



What's new
in AND 4.21

Table of Contents

Colours for Ducts	- 3 -
Auto-Splicebox Invert Command.....	- 5 -
Expanders excluded from “to centre” length calculation in site plans	- 6 -
Microduct Block Deploy improvements	- 7 -
The geographic path of existing subducts is kept when the block deploy is called repeatedly for existing micro-duct packages (ANDEV-3594)	- 7 -
Optional start with other sub-duct in micro-duct automatic deployment feature	- 8 -
New Assignment Feature for NCW Trenches	- 10 -
Special case “two crossed trench lines”	- 12 -
Special case “cables already assigned”	- 12 -
Synchronization between Meta-Data and AND attributes of Trenches.....	- 13 -
Splice Report Improvements	- 14 -
New SideSort and improved SideOrder	- 14 -
Hierarchy.....	- 15 -
AND / Library Object.....	- 15 -
Summary	- 15 -
New dynamic label accessor i\$	- 15 -
Improvement for splice name sorting	- 16 -
Return Path Slope Improvements (ANDEV-3570)	- 17 -
Testpoints.....	- 17 -
Manual Calculation	- 18 -
Netcheck	- 18 -
Equalizer behind bridge point.....	- 19 -
Sub-Ducts via Sheet Connector	- 20 -
Export to SHP (ANDEV-3670)	- 22 -
Block Export	- 24 -
GIS Import (ANDEV-3589).....	- 25 -

Colours for Ducts

Colours for conventional ducts for use in New-Civil-Works module (ANDEV-789)

Conventional pipes used in New Civil Works Routes can now have a colour code similar to cables and micro-duct packages. The colour code can be assigned to a duct object in the LibEdit in the library.

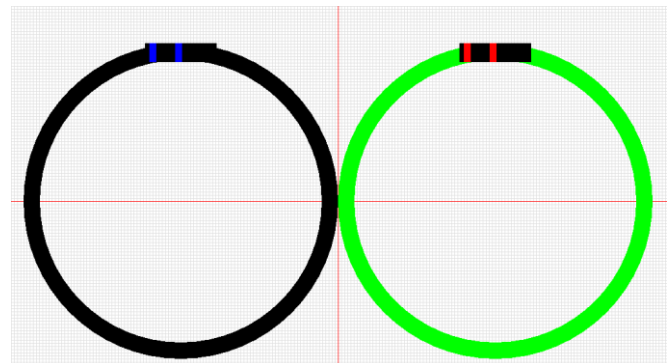


Example from a manufacturer's catalogue

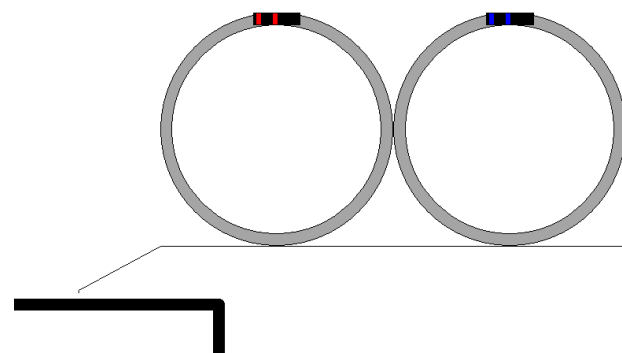
Language dependent Data:			
Object No.:	1	DB-ID:	
Name:	Duct(blue)	Color:	
Order No.:		Price:	0.00 € / 100m
Article No.:		Version:	1

These colour codes can be made visible as labels in:

1. the dialogue for editing the trench cross-section



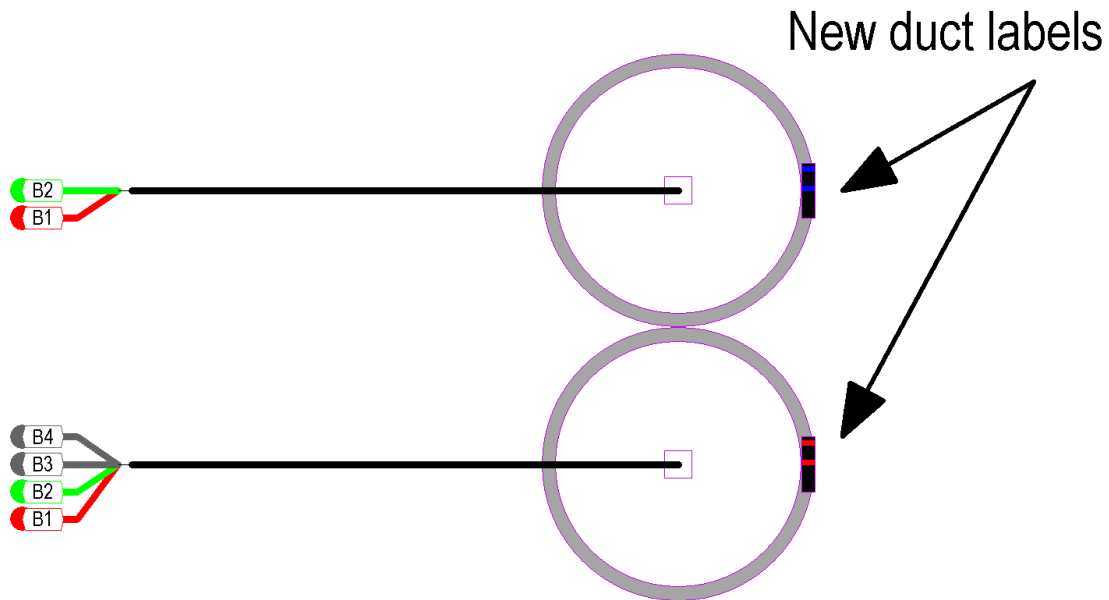
2. in the trench cross-section



3. as label to single drawn duct

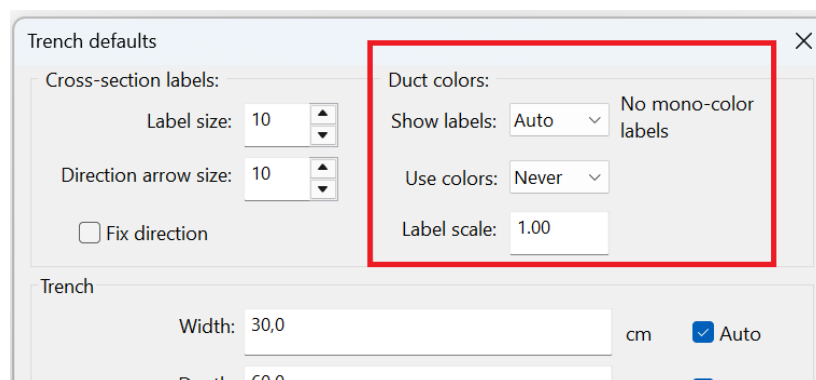


4. in the trench expander (sheet connector)



In case 1, all colour labels are always shown, regardless of the settings (see below) and for both mono- and multi-colour codes.

Cases 2. and 3. are controlled by settings (see **Trench defaults** dialogue):



Show labels can be 'Always', 'Never' or 'Auto', which means that multi-colour labels are drawn, mono-colour labels are not.

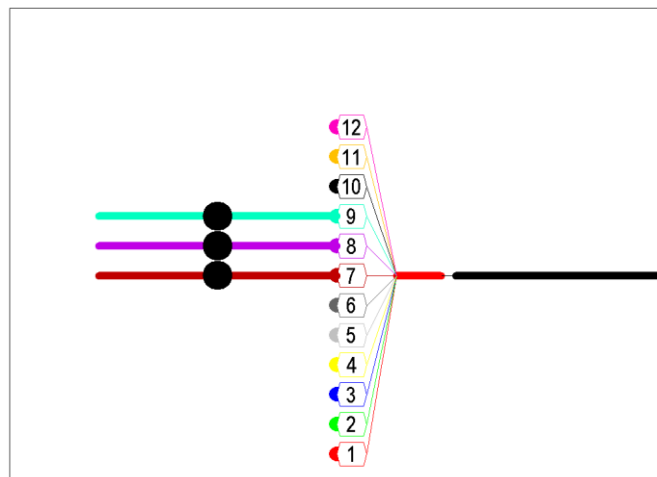
Use colours can be 'Always', 'Never' or 'Auto', which means the old behaviour: ducts are grey, DPs and MDs are coloured.

By default, the options are set to 'Auto' and 'Never', so without changing the settings it should look the same as it does in former versions.

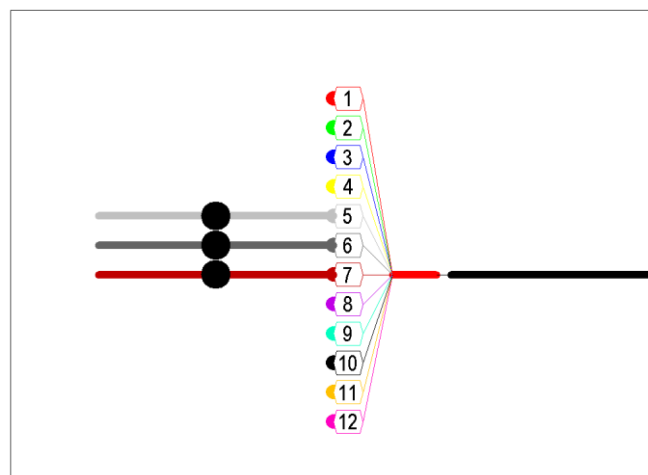
Auto-Splicebox Invert Command

New shortcut # to swap pins of splice-box(ANDEV-2902)

The '# Invert Expander' command was previously available for all other types of expanders and can now also be used for the auto-splice box via the context menu or the shortcut #. This command inverts the auto-splice box, reversing the pin order on the connector side. The colour and thickness of the connected bunches are updated automatically. It is possible that the reverse command will disconnect some bundles. Therefore, it is recommended to perform the flip before connecting the bundles.



Before swap



After swap

Expanders excluded from “to centre” length calculation in site plans

Expanders excluded from length calculation "To Centre" (ANDEV-3753).

For lines (cable/duct package/micro-duct/trench) in the site plan with GIS in scale, the length is usually calculated automatically based on the line points. The calculated length is extended to the centre of the symbol/sheet if a symbol or sheet is connected to the end of the line to compensate for the non-real extent of the (sheet) symbol.

AND Version 4.21 and higher excludes expanders of cable/duct packages, hybrid and twisted pair cables from this extension rule. Calculating the length to the centre of the expander makes no sense. Expanders are logical objects commonly used in schematic sub-sheets. It should rarely be necessary to draw an expander in the site plan. In some cases, the effective length may change.

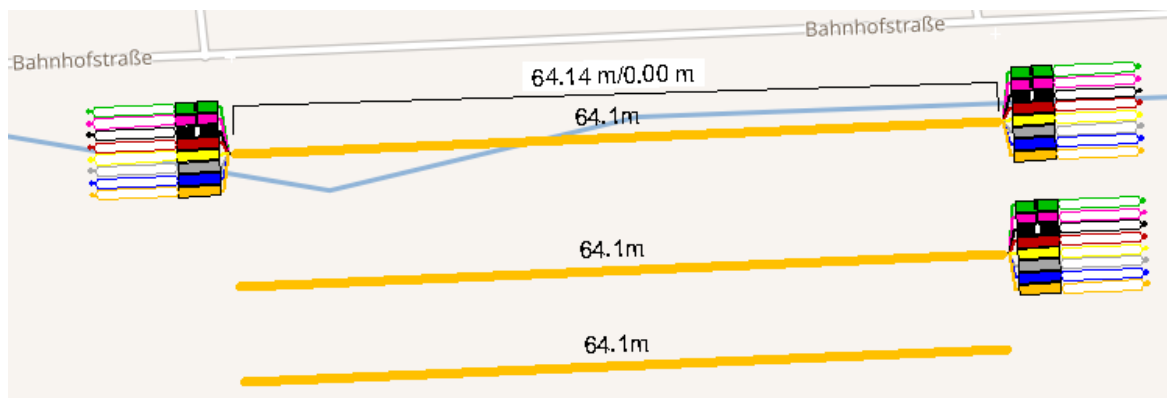


Illustration regarding the length calculation

When loading a project with such an object, a warning message will appear indicating the old and new lengths.

Example warnings output:

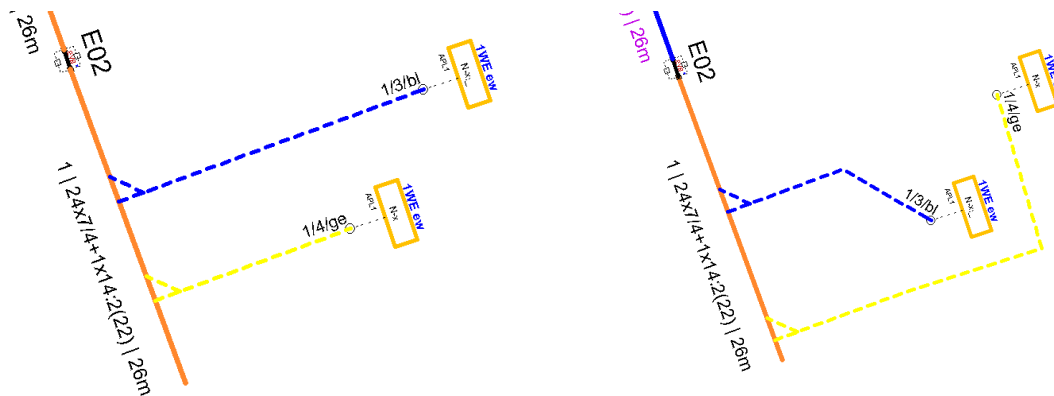
```
Expanders as terminators of lines in site plan don't enlarge the line length any more. old length = 67.7m, new length = 64.2m
Expanders as terminators of lines in site plan don't enlarge the line length any more. old length = 71.2m, new length = 64.2m
```

Microduct Block Deploy improvements

The geographic path of existing subducts is kept when the block deploy is called repeatedly for existing micro-duct packages (ANDEV-3594)

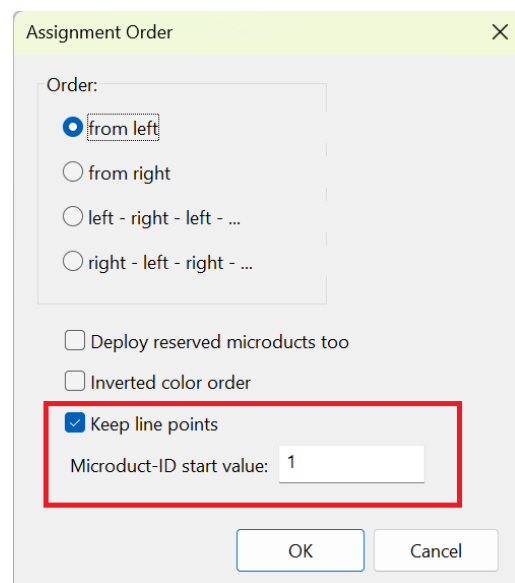
AND's micro-duct block deployment feature automatically creates the sub-ducts (legs) as a straight line to the target object to be connected.

The user can then adjust the graphical routes of these legs.



For the case that the automatic micro-duct block deployment feature should be applied again because for example the duct assignment should change then in AND versions prior to 4.21 the user-edited paths corrections were lost.

Now there is an option "Keep line points". When this option is enabled, existing legs keep their original line points.



Optional start with other sub-duct in micro-duct automatic deployment feature

Microducts: Block deployment can start with a duct other than 1 to keep the colours of the sub ducts free for later connected duct package (ANDEV-1419)

If the user wants to keep the same sub-ducts colours when connecting micro-duct packages later in the path, then this needed a manual effort for the sub-ducts before this connection point. If the automatic block deployment is used, then this starts with first duct (colour) which may be wanted to be kept free for the later duct package. So, the initial legs had to be deployed manually.

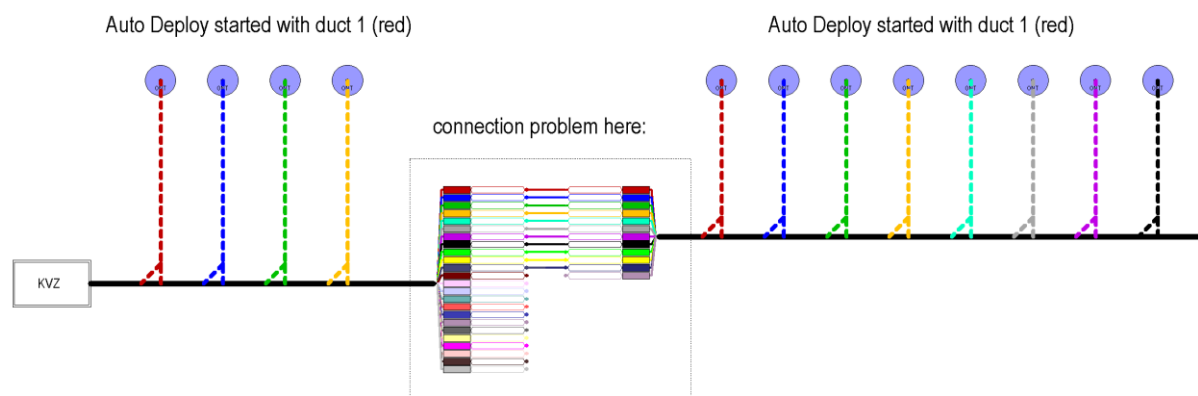


Illustration: sometimes the duct colour can change when different duct-packages are connected – this should practically be avoided

With AND 4.21 it's possible to set the duct (colour) which should be used as first duct for the auto duct deployment feature.

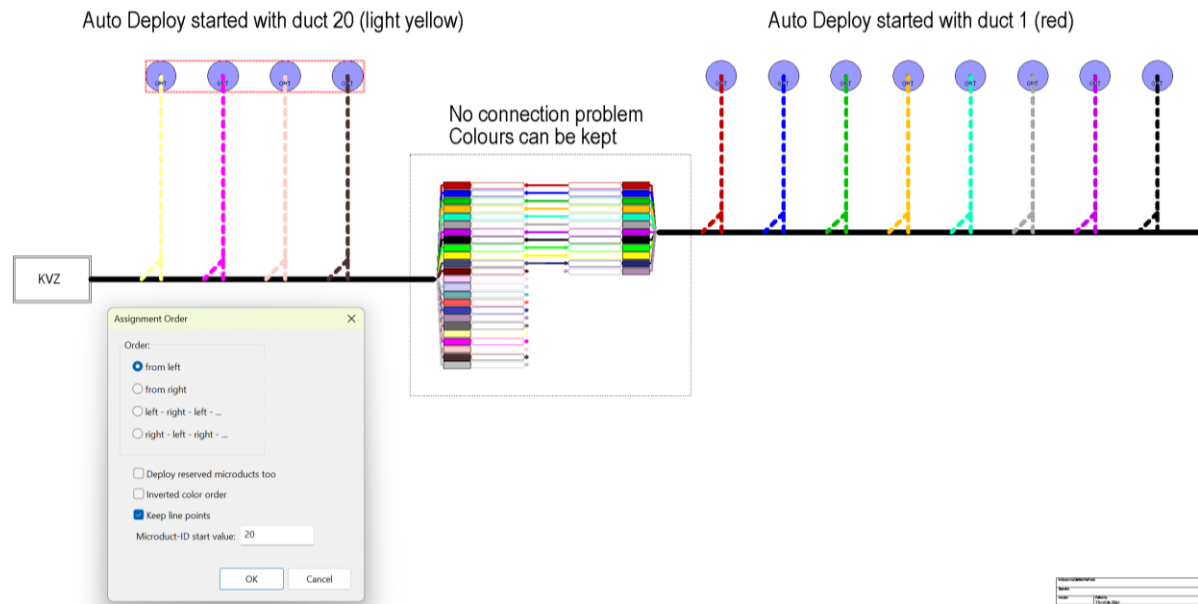


Illustration: keeping the duct colours when connecting different duct-packages

It is also now possible to enter a start ID value for the micro-ducts to be deployed. If a start value > 1 is entered, then all micro-ducts with an ID below this start value will not be deployed.

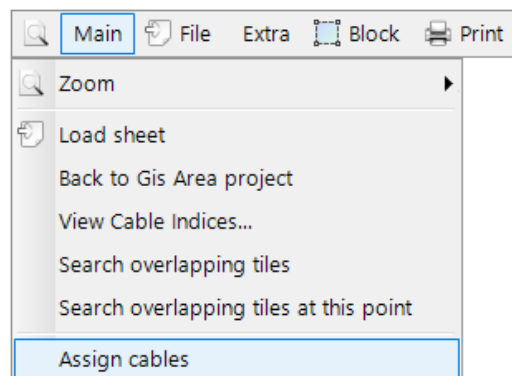
This can be very useful as the Auto Deploy can be set individually for each of the duct packs, especially if there are multiple packs to be supplied.

Together with the "Keep Line Points" option its now possible to repeat the auto deployment feature for each of the connected micro-duct packages without losing previous work.

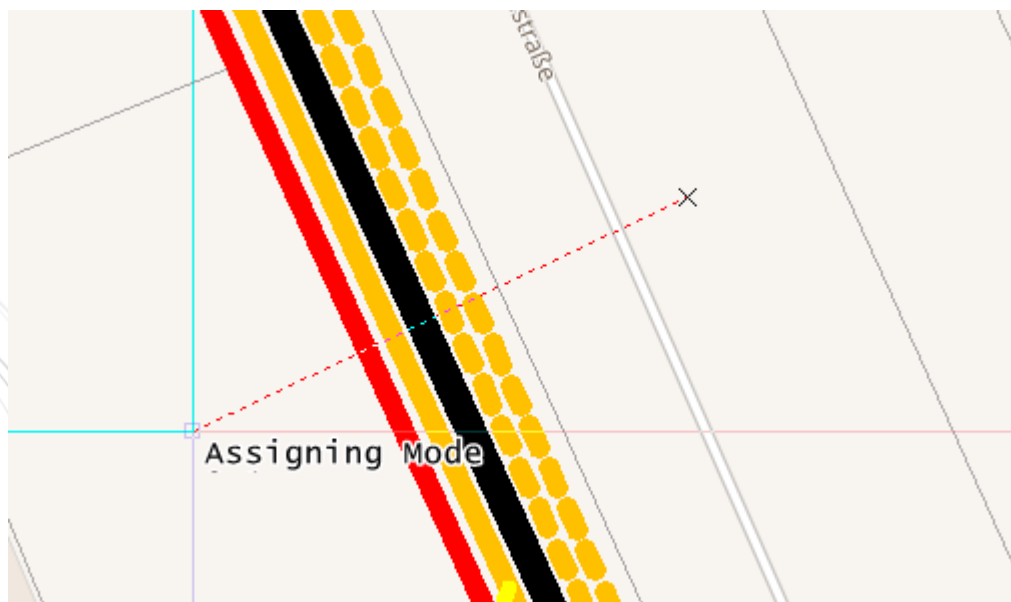
New Assignment Feature for NCW Trenches

Improved assignment feature for NCW: assign duct-packages and cables via crossline to trench (ANDEV-3607)

Assigning micro-duct packages or cables to NCW trenches is time-consuming and error prone. To make it easier, there is a new feature, the 'Assign Mode'.

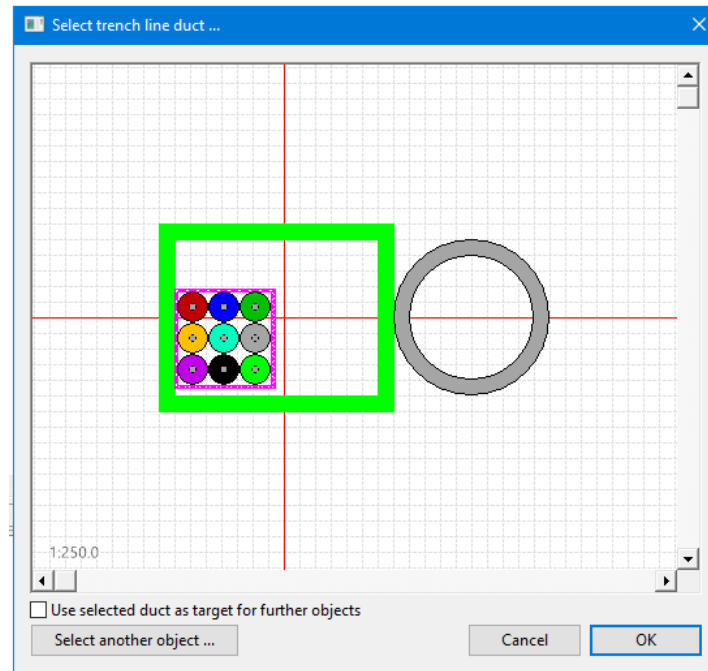


While working in a project, right-click on the main menu and select "Assign Cables". The cursor changes to the assigning mode. With the first click, the user begins drawing a temporary line.



Cables and micro-ducts crossing the temporary line after the second click are assigned to the trench line which also has been crossed ...

... and the placement window shown below appears to allow the user to place them in the correct position.



Also, a message is displayed in the Output window informing the user which cable (installation number) has been assigned to each duct.

Example:

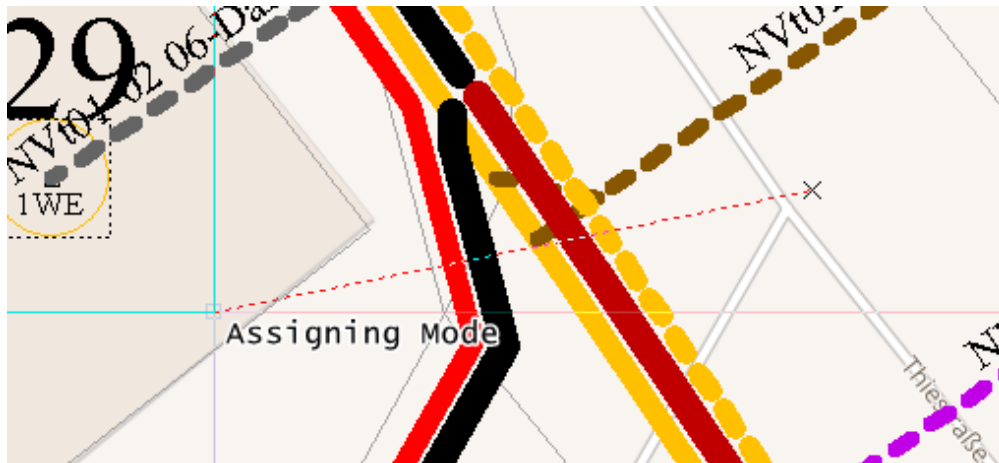
The object dp01 was assigned to the trench line :9.
The object hibrid01 was assigned to the trench line :1.
The object pigtail 01 was assigned to the trench line :10.

After that the temporary line disappears and the next allocation can be done with the marking-line, or the user can press ESC if no further allocation is needed.

If the line does not cross a trench line, a warning appears that no trench line was crossed.

Special case “two crossed trench lines”

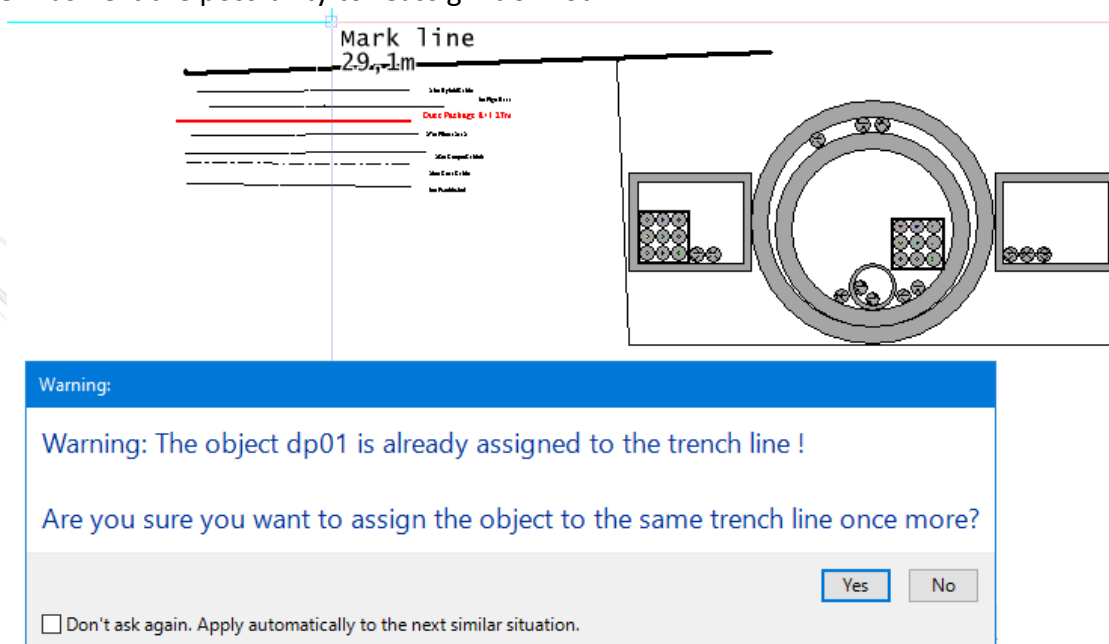
In the case the line crossed 2 trench lines, a warning should appear, that there are 2 trench-lines, and the assignment-line should be drawn more accurate, see figure below.



Special case “cables already assigned”

In case cable(s) are already assigned and the user calls "Assign cable" function again on that cable(s), then a warning is displayed informing the user that the cable is already assigned and the cable is blinking red.

User has next the possibility to reassign it or not.



A message is also displayed in the output window when cables are reassigned.

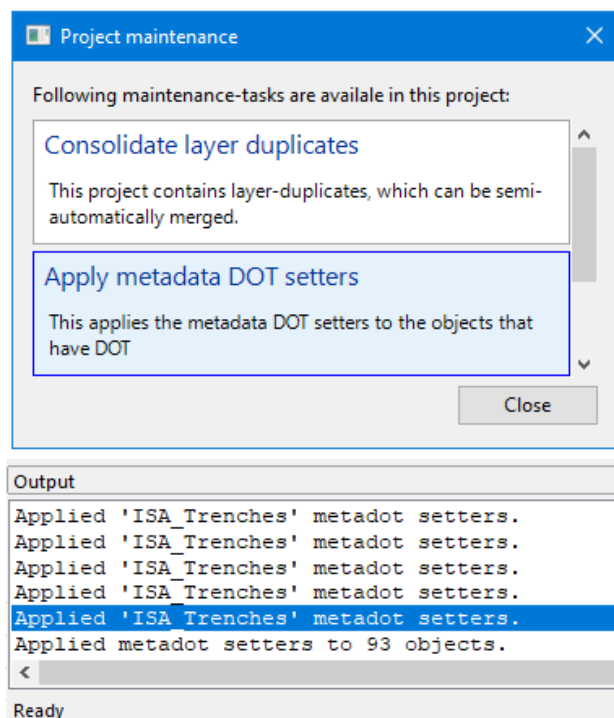
Synchronization between Meta-Data and AND attributes of Trenches

Improved synchronisation between Trench-line metadata and AND attributes (ANDEV-3656)

Synchronisation must be defined in the metadata and takes place in the direction DOT attributes → object attributes.

Triggers for synchronization:

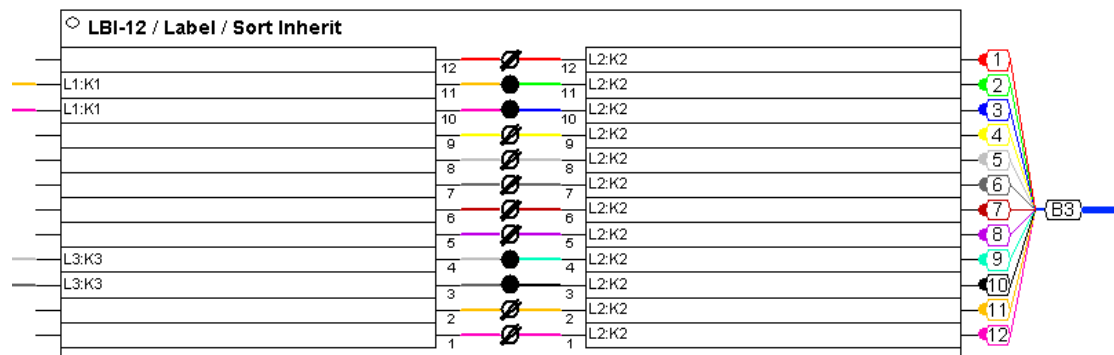
- when the user creates an object
- after the user modifies an object using the Object Properties dialog
- using Extras → Project Maintenance → Apply metadata DOT setters; this applies the metadata DOT setters to all objects in the project that have a DOT with metadata DOT setters (defined in metadata).



Splice Report Improvements

Support cut bundles and improve tray ordering in splice report (ANDEV-3699)

A splice report option has been added for including cut bundles in the splice report. Also, the splice report side ordering (SideOrder) and row sorting (SideSort) of symbols with tray/cassette flag has been improved.



When the library object's tray flag was set in logical splice and label objects were already included in the location hierarchy which detects the bundles inside.

The motivation for such objects is to display data from a remote and routed object while having the bundle state that user splice boxes and label objects do not have.

With these objects in the hierarchy, "left" and "right" sides also make sense in drawing (as opposed to general freehand bundles).

So, in addition to the ability to list cut wires, we now support fixed side order options (SideOrder) and also row sorting options (SideSort) for such objects.

Status in 4.20:

SideOrder: The user splice boxes offered the possibility to order the sides (left/right side swap) using a dynamic label (e.g., %L[i+TH]).

This label is evaluated on both pins of the splice report row and the side with the higher value is displayed on the left side of the report.

The dynamic label accessors i+, i-, w+, w-, etc. (libpinid, wirecount, .. see documentation) can be used to order splice sides.

New SideSort and improved SideOrder

Starting from AND 4.21 a new SideSort option has been added to control which side of the splices in a tray (location hierarchy context) is used for sorting within the tray.

This is useful if, as in the image above, the right side of the splice box is fully connected and only some wires are connected from the left side.

The possible values are "Inherited", "Left" and "Right".

Hierarchy

Inheritance (the default for existing and new library objects) works as follows for both SideOrder and SideSort.

The location hierarchy is checked from lowest to highest level. If a value exists at a lower level, it is used - this allows to overwrite the report default (left side) or values from a higher level, e.g. from a location rectangle with a value for the specific situation in location rectangles, frames, shelves, splice boxes/label objects.

AND / Library Object

Inside each level, first the SideOrder and SideSort values of the AND object are considered.



The screenshot shows a dialog box with two fields. The first field is labeled 'Splice side order:' and is an empty text input box. The second field is labeled 'Sorting side:' and is a dropdown menu with 'Inherit' selected and a downward arrow icon.

See in bottom section of the Location/InstNo. dialogue of the object

If there is no explicit value for

- SideOrder: "Splice side order" dynamic label specification is empty then the library object is checked for an explicit value
- SideSort: "Sorting Side" is "Inherit" then the library object is checked for an explicit value

Summary

SideSort sorting has been introduced for splice-box objects.

SideOrder and SpliceOrder are both new for label objects.

Note that MUC (Multi-Connector) objects only have this data within the AND object instance, not in the library object.

If the general splice report sorting setting is by splice label position, an explicit SideSort value overrides this and skips the position comparison and compares only the splice names. This allows the ordering and sorting to be enforced no matter how precise the drawing of the bundles is.

New dynamic label accessor i\$

In the above screenshot with the LBI-12 Label object the pin names are

- labelled with text 1,..12
- for the left side are L01, L02, .., L12
- for the right side are R01, R02, .., R12

The SideOrder in the library is %L[OTP|"M"TH] (O = pinname, TP = traverse symbol, go to partner: from bundle pin to other side, to label object pin).

This means that when evaluating the dynamic label, e.g., R03 vs. L03, the L03 side is placed on the left. The pin with the higher value goes to the left side in the report, but we use the TP traverse partner (and not the PP partner pin) to reverse this and get what we want.

For bundles connected to the label object on one side only, the value on that side is empty then the value "M" is used in that example, so that L01,... pins remain on the left and R01,... pins remain on the right, e.g. "L03" < "M" < "R03".

If for some reason the pin names cannot or should not be used for the splice side order (i.e., the name should be 01, not L01, ...) the new dynamic label accessor i\$ (e.g., %L[i\$TH]) has been introduced.

This returns "E" for even libpinid numbers (2,4,..) and "O" for odd libpinid numbers (1,3,..). This allows to put a value "M" for the unconnected side in the middle and still have the intended order.

Improvement for splice name sorting

- if the bundle partner pin is connected to an object marked as tray object in the library editor, the left side partner pin name is prefixed if present (sorting by L01, L02, ...: L03:[Cab1.B2.F3:Cab2.B2.F3], L04:[Cab1.B2.F4:Cab2.B2.F4], ..)
- Right splice side is not inverted when sorting splice report so, that unconnected left sides are sorted better: [_,Cab1.B01.F02] , [_,Cab2.B01.F01], .. (not: [_,F01.B01.Cab2] , [_,F02.B01.Cab1], ..)

Note for Library Editor users:

By using the pin names of the "inner" pins of the splice-box, you can also force "12" to be listed first (see the example LBI-12) by renaming the pins with text labels "12" to L01, R01; "11": L02, R02; ...

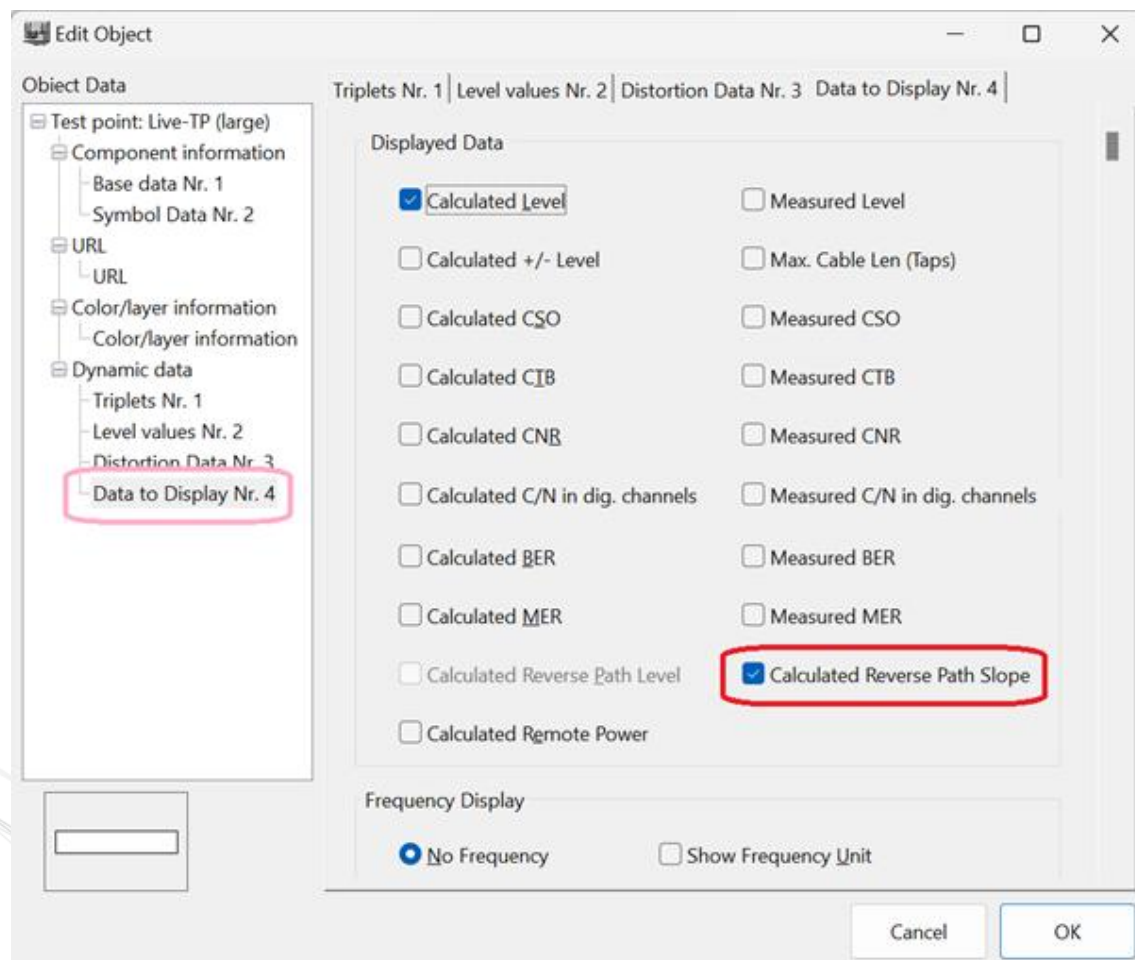
Normally these inner pins would not be displayed anywhere, but are probably only used for sorting purposes, so this should not confuse the end user.

The information above has also been added to the manual
"AND_DynamicLabels_Customer_Documentation.pdf"

Return Path Slope Improvements (ANDEV-3570)

Testpoints

A new test point setting allows the return path level to be displayed at lower frequencies as well.



You will see a line like this in the test point when this setting is on:

RL: 72.3 / 78.7

The first value shows the return path level at the lowest return path frequency, here 72.3 dBμV.

The second value shows the return path level at the highest return path frequency, which was already shown in previous versions.

Displaying both values gives information about the slope, which is the level difference between high and low frequencies.

The levels displayed here are nothing more than the flat level at the upstream receiver plus the frequency-dependent attenuation at the test point.

For example, if the level at the receiver is 70 dB μ V, the test point display above means that the attenuation to the test point is 2.3 dB at low frequency and 8.7 dB at high frequency.

The cutoff frequencies are those of the return path master channel. (Adjustments to return path amplifiers and live test points always refer to the master channel).

The new settings can be applied to all test-points of the project via the menu Extras - Program Settings - Testpoint Settings

In the following dialog, press the button Testpoint Label Settings and make sure that the checkbox Apply to all Testpoints in the bottom left corner is checked.

Be careful before pressing OK because the new settings will be copied to all test-points.

Manual Calculation

When calculating the return path level manually (CTRL+B for selected pin), both levels are now displayed.

Netcheck

The return path checks during Netcheck feature now also shows the endpoints (modem/BP) with maximum/minimum slope.

The extreme attenuations are shown at minimum and maximum frequency (before in 4.20 and lower, the extreme value was shown over all frequencies, and if that extreme frequency was in the middle, it was a bit confusing).

Example output of Netcheck:

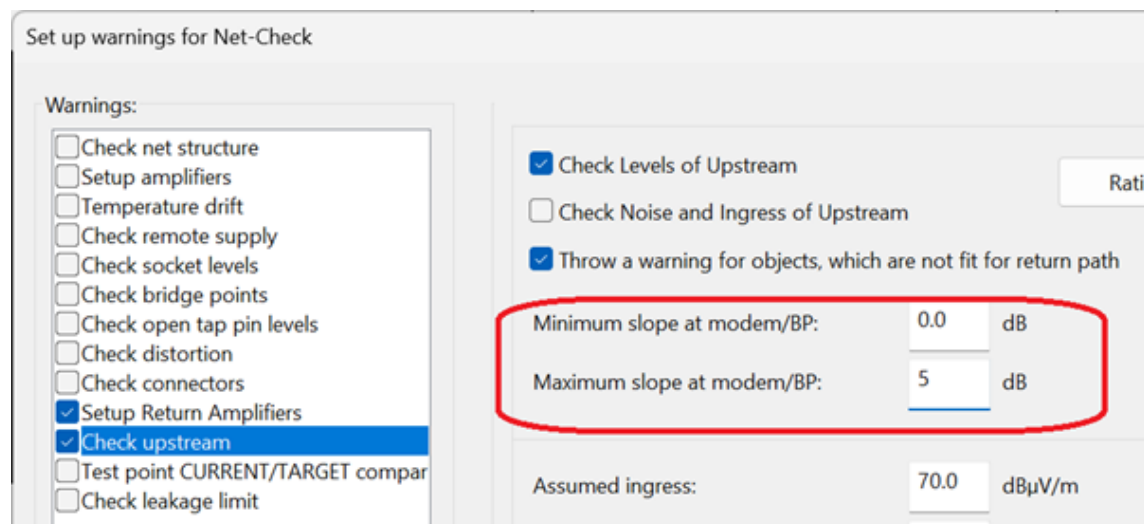
Reverse Path	Object with highest attenuation: 38.0 dB at 21 MHz; 38.0 dB at 204 MHz
Reverse Path	Object with lowest attenuation: 31.9 dB at 21 MHz; 32.0 dB at 204 MHz
Reverse Path	Return path bridge point with minimum sending slope = 0.1dB (79.9-80.0 dB μ V)
Reverse Path	Return path bridge point with maximum sending slope = 6.4dB (69.2-75.7 dB μ V)

It is possible to set limits for the return path slope at the modem or bridge point.

These limits are entered in the Netcheck settings.

If the maximum slope is greater than the minimum value, the return path slope for each modem and bridge point is calculated during Netcheck and a warning is issued if the slope is out of limits.

If the maximum gradient in the setting is equal to or less than the minimum then the Netcheck feature doesn't check the slope.

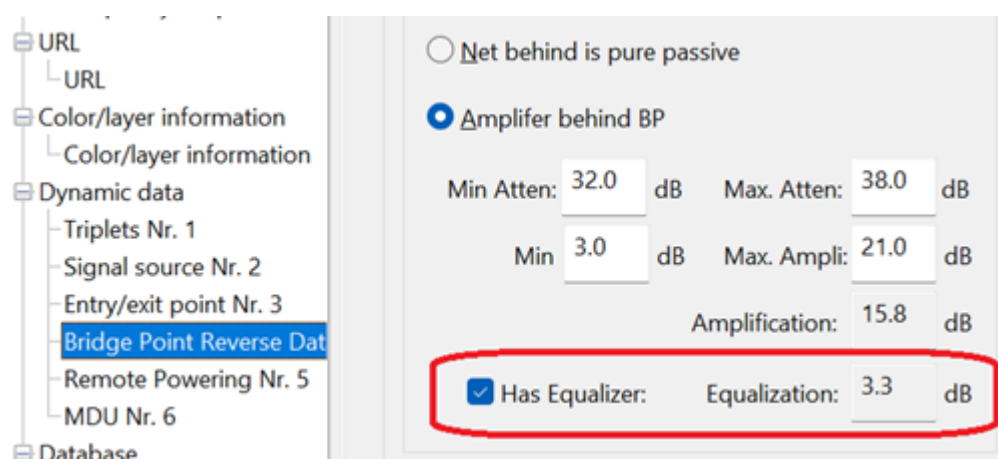


Equalizer behind bridge point

For bridge points (EEPs) with a reverse amplifier behind them, there is from AND 4.21 starting an option called "Has Equalizer". If it is turned on, the equalization behind the bridge point is also adjusted when adjusting the reverse amplifier (Alt+V).

This equalization is just the value needed to compensate the slope to the previous amplifier. It is assumed that the turning point of this equalizer is the top frequency of the master channel.

Only positive equalization is adjusted.



The screenshot above shows part of the edit dialog for a bridge point.

"Has Equalizer" is on and AND has set 3.3 dB because the slope to the previous amplifier in this project is 3.3 dB.

The "Has Equalizer" option is off by default. It can be switched on for all bridge points of the project via the menu Extras - Program Settings - Exitpoints Defaults..., if the checkbox "Apply to existing bridgepoints too" is checked.

Before pressing OK, note that the dialog copies all displayed data to the bridge points, not only the "Has Equalizer" flag.

A more explicit way to automatically set "Has Equalizer" is through the library defaults. The library defaults of a bridge point are copied to the AND bridge point when it is created and also when a library object is replaced by another (shortcut G).

Sub-Ducts via Sheet Connector

When connecting a Microduct (-sub-duct) with a sheet then all important use cases are possible. This information is in addition to the NCW trench connectors and conduit shafts.

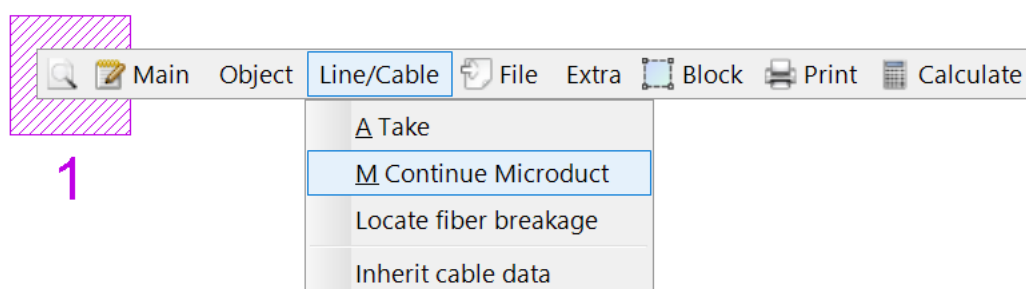
Valid and supported use-cases at sheet connector in regards to Multiduct-Packages and Sub-Ducts	
Object connected to pin of sheet symbol	Object connected to related sheet-connector inside sheet
Microduct Package	Microduct Package
Microduct sub-duct (leg)	Microduct sub-duct (leg)
Microduct sub-duct (leg)	Cable which is deployed via sub-duct (leg)
Cable	Cable

The network-check feature supports all these as valid.

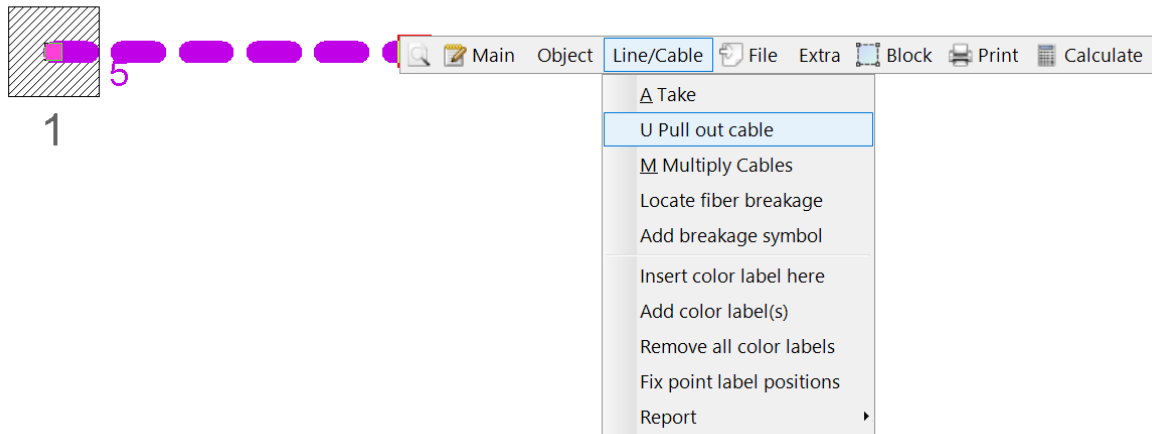
How to

Prerequisite: A microduct sub-duct (sub-pipe / leg) is connected to the outside pin of the worksheet.

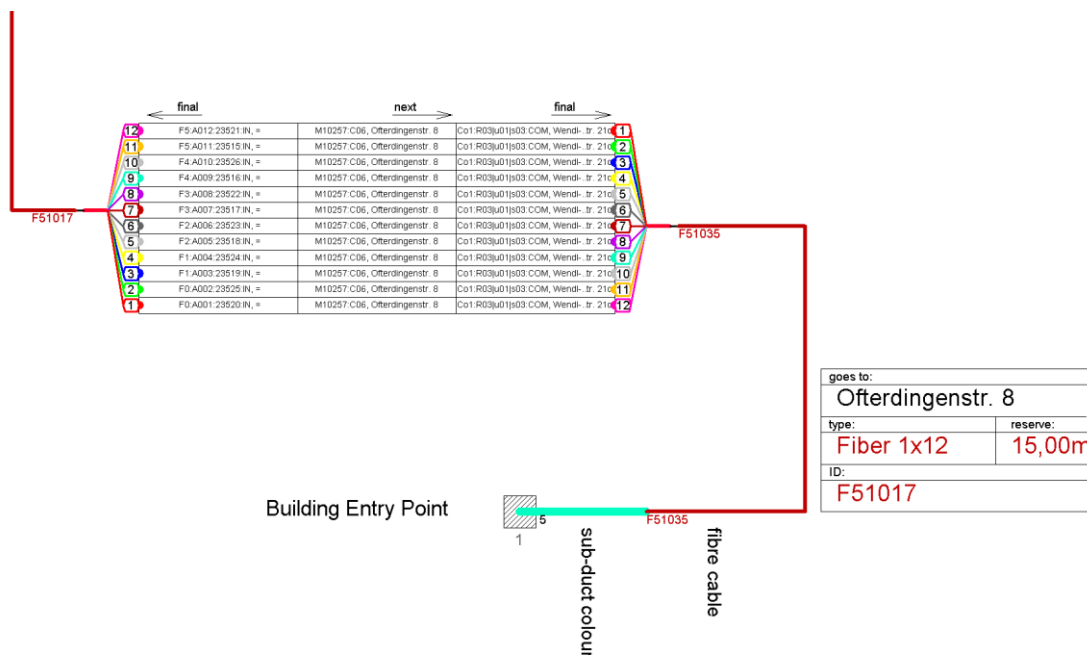
Within the worksheet, open the context menu with the right mouse button on the sheet connector and either 'Take' to pull out the cable directly or 'Continue Microduct' to pull out the sub-duct. Alternatively use the shortcuts 'A' or 'M'.



The cable can then be pulled out with key 'U' or by using 'Pull out cable' in the context menu of the end point of the sub-duct.



The example below shows the microduct pipe (turquoise sub-duct) connected to the outside of this sheet with the fibre cable continuing.



Updates of the leg outside will also be reflected in a change inside the sheet.

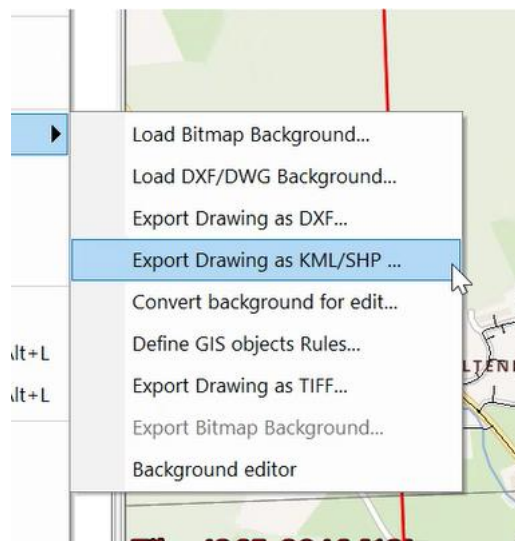
So, a comprehensive documentation can therefore be produced using all variants within the worksheets.

Export to SHP (ANDEV-3670)

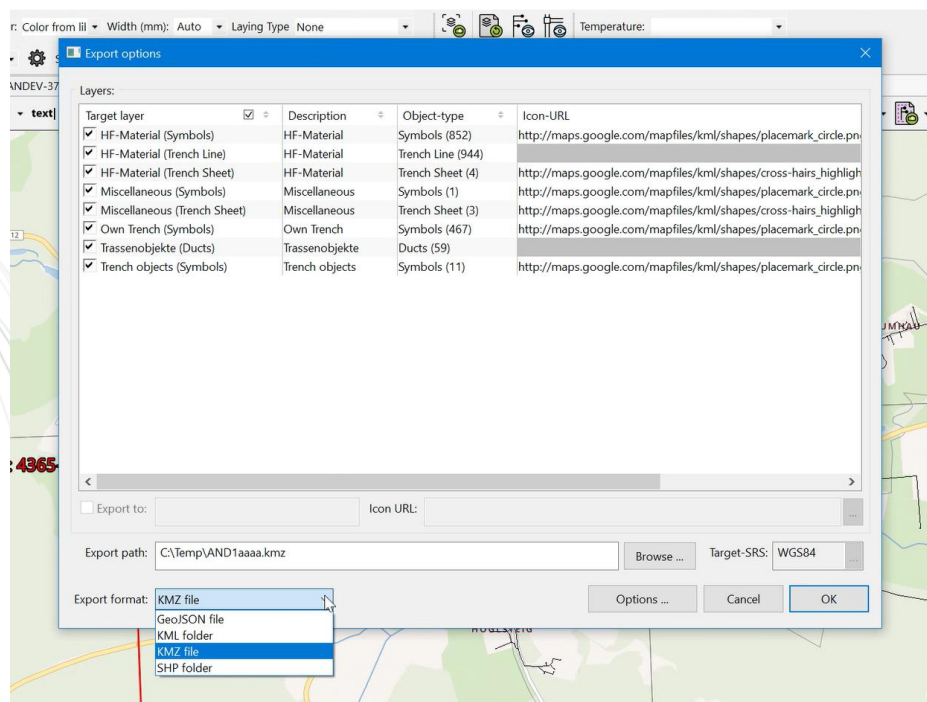
Starting with Version 4.21 AND supports now also exports in Shape-Format (SHP). The already existing KML-export feature has been extended by the SHP-export.

The KML/SHP export feature is available for GIS sheets only.

Select **FILE -> Background -> "Export drawing as KML/SHP" ...**



to open the following dialog:



New options are available in the lower left area of the dialog,

- "Export format" where the destination format can be selected and on the lower right
- "Target-SRS where the target coordinate system can be selected

The previous KML export workflow has not changed.

The items in the layer list represent the different object types used per layer (cables, duct packages, trenches, conduits, symbols, sheets, and trench points).

The entries in the first column specify the names of the layers to which the objects are exported. To edit an item, either click on it, or open the **Export to** edit dialog, which can also be used to rename multiple selected entries at once.

The second and third columns are read-only and show information about the source layer and the object type.

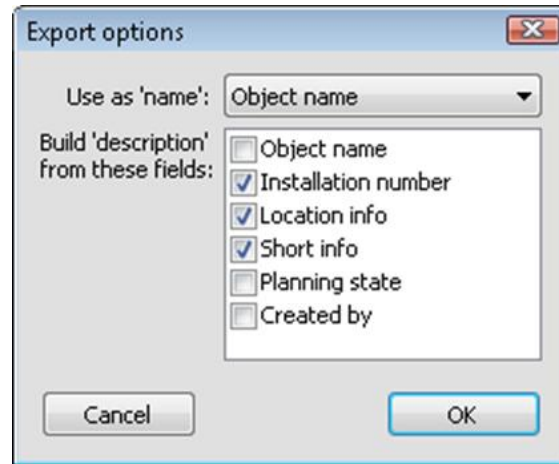
The fourth column can contain a link to a bitmap which can be used by other software (e.g. Google Earth) to display the respective item as a point object. Such objects can only be defined for symbols, sheets, and trench points. A double click on an entry in the fourth column (or pressing the ... button next to the **Icon URL** field) opens a dialog in which users can select an URL from a set of pre-defined symbols.

The **Export path** is the target path and can be edited manually or selected using the **Browse** button. The **Export format** dropdown menu contains these two options:

- **KMZ file:** The target is a single file with the file extension ".kmz". This file is in fact a compressed directory structure containing KML files for each exported layer, and the two default files *kml* and *styles.kml* defining the layer structure and the element styles
- **KML folder:** The target is a folder to which the uncompressed KML files are exported (including the *kml* and *styles.kml* files.)
- **SHP folder:** The target is a folder where the set of SHP files are exported to.

IMPORTANT NOTE: If "KML folder" is selected, the folder name entered in the *Export to:* field cannot be the name of an existing directory, otherwise the export will fail!

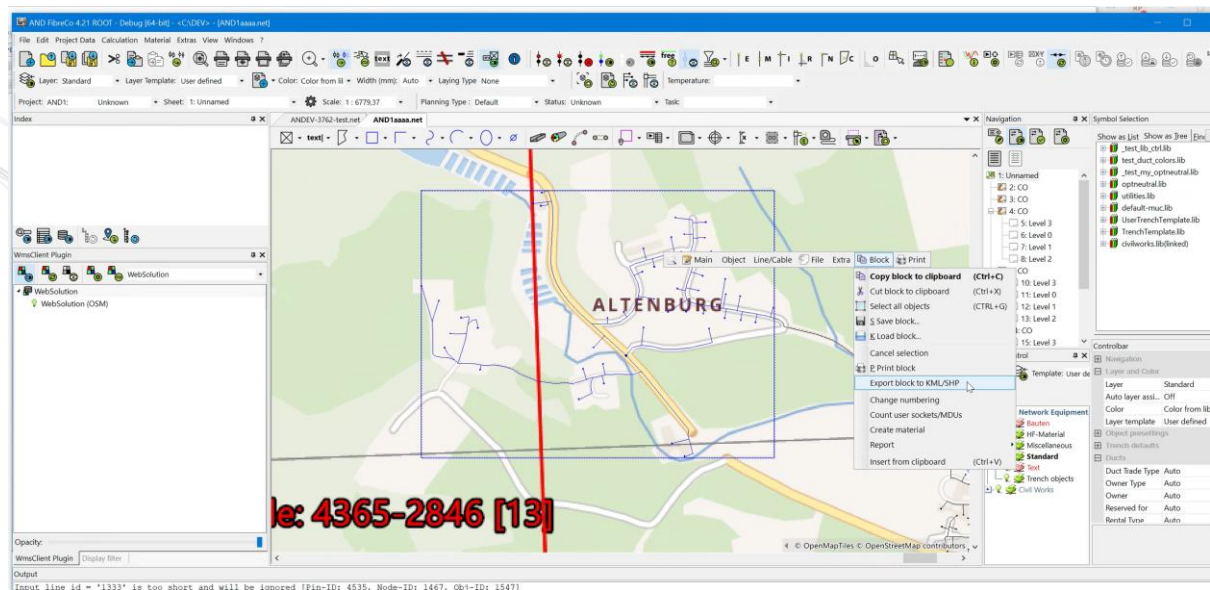
The **Options...** button opens this dialog:



Users can choose which of the listed data should be used as <name> and <description> for entries written to KML. For the <name> (which is shown as a text label in Google Earth, for instance), only one field can be selected. The <description> can include multiple fields (which are shown in Google Earth when a point is clicked on, for instance).

Block Export

Also starting with AND version 4.21 the functionality to export only objects selected in a block got added. To start an export, select an area with a block and right click on "Export block to KML/SHP".



GIS Import (ANDEV-3589)

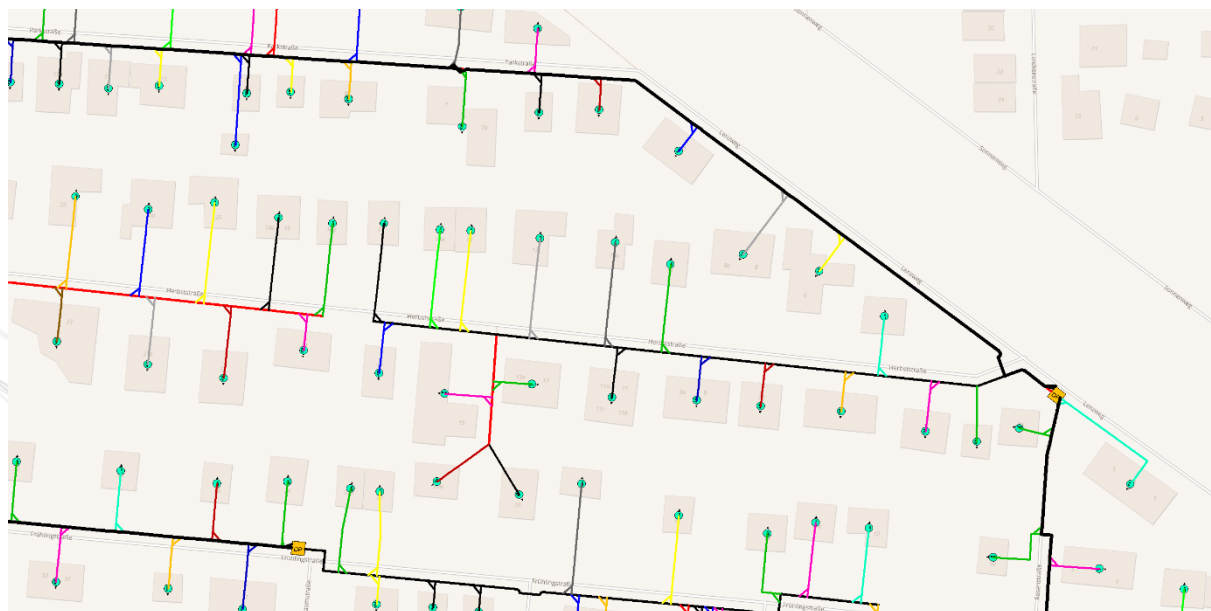
We have created a new way of importing GIS data into the AND to create usable networks based on AND objects. Initially, the infrastructure consisting of sites, routes and ducts as well as multi-duct packages is supported.

The concept is to accept a defined format (GeoJSON) which is documented and for which there is the possibility of validation.

The user can conveniently define detailed assignments/mappings to AND library objects based on object types and their attributes. Attributes of the objects to be imported are transferred into the AND objects.

Prior to this, various import formats are converted into this GeoJSON using so-called "Wrappers". These can be customized by the customer or created by AND Solution. We supply a wrapper for the import from WebSolution.

We intend to replace already existing atesio and shape import functions by this new feature and remove them in one of the next versions.



Example of an import which generated buildings with their addresses, central office and cabinets and the distribution via microduct-packages

For detailed documentation see the document "AND_GisImport.pdf".

A HTML documentation regarding the GeoJSON import format and JSON schema files for validation can be found in your installation folders.