Avalanche- Diode

The interference data:
„CSO: …dB"
„CTB: …dB"

refer to the following settings:

Values for:

level-aligned input

level-aligned output

„Preemphasis: ... dB"

Set preemphasis at the output.

"Channel Modulation: ... %"
Channel modulation at the receiver.

"Input Power: ... dBm"
Input power at the receiver.

You select the raster here.

Under the heading “Insertable Components”, the socket options in the device are defined.

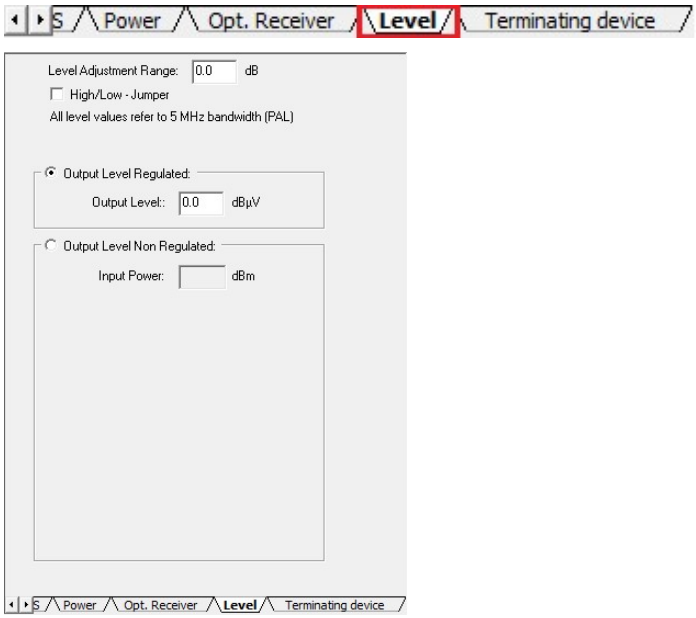
☐ Socket for Pad or Equ	This optical amplifier packet has a socket for an attenuator or equalizer.
☐ Socket for Pad	This optical amplifier packet has a socket for an attenuator .
☐ Socket for Equ	This optical amplifier packet has a socket for an equalizer.
☐ Pilot built in	This optical amplifier packet has a pilot control permanently integrated.
☐ Pilot pluggable	This optical amplifier packet has a socket for a pilot control..

Here you define the groups from which AND can select the components for the corresponding sockets.

The components selected in AND then appear both in the amplifier list and in the bill of materials.

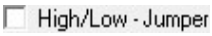
„Validity Range“

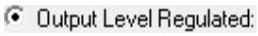
Clicking the “Level” tab takes you to the level settings of the opt. receiver.



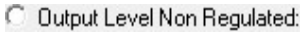
„Level Adjustment Range: ... dB“

Adjustable attenuation at the output of the receiver.

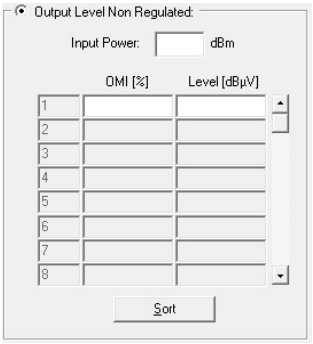
If you want to set up a „High/low jumper", select this check box 

.
This allows AND to switch between two level states.
If you select this button 

the output level entered here


remains constant irrespective of the input power.
Selecting the 

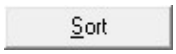
button activates further entry fields.



„Input Power ... dBm"

Here you enter the input power of the receiver.

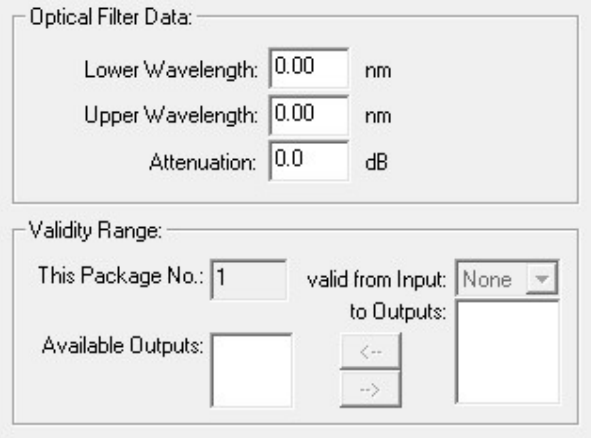
The value pairs for "OMI [%]" and "Level [dBµV]" refer to the input power entered.

The 

button sorts the value pairs.

By clicking the "Terminating device" tab, you can make entries for the logic signal.

3.12 Creating a data packet for an optical filter



When defining an optical filter, only the pass band is specified.

Data:

"Lower Wavelenght: ... nm"
Here you define the beginning of the transmission range.

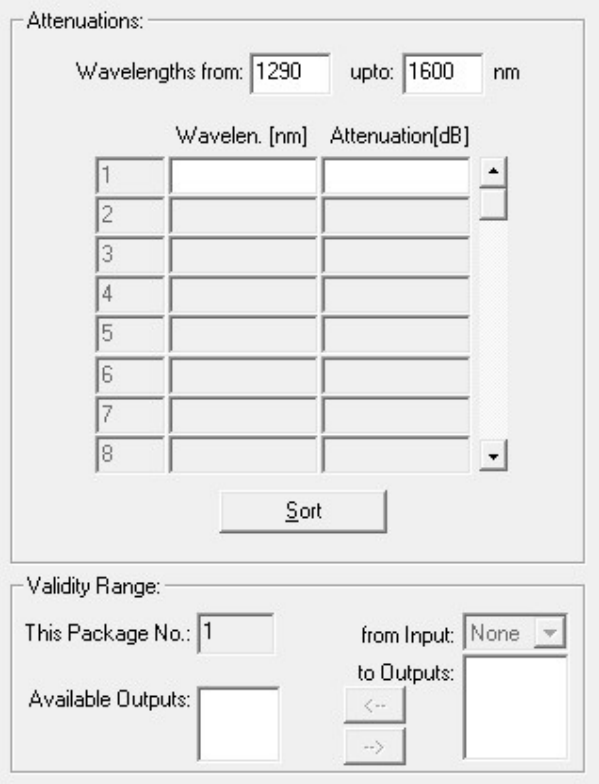
"Upper Wavelenght: ... nm"
Here you define the end of the transmission range.

"Attenuation: ... dB"
Here you define the through loss of the transmission range.

„Validity Range"

Please note that only opt. filter packets are possible here and not supplementary data packets, such as RF standard packets.

3.13 Creating a data packet for an optical splitter



Optical splitters are defined by the creation of optical standard packets.

Data:

"Wavelengths from: ..... upto ..... nm"
Here you define the beginning and end of the transmission range.

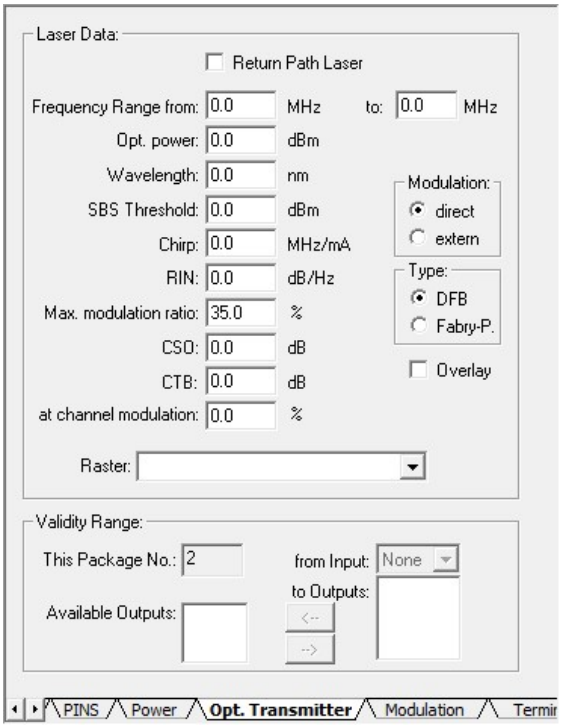
"Wavelen. (nm) Attenuation(dB)"
Here you enter value pairs that exhibit the relevant through loss, depending on the wavelengths.

The Sort button sorts the value pairs.

„Validity Range"

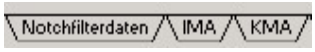
Please note that only opt. standard packets are possible here and not supplementary data packets, such as RF standard packets.

3.14 Creating a data packet for an optical transmitter



You can create additional data packets of type opt. Standard, opt. Transmitter and RF Standard.

After you have set the check mark, you define whether it is a reverse-path laser. Additional input options for reverse-path lasers are now activated.



This also deactivates the input options for “CTB”, “CSO”, “at channelmodulation” and raster selection.

Data:

"Frequency Range from: ..... MHz to ..... MHz"
Here you define the beginning and the end of the RF transmission range.

"Opt. Power: ..... dBm"
Here you define the output power of the laser.

"Wavelenght: ..... nm"
Here you define the output wave length of the laser.

"SBS Threshold: ..... dBm"
Here you enter the SBS threshold value at the laser..

„Chirp: ..... MHz/mA"

Chirp value of the laser for 

modulated lasers.

„Chirp: ..... MHz/Channel"

Chirp value of the laser for 

modulated lasers.

„RIN: ..... dB/Hz"
Here you enter the value for the noise behavior of a signal source.

"Max. modulation ratio: ..... %"
Here you enter the value for the maximum modulation depth.

The interference data

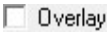
"CSO: ..... dB"
"CTB: ..... dB"

refer to the following settings:
"at channel modulation: ..... %"

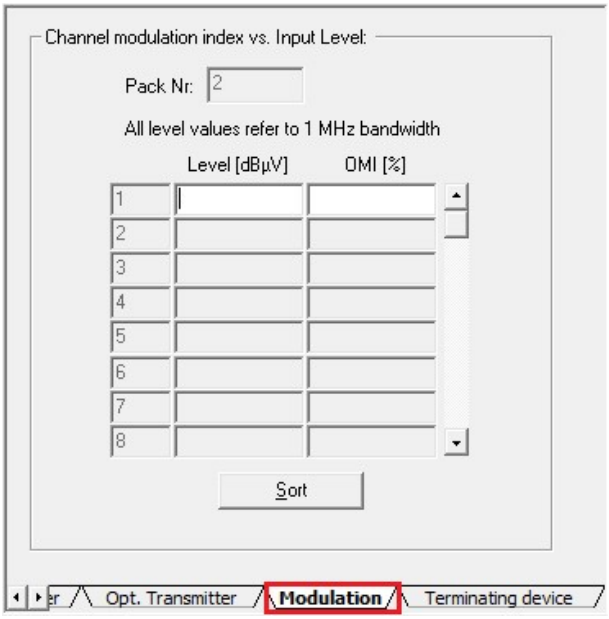
Channel modulation at the transmitter.

You can select the channel raster here.
„Validity Range"

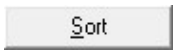
Here you can select the laser type.
This check box is currently for information only.



If it is an overlay laser, set the check mark here. This check box is currently for information only. After clicking the "Modulation" tab, you can make entries.



The values pairs establish a relationship between the RF input level "Level [dBµV]" and the modulation index "OMI[%]".

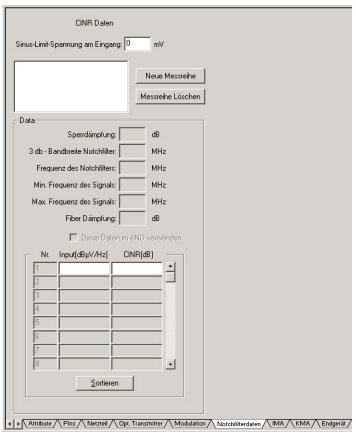
The  button sorts the value pairs.

By clicking on the "Terminating device" tab, you can make entries for the logic signal.

3.14.1.1 Reverse-path laser

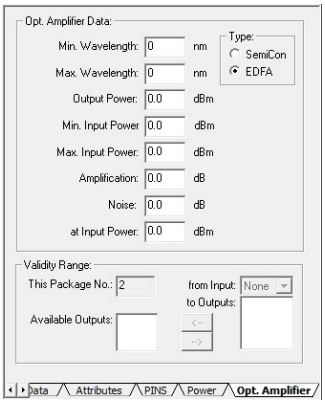
For a reverse-path laser, set the check mark accordingly in  .

After you have selected "Notch filter data" , the dialog box as follows appears:



The input forms for notch filter data, IMA and KMA measurement series are currently for information only.
The measurement series are NOT used for interference calculation in AND.
There are plans to remove the forms from the component editor.

3.15 Creating a data packet for an optical amplifier



Component type "Opt. amplifier" has been selected.
You can create additional data packets of type opt. Standard and RF Standard.

You now have the following input options in the input form shown in the figure above.

"Min. Wavelength: ..... nm"
Here you define the beginning of the transmission range.

"Max. Wavelength: ..... nm"
Here you define the end of the transmission range.

"Output Power: ..... dBm"
Here you enter the max. output power of the amplifier.

"Min. Input Power: ..... dBm"
Here you enter the min. input power of the amplifier.

"Max. Input Power: ..... dBm"
Here you enter the max. input power of the amplifier.

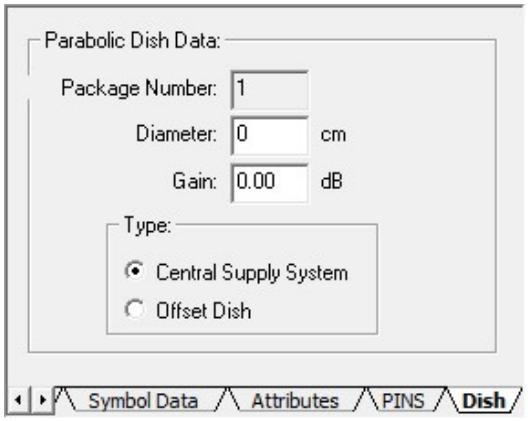
"Amplification: ..... dB"
Here you enter the max. gain.

"Noise: ..... dB"
Here you enter the noise figure of the amplifier.

"at Input Power: ..... dBm"
Here you enter the input power, with reference to the noise figure.

Here you select the amplifier type. At the moment, this check box is for information only.
„Validity Range"

3.16 Creating a data packet for a parabolic antenna



Component type "Parabolic antenna" has been selected.
You now have the following input options in the input form shown in the figure above:

"Package Number:"
This field is not active and cannot be edited.

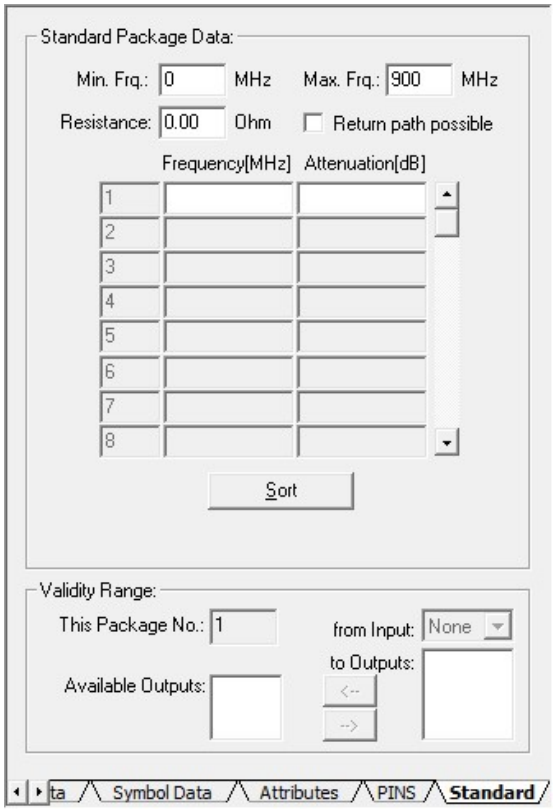
"Diameter: ..... cm"
Here you define the beginning of the receive range of the antenna.

"Gain: ..... dB"
Gain of the parabolic antenna.

Here you select the type of parabolic antenna.
At the moment, this check box is for information only.
You can only create one data packet of type "Parabolic antenna".

Please note that no supplementary data packets, such as standard packets are possible.

3.17 Creating a data packet for a passive component

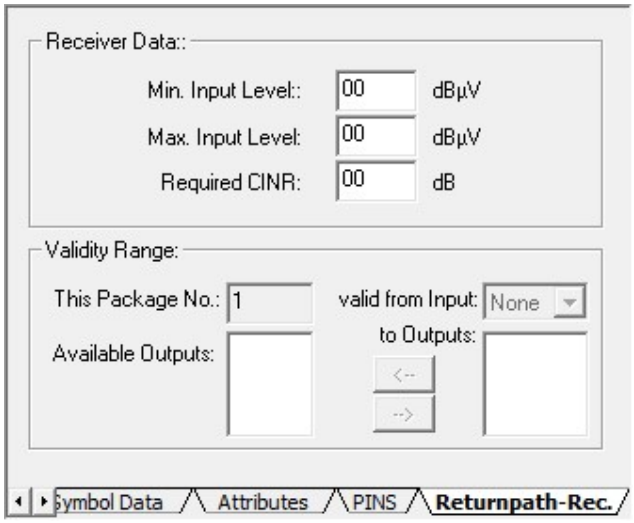


Component type “Passive Component” has been selected.
This component type is specified with “Standard” data packets.

You can create additional data packets of type Standard.

„Validity Range“

3.18 Creating a data packet for a reverse-path receiver



Component type "Reverse-path receiver" has been selected.

You now have the following input options in the input form shown in the figure above:

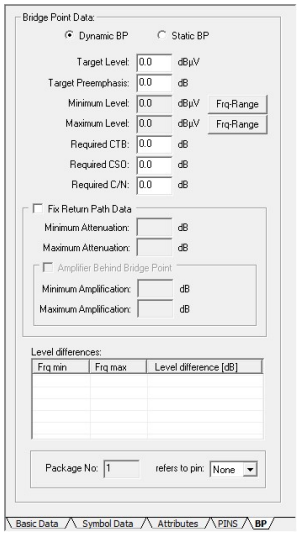
"Min. Input Level: ..... dBµV"
Here you enter the minimum receive level at the receiver.

"Max. Input Level: ..... dBµV"
Here you enter the maximum receive level at the receiver.

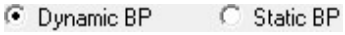
"Required CINR: ..... dB"
Here you enter the CINR ratio at the receiver.
You can only create one data packet of type "Reverse-path receiver".

Please note that no supplementary data packets, such as standard packets are possible.

3.19 Creating a data packet for a bridge point (NTU)



Component type "bridge point (NTU)" has been selected.



With this check box, you define whether this network termination unit is to be static or dynamic. If selection was "static", the entry fields are not active.

If selection was "dynamic" you have the following input options:

"Target Level: ..... dBµV"
Here you enter the recommended level at the NTU.

"Target Preemphasis: ..... dB"
Here you enter the recommended preemphasis at the NTU.

"Minimum Level: ..... dBµV"
Here you enter the minimum permitted level at the NTU.

"Maximum Level: ..... dBµV"
Here you enter the maximum level difference at the NTU.

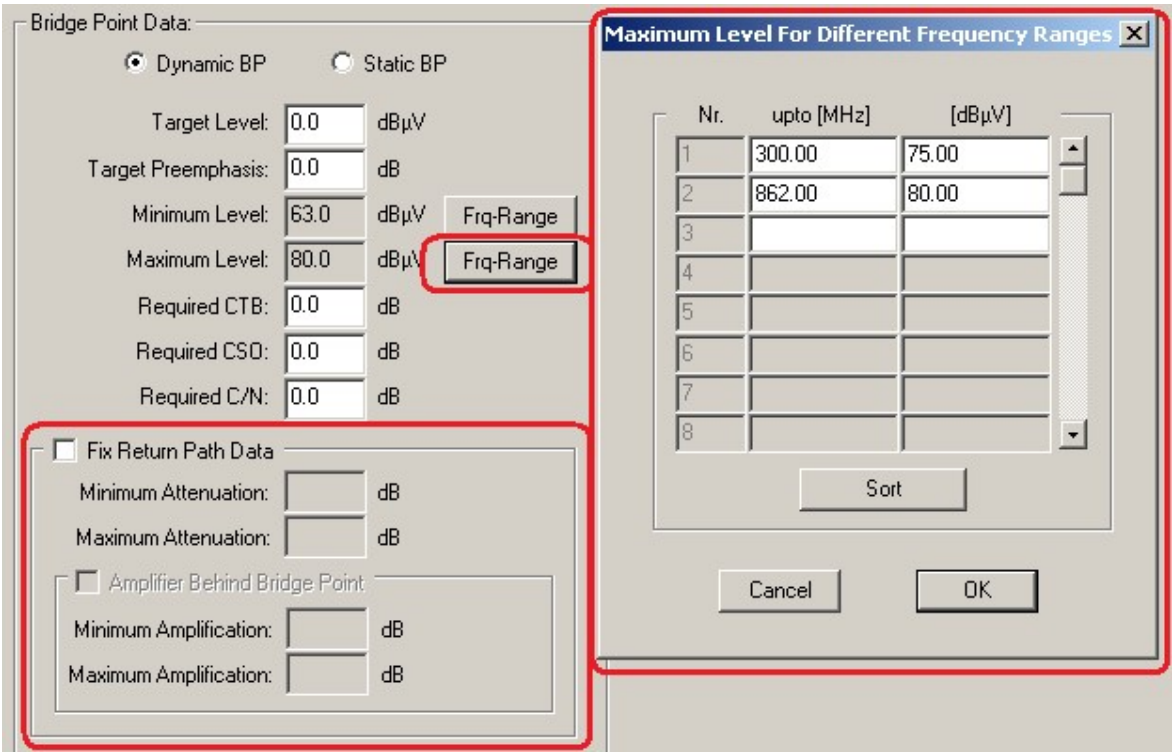
"Required CTB: ..... dB"
Here you enter the necessary CTB at the NTU.

"Required CSO: ..... dB"
Here you enter the necessary CSO at the NTU.

"Required C/N: ..... dB"
Here you enter the C/N at the NTU.
You can only create one data packet of type "bridge point (NTU)".
Please note that no supplementary data packets, such as standard packets, are possible here.

3.19.1 Library Extensions For Bridge Points

The extension is implemented from Build 4.0.765.57 on and from Build 4.1.835.0 on
The extensions in Libedit are marked in red below:



Picture 1: Screenshot from Libedit.

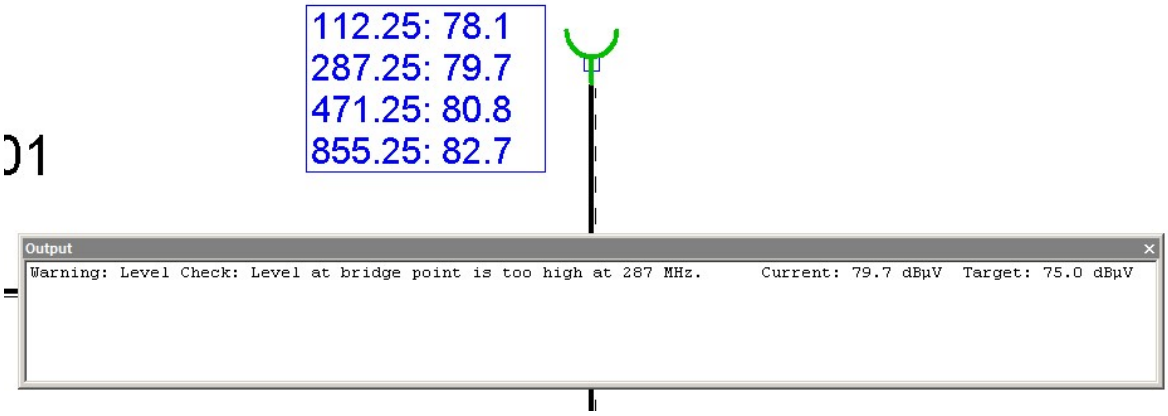
Extension 1: Maximum level of a bridge point can be frequency dependent.

The button "Frq-Range" was added at the right side of the editbox for the maximum-level.
If the button is pressed, the dialog "Maximum Level For Different Frequency Ranges" opens, where you can enter the maximum allowed levels for different frequency ranges.

Note: The entered frequency is the upper border of the range.
I.e. in the screenshot above we have:

For all frequencies f ≤ 300 MHz: Maximum Level = 75 dBµV
For all frequencies f > 300 MHz: Maximum Level = 80 dBµV

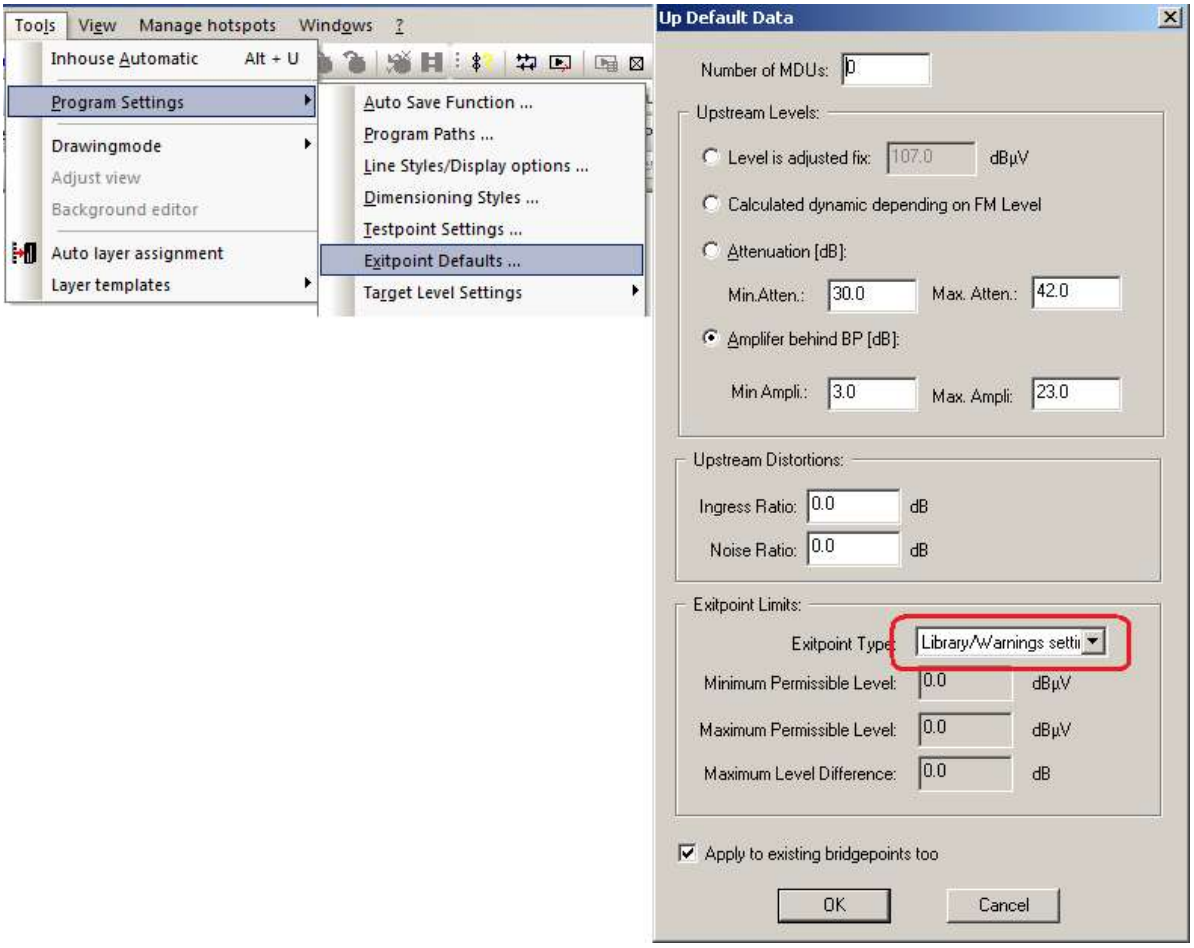
The entered maximum values are used by the level-check-function of AND's netcheck.
In our example a warning will be generated by the level-check-function,
if for a channel with frequency ≤ 300 MHz the level is higher than 75 dBµV or
for a channel with frequency > 300 MHz the level is higher than 80 dBµV.



Picture2: In the outputwindow you see the warning of the level-check-function for the green bridgepoint with limits 75/80 like in Picture 1.

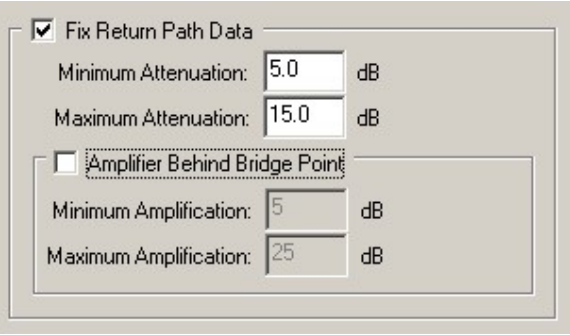
The warning refers always to the channel with the highest deviation, which is here 4.7 dB at 287 MHz.

Hint: The limit from library is only used, if you have set exitpoint type to "
Library/Warning Settings":



Extension 2: Upstream

The Bridgepoints in the library are extended by the following data:



- Checkbox "Fix Return Path Data". If this is unchecked the upstream data (attenuation range, amplifier behind, amplification range) are entered in AND.
  If this checkbox is checked, the upstream data are defined in Libedit and cannot be edited in AND.
- Editbox "Minimum Attenuation": The minimum passive return path attenuation behind this bridge point
- Editbox "Maximum Attenuation": The maximum passive return path attenuation behind this bridge point
- Checkbox "Amplifier behind Bridge Point".
  If this is checked, the adjustment of reverse amplifiers in AND will assume that directly behind the bridge point is an amplifier with amplification range entered below.
  The attenuation range is then interpreted as the passive attenuation behind the amplifier.
  If this is unchecked, the network behind the bridge point is pure passive with attenuation range entered above.

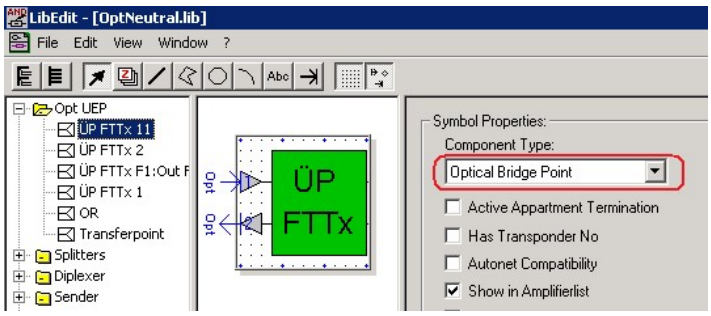
3.19.2 Signal source

Signal sources are defined directly in AND.

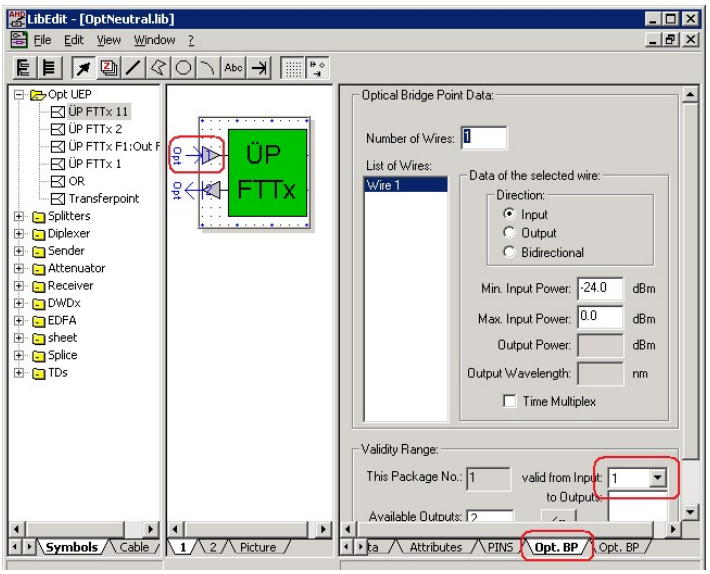
There is no way of specifying signal sources using data packets.

3.20 Creating a data packet for an Optical Bridge Point

With the component editor, you can create an "Optical bridge point” (BP):



An "Opt.BP-Data packet" should be created for every optical connection of an optical BP:



The first data packet is created automatically by the component editor.
Any further packets have to be created manually using the context menu for basis data.
In the data packet, the number of fibers (number of wires) of the connected glass-fiber cable is defined.

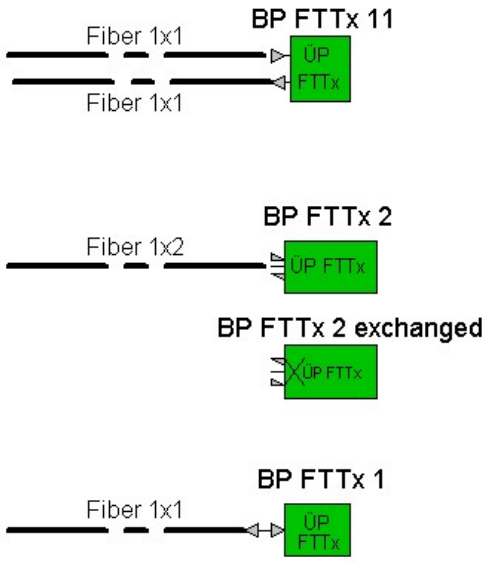
The standard value is number of wires = 1. For each fiber, a signal direction is entered in the “Direction” groupbox, defining whether the signal will flow in (input), out (output) or whether it is bidirectional, i.e. one signal flows in; another signal with a different wavelength or another time window flows out.

The first two enter boxes (Min./Max. Input Power) describe the permissible level window of the incoming signal.

The AND network check function "Check bridge points" checks, for each opt. BP, whether the input signal is in the permissible range.

The bottom two enter boxes ("Output Power" and "Output Wavelength") describe the outgoing signal. Each optical BP with the signal direction "Output" or "Bidirectional" is an optical transmitter in AND.
The values entered in the library for the power and wavelength of the transmitter are the standard values for the state directly after creation of an object in AND.
These values can be overwritten in AND.

The "Time Multiplex" checkbox is not evaluated by AND and is for information only.
The example drawing "Demo OptUEP.net" shows the typical FTTx applications.
The optical BPs all originate from the "OptNeutral.lib" library "OptNeutral.lib".



The figure shows an excerpt from "Demo optUEP.net".
The component "FTTx 11" has two connection pins for single-core cables:
the upper pin for the incoming signal, and the lower pin for the outgoing signal.
The component "FTTx 2" has a connection pin for a two-core cable.
The incoming downstream signal is expected on fiber 1;
the upstream signal is transmitted on fiber 2.
This signal assignment is supposed to be ensured by connection in the incoming splice box.
If the fibers at the BP have to be connected with swapped-round contacts,
a second library object "FTTx 2 exchanged" is used
The component "FTTx 1" has a connection pin for a single-core cable.
The data packet in the library is set to bidirectional. An incoming and outgoing signal flow on the same fiber, but with different wavelengths or time windows.
The components of type "Optical BP" behave in many respects like known optical signal entry or exit points, e.g. two projects can be connected with optical BPs and
it is possible to jump between projects.

The differences between the two object types are:

- Entry/exit points take over the number of fibers from the connected cable.
  For optical BPs, the number of fibers is defined by the library object.
- Only cables with one bundle can be connected to an optical BP.
- A level window for outgoing signals can only be defined in the optical BP.

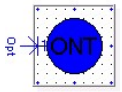
3.20.1 Optical Bridge Point - Type ONT, ONU, OLT

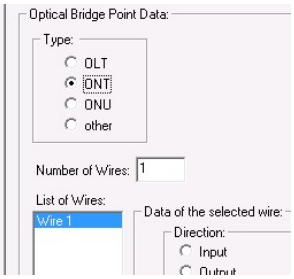
Implemented from Build 1213 on.

Components of Type „Optical Bridgepoint" have an attribute „Type".

Possible values are:

- OLT
- ONT
- ONU
- other.

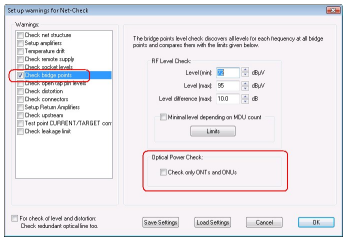




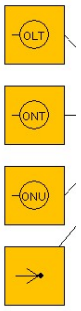
Consequences in AND

1. If the option „Check only ONTs and ONUs" is activated, the netcheck will check the input power only for this types.

Dialog for editing the warning setup:



2. Each type will be displayed with different graphics in NIS:

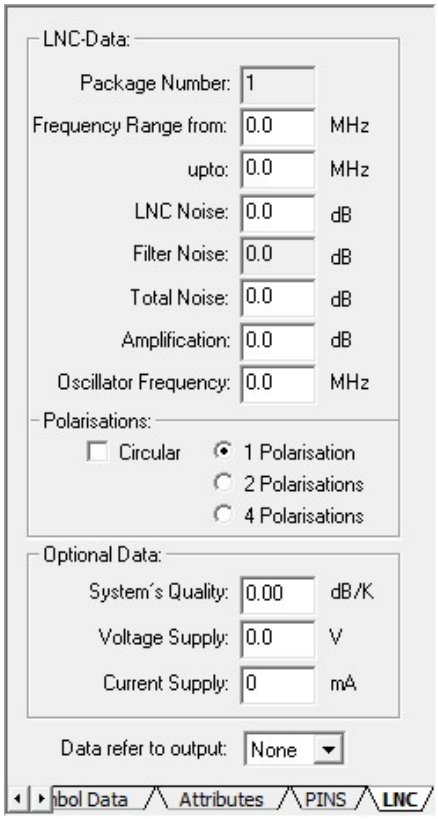


The lowest node is of type „other".

The types correspond with the following database values of NIS_NODES::NODE_TYPE:

Type	Value
ONT	772
OLT	1045
ONU	1046
other	1047

3.21 Creating a data packet for the feed (LNC)



Component type "Feed (LNC)" has been selected.
You now have the following input options in the input form shown above.

LNC data:

"Package Number:"
This field is not active and cannot be edited.

"Frequency Range from: ..... MHz"
Here you enter the beginning of the transmission range.

"upto: ..... MHz"
Here you enter the end of the transmission range.

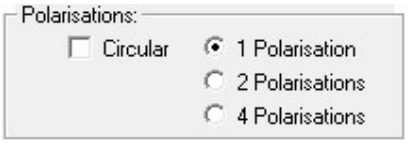
"LNC Noise: ..... dB"
Here you enter the background noise of the LNC without the polarization noise.

"Filter Noise: ..... dB"
Noise of the polarization filter:
This value results from the "LNC noise" and the "total noise".
It is for information only and cannot be edited.

"Total Noise: ..... dB"
Here you enter the total noise of the LNC incl. the polarization filter.

"Amplification: ..... dB"
Here you enter the gain of the LNC.

"Oscillator frequency: ..... dB"
Here you enter the oscillator frequency of the converter.



This check box is not currently evaluated in AND and is therefore for information only.

Optional data:

This date is currently not evaluated in AND and is therefore for information only.
"System Quality: ..... dB/K"

Here you enter the figure of merit for a complete system.
"Voltage Supply: ..... V"

Here you enter the feed voltage of the LNC.
"Current Supply: ..... mA"

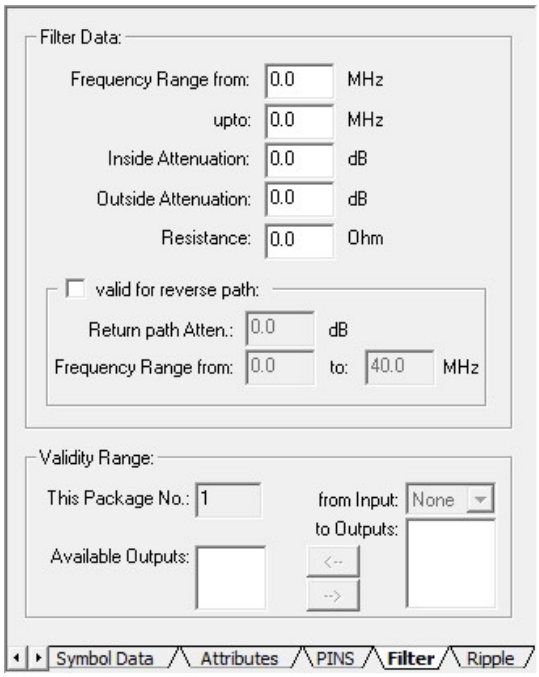
Here you enter the maximum feed current of the LNC.

Here you define the output to which the LNC packet is to apply.

The output is selected using the button.

Please note that no supplementary data packets, such as standard packets are possible.

3.22 Creating a data packet for a trap



Filter Data:

"Frequency Range from: ..... MHz upto ..... MHz"
Here you define the beginning and the end of the stop band.

"Inside Attenuation: ..... dB"
This field can be ignored.
You cannot influence the rejection.
Various inputs are not taken into account.

"Outside Attenuation: ..... dB"
Attenuation outside the specified stop band.

"Resistance: ..... Ohm"
Remote feed impedance for this packet (for component power passing).

If you activate the valid for reverse path: field, the object is suitable for reverse path.

The following input fields then become active:

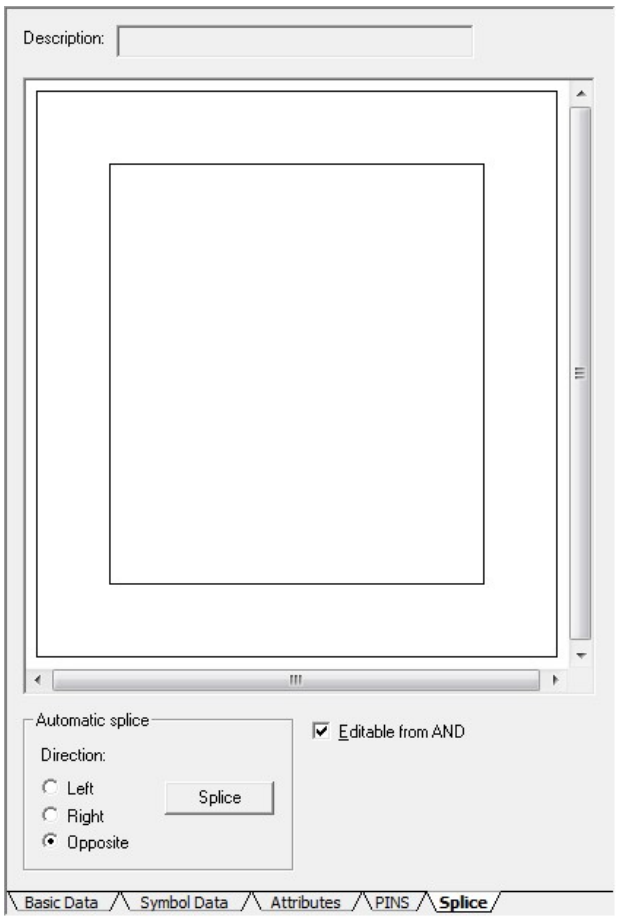
"Return path Atten: ..... dB"

Here you can enter the attenuation of the reverse-path frequency range.
This is defined under
"Frequency Range from: .....to..... MHz"
Entry of the reverse-path frequency range.

„Validity Range"

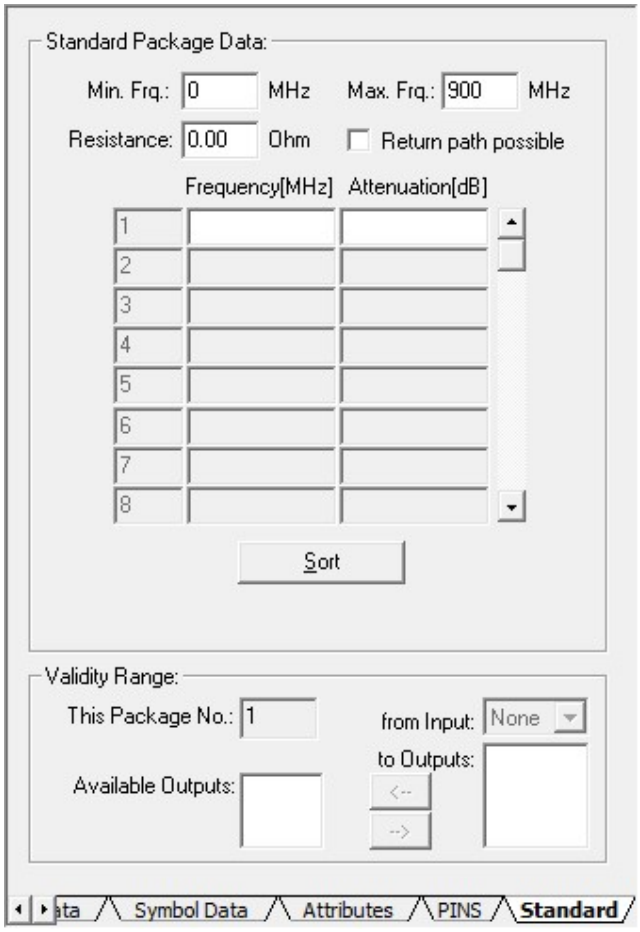
3.23 Creating a data packet for a splice box

You have selected the component type “Splice” .



With this data packet, you can generate splice box objects.
The only relevant elements of these objects are the mechanical cable connections.
In addition, supplementary connections can be generated that have no effect on the splice connections.

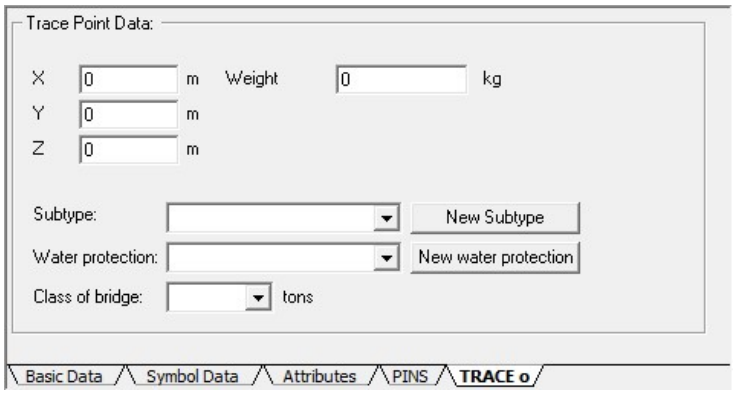
3.24 Creating a data packet for a wall outlet



Component type “Wall outlet” has been selected.
This component type is selected with “Standard” data packets.

3.25 Creating a data packet for a trace

Components of this type are used to represent real objects that normally create connections and permit access to traces.
Cable trace objects are used for documentation, costing and routing traces.



Component type "Trace" has been selected
You now have the following input options in the input form shown in the figure above:

Trace Point Data:

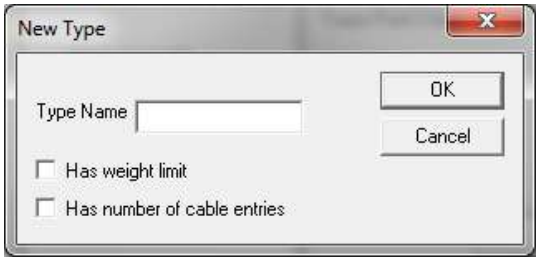
"X, Y, Z ..... m"
Dimensions of the trace point.

"Weight: ..... Kg"
Total weight at the trace point.

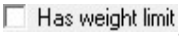
"Subtype:"
Subtypes are, for example, components or parts of a supply duct.

Clicking the 

button opens a new dialog box.



Here you can define the name of the new subtype.

If you place a check mark in 

,

then the entry field 

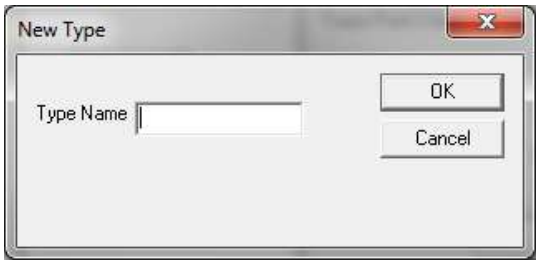
is generated.

For example, the maximum load of a supply duct cover can be stated here.

If you place a check mark in ☐ Has number of cable entries,
then the entry field Number of cable entries [ ] is generated.
The number of possible cable entries is documented here.

„Water protection:"
specifies the degree of protection of the cable trace object.

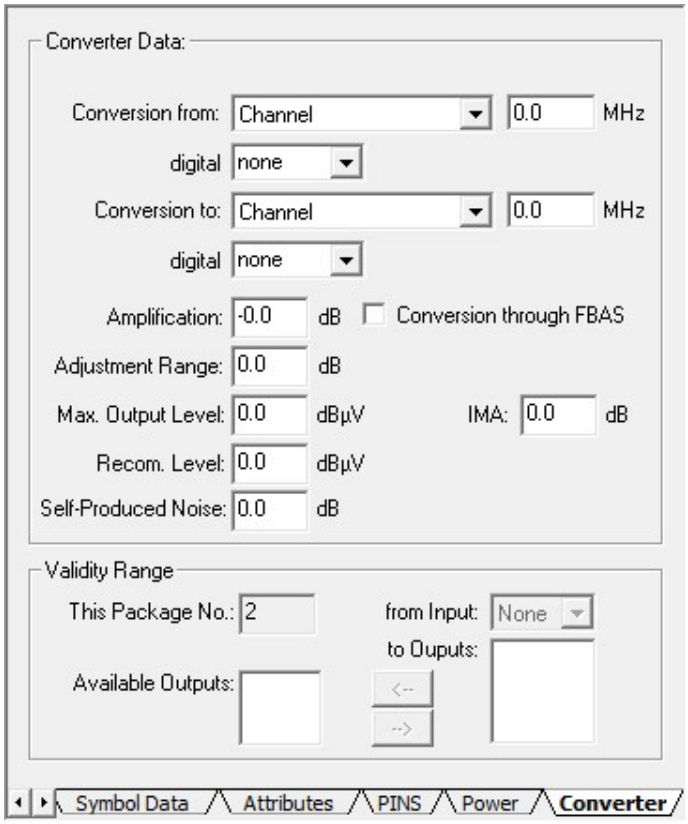
By clicking the New water protection button you can create new water protection types.
The following dialog box then opens:



After confirming the appropriate entries, a new button appears Add,
that you can use to change or delete the entries at any time.

"Class of bridge:…..tons"
((To be completed))

3.26 Creating a data packet for a converter



Component type "converter" has been selected.
You now have the following input options in the input form shown in the figure above.

Converter Data:

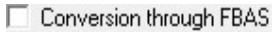
"Conversion from: ..... MHz"
Here you select the input frequency of the converter.
You can enter the band combinations or individual frequencies.

"digital"
Here you define whether the input signal is received digitally.

"Conversion to: ..... MHz"
Here you define the output frequency of the converter.

"digital"
Here you define whether the output signal is to be digital.

"Amplification: ..... dB"
Here you enter the maximum gain without control.



After you have selected the check box, frequency translation is not performed directly but via a video signal (rec. output level is applied in AND).

"Adjustment Range: ..... dB"
Here you define the control range of the adjustable attenuator.

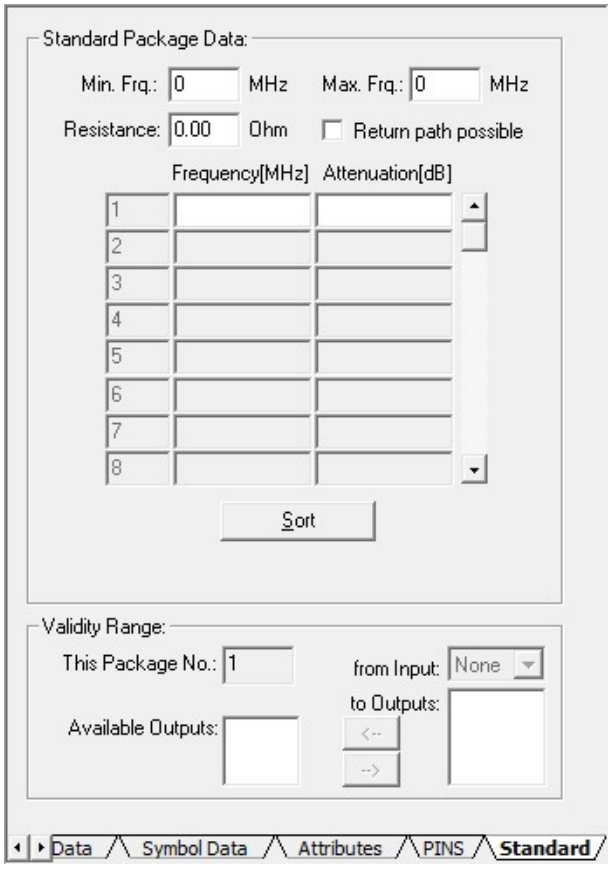
"Max. Output Level.: ..... dBµV"
" IMA: ..... dB"
The maximum output level refers to the IMA value according to DIN entered here.

"Recom. Level: ..... dB"
Here you enter the recommended output level (target level).

"Self produced Noise: ..... dB"
Background noise of the converter module.

„Validity Range"

3.27 Creating a data packet for a connecting element

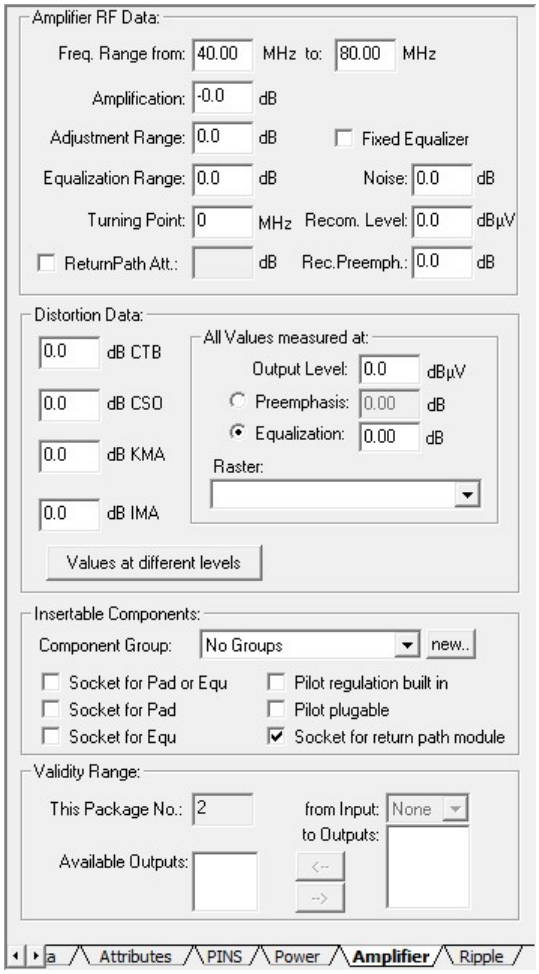


Component type "Connecting Element" has been selected.
This component type is selected with "Standard" data packets.

You can create additional data packets of type Standard.

„Validity Range"

3.28 Creating a data packet for an amplifier



Component type "Amplifier" has been selected.
You can create supplementary data packets of type Standard.
You now have the following input options in the input form shown in the figure above.

Amplifier RF Data:

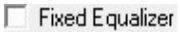
"Freq. Range from: ..... MHz to ..... MHz"
Here you enter the beginning and end of the transmission range.

"Amplification ..... dB"
Here you enter the maximum gain without control.

"Adjustment Range: ..... dB"
Here you enter the maximum attenuation value of the control range.

"Equalization Range: ..... dB"
Here you enter the maximum equalizer value of the control range with reference to the lowest transmission frequency.

If you place a check mark in.



the value entered in "Equalization Range" is taken over

as the fixed equalizer value.

"Turning Point: ..... MHz"
Here you define the base for equalization.

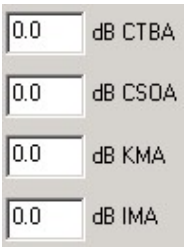
If you do **not** place a check mark in.

the amplifier is classified as not suitable for reverse path in AND.
If you select the check box, the entry field becomes active and you can enter the through loss for the reverse path.
The through loss entered refers to the highest transmission frequency of the reverse-path function entered in AND.

"Noise: ..... dB"
Here you enter the background noise of the amplifier.

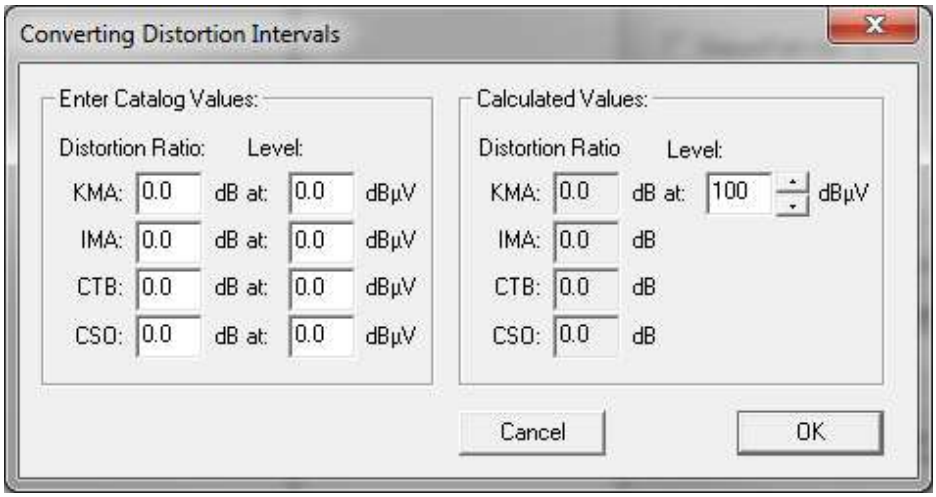
"Recom. Level: ..... dBµV"
"Rec. Preemph.: ..... dB"
The values entered here are applied in AND and are the target values for the level and preemphasis of the amplifier.

Distortion Data



Here you can enter the catalog values for CTBA, CSOA, IMA and KMA of the amplifier.
The CTBA and CSOA values are always used for calculation because these values are more meaningful.
If no CTBA and CSOA values are available, the KMA and IMA values are used instead.
If no data is entered here (value 0.0 dB), an error message is generated in AND.

Clicking this button opens a new dialog box:



You can use this dialog box to adapt the signal-to-noise ratios entered to a changed level. These values are automatically applied when you exit the dialog box.

All values entered refer to the

"Output Level: ..... dBµV"
Here you enter the output level, with reference to the interference data.

"Preemphasis: ..... dB"
Slope in the selected channel bandwidth (raster) or

"Equalization. ..... dB"
Value of the equalization for the entire transmission range.

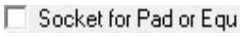
"Raster"
From this combo box 

you select the frequency raster.

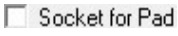
Insertable Components:

Under the heading "Insertable Components", the socket options in the device are defined.

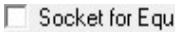
Here you define the groups from which AND can select the components for the corresponding sockets.



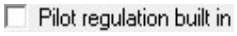
This amplifier packet has a socket for an attenuator or equalizer.



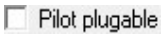
This amplifier packet has a socket for an attenuator.



This amplifier packet has a socket for an equalizer.



This amplifier packet has a pilot control permanently integrated.

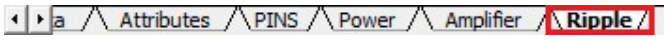


This amplifier packet has a socket for a pilot control.

The components selected in AND then appear both in the amplifier list and in the bill of materials.

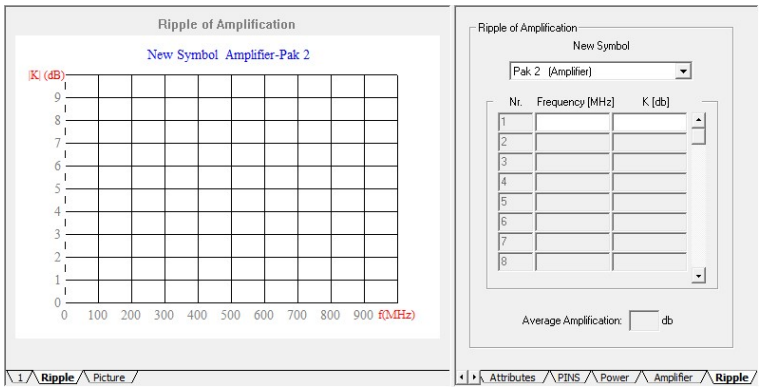
„Validity Range"

3.28.1 Entering frequency slope measurement data



Selecting this tab gives you a new input form for value pairs.

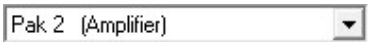
Caution:
If value pairs are placed here, AND will use these values for calculation.
That means that all other values are **no longer** used in the calculation!



The window in the right-hand section is used for editing value pairs to document any ripple.
You can enter the value pairs in any order.
The value pairs will automatically be sorted by frequency.

You can most quickly enter the value pairs if you press the  key after each entry.

At the same time, the data entered is shown in the center section.
After each entry, the calculated average is the shown accordingly.

By operating this combo box 

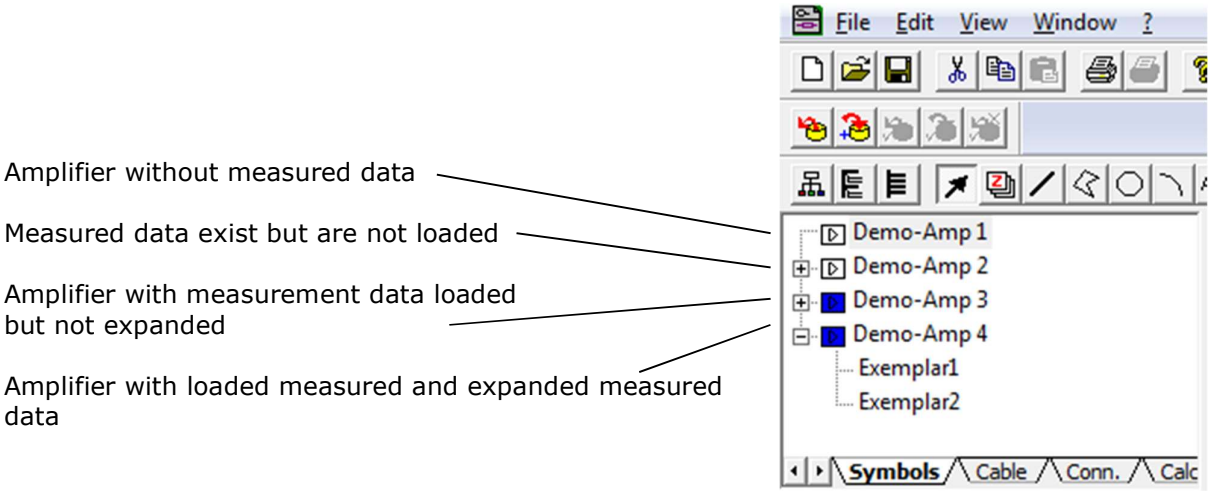
you can store value pairs for each amplifier packet.

3.28.2 Calculating using ARD

Using the program "*Amplifier Raster Design*" (ARD) , you can calculate the entire CSO/CTB spectrum of coaxial CATV amplifiers for any frequency raster based on a few measured CSO/CTB measured values. It is immaterial for which raster measured values are available. As an alternative to CSO/CTB measured values, you can also use a series of KMA/IMA data, e.g. from a DIN measurement station.

The amplifier symbols have a special function here.
If there is measured data for an amplifier, a + box appears to the left of the amplifier.
After you have started the program, the measurement data are not loaded and the amplifier symbol is white.
Double-clicking the amplifier or single-clicking the + box loads the measurement data – you may have to search for the relevant data – and the amplifier symbol is now blue.
If the amplifier is expanded, all the instances of the amplifier will appear as its "children".

The following figure shows the possible states of an amplifier:



The measurement data is stored in files with the extension *.amp.
There is one file for each amplifier. The name of the file is normally *amplifier_name.amp*.

The working directory of the program indicates where the measurement data is stored and where the measurement data is sought on loading.
If the measurement data is not found on loading, you can look for the measurement data file yourself in a dialog box.

The path of the working directory can be set with menu item "Choose Directory".

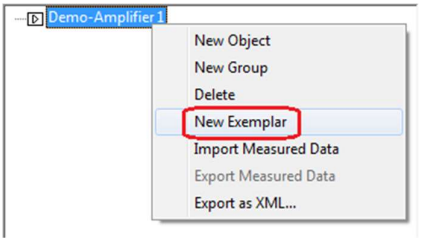
The measurement data of an amplifier can be imported and exported using the context menu (right-hand mouse button).

The import function can also be used to add further data to the existing measurement data.

The context menu is also used to create a new instance.

3.28.3 Entering CSO/CTB measurement data

If there is not yet any measurement data for a particular amplifier,
you must first create a new instance of the amplifier.
This is done by selecting the relevant amplifier, opening the context menu with the right mouse button, and choosing the menu item "New Exemplar" from it.

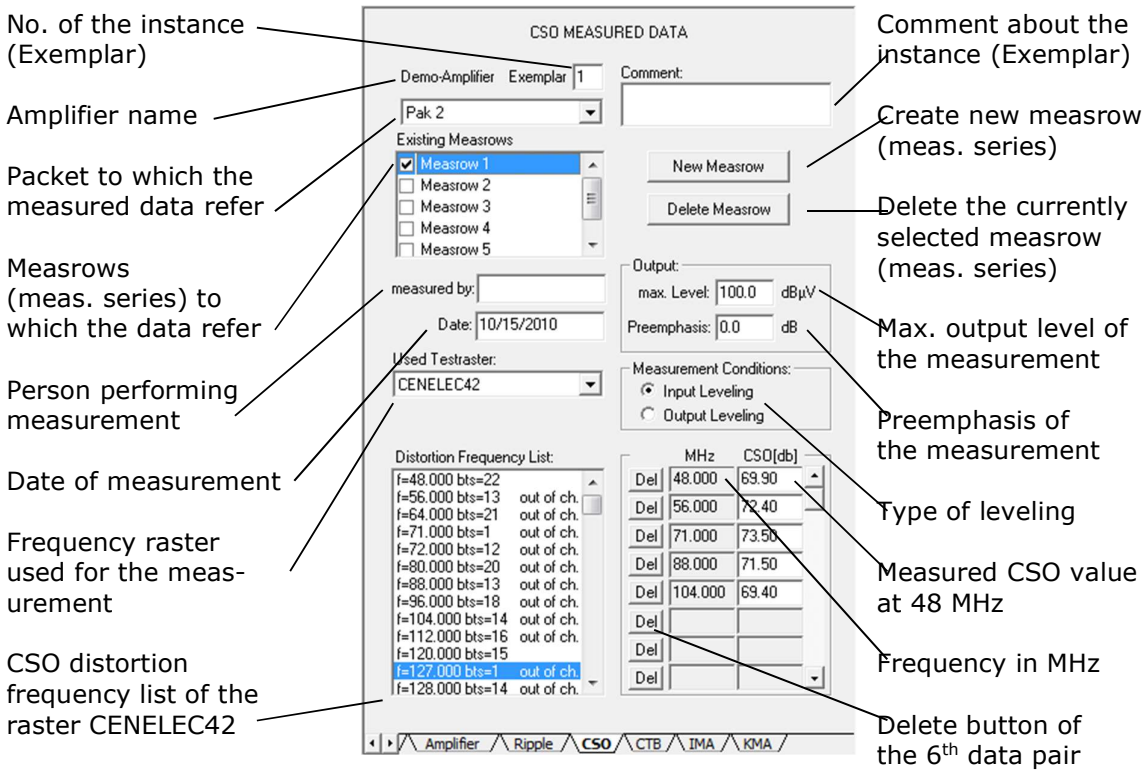


If you want to add further instances, proceed as described.

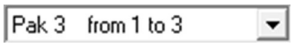
The center section contains the tabs CSO and CTB for the calculation.

The right-hand section contains four tabs for the measurement data:
CSO, CTB, IMA, KMA.

For example, if you want to enter CSO measurement data,
choose the CSO tab in the right-hand section.
You will then see the following display.

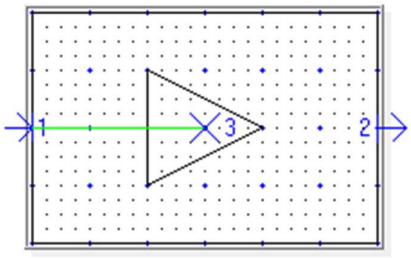


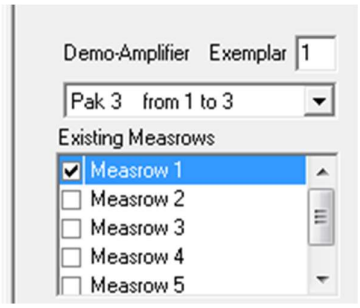
If the amplifier consists of more than one amplifier packet,

you first have to select the correct input/output pair in the 

combo box.

To do this, select the "1" tab in the center section to obtain a clearer view.
This current input/output pair is highlighted in color there.





If you want to enter a new measurement series,

you first must create a new measurement series by clicking the 

button.

To change an existing measurement series, choose the relevant measurement series from the available measurement series.
First select the measurement raster in which the data were measured from the list.
If the required raster is not yet in the list, you must define a new raster first.

In the dialog box that then appears, you can also view and modify the channel assignment of each raster.
After you have selected the raster, the interference frequency list of the raster is displayed.
If measurement data is already entered, it will be cleared.

You are first warned of this by a message box.
Now you can enter the frequencies and CSO values.
To facilitate selection of the measurement channels and to avoid incorrect frequency selection, entry of frequencies is only possible by double-clicking the entry for the interference frequency list.

It is convenient to select the measurement channels evenly across the frequency range of the measurement raster and within the channels to select the measurement frequencies with the largest number of "beats" (bts).

The CSO/CTB values must be between 30 and 90 dB.
To complete the data, you must enter the maximum test output level and the type of level alignment.

The *"Comment"*, *"Date"* and *"measured by"* fields are to facilitate administration of the measurement data.
These data are not used in the calculation algorithm.

To delete a value pair, click the 

button to the left of the corresponding frequency field.

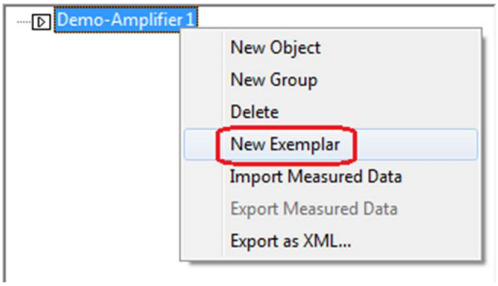
To delete a measurement series, select the measurement series to be deleted and then click the 

button.

The measurement data is now complete.
The measurement series still has to be selected before calculation.
The CTB data is entered in exactly the same way.

3.28.4 Entering IMA/KMA measurement data

If there is not yet any measurement data for a particular amplifier,
you must first create a new instance of the amplifier.
This is done by selected the relevant amplifier, opening the context menu with the right mouse button, and choosing the menu item "New Exemplar" from it.



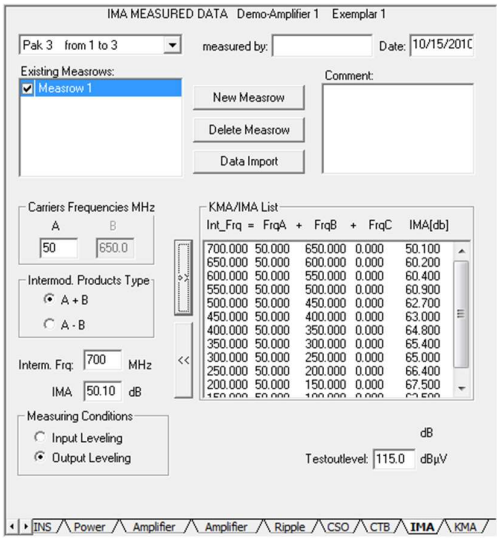
If you want to add further instances, proceed as described.

The center section contains the tabs CSO and CTB for the calculation.

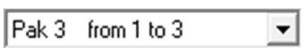
The right-hand section contains four tabs for the measurement data:
CSO, CTB, IMA, KMA.

For example, if you want to enter IMA measurement data,
choose the IMA tab in the right-hand section.

You will then see the following display:

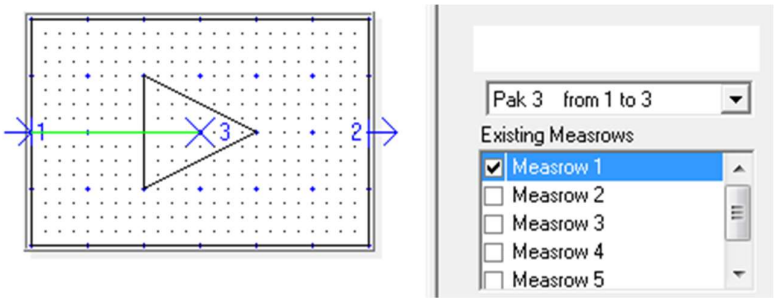


If the amplifier consists of more than one amplifier packet,

you first have to select the correct input/output pair in the 

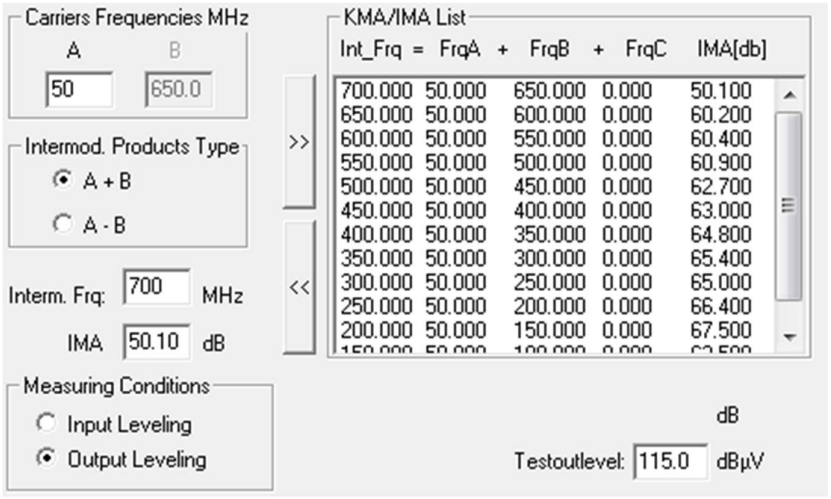
combo box.

To do this, select the "1" tab in the center section to obtain a clearer view.
This current input/output pair is highlighted in color there.



If you want to enter a new measurement series, you first must create a new measurement series by clicking the New Measrow button.

To change an existing measurement series, simply select the measurement series in question.
Entering the data:



The window contains the fields for carrier frequencies A in the case of IMA or A and B in the case of KMA (B is calculated automatically for IMA with correct sign and C, for KMA).

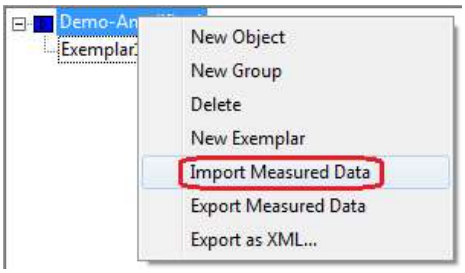
Below this, you will find the buttons for the intermodulation types (**A ± B** for IMA and **A ± B ± C** for KMA) and the entry fields for the intermodulation frequency **Interm. Frq.** and the KMA/IMA value.

To ensure that the measurement series are representative, interference products have to be entered for the main intermodulation types **A ± B** for IMA and **A + B - C** for KMA.
To be compatible with common DIN measurement stations, only the usual combinations are listed for selection.
If unsupported frequency combinations are required, a negative offset can be entered for the B value.

On the right-hand side, the interference measurement list appears with the carrier frequencies.
To transfer data from the entry fields into the interference list,
you must click the add button ">>" with the mouse.

All frequencies are then written with the correct sign for this intermodulation type. T
o remove data from the interference list again, mark the measurement data and
click the remove button "<<".

You can import data with the context menu of the left-hand view.



In the case of IMA/KMA measurement data, it is also possible to import data from a data medium in the format of a Rohde&Schwarz measurement station.

This is done by clicking the Data Import. button.

Select the appropriate file from the menu that then appears.

Please do not confuse the KMA and IMA measurement series because the program cannot distinguish between KMA and IMA measurement data at this point.

After confirmation with Open the selected measurement series will appear on the right-hand side of the interference list.

In the case of KMA data, you still have to enter the reduction of the auxiliary test carrier (e.g. 6 dB for two out of three test carriers).

You also have to specify the test output level of the CATV amplifiers.
If the standard table or your data on the data medium is different from usual (as, for example, on the measurement station of Rohde&Schwarz), you have to enter the data manually.

To reduce the influence of measurement errors and to improve the accuracy of determination of the nonlinearity of the amplifier, the amount of data must be sufficient.
(It must be entered at 1 MHz intervals and there should be more than 5 KMA/IMA data items for each intermodulation frequency).

The “Comment”, “Date” and “measured by” fields are to facilitate administration of the measurement data.
These data are not used in the calculation algorithm.

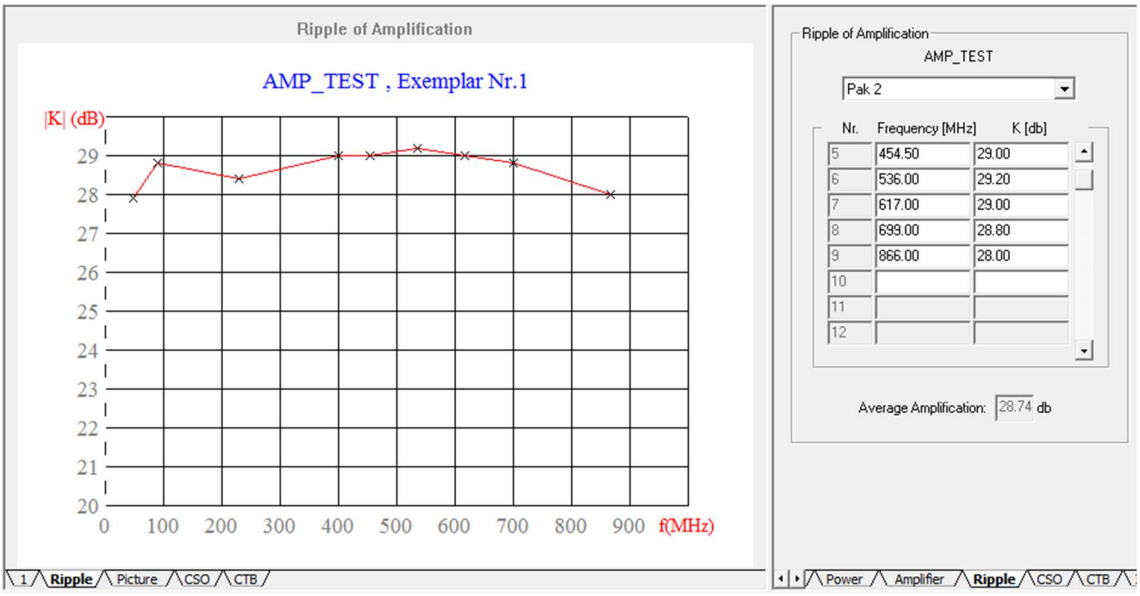
To delete a value pair, click the Del button to the left of the corresponding frequency field.

To delete a measurement series, select the measurement series to be deleted and then click the Delete Measrow button.

The measurement data is now complete.
The measurement series still has to be selected before calculation.

The KMA data is entered in exactly the same way.

3.28.5 Measurement data for the frequency slope

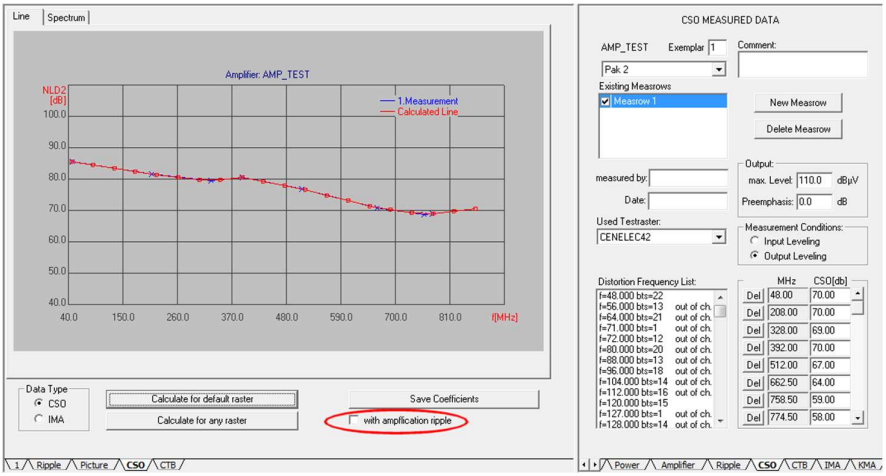


Entry of value pairs for each amplifier packet is explained in section 3.28.1 *Entering frequency slope measurement data.*

You have to enter the frequency and the gain of the amplifier over the entire transmission bandwidth in the case of a shallow, level-aligned input.
However, it is enough to enter just the characteristic points of the gain,
at which there are considerable changes in the gain.

To enable the ripple to be considered in the calculation,
the ripple of each instance must be entered for each instance of the amplifier.

The ripple is taken into account before the calculation if you select the check box



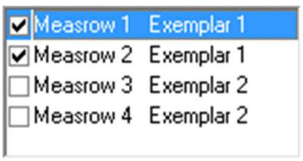
3.28.6 CSO/IMA –CTB/KMA calculation

The **CSO** and **CTB** tabs of the center section are responsible for the calculation. These windows only appear if an amplifier with loaded measurement data or an instance is selected.

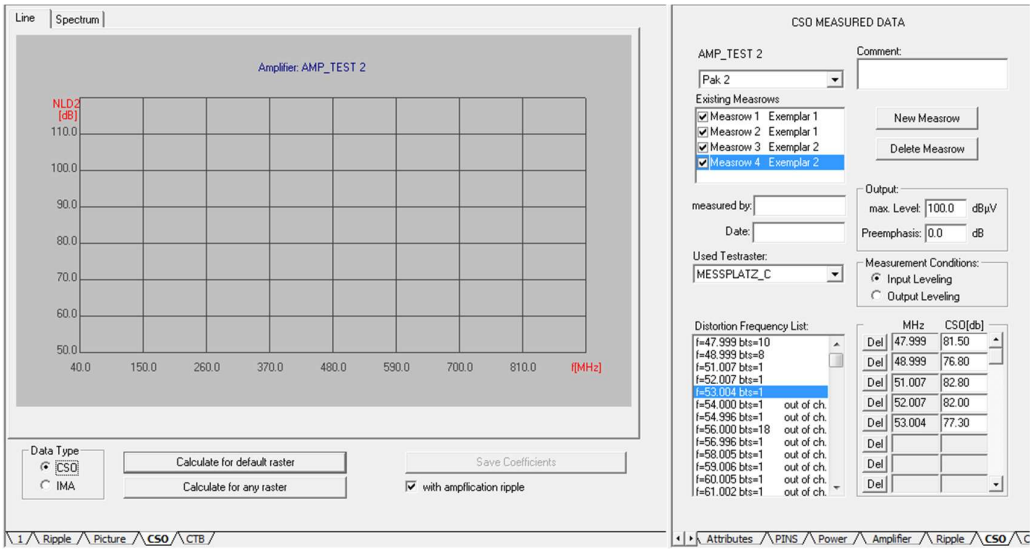
See also the figure in section 3.28.2 Calculating using ARD.

Depending on the selected measurement data form selected, the CSO calculation window refers either to the CSO measurement data or to the IMA measurement data of the right-hand section.

The CTB calculation window refers either to CTB measurement data or to the KMA measurement data of the right-hand section.
Only the selected measurement series are used for the calculation.



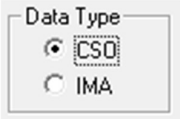
If you have selected the **CSO** tab, the following screen will appear:



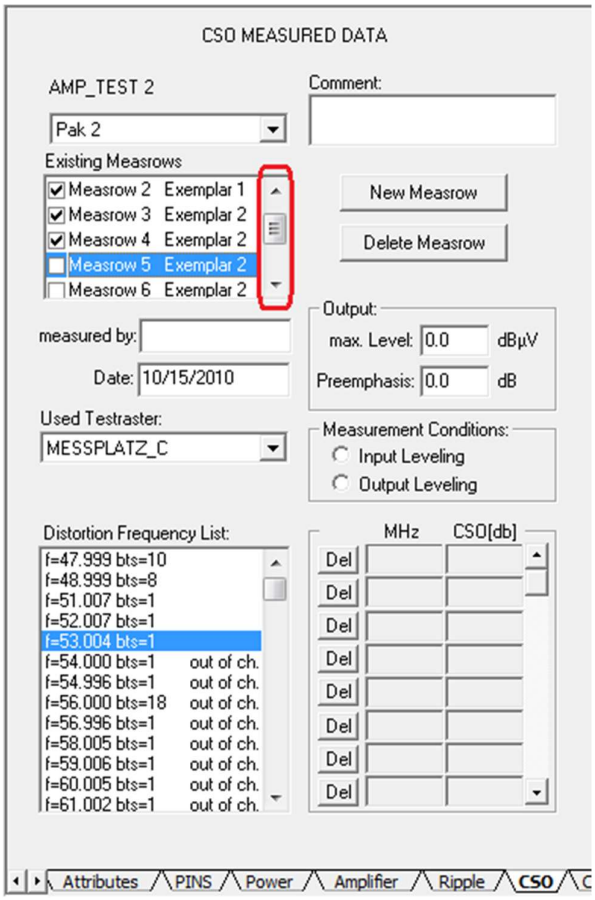
You can access the CTB calculation window by pressing the **CTB** tab.

3.28.6.1 Steps to perform before calculation

- You have entered the measurement data as described.
- Now select the form for your measurement data.



- Select the measurement series to be used in the calculation from the available measurement series in the right-hand section by selecting the check box in front of each measurement series.
  Deactivate the check box in front of the measurement series if you do not want these to be considered in the calculation. (see figure below).



- The picture shows the measurement series that have been selected for the calculation.
  If more measurement series have been created than can be displayed,
  you can make further measurement series visible using the scrollbar (marked red)
  to select them for the calculation.
- You should select a measurement series that covers the entire amplifier bandwidth for the calculation.
- You can perform also the calculation with all measurement data for different rasters.

- You can activate the option

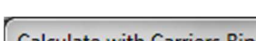
  (see picture at the bottom of page 137) before the calculation.
  If this option is active, the ARD program can calculate the CTB/CSO spectrum, taking the amplifier ripple and preemphasis into account.

  If this option is not activated or if there is no data on the frequency slope, calculation will be performed at a constant level.
- To start the calculation, click one of the two buttons:

- Calculation is performed for the catalog raster and the output level that are entered for the amplifier packet.

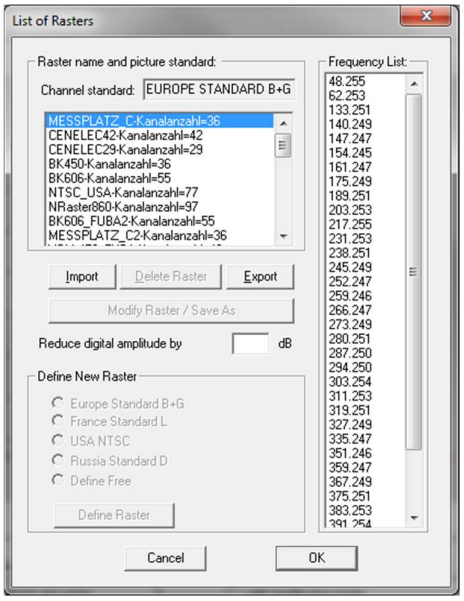
  

- Calculation is performed for any selected raster.
  The selected raster can be different from the measurement rasters.
  The selected raster on no account have a larger bandwidth than the amplifier bandwidth.
- After confirmation of one of the two calculation buttons, the following dialog box opens (

  must be selected):

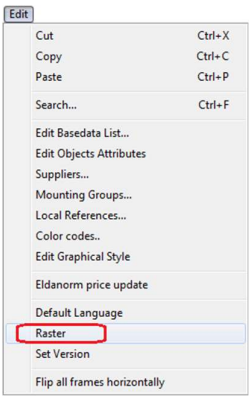
- Here you can set whether the calculation will be performed with a level-aligned input or level-aligned output.
- You can enter a preemphasis.
- If you activate the "assume, that equalizer is linear" check box,
  the frequency slope of the amplifier equalizer simulated linearly,
  otherwise it is simulated as equalization of the non-linear frequency slope of a typical coaxial cable, that is, a sort of cable equalizing characteristic.
- After entry of the required data, you can now click one of the two calculation buttons.

- If you decided on the Calculate for default raster button, the calculation results will be displayed to you immediately, either as characteristic curves or in the spectrum.
- See page 142 Calculation result curve or
- section 3.28.7 Calculation result for CSO/CTB spectrum
- If you have selected the Calculate for any raster button, a further dialog box appears for selecting the raster. Here you can only select the raster.



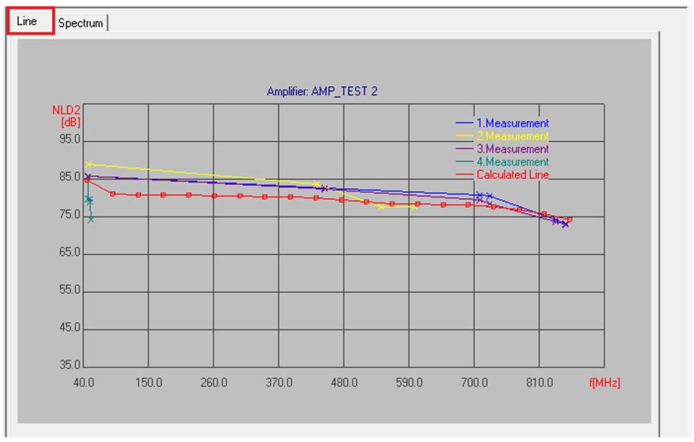
If you want to change a raster or define a new raster, choose menu item Raster (see next figure), change the raster data there and start the calculation again.

See also 4.1 Defining a new raster



- The calculation window for the CTB calculation has an analogous layout.

3.28.6.2 Calculation result curve

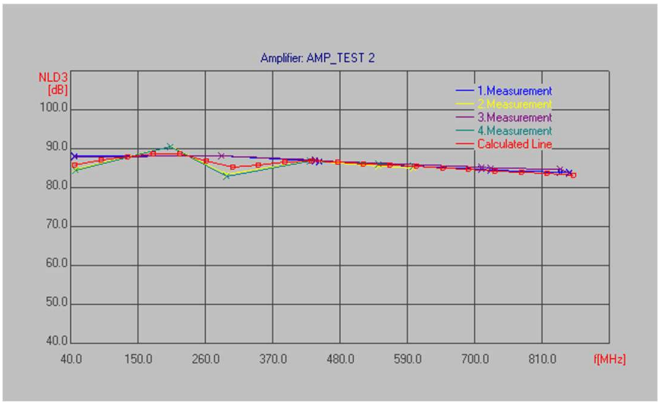


The characteristic curve is a plot of the calculated frequency dependency of NLD2 or NLD3 (2nd and 3rd order coefficients of **N**on**l**inear **D**istortion) for each selected measurement series and for the representative CATV amplifier.

If the measurement error is relatively small, the curves for the measurement error analysis can be helpful even if they have been measured with different frequency channel assignments.

That especially applies to CTB measurement data because it usually contains smaller measurement errors and better measurement data repetition than the CSO measured values.

Randomness has a large influence on measurement of CSO because the number of "beats" is relatively small in this case, which makes measurement data repetition poorer.
If the calculation is based on a statistical value, measurement points with a larger number of "beats" for representative amplifiers can be better defined.
The following figure shows the CTB curve of the same amplifier.



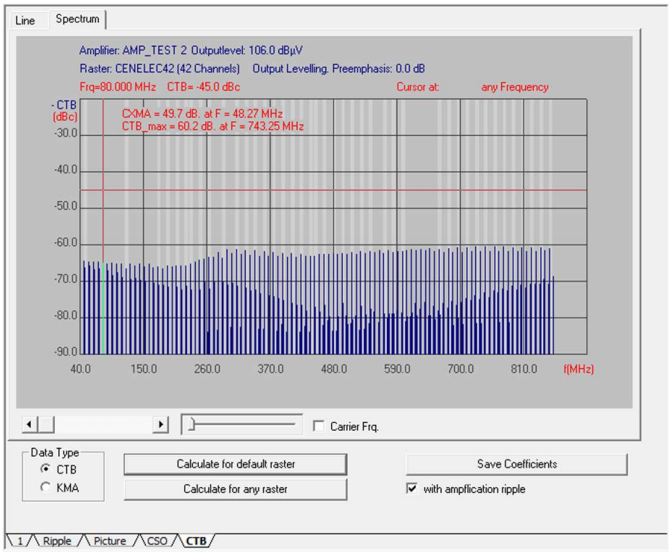
3.28.7 Calculation result for CSO/CTB spectrum

The CSO/CTB value at a frequency is the difference between the video carrier level and the interference product level.
If the video carrier level changes within the bandwidth of the amplifier, the CSO/CTB ratio also changes, even without any change to the interference product level.

If the measurement data of the CATV amplifier have been measured, or if the CATV amplifier exhibits a large ripple, a difference results between the calculated values and the measured values, e.g. on conversion to another frequency channel assignment.

A change in the ripple or the preemphasis can either improve or worsen the CSO/CTB values for a certain raster and a certain amplifier.
Without special numeric analysis, activated with the ☐ with ampflication ripple check box, no prediction is possible.

The following figure shows the CSO/CTB spectrum of the selected raster.



If both the raster and the type of level alignment for a measurement series match the parameters selected for calculation, the data of this measurement series is plotted in short, horizontal dashes.

The figure above shows the result of a CTB calculation with the CENELEC29 raster and a level-aligned input.

Measurement series 2 was also measured with the CENELEC29 raster and a level-aligned input.
Consequently the measurement data for measurement series 2 appear as horizontal dashes in the spectrum. The color coding is the same for the curve display and the spectrum display, that is, lines of the same color belong to the same measurement series.

Using the boxes for changing the scale and offset

You can move the measurement cursor simply by operating the mouse.
You only have to move the mouse pointer to the required position in the image grid, c
lick this position once and release the mouse button.
The mouse pointer will then be at the required position.

This check box currently has no function

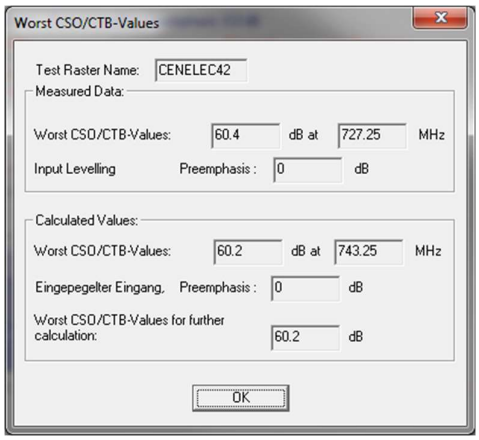
3.28.8 Writing calculated data into the library

After calculation you can store the calculated CSO/CTB values in the library with the corresponding amplifier.

For this purpose, the Save Coefficients button becomes active immediately after calculation.

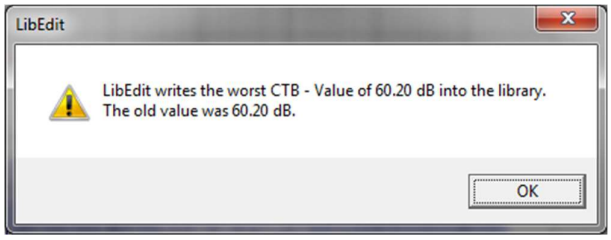
The program will work with the data last calculated.
If the input data of the calculation changes in some way or other, the button is deactivated again.

After you have clicked the button, an info box opens.



Here you can compare all measurement and calculation results again.

You exit the info box by clicking OK.
Now the calculated data is stored and the catalog data is written for CSO/CTB.



Confirming this info box finalizes the operation.
Storage of the CSO data is performed completely analogously.
On storage, only the product of the calculation results, the characteristic curve with the library, is stored.
The stored measurement data are placed in a separate file (amplifier_name.amp) for each amplifier in the Lib folder.
These files are not required for planning in AND.
However, they should remain in the Lib folder because this file is accessed on every change to the amplifier.

3.28.9 Printing calculation results

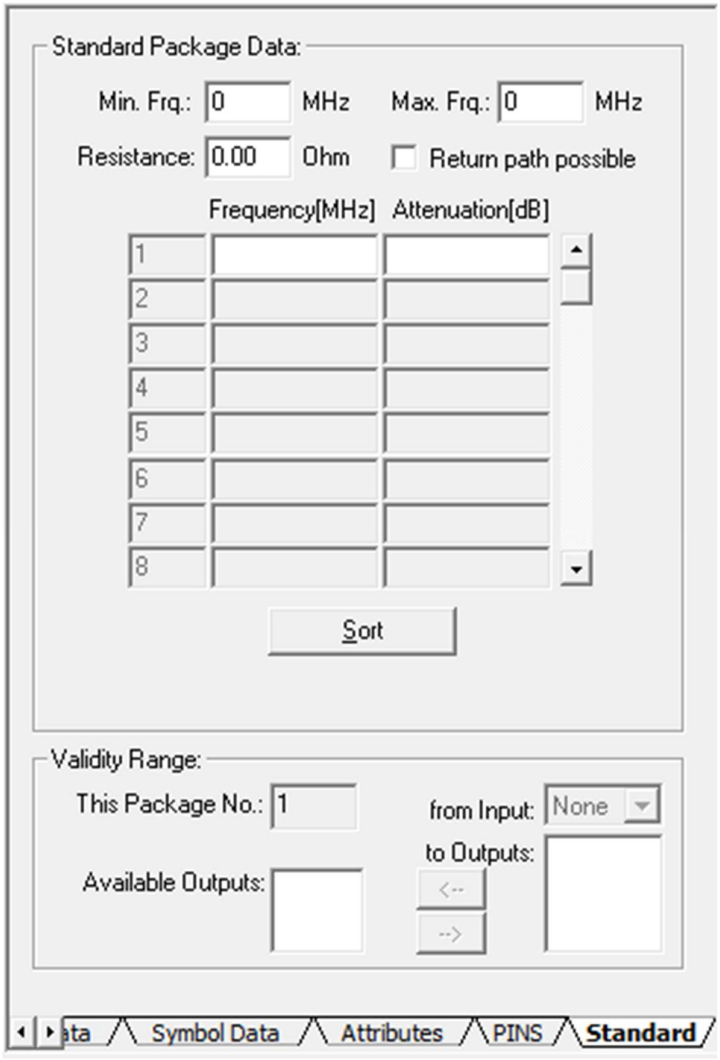
With the button (marked red)  of the toolbar,
you can print out the graphics currently visible in the calculation window.

If the button is not active, a mouse click on the image grid in the center section is enough.

If you have previously clicked the 

button,
the graphic will be printed out with the most important amplifier data.

3.29 Creating a data packet for a splitter

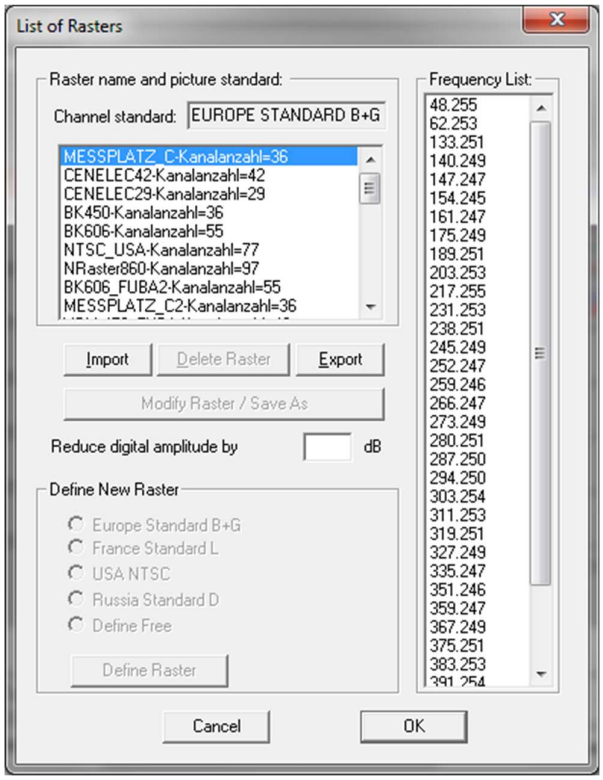


The component type "Splitter" was selected.
This component type is specified with "standard" data packets.
You can create additional data packets of type Standard.

„Validity Range"

4 Frequency Rasters

A raster consists of a name, a list of video carrier frequencies and a TV standard.
The frequencies can also be digital.



The upper part of the raster list contains the channel standard of the selected raster.

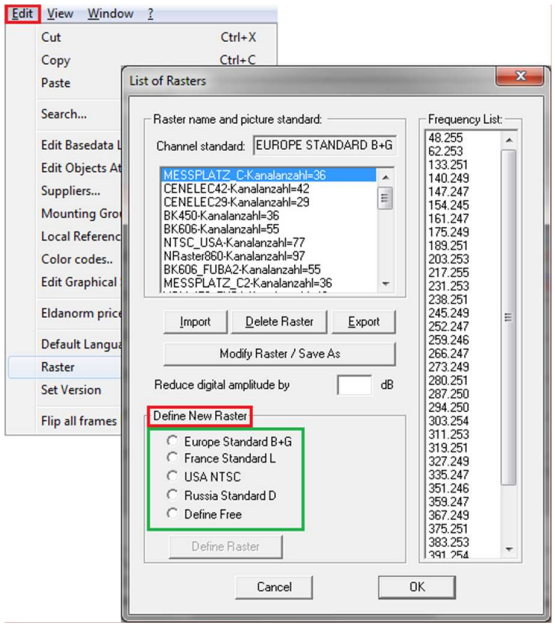
The upper raster is marked (“MESSPLATZ_C-Kanalzahl=36”).

The right-hand window shows the video carrier frequency list of the selected raster.

Level lowering for rasters with digital channels is specified here (usually 10.0 dB).

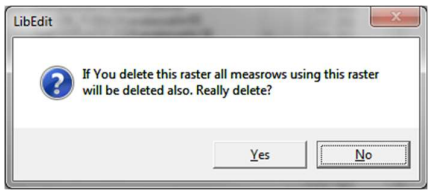
The standard measurement procedure prescribes unmodulated carriers for CSO/CTB measurement and determination.
Rasters with digital channels are used to check the CSO/CTB spectrum for assignment of the channel to a digital carrier frequency.
They can also be used to detect what interference PAL-unmodulated carriers are causing in these channels.

4.1 Defining a new raster



These buttons allow you to import and export existing raster file.

With this button, you can delete the currently selected raster.



A message informs you that measurement series that refer to this raster will also be deleted.

With this button, you can store or modify the selected raster with a new name.

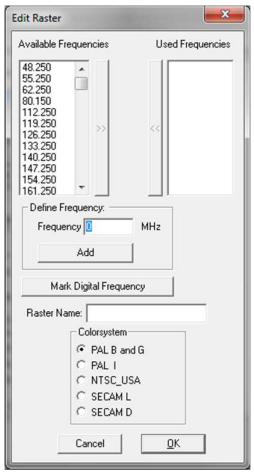
„Reduce digital amplitude by: ..... dB"

Channels marked as digital are lowered by the value entered here.

4.1.1 Freely defining a raster

To define a new raster, you must perform the steps shown in the figure at the top of page 149.
You are now in the "List of rasters" dialog box ".
First select the standard of the frequency list
(marked green in the figure at the top of page 147.).
You have selected "Define free" within the section "Define New raster".

A click on the activated button Define Raster takes you to a new dialog box "Edit Raster".



"Frequency ..... MHz"

Enter the required frequency in the intended field.

Clicking the „Add" button puts the frequency in question in the frequency list of used frequencies.

If you want to delete individual frequencies from the list of used frequencies,

select the required frequency and click the << button.

If you want to define individual frequencies of the list as digital,

first select the required frequency, and then click the Mark Digital Frequency button.

Clicking this button lowers the previously selected frequencies by the value defined in the list of rasters in the entry field

"Reduce digital amplitude by: ..... dB"

The frequency in question is marked.

"Raster Name:"

Enter a unique name for the new raster here.

"Colorsystem"

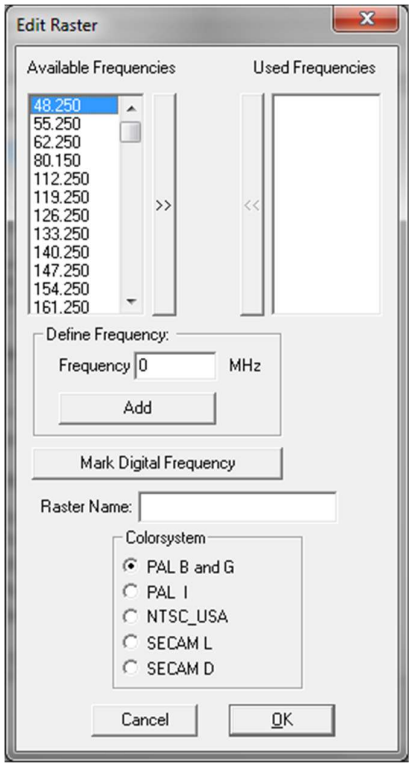
Choose one of the available colorsystems.

4.1.2 Defining a frequency raster according to a TV standard

To define a new raster, you must perform the steps shown in the figure on the top of page 149.
You are now in the "List of Rasters" dialog box.
First select the standard of the frequency list
(marked green in the figure on the top of page 149).

In the "List of Rasters" You have selected the raster "Europe Standard B+G".

Clicking the Define Raster button takes you to a new "Edit Raster" dialog box.



"Available Frequencies:"
List of all frequencies that are available for transmission.

"Used Frequencies:"
In this list, the required frequencies are selected from the list of available frequencies.

That is done by double-clicking or

single selection followed by transmission with the >> button.

"Define Frequency:"
Here you can also add freely defined frequencies to the list of used frequencies.
Clicking the „Add" button puts the frequency in question in the frequency list of used frequencies.

The list of used frequencies is automatically sorted by frequency.
If you want to remove individual frequencies from the list,

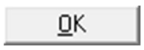
select the frequency in question and click the  button.

Clicking this  button lowers the previously selected frequencies by the value defined under

"Reduce digital amplitude by: ..... dB"

The frequency in question is marked.

After you have entered a unique raster name, you can select the color system.
However, you should not use the NTSC-USA standard for a European frequency list or vice versa.

Finalize your entries with 
After you have exited the dialog box, the interference frequency list is calculated for the new raster.

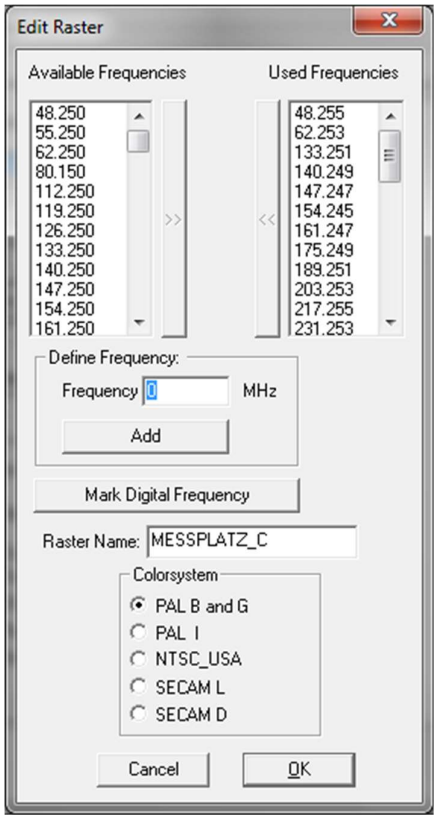
If you exit the dialog box with  the data you entered will be discarded.

For rasters with a large number of channels, calculation may take some time.

4.1.3 Modifying a raster / using an existing raster as a template

An existing working raster can be modified or renamed at any time.

This is done by clicking the Modify Raster / Save As button.
The following dialog box then opens again



You can edit the list in this dialog box.
If you leave the original raster names unchanged, the existing raster will be modified, but if you change the raster name, you will create a new raster.

Finalize your entries with OK.

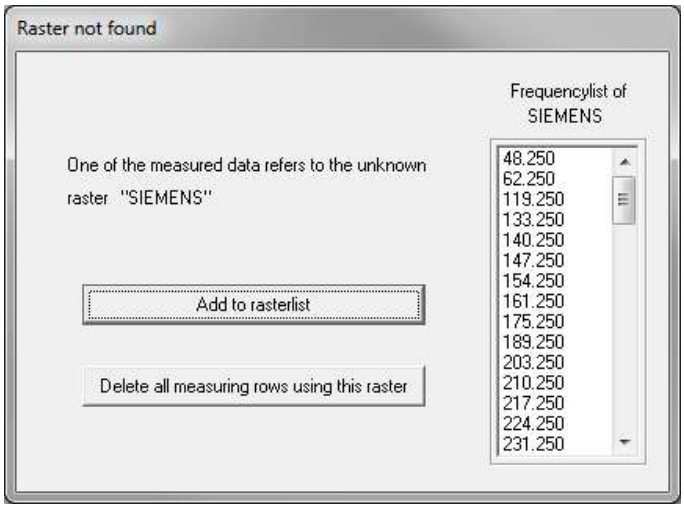
After you have exited the dialog box, the interference frequency list is calculated for the new raster.

With Cancel you can exit this dialog box and the data you entered will be discarded.
In rasters with a large number of channels, calculation can take some time.

4.1.4 Importing/exporting a frequency raster

The rasters used are also stored in a measurement data file.
You can load a new raster indirectly by loading the measurement data that this raster uses.
To import a complete raster list, overwrite the file "Raster.rst",
which is located in the same directory as the program, with the new file.

This file must have the same name. Your old raster file is deleted by this.
A message is displayed if you load later measurement data for a new amplifier
that uses the old raster.



If you then select this 

,
button, the old raster is imported again with the measurement data.

With a click on the "Export" button, you can export the existing raster list into
an existing directory.

5 Cables

Each library can contain an unlimited number of cables.

We make a distinction between two basic types.

- Coaxial cables
- Fiber-optic cables

5.1 Multicolours

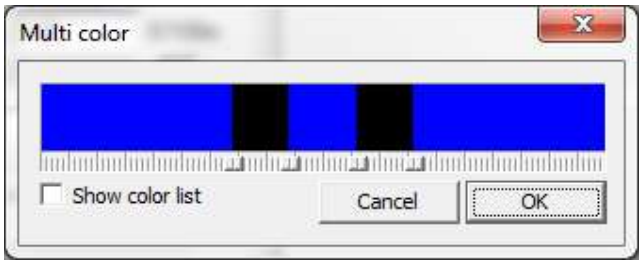
Cables, wires, bundles, duct packages and microducts can be multicoloured.

A "multicolour" consists of a background colour and an optional superimposed ring pattern.

Multicolour consisting of a blue background, a red and a black ring

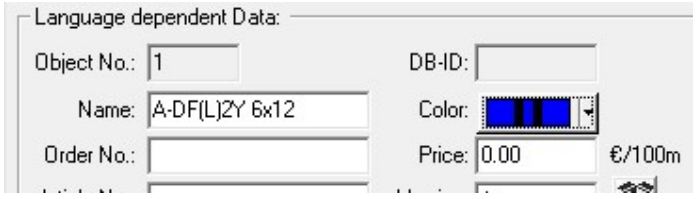
A plain colour is simply a multicolour without a superimposed pattern.

The number, positioning, width and colour of the rings is editable.



5.1.1 Multicoloured cables and duct packages

The multicolour for these object types is displayed in the base data view:

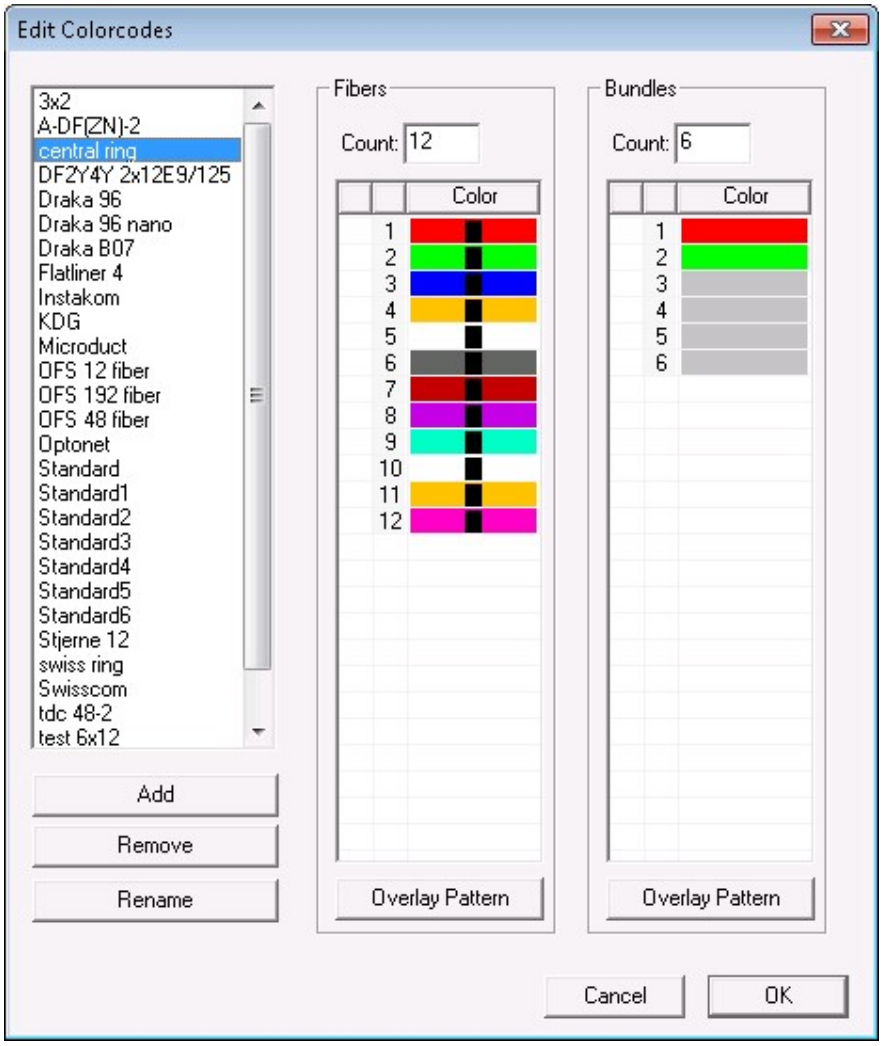


To edit a multicolour, click on the *Color* button.

5.1.2 Multicoloured colour code

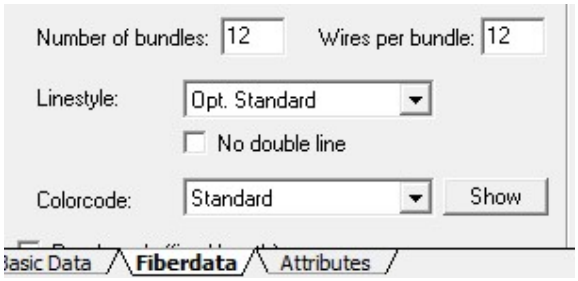
Colour code entries can be multicoloured.

In the *Edit Colorcodes* dialog (*Edit > Color codes*), you can edit the multicolour of a single wire or bundle by double-clicking on the multicolour cell. Using the *Overlay Pattern* button, you can assign a pattern to all selected items. If no item is selected, the pattern is assigned to all items. This batch mode applies to ring pattern edits only. The background colour is ignored in this batch editing mode.



5.1.3 Colour codes for fibre cables

In AND 4.9 and earlier versions, there was a global colour code list. Each fibre cable would reference a colour code in that list. From AND 4.10 on, each fibre cable has its own instance of a colour code. The global colour code list still exists, but is used only for copying purposes.



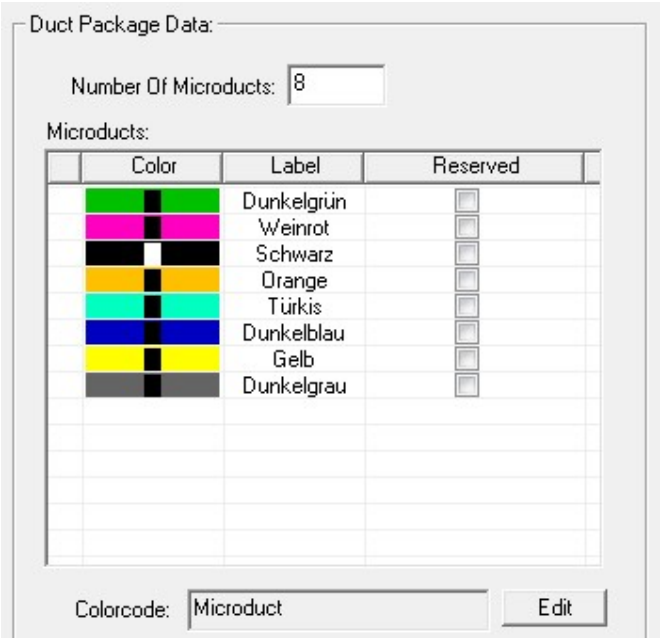
Fiberdata dialog in LibEdit

The *Show* button is for both viewing and editing. The colour code of the cable (e.g. "Standard") may differ from the "Standard" code in the global list. In this case you see two colour codes in the *Show* dialog called "Standard". The one assigned to the cable is selected, and the one from the global list is displayed with the suffix "(global)".

If there is no second entry with the suffix "(global)", the cable's colour code and the global code are identical. In this case, if you edit the global code in that dialog and select *OK*, the cable's code is a copy of that global code.

5.1.4 Multicolours for microducts and copper cables

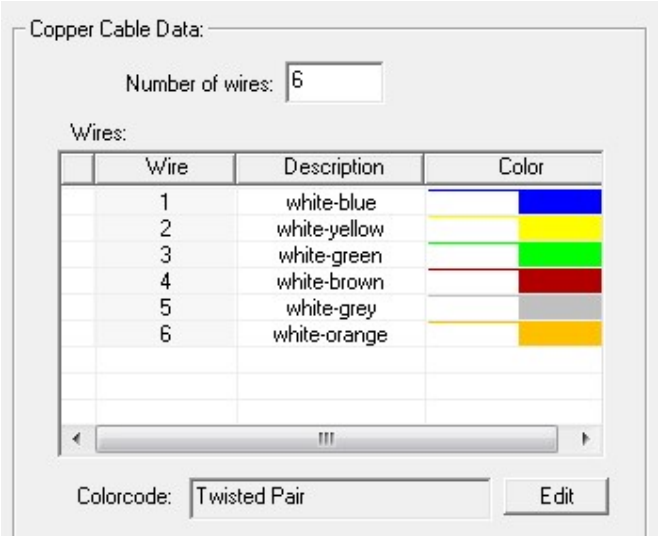
Microducts and wires of copper cables can also be assigned multicolours.



Edit dialog for Duct Package Data in LibEdit

You can edit individual multicolours by double-clicking on the respective cell. Select the *Edit* button to modify or replace the entire code.

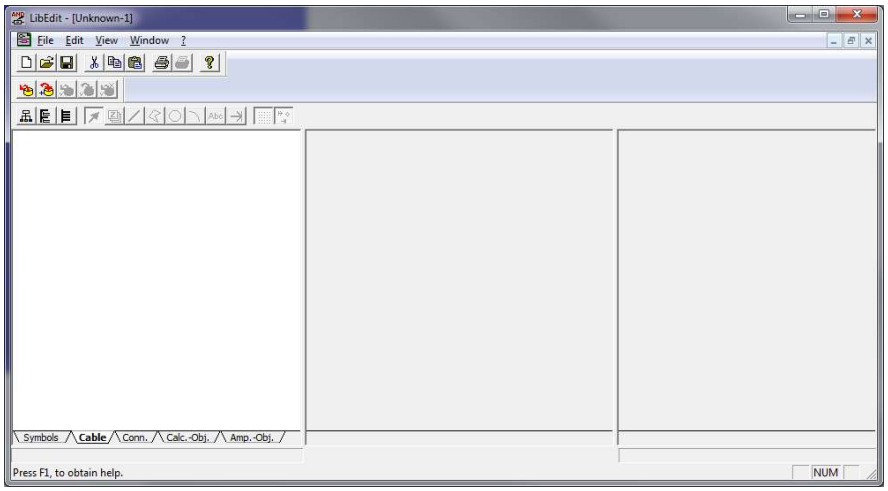
The same multicolour functionality is available for the wires of a copper cable.



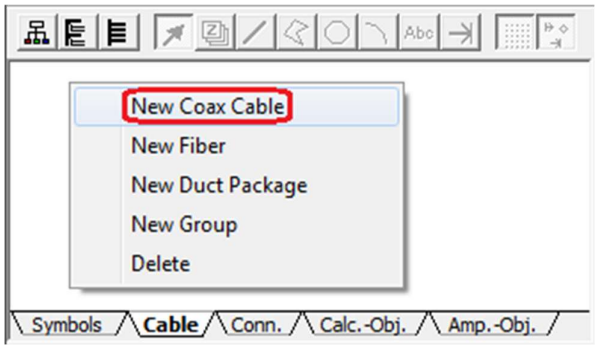
Note that in LibEdit and AND, multicolours are displayed as vertical patterns because a vertical layout is the best method to display several colours. For microducts, the pattern is often horizontal. For copper cables, multicolours might be used to represent the 2 colours of the twisted wire pair.

5.2 Coaxial cables

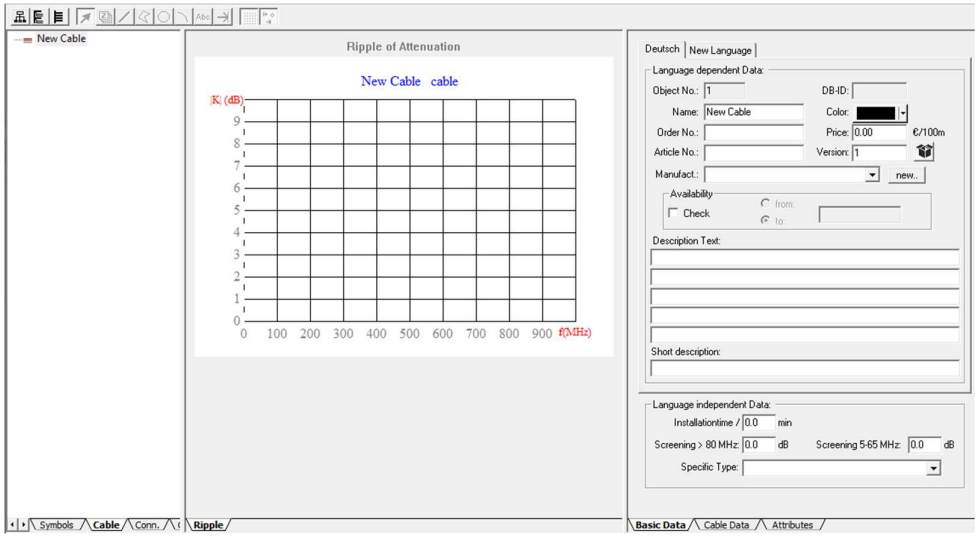
After you have selected the "Cables" tab in the left-hand section, the following screen will appear:



You will find information on each individual section in section **1.3 Screen layout**
If you want to create a new coaxial cable, move the mouse pointer into the left-hand section. Press the right mouse button once here to obtain a context menu.



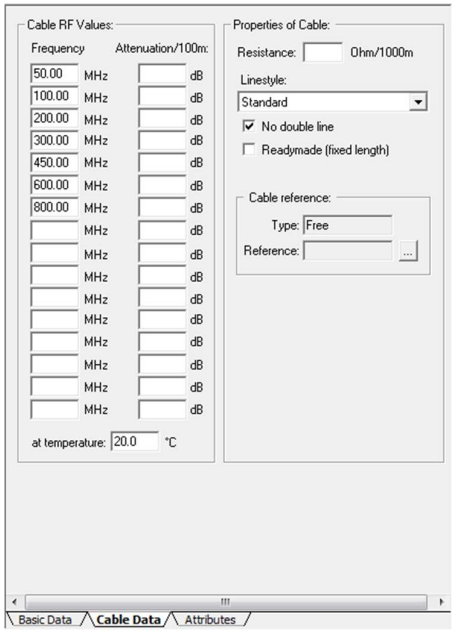
After you have selected the required cable, the relevant screen is displayed:



The basic data is entered in an analogous way to the symbols.

See also 1.5.1 Basic data

You can create cable groups.
When a cable is replaced, AND only suggests cables that belong to this group.
You now switch to the "Cable Data" tab in the right-hand section.
The following window then opens:



The figure shows the input form for the RF data.
Values are already suggested to you by default.
These can be overwritten at any time.

5.2.1 RF values

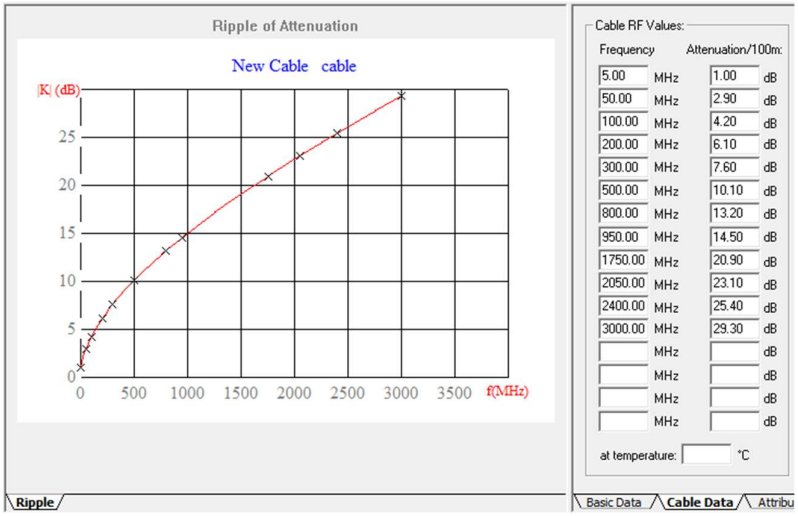
„Frequency ..... MHz ..... dB"
Value pairs are created here.
These refer to a frequency and the associated attenuation/100m for the temperature specified here.

The attenuation value for a cable with integral connectors is absolute.

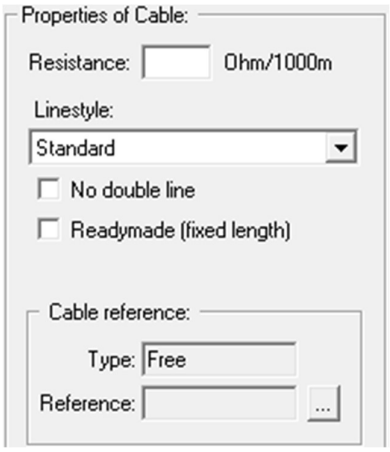
Values can be entered quickly by clicking on the first entry field for frequency with the left mouse button and then moving on field by field with the  key.

Please note that these must be as evenly spread as possible over the entire frequency range of the cable.
The program interpolates between the individual data according to theoretical knowledge of the attenuation curve of a coaxial cable.

You can create up to 16 value pairs for one coaxial cable.
At the same time, the frequency curve is graphically displayed in the center section.

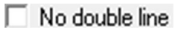


5.2.2 Cable properties



„Resistance: ..... Ohm/1000m"
Enter the loop impedance for 1000 meters of the cable without connectors fitted.

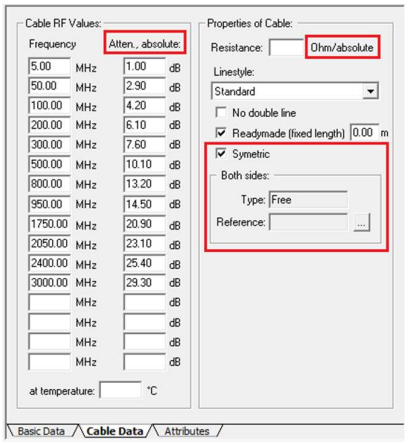
"Linestyle:"
You can choose between 7 other drawing styles in addition to the standard drawing style.



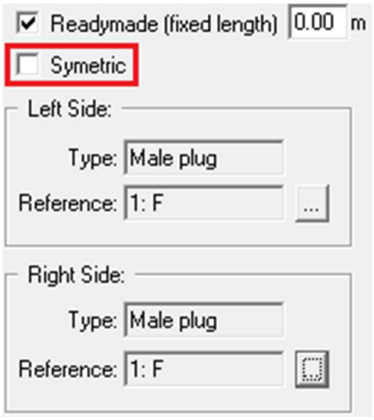
If you select this check box, AND will not permit a double line for this cable
(the second continuous line is not permitted).

With this check box, you can define whether the cable has integral connectors.
If you select this check box, the input form changes as shown in the figure below.
The attenuation and impedance values are absolute values for the fixed length of the cable with integral connectors.
Because a fixed length of cable is assumed, the attenuation values should correspond to the length of the cable created. AND considers the real attenuation values
for each calculation frequency.

If the cable is created with integral connectors, a second check box appears.
If you select this check box, it is assumed that both cable ends have the same connector (see the following figure).



If this check box is cleared, a distinction is made between the two cable ends (see picture below).



For cables having different connectors at each end, it is possible to create different references. Because the position of the cable is relevant to the search for a connector in AND, the ends are shown with symbols, so that planners can distinguish between the two ends).

It is mandatory to select the connector references to match the integral connectors when defining a cable with integral connectors.
If this is not done, the program automatically clears the check box for integral connectors.

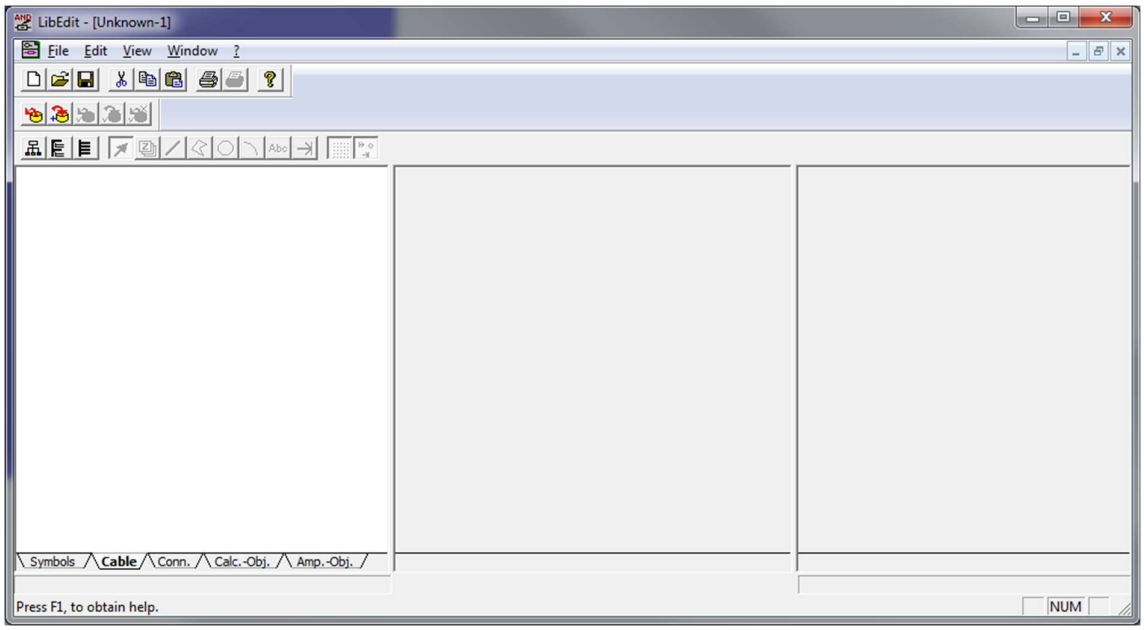
You will find information on references in section 2.4.3 Cable and connector references.

You can define the object properties under the “Attribute” tab.

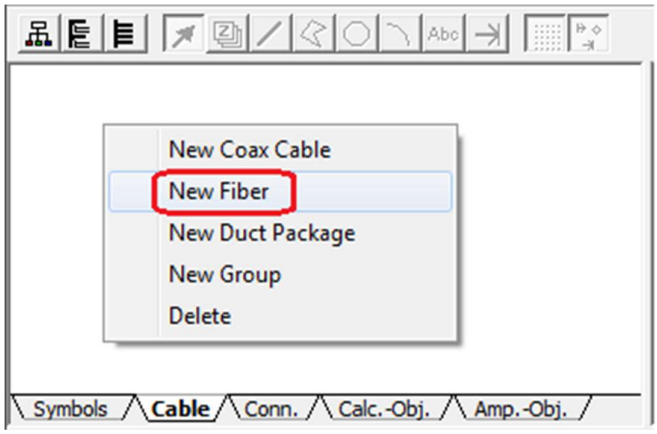
You will find information on attributes in section 1.6.4 Attributes.

5.3 Fiber-optic cables

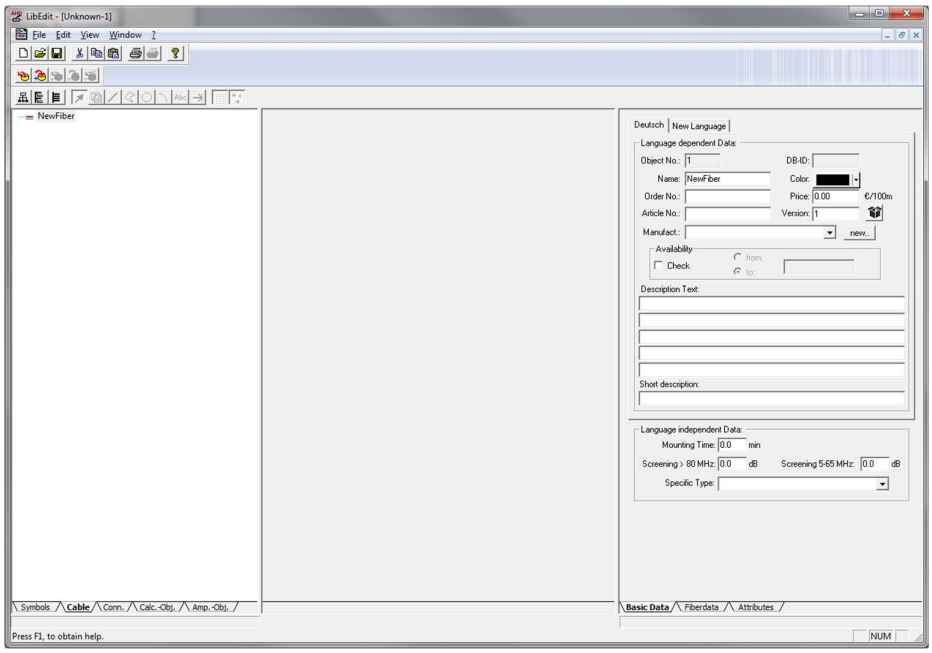
After you have selected the "Cable" tab in the left-hand section, the following window will appear:



If you want to create a new fiber-optic cable, move the mouse pointer into the left-hand section.
Press the right mouse button once here to obtain a context menu.



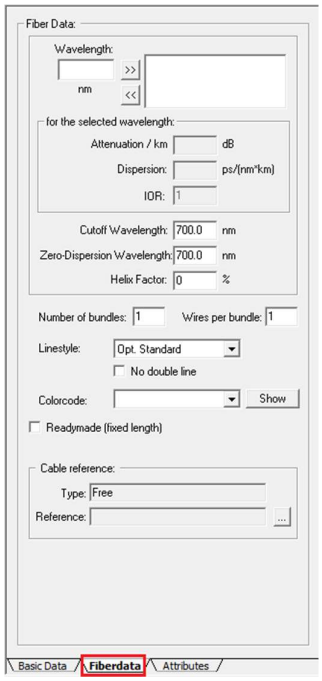
After you have selected the required cable, the relevant window is displayed:



The basic data is entered in an analogous way to the symbols.

See also 1.5.1 Basic data

You now switch to the "Fiberdata" tab in the right-hand section.
The following window then opens:



The figure shows the input form for the fiber-optic data.
Values are already suggested to you by default.
These can be overwritten at any time.

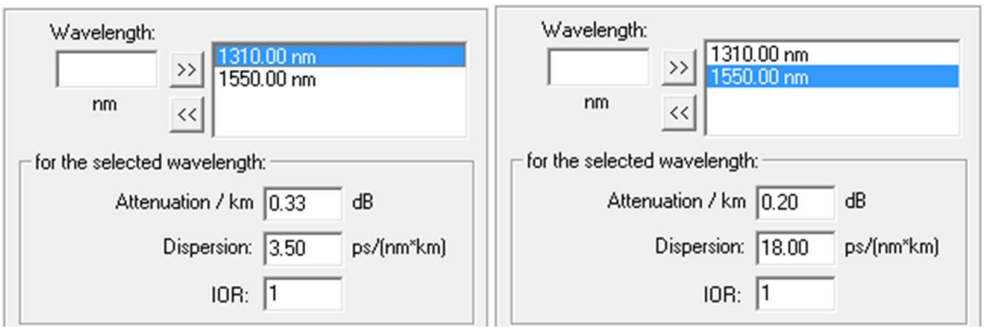
5.3.1 Fiber data

„Wavelenght: ..... nm"
Here you define the wavelengths for the fiber-optic cable.
These refer to all fibers of the cable.
You can enter multiple wavelengths.
Move each entered wavelength individually into field for the transmission range

of the fiber-optic cable with the >> button.

Use the << button to remove a selected wavelength from the window again.

This input form is generated afresh for each defined wavelength.



That means, if you select the wavelengths entered one after the other,
the data for attenuation/km and dispersion will only be displayed for the selected wavelength.

"Attenuation / km ..... dB" for the selected wavelength
Here you enter the attenuation for 1 km of the fiber-optic cable with reference
to the selected wavelength.

*"Dispersion: ..... ps/(nm*km)" for the selected wavelength*
Here you enter the dispersion of the fiber-optic cable with reference
to the selected wavelength.

"Cutoff- Wavelenght ..... nm"
Here you enter the cutoff wavelength.

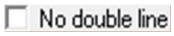
"Zero- Dispersion Wavelenght ..... nm"
Here you enter the zero-dispersion wavelength.

"Helix factor …….%"
((To be completed))

"Number of bundles ..... "
Here you enter the number of bundles of the fiber-optic cable.

"Wires per bundle ..... "
Here you enter the number of fibers of the bundle.

"Linestyle:"
You can choose between 8 other drawing styles in addition to the standard drawing style.



If you select this check box, AND will not permit a double line for this fiber-optic cable (the second continuous line is not permitted).

"Colorcode" :

See 5.2.2 Color code

"Show"

See 5.2.2 Color code

With this check box, you can define whether the fiber-optic cable has integral connectors.
If you select this check box, the input form changes.

The attenuation values are absolute values for the fixed length
of the cable with integral connectors.
Because a fixed length of cable is assumed, the attenuation values should correspond to the length of the cable created. AND considers the real attenuation values for each wavelength.

If the fiber-optic cable is created with integral connectors, a second check box appears.
f you select this check box, it is assumed that both cable ends have the same connector.
If this check box is cleared, a distinction is made between the two cable ends.
The fiber ends of a cable with integral connectors is shown thickened in AND.
It is mandatory to select the connector references to match the integral connectors when defining a cable with integral connectors.

If this is not done, the program automatically clears the check box for integral connectors.

You will find information on references in section 2.4.7 Cable and connector references

You can define the object properties under the "Attribute" tab.

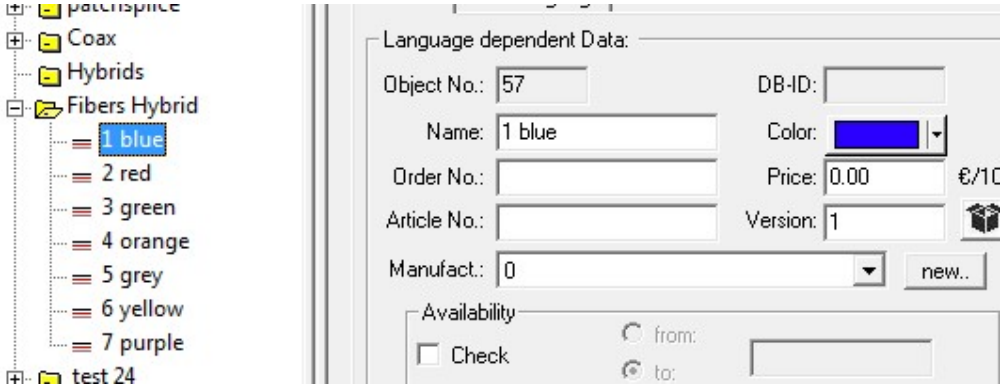
You will find information on attributes in section 1.6.4 Attributes.

5.3.2 3-staged fibre cables

AND supports fibre cables which are organised in three hierarchical levels. The first level is referred to as a "pack", the second level as "bundles" and the third level as single "wires". Example: A cable consists of 7 packs. Each pack comprises 12 bundles, and each bundle contains 12 wires. The cable thus has a total wire count of 7*12*12 = 1,008.
In LibEdit, a 3-staged fibre cable is designed as a formal hybrid cable with subcables of type "fibre".

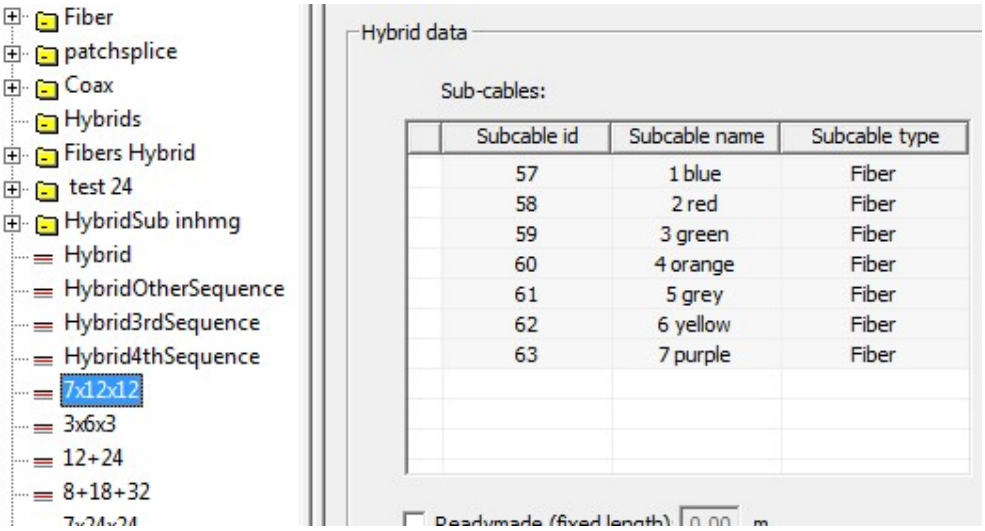
In AND, you can use 3-staged fibre cables in the same manner as any other hybrid cable. AND recognises pure-fibre hybrid cables and offers the same functions as for normal fibre cables, e.g. the "r" function.

In order to design a 7x12x12 cable in LibEdit, create seven 12x12 subcables:



All subcables have identical technical data. They differ only in name and colour.

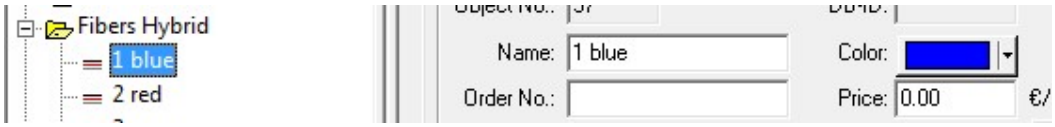
In LibEdit, a 3-staged fibre cable is designed as a hybrid cable. To create a new hybrid cable, select the *New Hybrid Cable* command in the context menu of the tree view. Enter the name "7x12x12", and add the seven subcables to it:



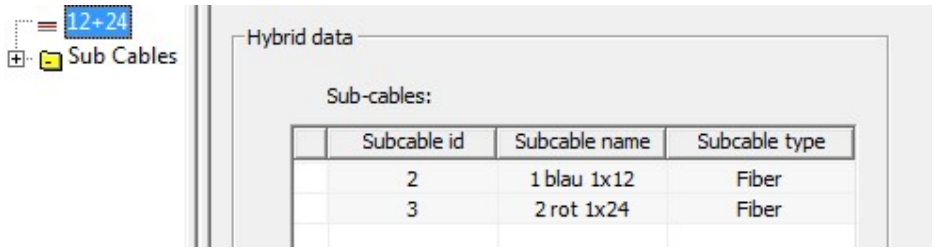
A 3-staged fibre cable with 7 packs. Each pack represents a 12x12 fibre cable.

Use the list control context menu to add and delete subcables.

The library name of a pack appears in the hybrid expanders in AND. It is recommended to assign a suitable colour to each pack. If the cables of a pack are to be ignored in the BOM, make sure there is no price, orderNo or ArticleNo entry for them in the library:



The subcables normally have identical physical properties. But it is also possible to use formal hybrid cables to design inhomogeneous fibre cables, e.g. a 2-staged cable with 12 wires in the first bundle and 24 in the second:



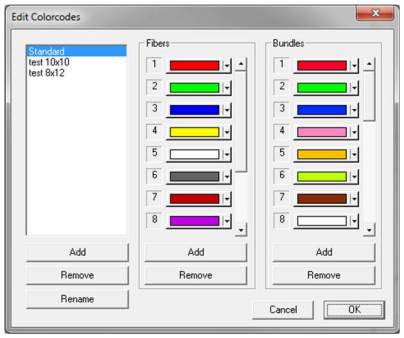
5.3.3 Color code

„Colorcode:"
Fiber-optic cables have a certain code by which the sheath color of bundles and fibers is determined.
The standard color code is suggested here.

You can also select other color codes, display them with the Show button, edit them, or create new color codes.

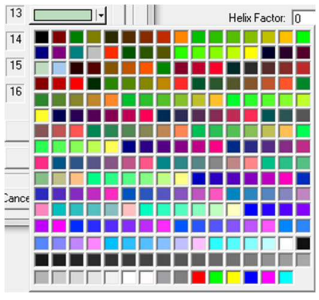
5.3.4 Editing of color codes

You can obtain the window for editing color codes via the Edit/Colorcodes menu item.



Standard colors are stored for 12 fiber colors and for 4 bundle colors.
The fiber and bundle colors recur in the same way.
That means that the sheath color of the 5th bundle of the cable to which this code has been assigned, is red again, the 6th bundle green, etc.
With the Add and Remove buttons, you can define or delete fiber or bundle colors.

If you want to add a fiber color, for example, click on "Add", and a new button will appear with the number 13. This button initially has the color gray.
If you click the new button, you can select a color for the new fiber No. 13 from the color palette that is now open

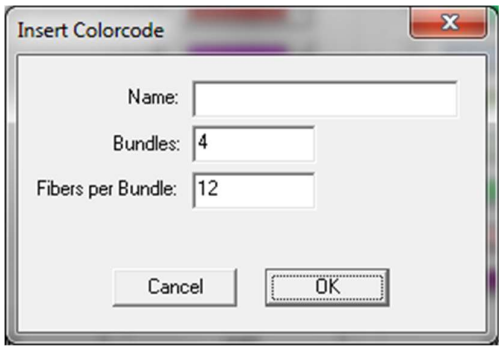


Clicking OK finalizes your entries.

Editing of the bundle colors is analogous.

5.3.5 Adding color codes

If you want to add a new color code, click on “Add” in the list of color codes.
A new dialog box opens.



„Name:“
Enter the name of the new color code here.

“Bundles:”
Enter the number of fiber bundles here.

“Fibers per Bundle:”
Enter the number of fibers per bundle here.

After you have clicked 

the newly created color code appears in the list of color codes.

5.4 Twisted pair networks in AND

In LibEdit, it possible to create twisted pair cables, network components and twisted pair splice boxes. Twisted pair pins can be added to all kinds of optical and coax termination devices.

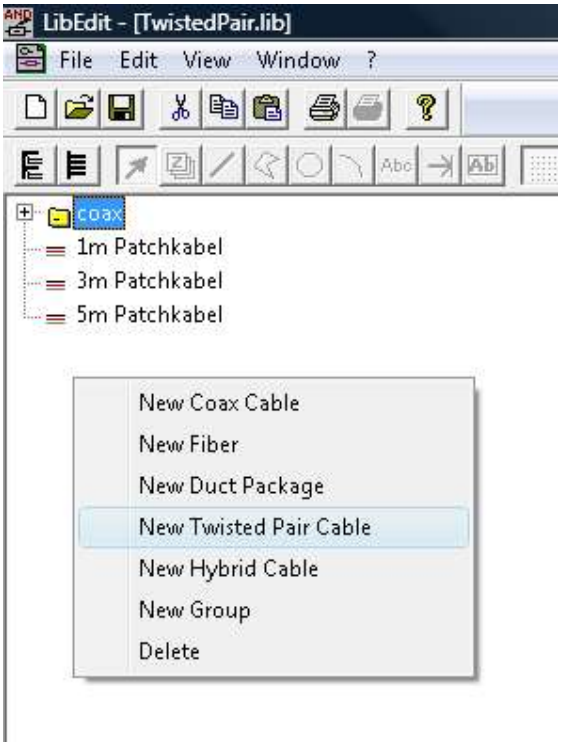
In AND, twisted pair cables are treated like other cables, but can be connected only to pins of type "twisted pair".

If you press "R" for a twisted pair pin, AND shows data of both ends of the route. When routing through a network component AND uses the uplink flag to determine the next pin.

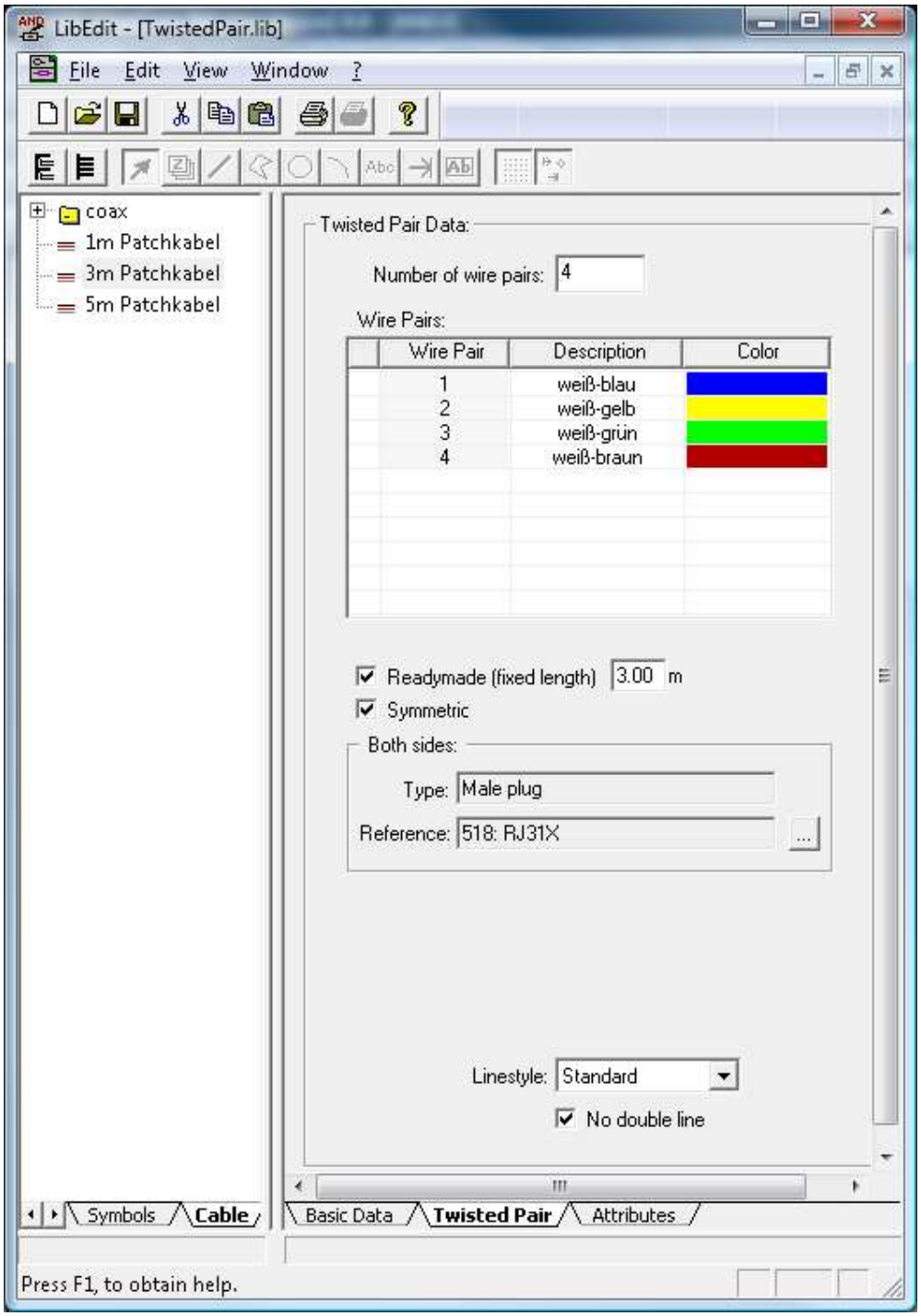
There are 2 distinct use cases. For computer and data networks, you use symbols of type "network component" and connect the pins of the network components with twisted pair cables. For the documentation of telephone networks, you split the twisted pair cables into subordinate wire pairs and distribute them using expanders and splice boxes.

5.4.1 Twisted pair cable

The context menu of the cables view includes a *New Twisted Pair Cable* item:



The *Twisted Pair* tab in the right panel shows the data of a twisted pair cable:



Number of wire pairs: The number of twisted subordinate wire pairs

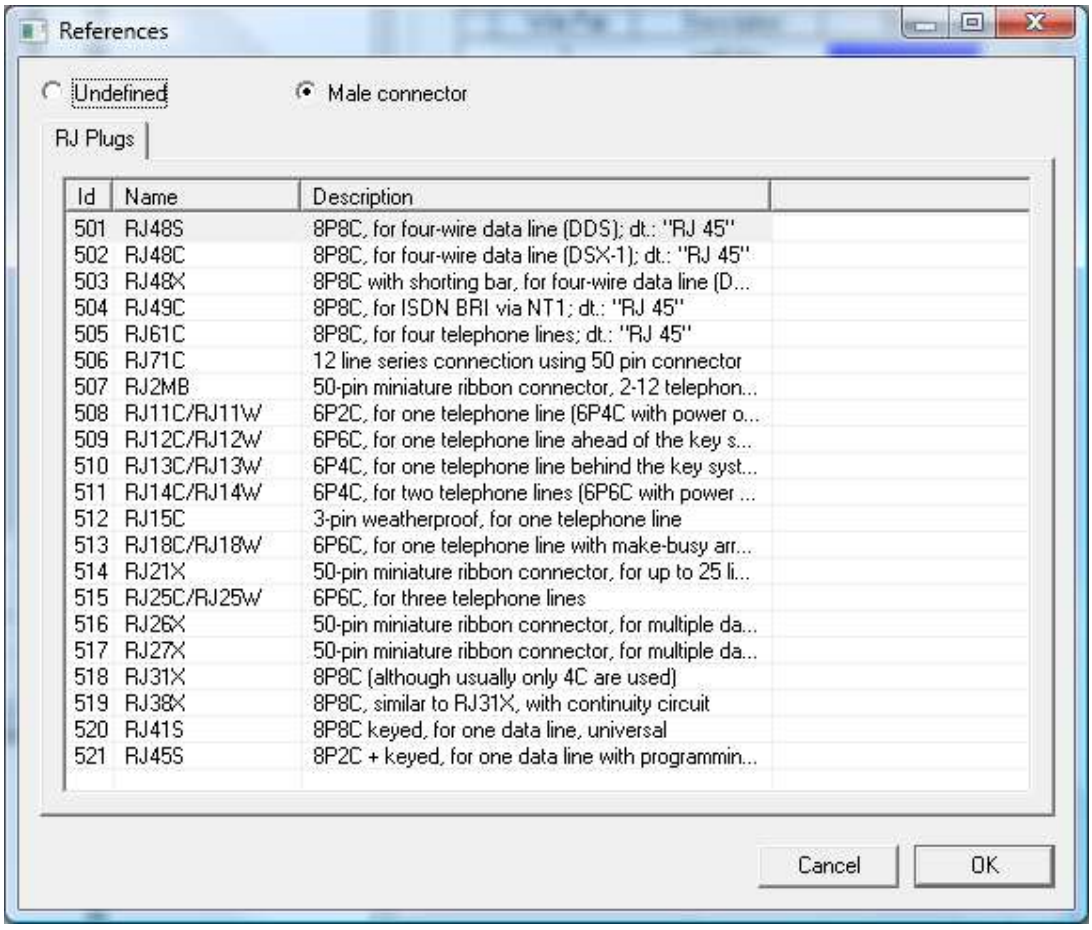
List of *Wire Pairs*: Description and colour of each subordinate wire pair. As AND cannot render striped colours, you can enter only one colour in LibEdit.

Readymade: Enabled if the cable has a fixed length and, if so, the length and connectors.

Linestyle: The line style in which the cable is drawn in AND.

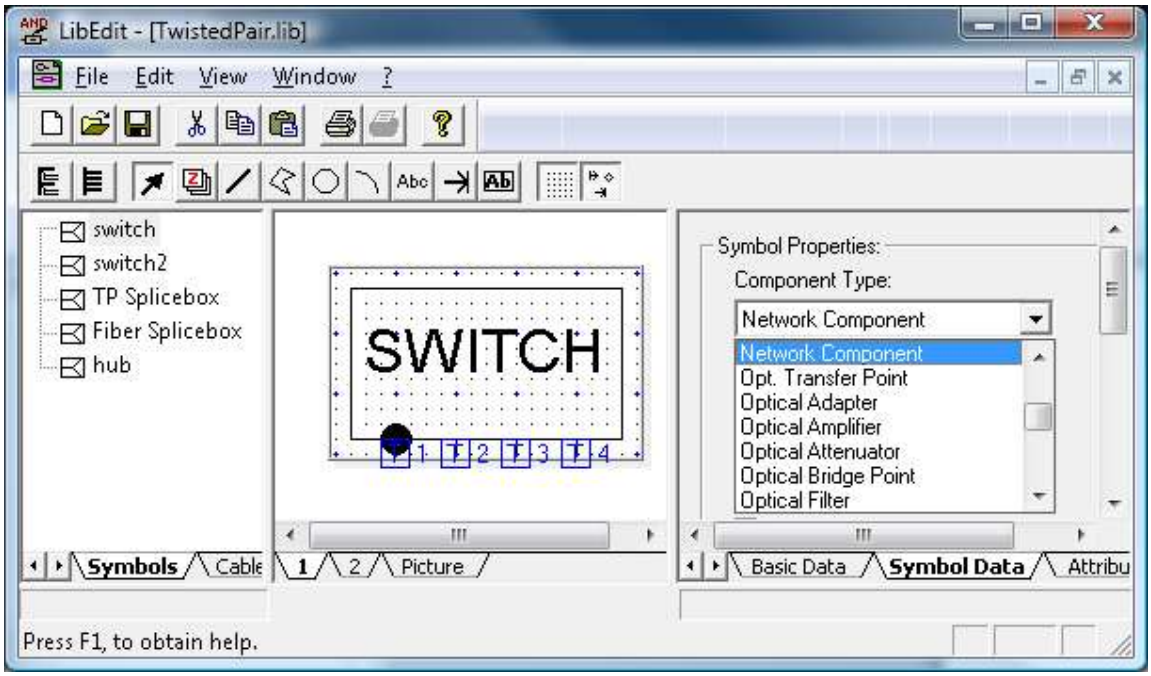
No double line: Do not draw as double line in AND.

The following connector reference types are supported:

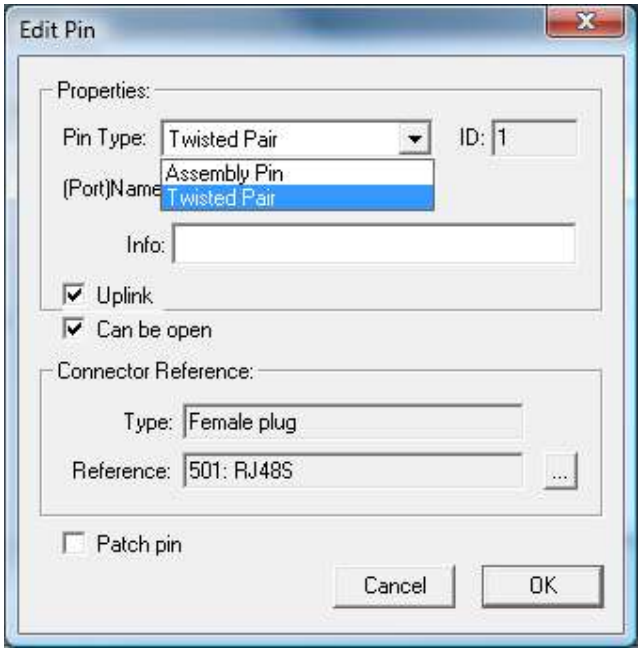


5.4.2 Network components

Switches, hubs etc. are designed as symbols of type *Network Component*:



The available pin types for network components are *Twisted Pair* and *Assembly Pin*; *Twisted Pair* is the default.



Uplink checkbox: If a network component has more than 2 twisted pair pins, one of the pins should be defined as an uplink. If routing in AND enters a network component at the non-uplink-pin, the uplink pin is regarded as the only possible pin to continue routing.

There can be only one uplink pin per symbol. If a pin has already been defined as the uplink, the *Uplink* checkbox is hidden.

The connector type is always *Female plug;* only twisted pair types are possible.

5.4.3 Twisted pair pins for other symbol types

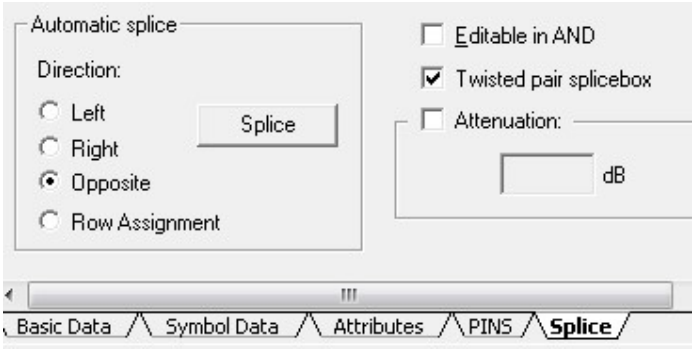
Twisted pair pins are also supported for the following symbol types:

- Optical receiver
- Optical transmitter
- Optical bridge point
- Optical transfer point
- Terminating device
- RF signal source
- RF exit point

5.4.4 Splice boxes for a twisted pair

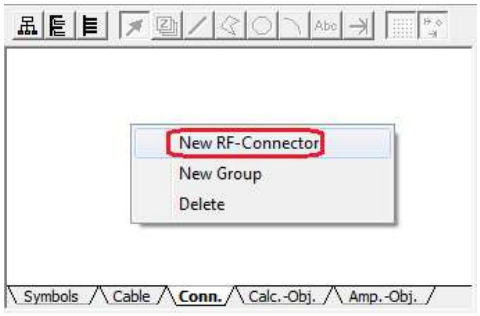
Symbols of type *Splice box* can be used for distributing the subwires of twisted pair cables, too.

Select the *Twisted pair splicebox* checkbox in the view for the splice box data pack.

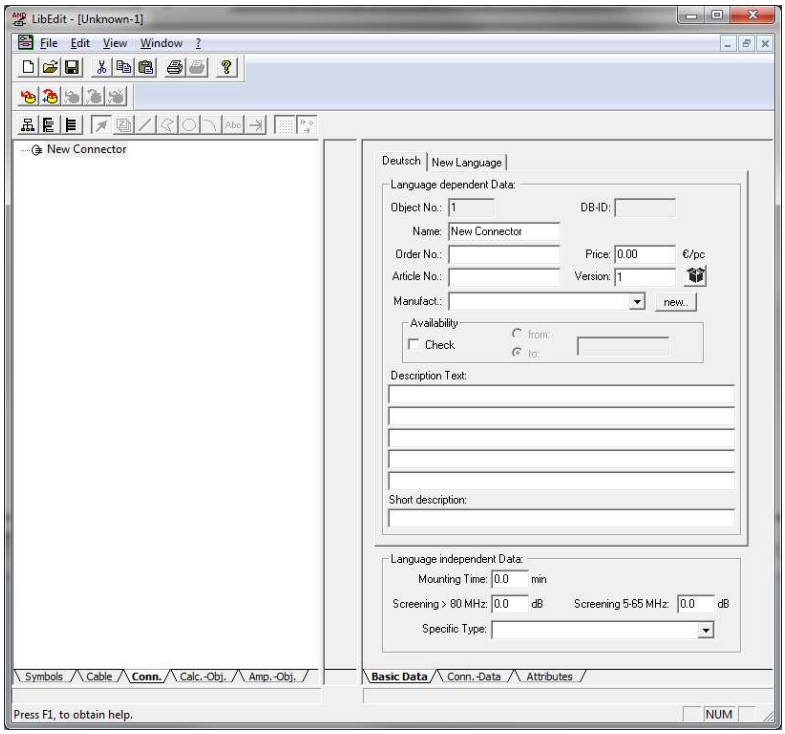


6 Connectors/Adapters

Each library can contain an unlimited number of connectors/adapters.
To edit RF connections or to define new RF connections,
click on the "Conn" tab in the left-hand section, move the mouse pointer into this section, and press the right mouse button.
The following screen opens.



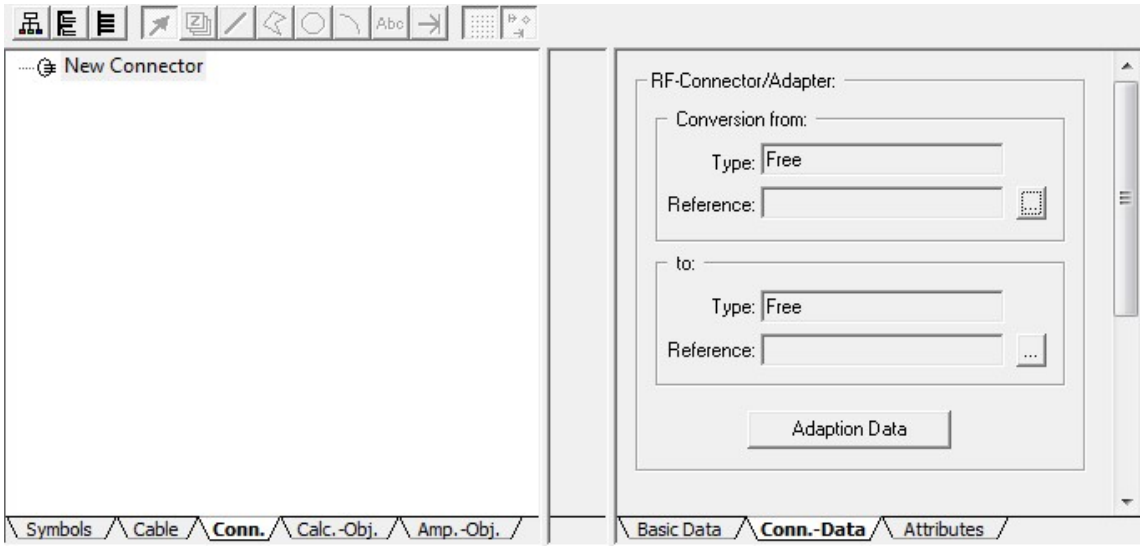
If you confirm this context menu, a new connector is created.



The basic data is entered in an analogous way to the symbols.

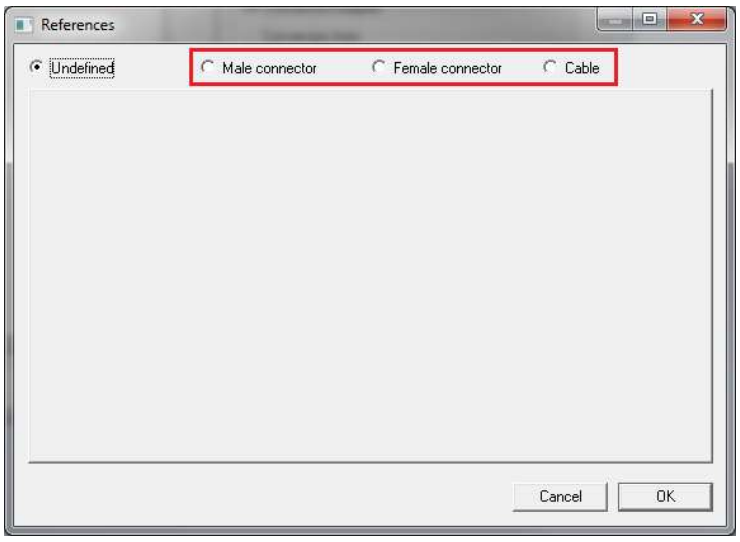
See also 1.6.1 Basic data

You can create connector groups.
These are only for organization of the connectors within the library and do not have any function in AND.
The center section is not used in this mode.
You now switch to the "Conn.-Data" tab in the right-hand section and the following screen appears:



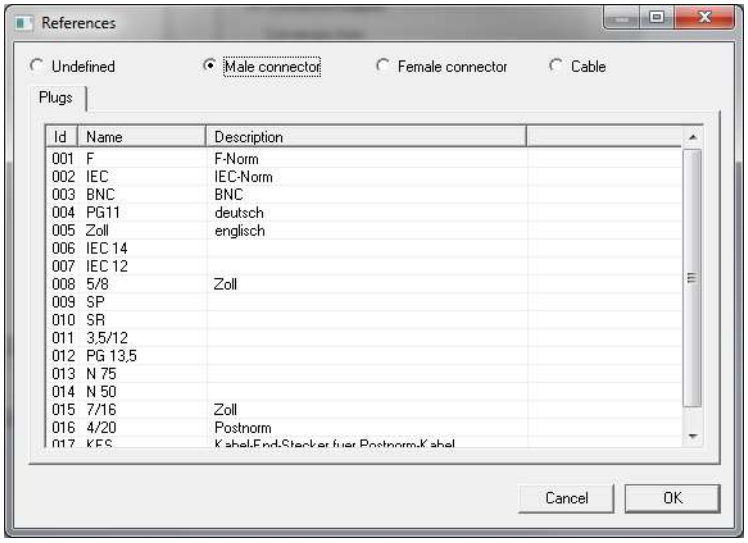
Every connector/adapter always has 2 references.
Here you can reference both ends of the connector/adapter.
If no reference is assigned to a connector/adapter,
this is not taken into account in the connector search in AND.

Clicking the ... button causes a new menu to appear for the first end of the connector/adapter.

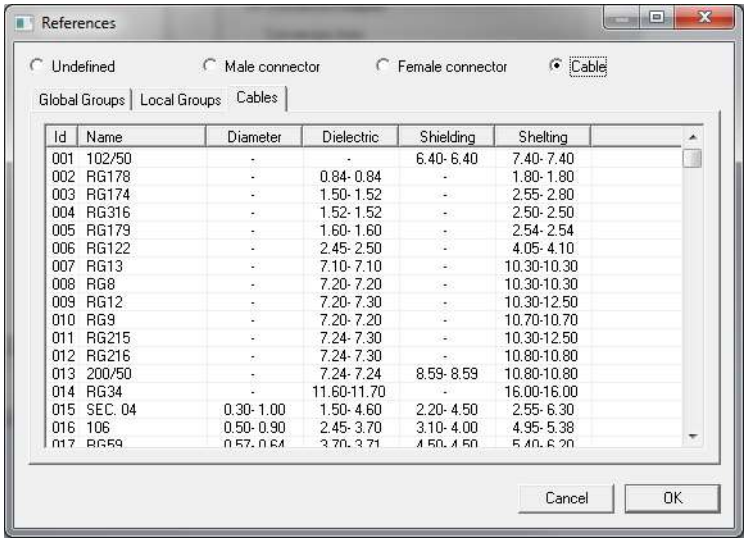


By default each newly created connector/adapter is created as undefined.
That means no referencing has been performed yet.
Each end of the connector/adapter is considered separately.
You therefore first define whether the first end of the connection element is a male or female connector and whether a cable is to be connected here (red marking in the figure above).

If you select this check box, for example, for a male or female connector, a new menu appears with all references for this type.



If you want to connect a cable, select the check box for cables



Select the relevant reference based on the cable dimensions.

See also 2.4.7 Cable and connector references.

Using the 

button, you can create adaptation data for this connector.

You will find information on adaptation data in section 1.6.6 Adaption.

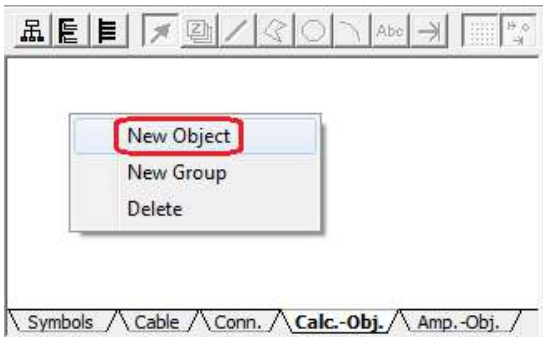
You will find information on attributes in section 1.6.4 Attributes.

7 Costing Objects

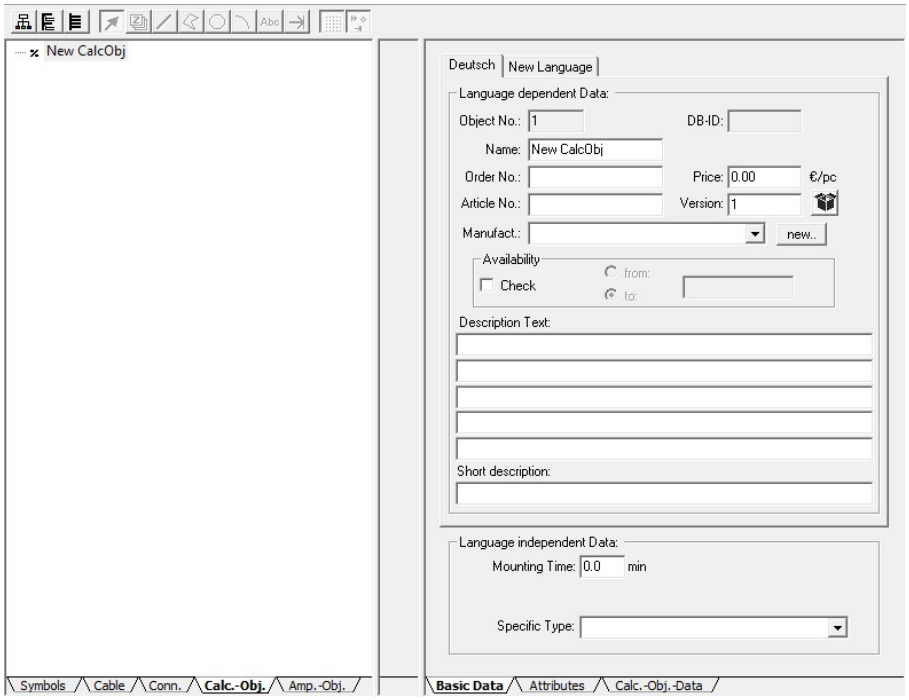
Costing objects are objects that can be assigned in AND either to discrete objects (e.g. antenna wall outlets) or length objects (e.g. cables).

These objects have no effect on electrical calculations but are included in the costing calculations.
Each library can contain an unlimited number of costing objects.
To edit costing objects or define new cost objects,
click on the "Calc.-Obj" tab in the left-hand section, move the mouse pointer into this section, and press the right mouse button.

The following screen opens.



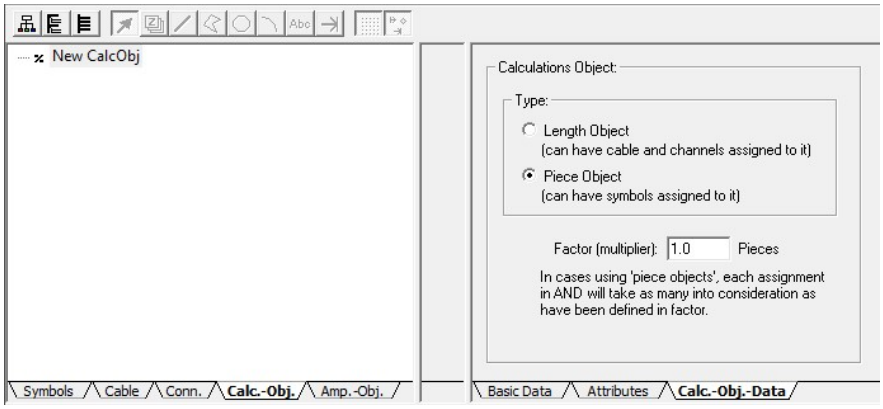
If you confirm this context menu, a new costing object is created.



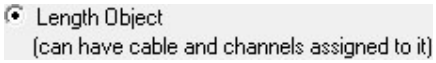
The basic data is entered in an analogous way to the symbols.

See also 1.5.1 Basic data.

You can create object groups.
These are only for organization of objects within the library and
do not have any function in AND.
The center section is not used in this mode.
You now switch to the "Calc.-Obj.-Data" tab in the right-hand section and
the following screen appears:

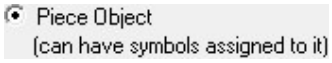


You define the type of costing object here.

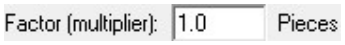


If you select this check box, it is a length object.
This means that this object can be assigned to cables and ducts in AND.
At the same time, the multiplier of this object is generated.

If you assign this costing object (a length object in this case) to a cable in AND, the length in meters of the active cable section results in number of meters of the assigned costing object. If the factor is not equal to 1 for length objects, the length of the assigned cable is multiplied by this factor and the unit "Pieces" is used instead of "m".



If you select this check box, it is a discrete object.
This means that the object can be assigned to symbols in AND.



If you assign this costing object (a discrete object in this case) to a symbol in AND,
the number of units of the costing object that are included in the costing calculation is
the number specified in "Factor".
You can define object properties under the "Attribute" tab.

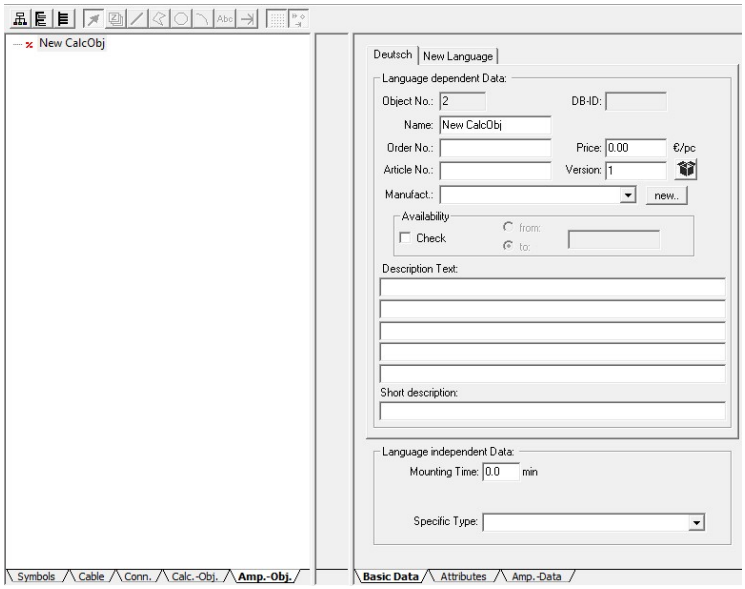
You will find information on attributes in Section 1.5.3 Attributes.

8 Amplifier Objects

Amplifier objects are components that can be used with amplifiers and optical receivers.
These are attenuators, equalizers, cable simulation modules, pilot controls,
passive or active reverse-path modules.
Each library can contain an unlimited number of amplifier objects.

To edit amplifier objects or define one amplifier object, click the "Ampliefier-Objects" tab in the left-hand section, move the mouse pointer into this section, and press the right mouse button. The following window opens.

If you confirm this context menu, a new amplifier object is created.

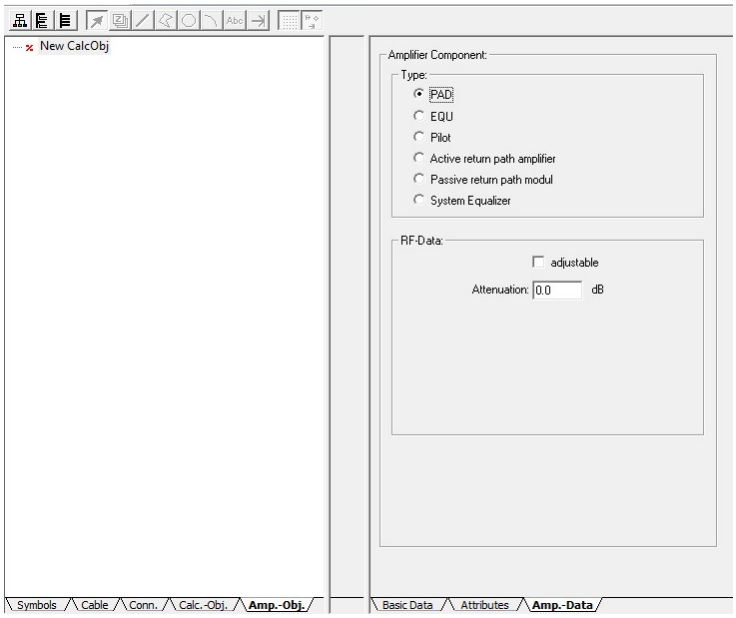


The basic data is entered in an analogous way to the symbols.

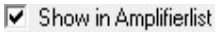
See also 1.6.1 Basic data.

You can create object groups.

Switch to the “Amp.-Data” tab in the right-hand section and the following screen will appear:



Amplifier Component:
Selecting the appropriate check box defines the type of amplifier component.



By selecting this check box, you define that the object will also be listed in the amplifier lists.
On the “Attributes” tab, you can define the object properties.

Information on attributes is given in section 1.5.3 Attributes.

On the “Ripple” tab, you can store data pairs for the frequency slope, for example, for system equalizers

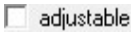
See also 1.6.7 Frequency slope measured data window.

See also 1.5.3 Frequency slope display window.

8.1 Amplifier object PAD (attenuator)

You want to create a PAD.

„Attenuation: ..... dB"
You can define the maximum attenuation value here.



The attenuator can be made adjustable with this check box.
0.... max. attenuation value in dB.

8.2 Amplifier object EQU (equalizer)

EQU

You want to create an equalizer.

adjustable

Selecting this check box makes the equalizer adjustable.
0.... max. equalization value in dB.

"Attenuation: ..... dB"
You define the residual attenuation value of the equalizer here.

"Equalization: ..... dB"
Here, you define the maximum equalization value.
This means that this value is taken into account in calculation of the lowest frequency.
If this value is negative, the value is taken into account in calculation of the highest frequency.

"Min. Frequency: ..... MHz"
You define the minimum frequency of the transmission range here.

"Max. Frequency: ..... MHz"
You define the maximum frequency of the transmission range here.

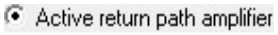
"Upper Turning Point: ..... MHz"
You define the base of the equalizer here.

8.3 Amplifier object pilot control

Pilot

You want to create a pilot control module.
At the moment, only the residual attenuation is considered in AND.
"Attenuation: ..... dB"
You can define the residual attenuation here.

8.4 Amplifier object reverse-path amplifier



You want to create an active reverse-path amplifier:

After you have selected this check box, the "Notch filter data", "IMA", and "KMA" tabs with the corresponding input forms will appear in the right-hand section.
Here you can create measurement series in a similar way to forward amplifiers and have the fault data calculated.
These values are graphically displayed accordingly in the center section.

Reverse-path amplifier data:

"Frequency Range from: ..... MHz"
You define the minimum frequency of the transmission range here.

"upto: ..... MHz"
You define the maximum frequency of the transmission range here.

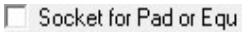
"Amplification: ..... dB"
You define the gain here.

"Attenuation adj. Range: ..... dB"
You define the control range of the integrated adjustable attenuator here.

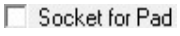
"Equalization adj. Range: ..... dB"
You define the control range of the integrated adjustable equalizer here.

"Self produced Noise: ..... dB"
Enter the value for the noise figure here.

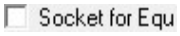
Insertable Components:
Under the heading "Insertable Components", the socket options in the device are defined



This amplifier packet has a socket for an attenuator or equalizer.



This amplifier packet has a socket for an attenuator.



This amplifier packet has a socket for an equalizer.

Here, you define the groups from which AND can select the components for the corresponding sockets.

The components selected in AND then appear both in the amplifier list and in the bill of materials.

After you have selected the “Notch filter data” tab, you can enter the data of the notch filter in the form.

After you have selected the “IMA”/”KMA” tab, you can store the data for IMA/KMA.

See also 3.27.2.2 Entering IMA/KMA measurement data

In the center section, you can calculate and store the CINR from the KMA+IMA based on data entered.

Parallel with that, you can display the calculated curves.

8.5 Stages of Reverse Amplifiers can be split to different parent components

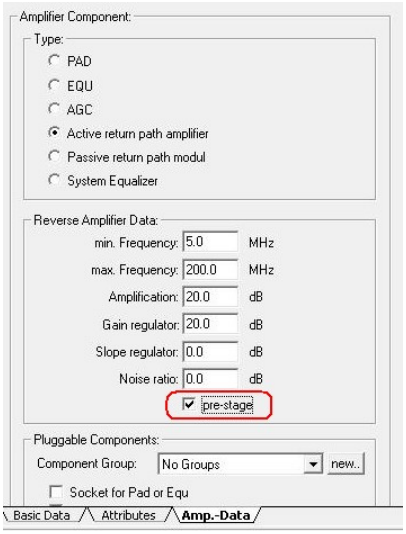
Implemented from Build 4.5.1277 on, Mantis 9217.

It is possible to divide the stages of a reverse amplifier into different library components.

Details

The reverse amplifier adjustment in AND recognizes all stages automatically as one reverse amplifier, if all stages are plugged into the same component (but different amplifier packages)

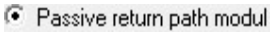
If the stages are plugged into different library components, the automatic detection won't work. For this use case there is a checkbox "pre-stage" in the library:



If you set the check „pre-stage", you force that this reverse amplifier will be adjusted together with the following one (in sense of upstream signal direction)

I.e. if you have a 3-staged reverse amplifier split over several components you set the check for the first 2 stages in sense of upstream signal direction.

8.6 Amplifier object passive reverse-path module



You want to create a passive reverse-path module.

"Frequency Range from: ..... MHz"
You define the minimum frequency of the transmission range here.

"upto: ..... MHz"
You define the maximum frequency of the transmission range here.

"Attenuation: ..... dB"
You define the through loss here.

"Attenuation adj. Range: ..... dB"
You define the control range of the integrated adjustable attenuator.

"Equalization adj. Range: ..... dB"
You define the control range of the integrated adjustable equalizer.

"Self produced Noise: ..... dB"
Please do not enter anything here.
The entry field is not taken into account.

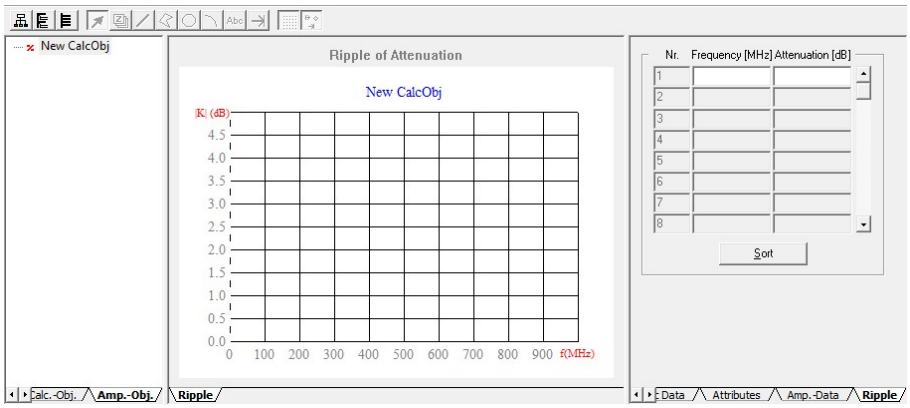
Insertable Components:

See 8.4 Amplifier object reverse-path amplifier

8.7 Amplifier object system equalizer

System Equalizer

You want to create a system equalizer.
If you select this check box, the "Ripple" tab is generated.
If you click on this tab, the following screen will be displayed:



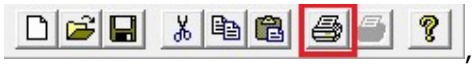
Now enter your data pairs in the right-hand section.
At the same time, the graphical display of the data is shown in the center section.

Sort

The data is sorted with this button.

9 Printing Libraries

To print the active library, confirm either with the button on the toolbar with the red border



,

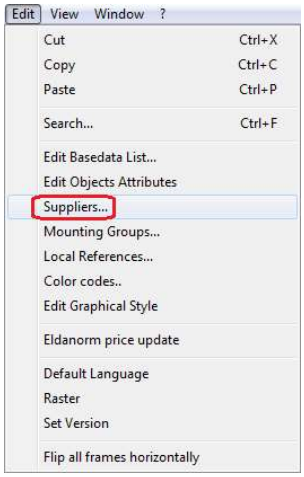
by pressing the shortcut Ctrl+P, or by choosing the “Print” menu item from the menu bar.

Make your settings here and confirm with 

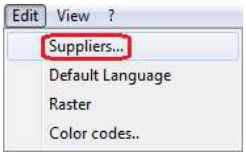
.

10 Supplier Data

You can access the supplier data using the menu.

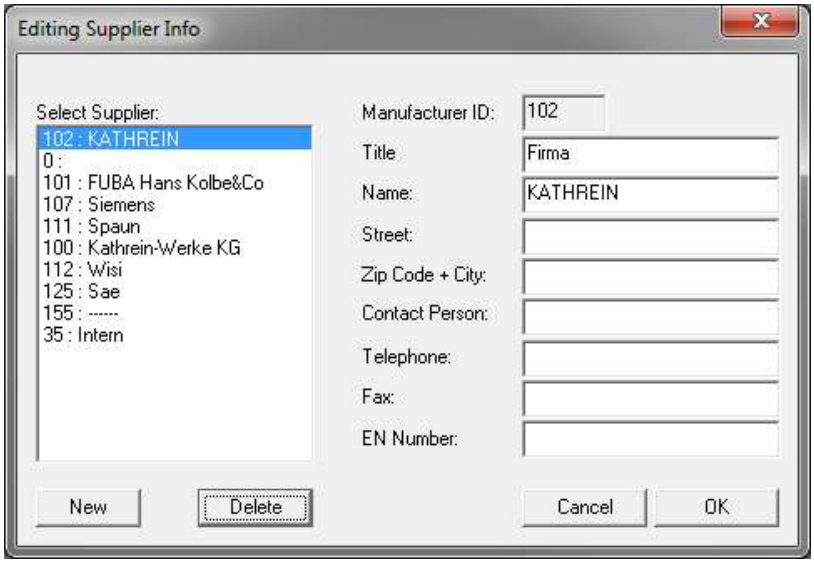


Context menu when the library is loaded.



No library loaded.

You can now edit the supplier data in the following dialog box.



The New button is used to create new supplier data.

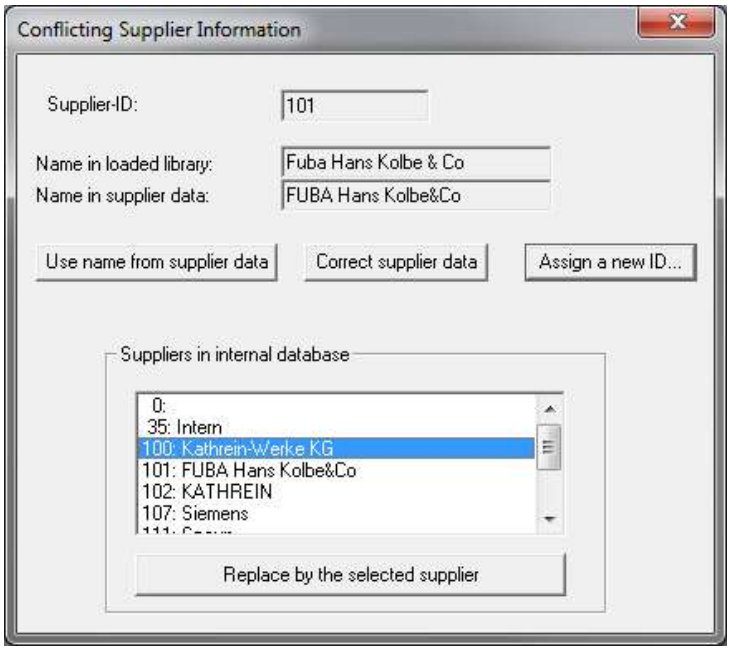
Each supplier is automatically assigned a manufacturer ID.

The Delete button is used to remove suppliers from the list if no library is loaded.

The OK button finalizes your entries.
At the same time, all changes are stored with the library and also in a separate file (supplier.and).
This file is located in your AND directory.

Resolving conflicts

Because the supplier data is stored both in the library and in a separate file, inconsistencies can arise on loading third-party or older libraries.
When a library is loaded, the data stored in it is compared with a separate file.
If this reveals conflicts, they are displayed here.



Using this dialog box, you can decide whether to retain the previous data, take over the new data, or assign a new ID or replace the supplier with another supplier.

Use name from supplier data

The data from a separate file (supplier.and) is used and the supplier is replaced from the loaded library.

Correct supplier data

The data of the loaded file are retained and the entry is replaced in the separate file (supplier.and).

Assign a new ID...

The supplier in the loaded library is given a new ID number and therefore available as an additional supplier.

Replace by the selected supplier

The supplier is replaced by the selected supplier.

Index

E

F

G

H

I

K

L

M

N

O

P

R

S

T

U

V

W

Glossary
(see Separate Manual)

A glossary and list of available keyboard shortcuts are combined in a separate manual that forms part of the AND documentation (see Section **Error! Reference source not found.**).