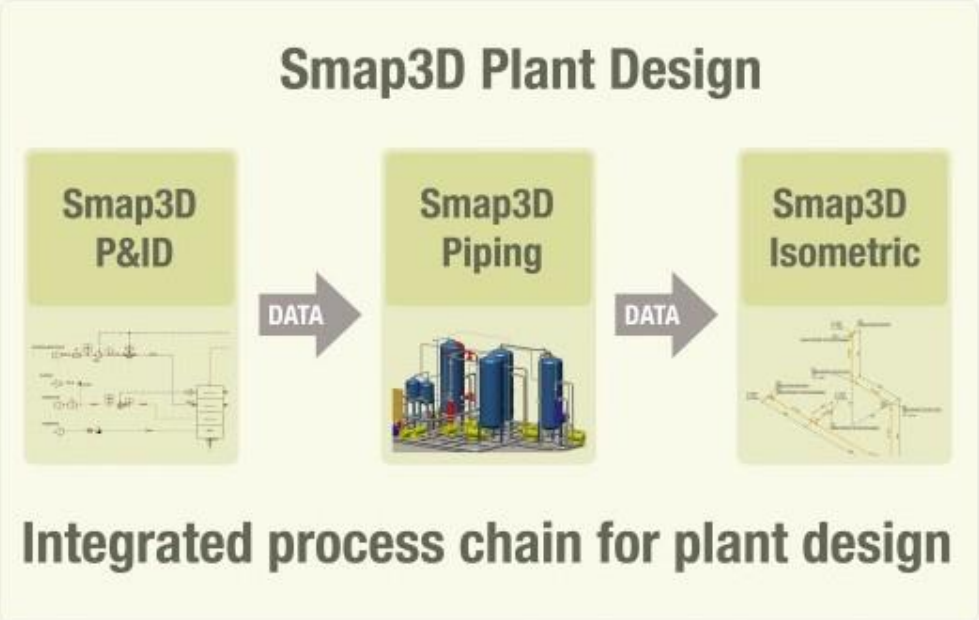
# This is new in Smap3D Plant Design 2017

## Welcome to Plant Design 2017 with SOLIDWORKS

This year as well, the new version of Smap3D Plant Design has gained many enhancements and improvements in functionality and applications.

### The individual chapters in this document are:


- General news
- New in P&ID
- New in the Component Wizard
- New in the Pipe Specification Editor
- New in 3D Piping
- New in the P&ID To-Do List
- New in Isometric

# General improvements

## Supported CAD versions

### SOLIDWORKS 2016 and SOLIDWORKS 2017

The release version of Smap3D Plant Design 2017 supports:

- **SOLIDWORKS 2016 with service pack 5.0 or later**
- **SOLIDWORKS 2017 with service pack 4.0 or later**

After its release, support for the next SOLIDWORKS 2018 version will be provided with a service pack for Plant Design 2017.

## New product - Industrial pipe specifications

A few weeks after the release of Plant Design 2017, we will launch the product **Industrial Pipe Specifications**.

With this supplemental product for Plant Design 2017, customers receive numerous pre-defined pipe specification files with associated 3D standard parts in accordance with current industry standards.

The following will be included with the delivery:

- Pipe specifications in accordance with **ASME B31.3** (various combinations)
    - Various materials (Carbon steel & stainless steel)
    - Various pressure levels (150, 300, 600,..)
    - Various fitting types (butt weld, socket weld, threaded)
    - Various media/states (water / gas …)
- **Water pipe in accordance with AWWA**
- **IS0 8434 Metallic tube connections for fluid power and general use**
- **Cooper press fittings** (similar to Mapress / Sanpress)

and more.

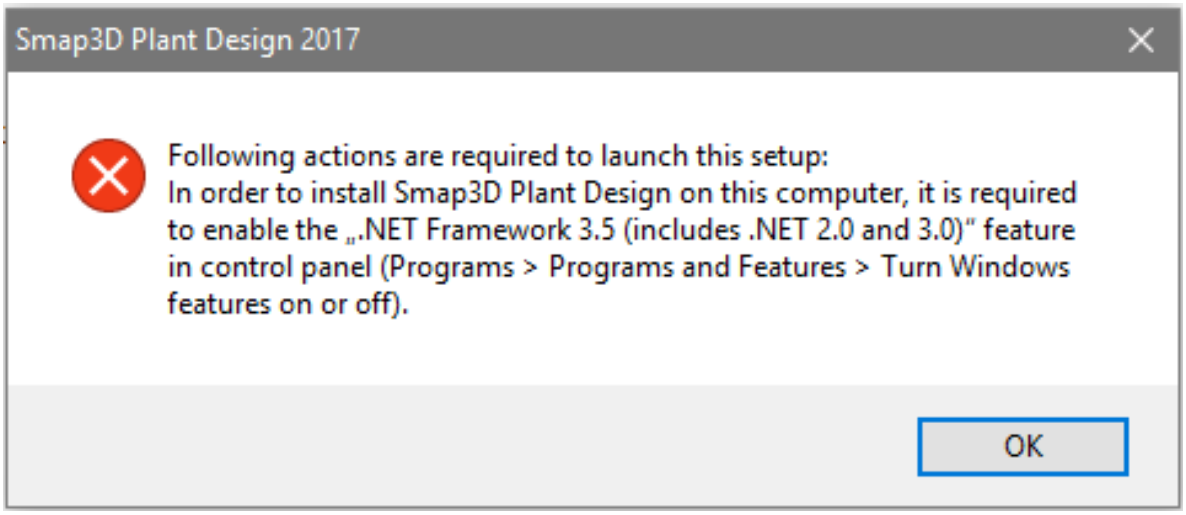
## Further general news

### Setup: The .NET Framework 3.5 at Windows 10 computers may need to be manually activated/installed before Plant Design 2017 installation.

Since Microsoft made changes to the old **.NET Framework 3.5** components in the **Windows 10 operating system** (Creator Update in April 2017), it is no longer possible to install or activate these components separately on a Windows 10 computer via external installation routines.

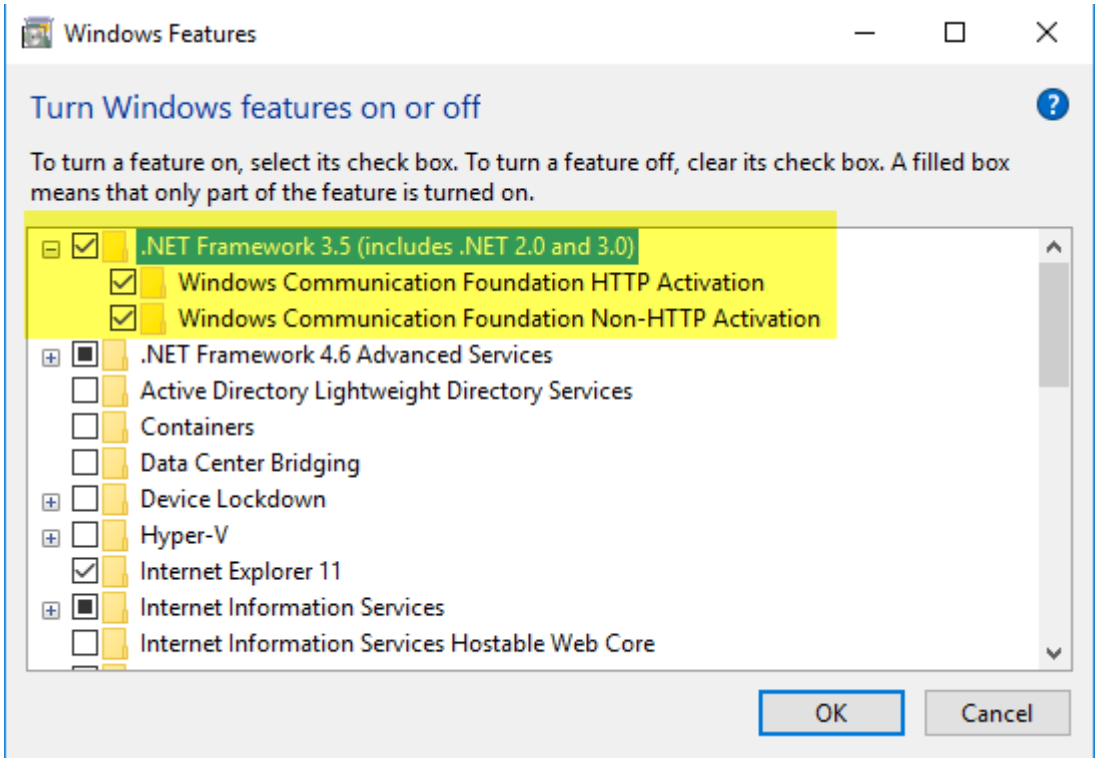
However, as Smap3D Plant Design requires these **.NET Framework 3.5** components for the Smap3D PartFinder (PDM 9.6), the Plant Design installation routine now has a built-in test to determine whether these components are already installed/activated on the active computer.

If the following message occurs during Plant Design 2017 installation on a **Windows 10** computer, the installation cannot be carried out until a user/administrator has manually activated or installed the required **.NET Framework 3.**5 components.



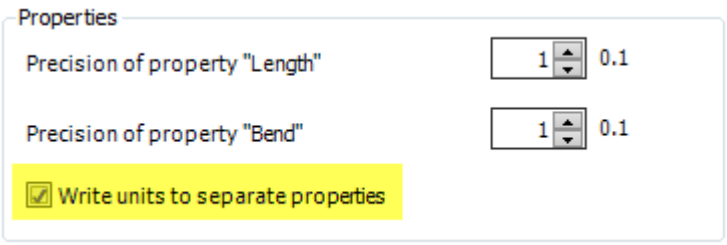

On a Windows 10 computer, **.NET Framework 3.5** is activated manually via:
***Control panel > Programs > Programs and Features > Turn Windows features on or off***

In the dialog shown, a checkmark must be placed for all three yellow-marked options. Complete the dialog with **OK** to fully install or activate .NET 3.5 for Windows 10.

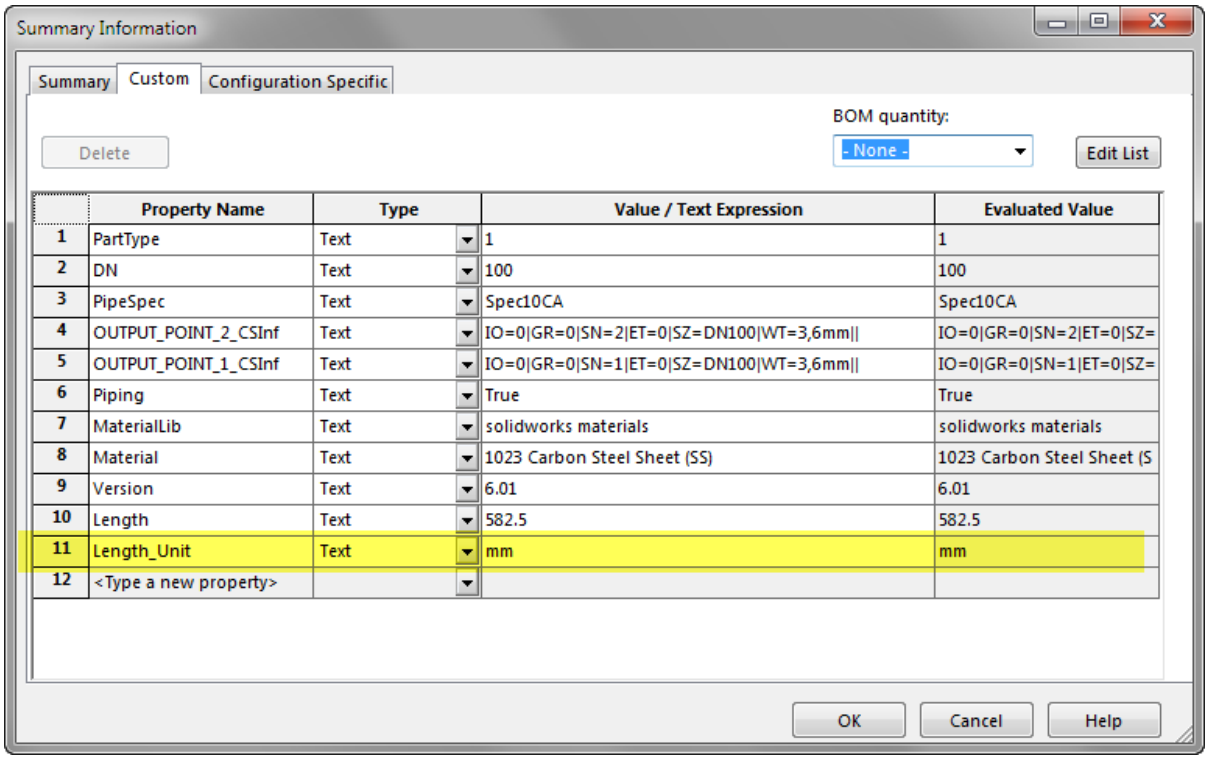


## Plant Design Administrator: New option for Length and Angle properties

The **Write units in separate properties** option (in the Piping common tab in Plant Design Administrator) can control whether Smap3D Piping writes the associated unit for the user-defined properties **Length** and **Angle** in separate properties when generating the pipes.




This option is designed for all users/customers who are analyzing and processing the file properties **Length** and **Angle** in other systems (e.g. PDM or ERP) and therefore require only the pure numerical value as the value of the property.
Previously the unit was always written as a component of value (e.g. 582.5 mm).



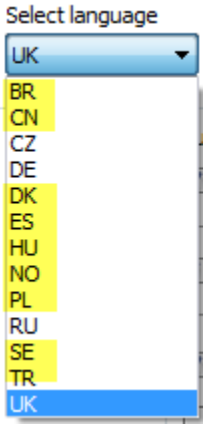

| | Property Name | Type | Value / Text Expression | Evaluated Value |
|---|---|---|---|---|
| 1 | PartType | Text | 1 | 1 |
| 2 | DN | Text | 100 | 100 |
| 3 | PipeSpec | Text | Spec10CA | Spec10CA |
| 4 | OUTPUT_POINT_2_CSInf | Text | IO=0\|GR=0\|SN=2\|ET=0\|SZ=DN100\|WT=3,6mm\|\| | IO=0\|GR=0\|SN=2\|ET=0\|SZ= |
| 5 | OUTPUT_POINT_1_CSInf | Text | IO=0\|GR=0\|SN=1\|ET=0\|SZ=DN100\|WT=3,6mm\|\| | IO=0\|GR=0\|SN=1\|ET=0\|SZ= |
| 6 | Piping | Text | True | True |
| 7 | MaterialLib | Text | solidworks materials | solidworks materials |
| 8 | Material | Text | 1023 Carbon Steel Sheet (SS) | 1023 Carbon Steel Sheet (S |
| 9 | Version | Text | 6.01 | 6.01 |
| 10 | Length | Text | 582.5 | 582.5 |
| 11 | Length_Unit | Text | mm | mm |
| 12 | <Type a new property> | | | |

# New in Smap3D P&ID

## New languages for the user interface

The available languages for the user interface were extensively expanded in P&ID 2017.
There are now a total of 13 languages available.

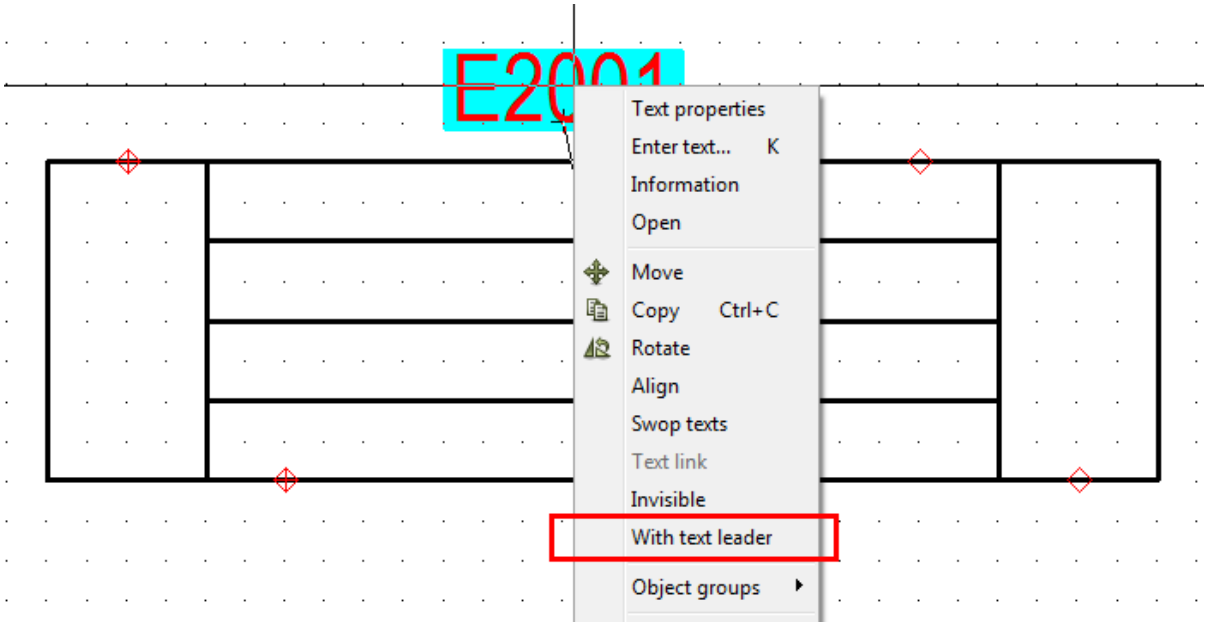
In addition to the languages available through many versions UK - English, DE - German, CZ - Czech, and RU - Russian,
the following languages have been added:




- **BR** - Brazilian/Portuguese
- **CN** - Chinese
- **DK** - Danish
- **ES** - Spanish
- **HU** - Hungarian
- **NO** - Norwegian
- **PL** - Polish
- **SE** - Swedish
- **TR** - Turkish

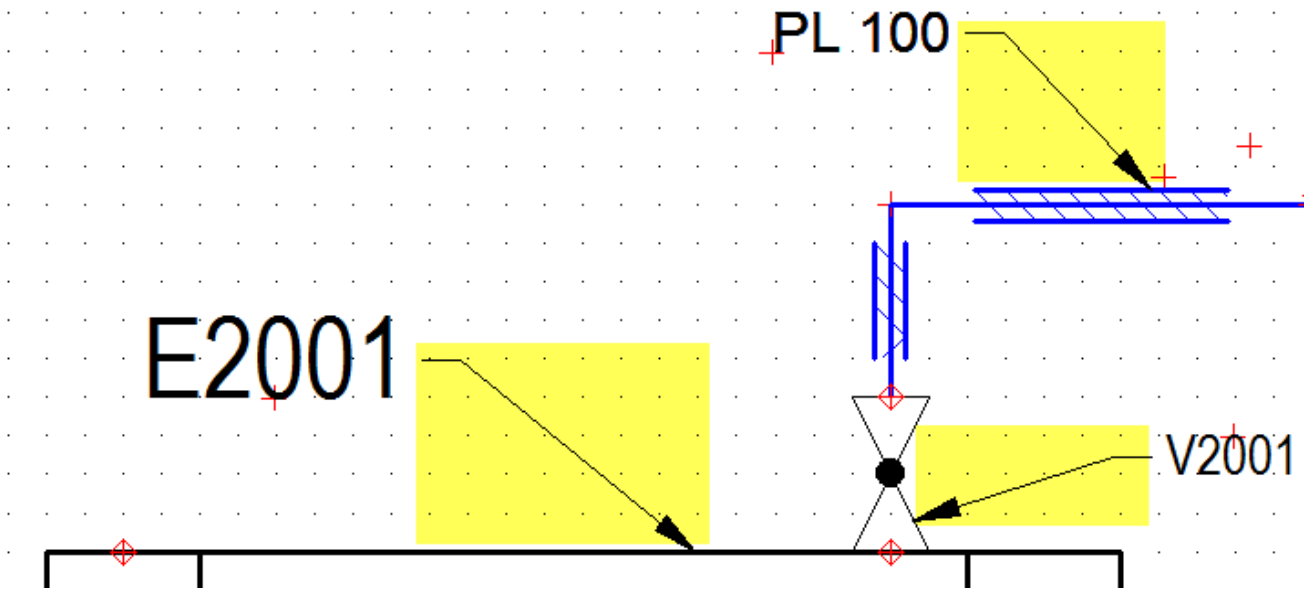
## New Text leader function available for all text types

The new **With Text leader** function is available in the context menu for text.
Depending on the status, a leader line can be switched on (**With text leader**) or off (**Without text leader**) for the selected text.




The arrow tip is positioned individually by the user.
This position can be subsequently changed in the **Text (T)** editing mode (by clicking on the arrow tip).

These context menu functions are available for **all visible text types** in Smap3D P&ID.



# Enhancements to the line data fields

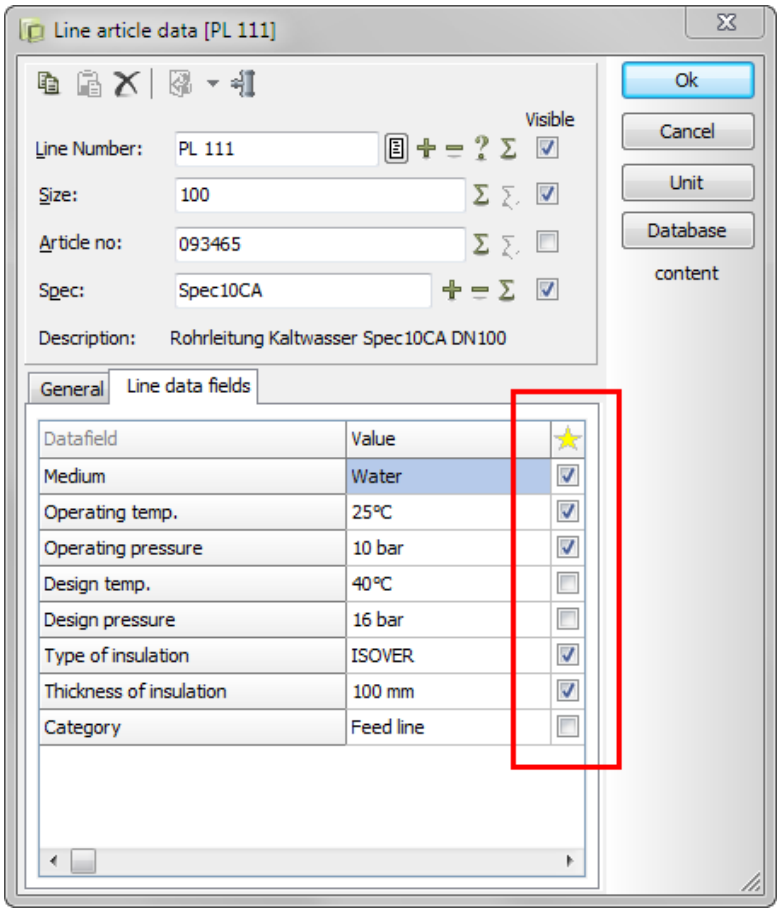
**Line data fields** were added to Smap3D P&ID with version 12.
Since that time, users can define any user defined attributes (process properties) on lines.

This range of functions was now significant enhanced.

## Visibly display line data fields on a drawing

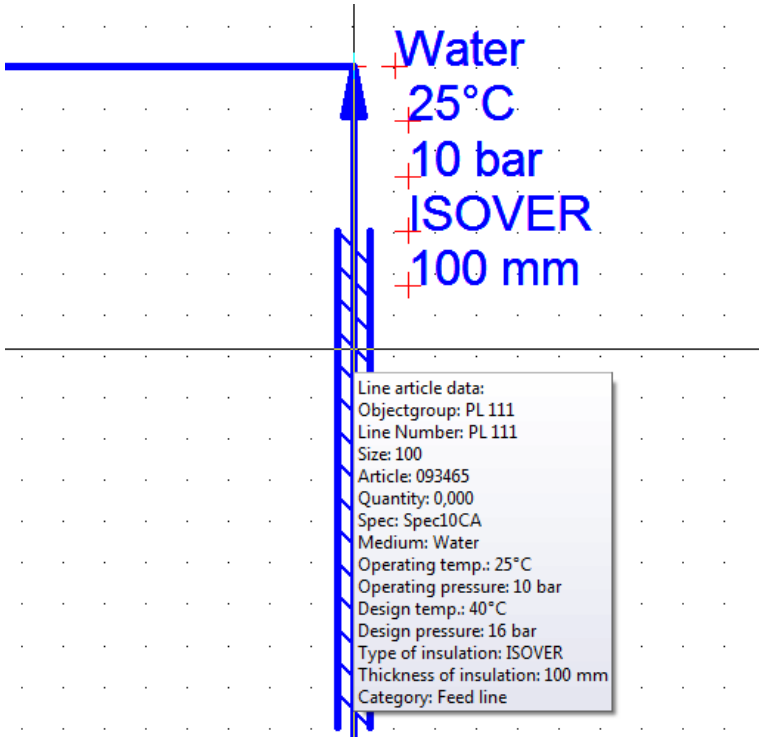
In response to many customer requests, individual line data fields can now be visibly displayed in a diagram.

In the **Line article data** dialog (**Line data fields** tab), there is a new column for individual control of the visibility.
By placing a checkmark in the respective row, the value of the line data field for the previously selected line is switched to visible.
Afterwards, exit the Line Data dialog by clicking **OK.**



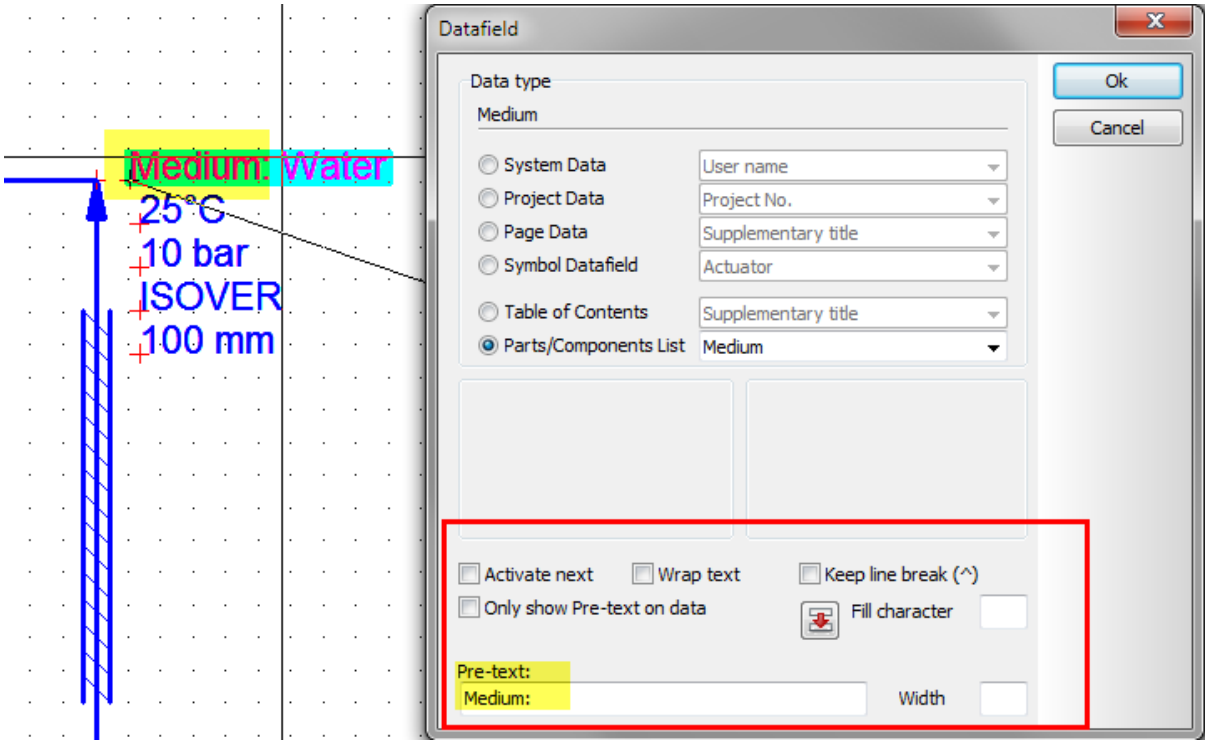

Thus, the **values of the selected line data fields** (only data fields with a value) are visible in the diagram.

The default position of the text is always at the start of the associated line.
Each text can be positioned individually on the active diagram page as needed.



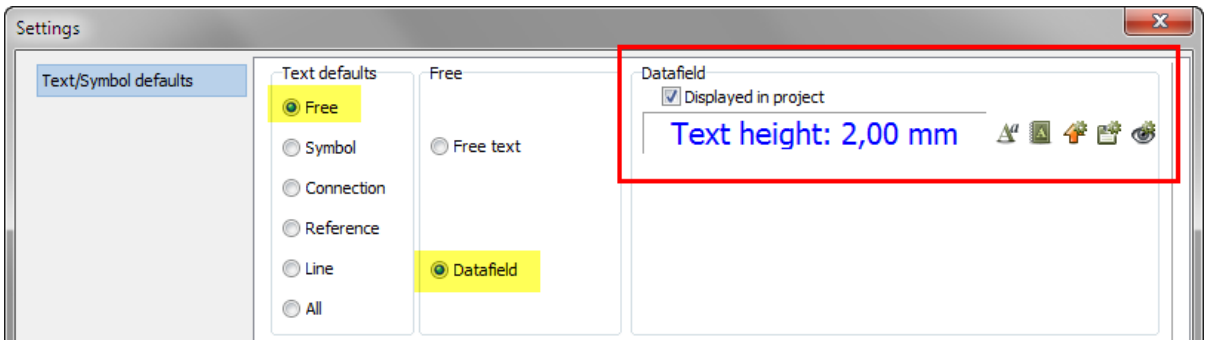

**Datafield** settings are available for each visible line data field.
This way, each displayed text can be individually adapted.

The new Leader line display is also fully supported.

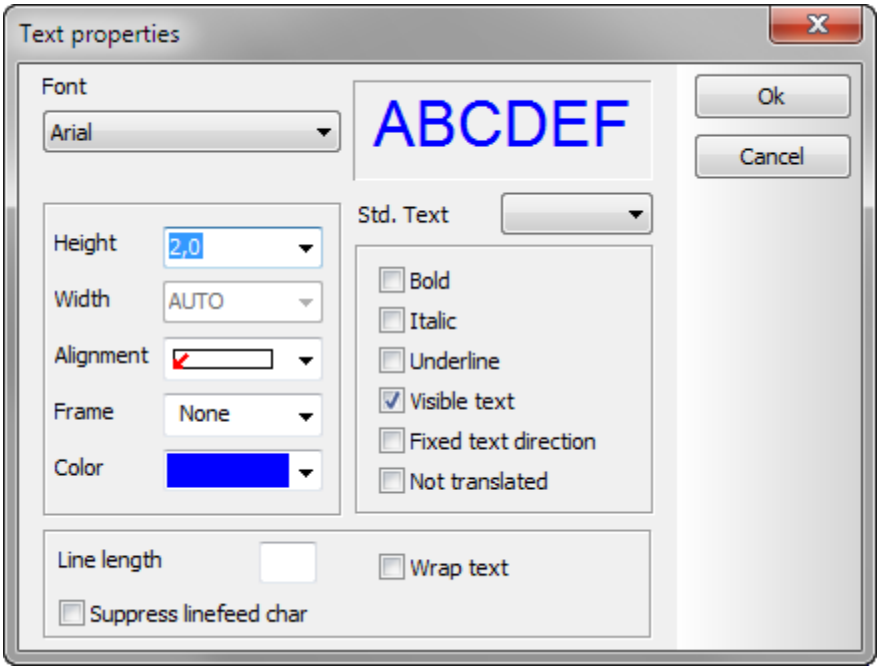


## Standard text format for visible line data fields from "Text/Symbol defaults"

The active system settings for **Text/Symbol defaults** > **Free** > **Datafield** are used as the standard setting for all visible line data fields.

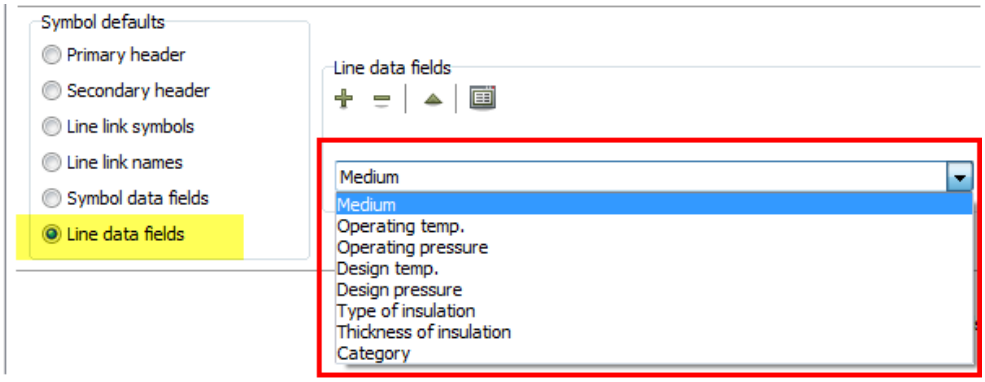



Each visible line data field can also be adapted individually in **Text properties**.



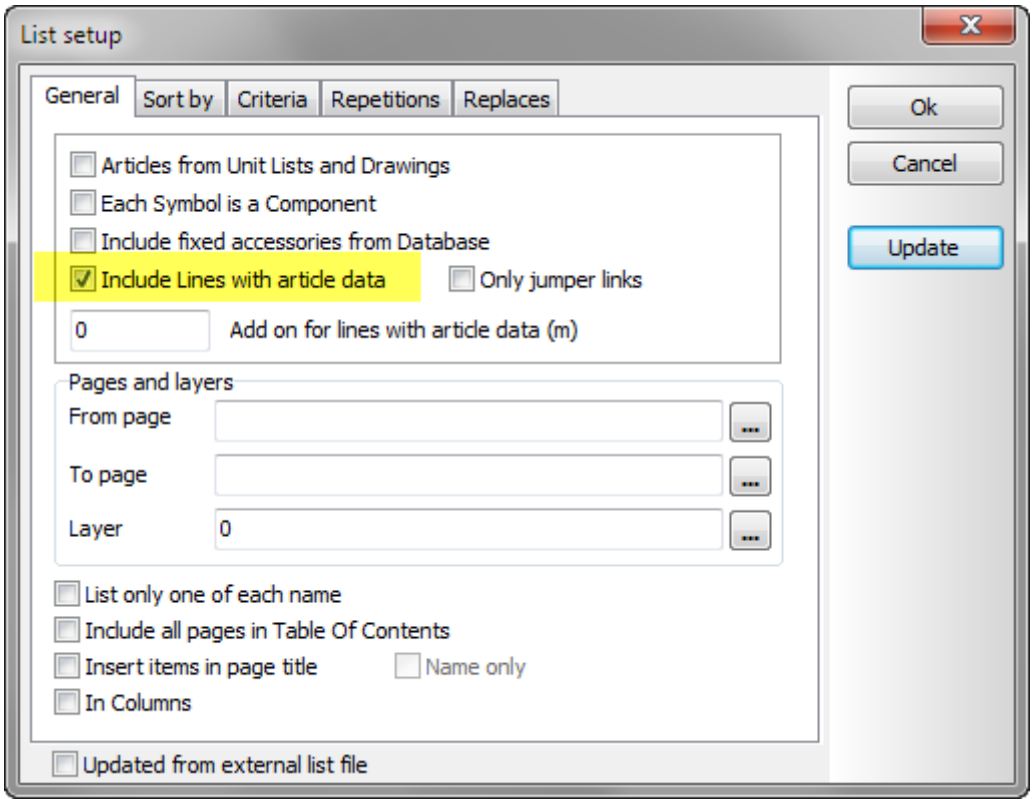
## Line data fields available for evaluation in lists

All line data fields defined in the system (**Settings** > **Text/Symbol defaults**) can be used for lists in which lines are evaluated as well.



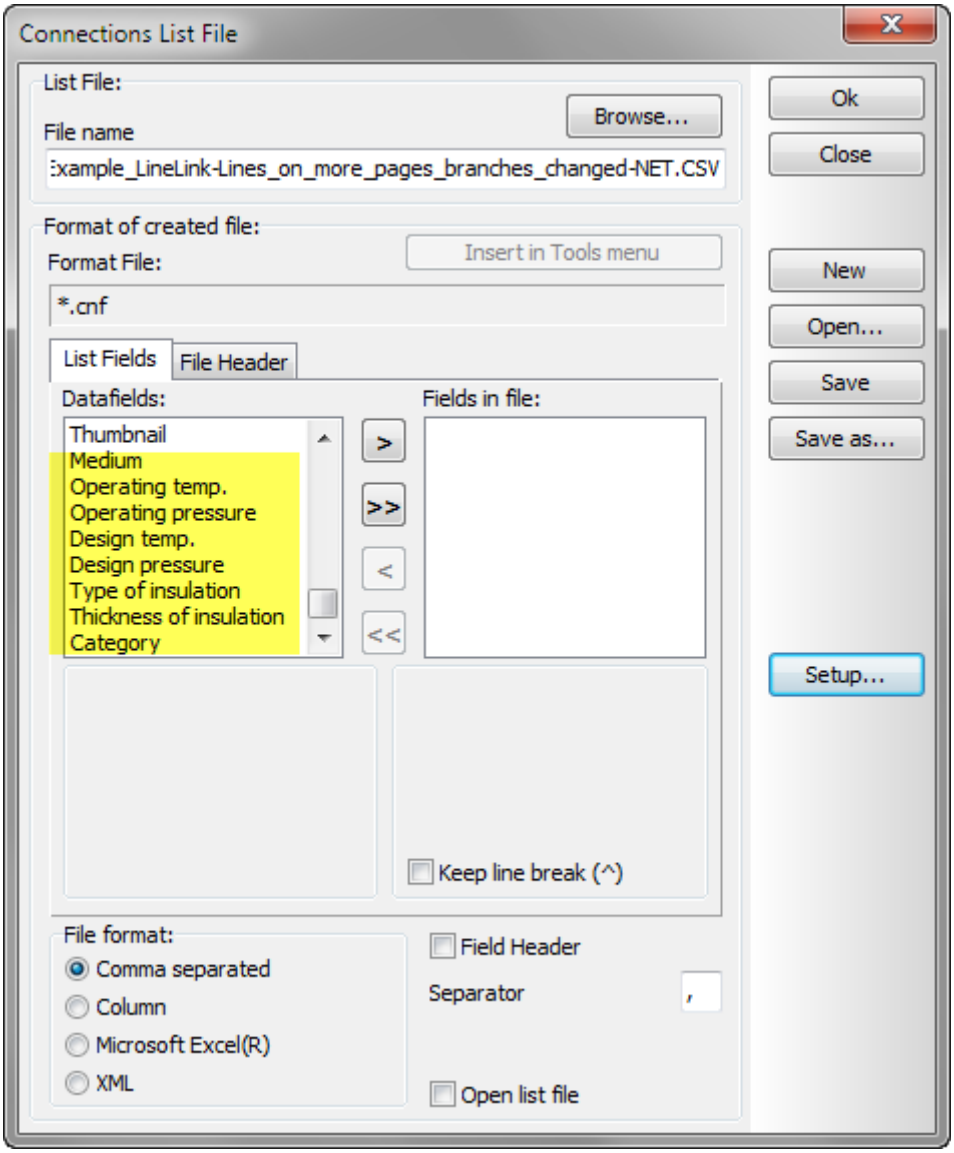

### For list pages in a PRO file

line data fields can only be evaluated/listed in **Component lists** (not Parts Lists).
In addition, the component list's **List setup** must have the **Include Lines with article data** option activated.



**For external lists in files**

line data fields can be evaluated with the functions **Components List File** and **Connections List File**. In this case as well, for a c**omponent list in file**, the **Include Lines with article data** option must be activated in the list settings.

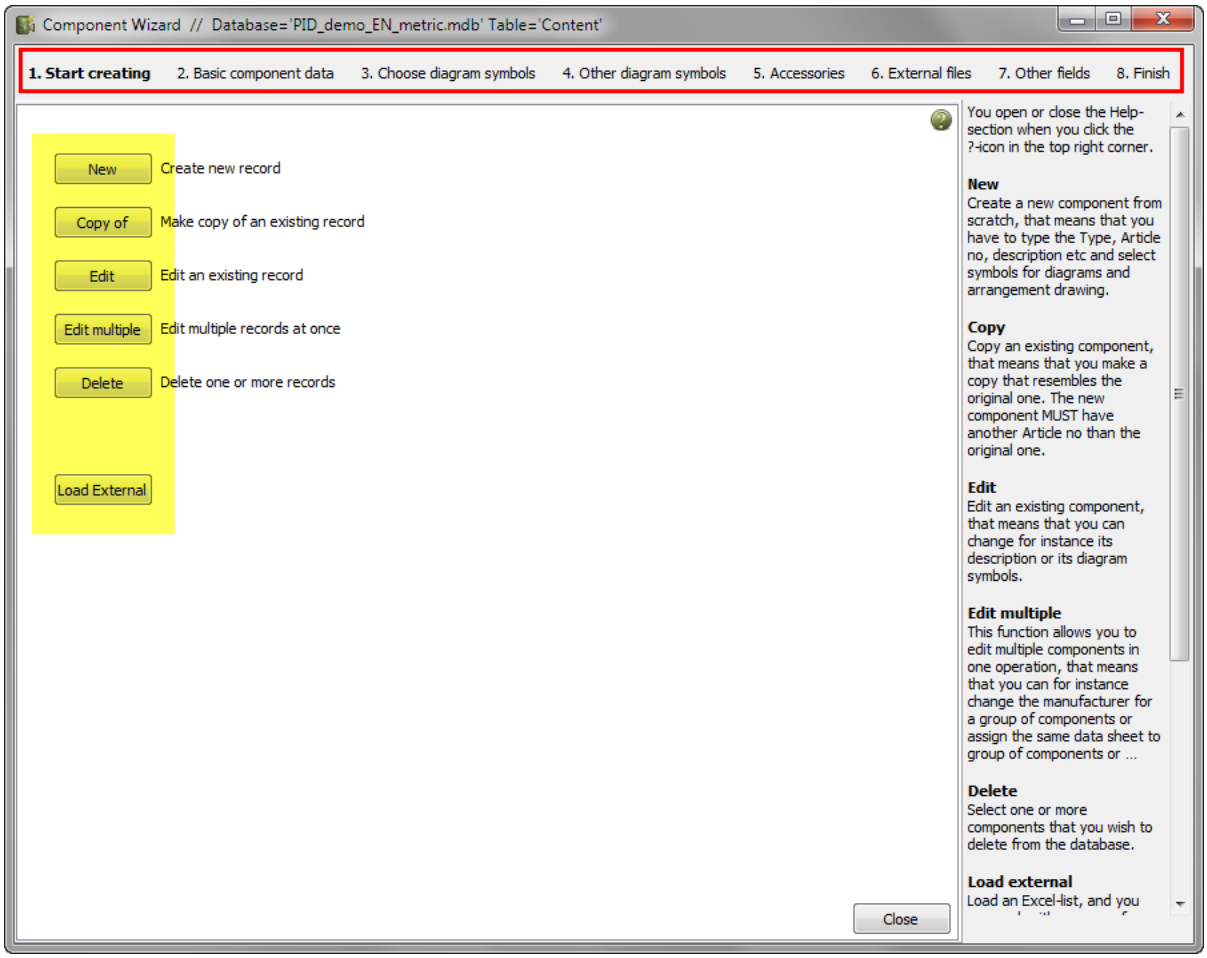


## New Component Wizard

The new **Component Wizard** enables the **creation and modification** of database entries in a clear and easy to use assistant.

In no more than 8 steps, a user can:

- Create new data records
- Copy, edit (once or repeatedly) or delete existing data records.
- Load an Excel table with external data records in order to quickly and easily edit or add the content.
  After processing in this Wizard, such an Excel table can be used for automatic import in the P&ID Database program.



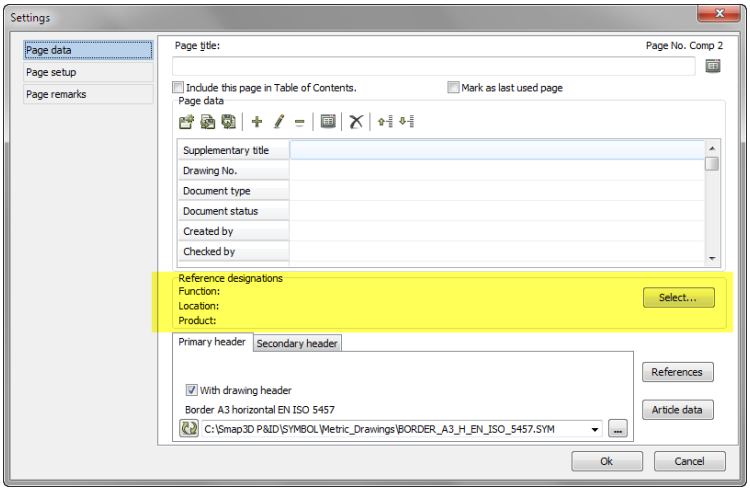
# News to reference designations

## Selection function always available/visible

As of P&ID 2017, all selection functions and options for **Reference designations** are always displayed, even if no definitions for reference designation are available in the active project.
In the previous version, such definitions had to be available in the active project before the function could become visible.

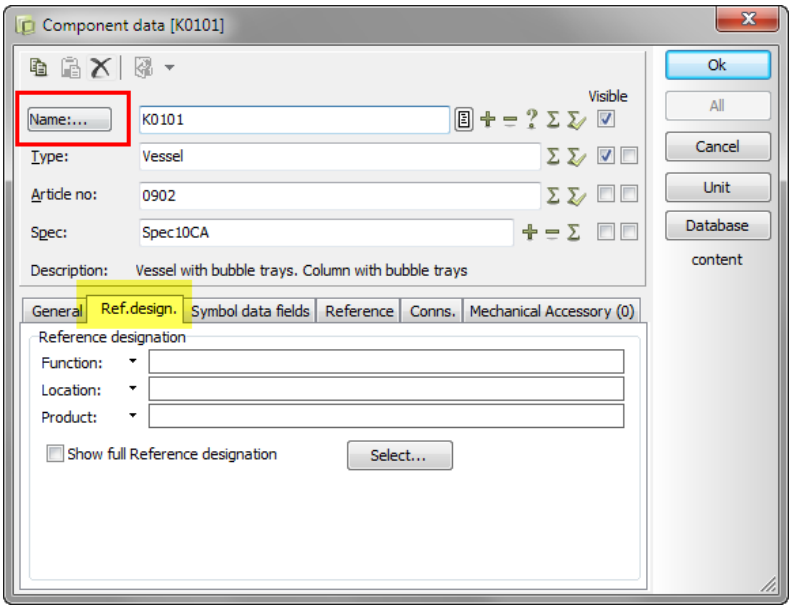
### Page data

The functions for **Reference designations** are now always visible in **Page data** dialog.

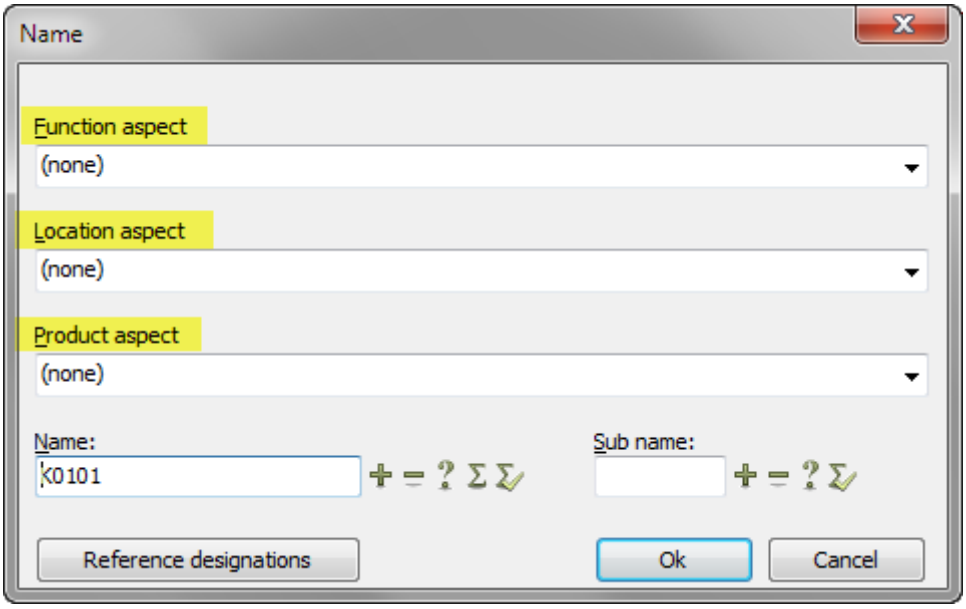



### Component data

The **Ref.design.** tab is now always available in the **Component data** dialog.

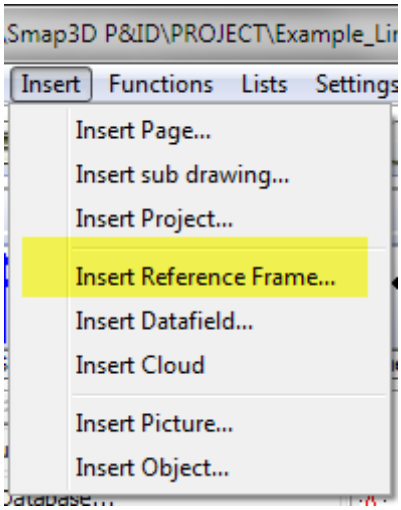


The functions for reference designations are now always available in the **Name** dialog (click on **Name:..** button).



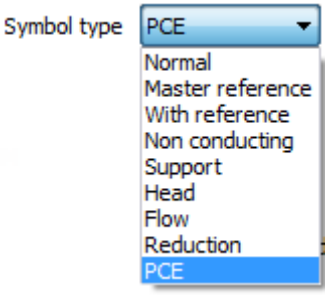

**Reference Frame**

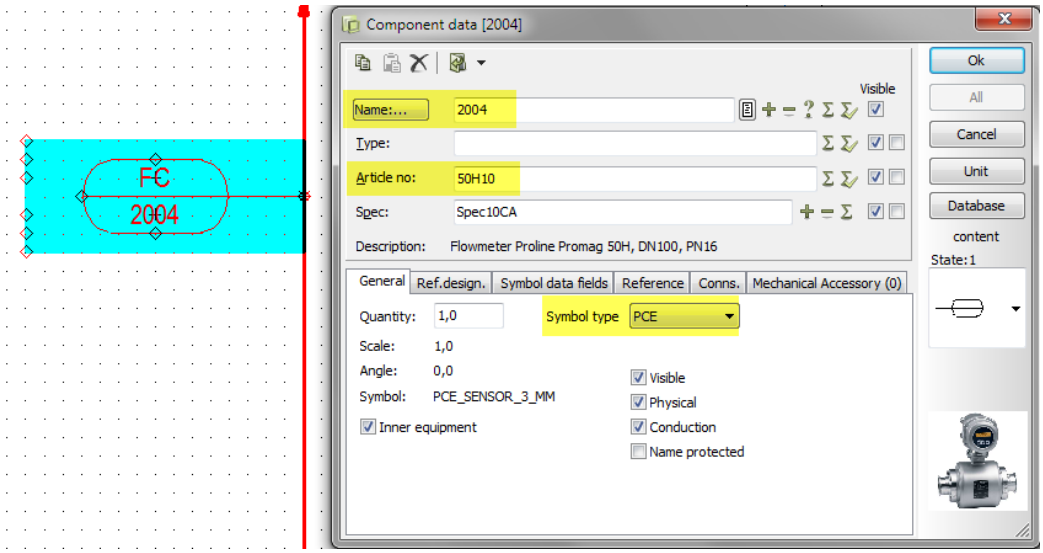
The **Insert Reference Frame** function is now always available in the **Insert** menu.



## New symbol type PCE

The new symbol type **PCE** is now available for creating symbols or using symbols in diagrams.




This new symbol type was introduced to give users the option of differentiating symbols for process control technology (PLT, for sensors and process control functions) from regular P&ID symbols.

This new symbol type was already set up for all symbols delivered with Smap3D P&ID 2017 and designated for this purpose.

- **EN 62424 - mm** (complete)
- **EN 1861 - mm** (the relevant symbols)
- **ISA 5.1 - mm** (the relevant symbols)
- **ISA 5.1 – Inch** the relevant symbols)

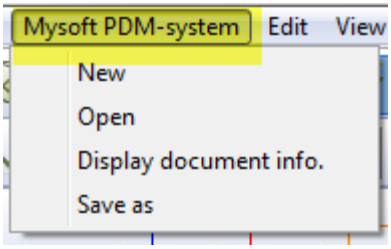
Differentiating the symbol types is primarily useful in analyzing **Component lists** (internally as a page or externally in a file) or when using the **P&ID To-Do List** application.
In both cases the symbol type (its value) can be used as a criterion for filtering and sorting.

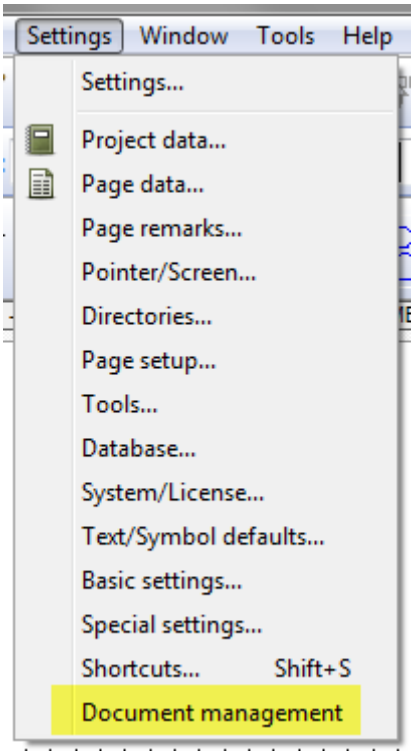
## New standard interface to PDM systems

With the development of P&ID 2017, a neutral standard interface for connecting Smap3D P&ID to arbitrary PDM systems was developed as well.

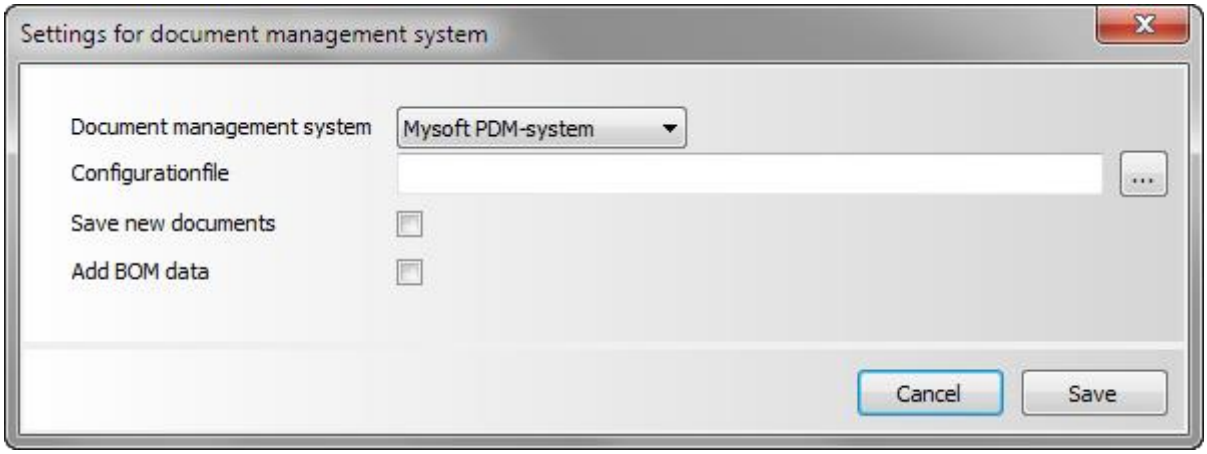
Although this is not included with a Plant Design 2017 standard installation, it can be easily installed subsequently with the required data.

When this interface is installed, an **additional menu** and **additional settings** are displayed in the P&ID user interface.



In the dialog **Settings for document management system**, the neutral Smap3D interface is linked and configured with the interface of any PDM system.




The basic communication of this neutral interface is based on the principle of an **XML handshake communication**.

This means that when P&ID functions like **New**, **Open**, **Save as**, etc. are accessed, our interface produces various neutral XML files in the background with structured information at a neutral storage location.
These can be read and processed by a **corresponding communication interface from a PDM system**.

Then, depending on the definition and configuration of the **communication interface** there, the PDM system sends corresponding replies and information back to the neutral storage location in XML format.
These are recognized and used by the Smap3D P&ID interface.

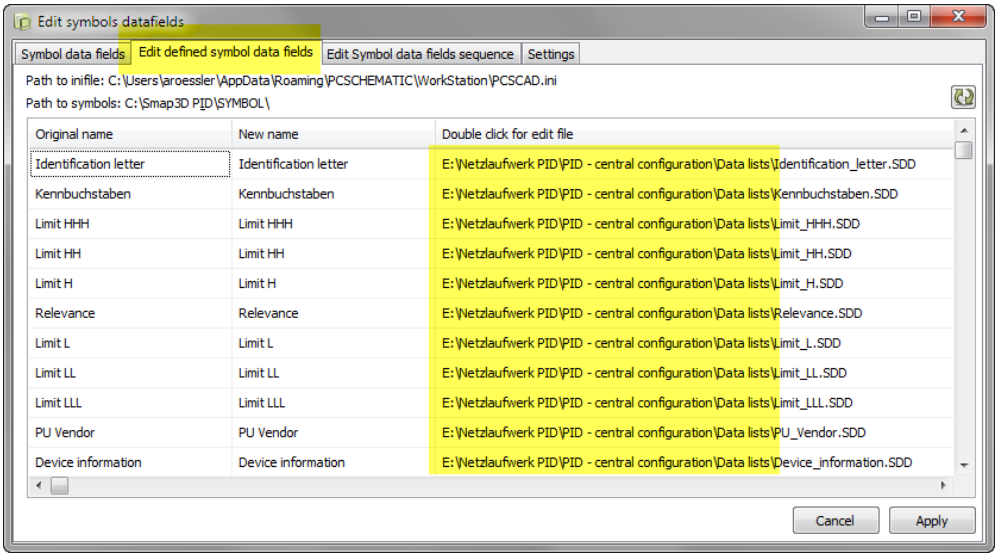
All the usual administration functions can be realized by using this interface (and the corresponding interface in a PDM system) with the PRO files from Smap3D P&ID:

- Create document in PDM
- Check in/check out document
- Exchange document information (e.g. fill out drawing header in P&ID)
- Exchange list information (e.g. automatically create parts lists in PDM).
- Revise/release document
- Etc.

# Further news in P&ID

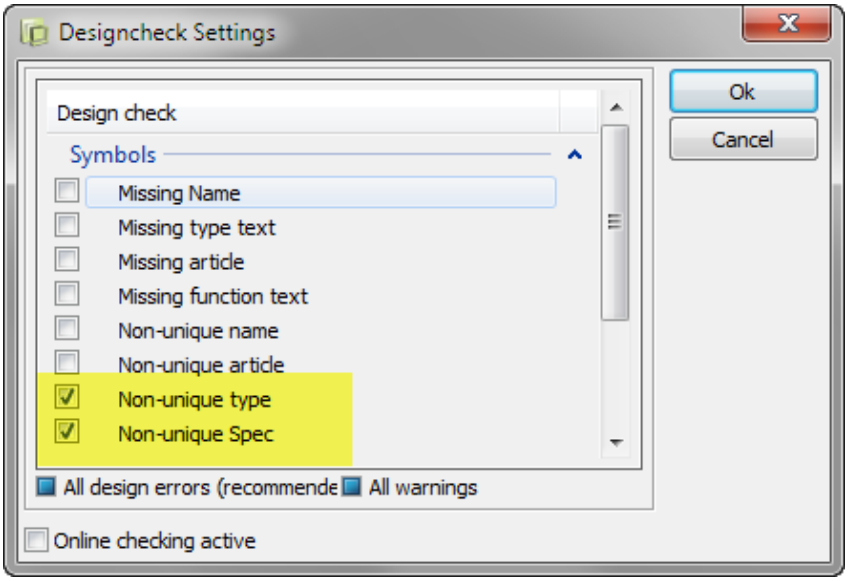
## Edit Symbols datafields uses centralized data lists

The **Edit symbols datafields** tool now also works with centrally stored data lists (*.SDD).
The path for this is defined in **Settings > Directories**.
Previously these files always had to be stored locally in the Installation directory of P&ID




## New Design Checks

In response to special requests from customers, more options for design checks were built in for symbols.
The new options **Non-unique type** and/or **Non-unique Spec** can test whether multiple symbols with the same value are present in the data field **Type** and/or **Spec** in the active project.
If this is the case, the corresponding notification appears in the test results.

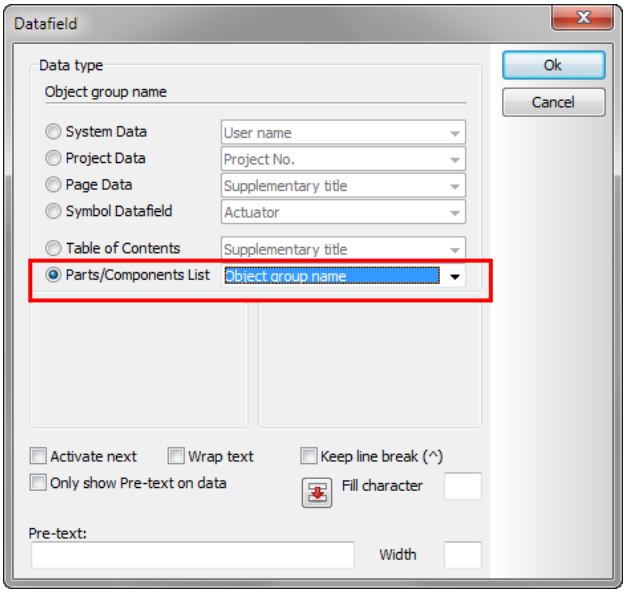


## Name of Object Group data field can be used in lists

The **Name of Object Group** data field can now be used for evaluation of lists.

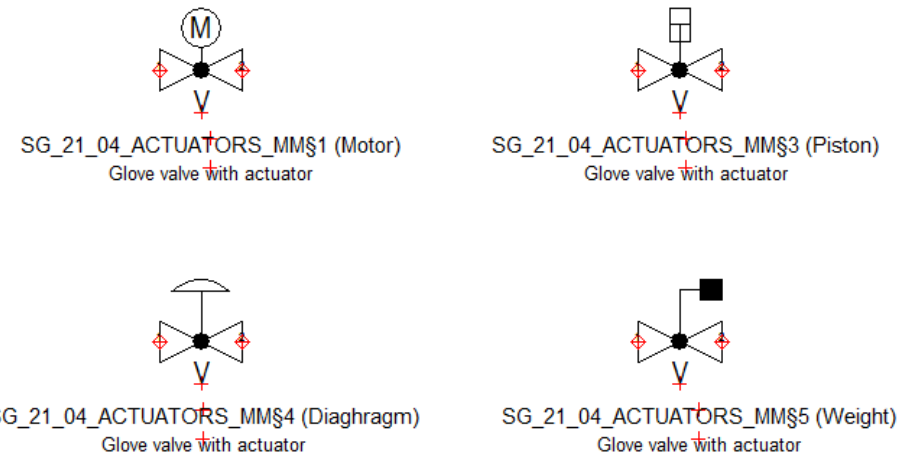
This is supported for:

- Parts lists
- Components lists
- Parts list file
- Components list file




## Symbol documentation recognize symbols with states/variants

The **Symbol Documentation** tool can now recognize **symbols with state/variants** and list them individually.
If the tool is used in the project, all symbol states/variants used on diagram pages are documented



# New in the Component Wizard

## Component Wizard supports assembly files

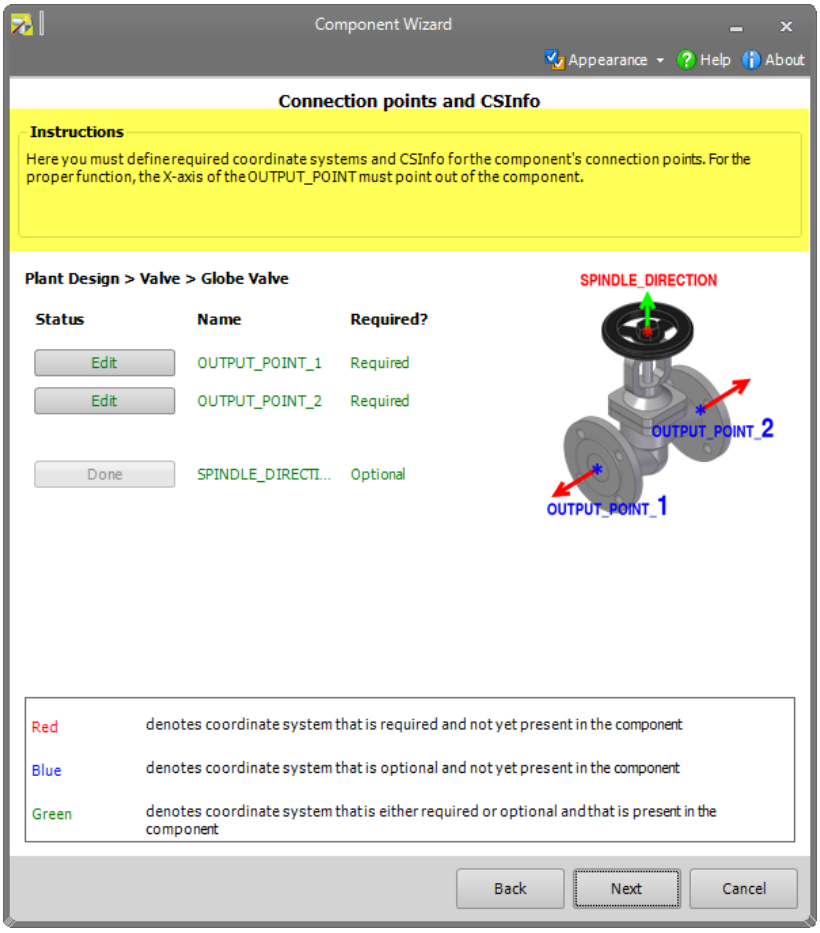
With Plant Design 2017, assembly files (*.SLDASM) can also be prepared in the Component Wizard. The process for the individual steps is the same as before for individual part files.

It only differs in the last step when saving a prepared assembly file:

- If the active SOLIDWORKS assembly file (and the associated individual part files) are already **in the standard parts folder for the FileSystemProvider**, the existing file is overwritten/updated.
- If the active SOLIDWORKS assembly is **outside of the standard parts folder for the FileSystemProvider**, the user will be asked to select a directory in the standard parts folder structure.
  Afterwards, a **Copy** of the SOLIDWORKS assembly file along with all associated individual parts is created in the selected folder.

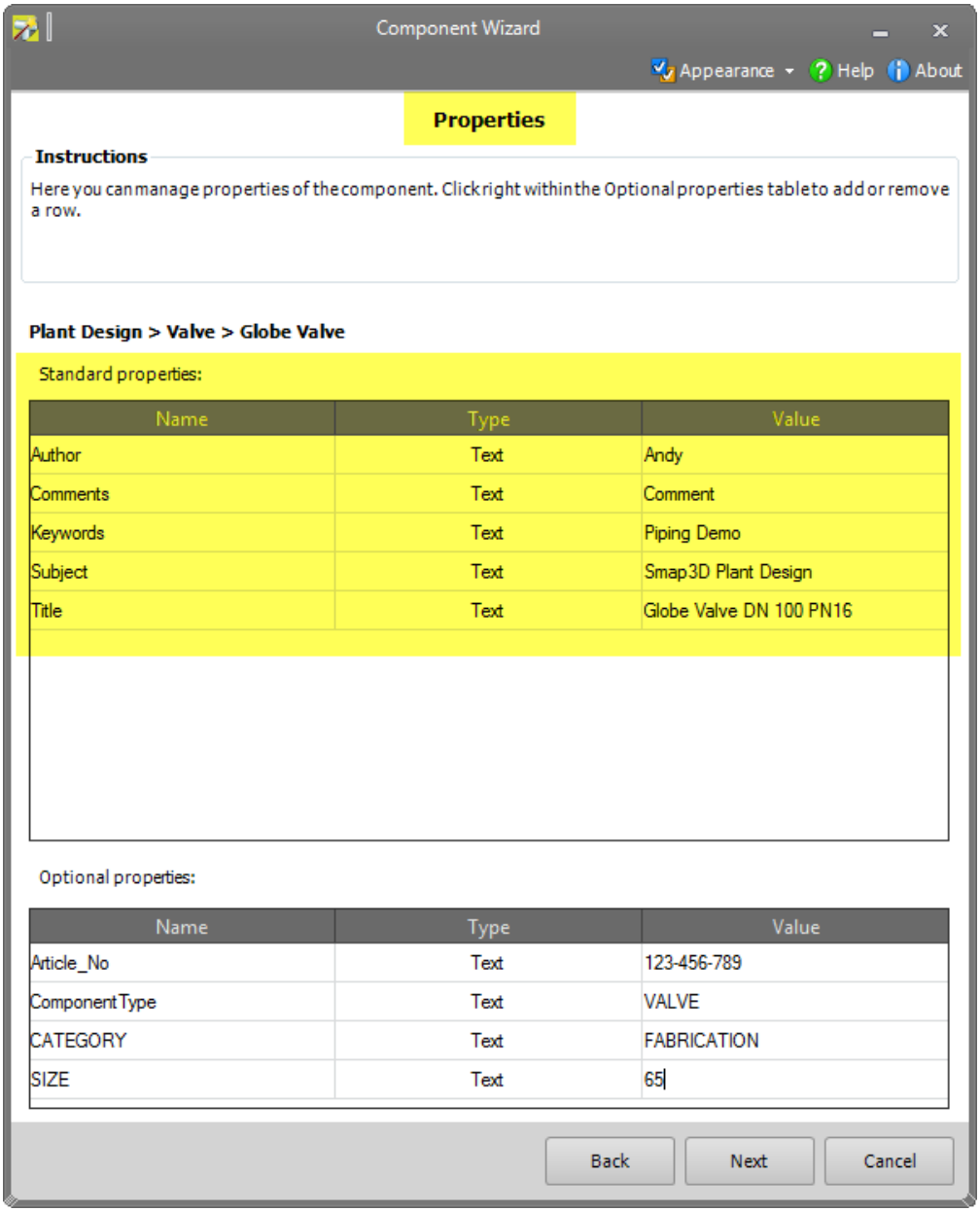
## Additional instructions

A specific instruction is now shown for each step of the Component Wizard.



## Edit standard SOLIDWORKS properties directly

In the **Properties** step, the **standard properties** of the active SOLIDWORKS file are automatically read and displayed.
In the **Value** column, values can be edited or new values can be added.
Previously, such standard CAD properties had to be edited separately with native SOLIDWORKS functions.
Now users can edit these quickly and conveniently directly during processing in the Smap3D Component Wizard.

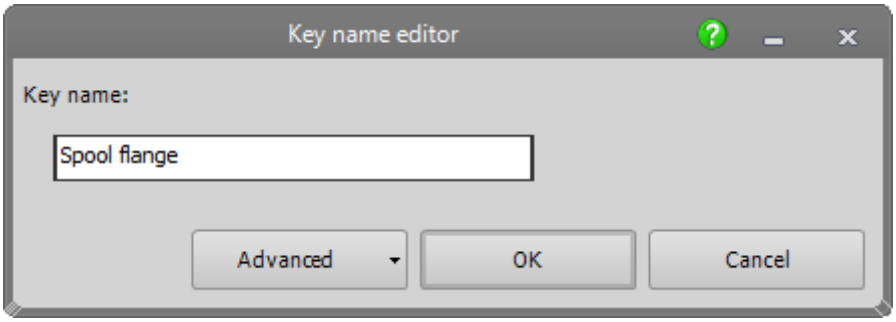


# New in the Pipe Specification Editor

## New Key name Spool Flange

With Plant Design 2017, the new key name **Spool Flange** with the associated QuickPlace method Spool Flange is available in the pipe specification editor.

***This new key name is also automatically added to existing pipe specification configurations (Common settings - PipeSpecificationCommon.pssx) with an update (from Plant Design 2016 to Plant Design 2017 - Existing data option).***

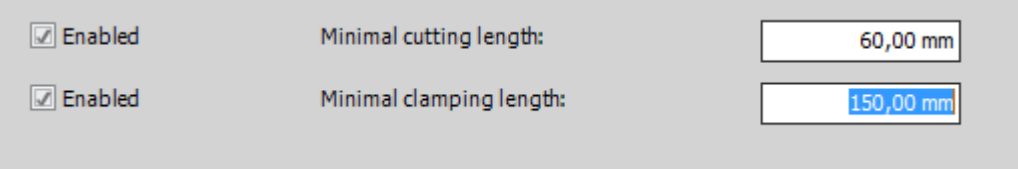



When generating a pipeline with Smap3D Piping, 3D components defined in a pipe specification with the key name **Spool Flange** are always placed on a free (unconnected) pipeline end in order to automatically create Spool pipelines.

If end components are also defined in the same pipe specification, these are ignored since the QuickPlace Method Spool Flange has the higher priority by default.

## New options for minimal cutting and clamping length

The new options **Minimal cutting length** and **Minimal clamping length** are available in the **Options** tab of the pipe specification editor.
The settings for these can be made **global** or individually for each **diameter**.




**These two settings are specially developed for bent pipes.**

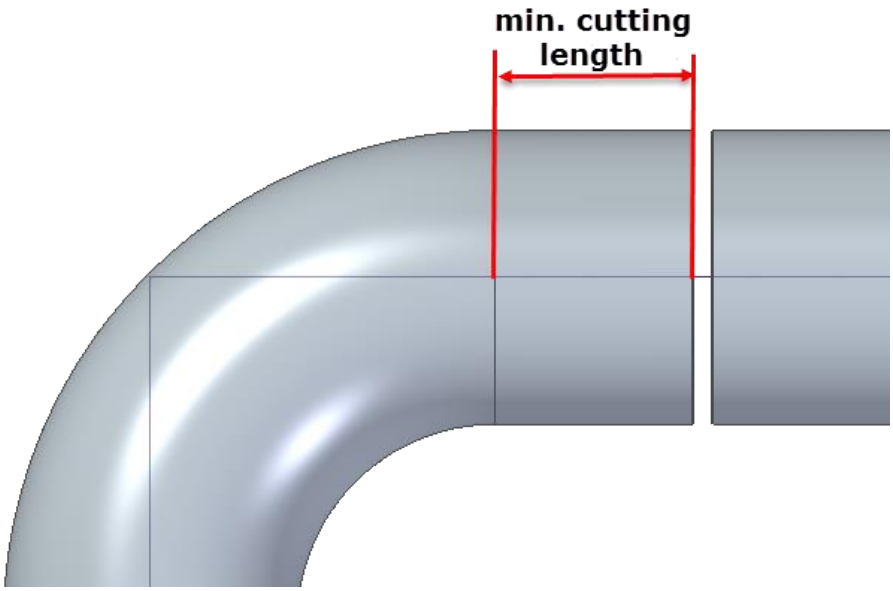
Therefore, these settings should only be used in pipe specifications in which **no bend/elbow components** are defined or in which the option **Do not place bends from pipe specification automatically** is activated.
Otherwise these settings do not provide any effect or feedback for the user.

These settings affect the geometry of bent pipes as well as feedback for problems when generating a pipe, if these are determined by the plausibility checks in Piping.

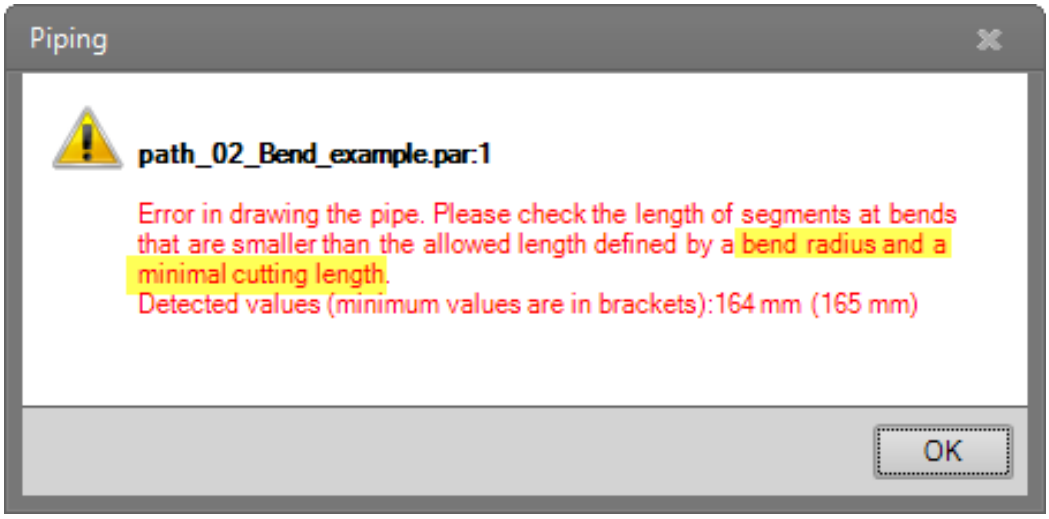
### Minimal cutting length

The minimal cutting length should be used to ensure the ensure the ability to manufacture bent pipes.
If a bent pipe will be generated with the active option **Divide**, so the bent part will be extended in both directions with the set value.

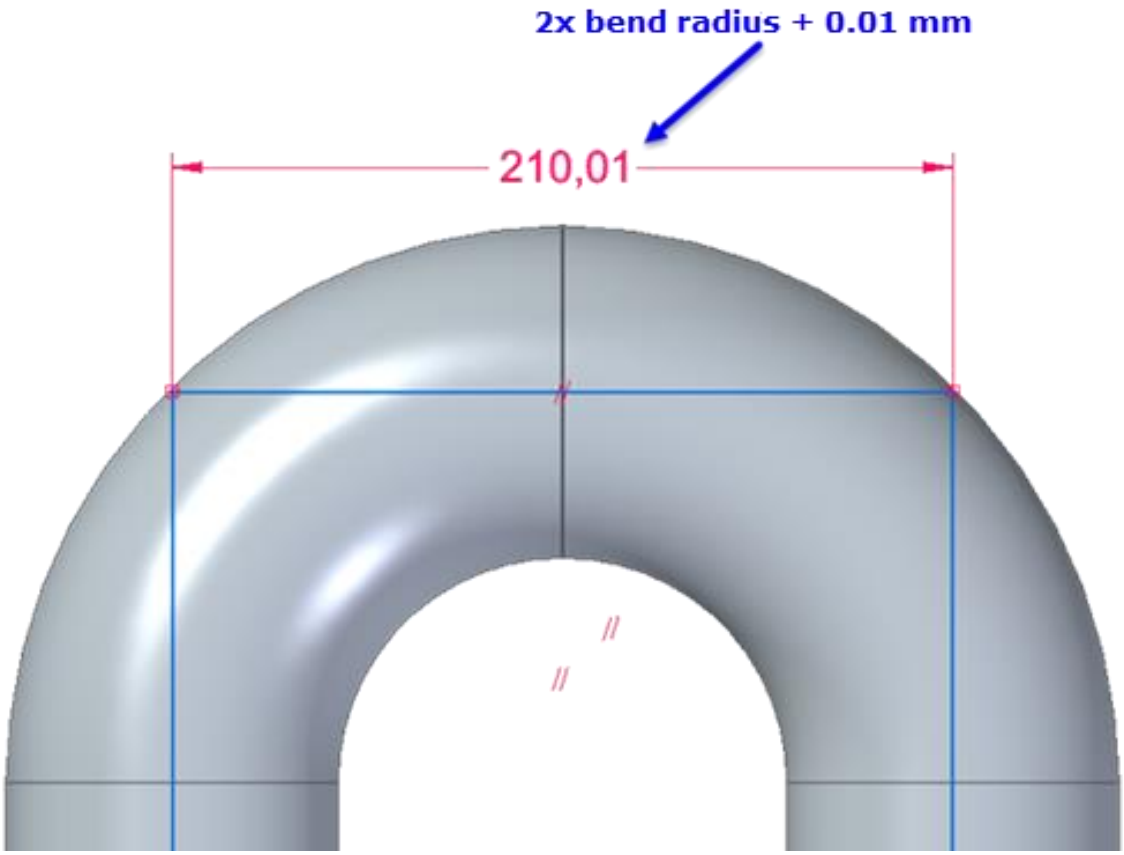
In practice this value describe the required cylindrical pipe on a bent pipe, that will be used to fix the pipe at the work process of cut to length.



If the current length of a bent pipe is shorter than the permissible length (defined by the bending radius and minimal cutting length), a notification is displayed and the pipe is not generated.

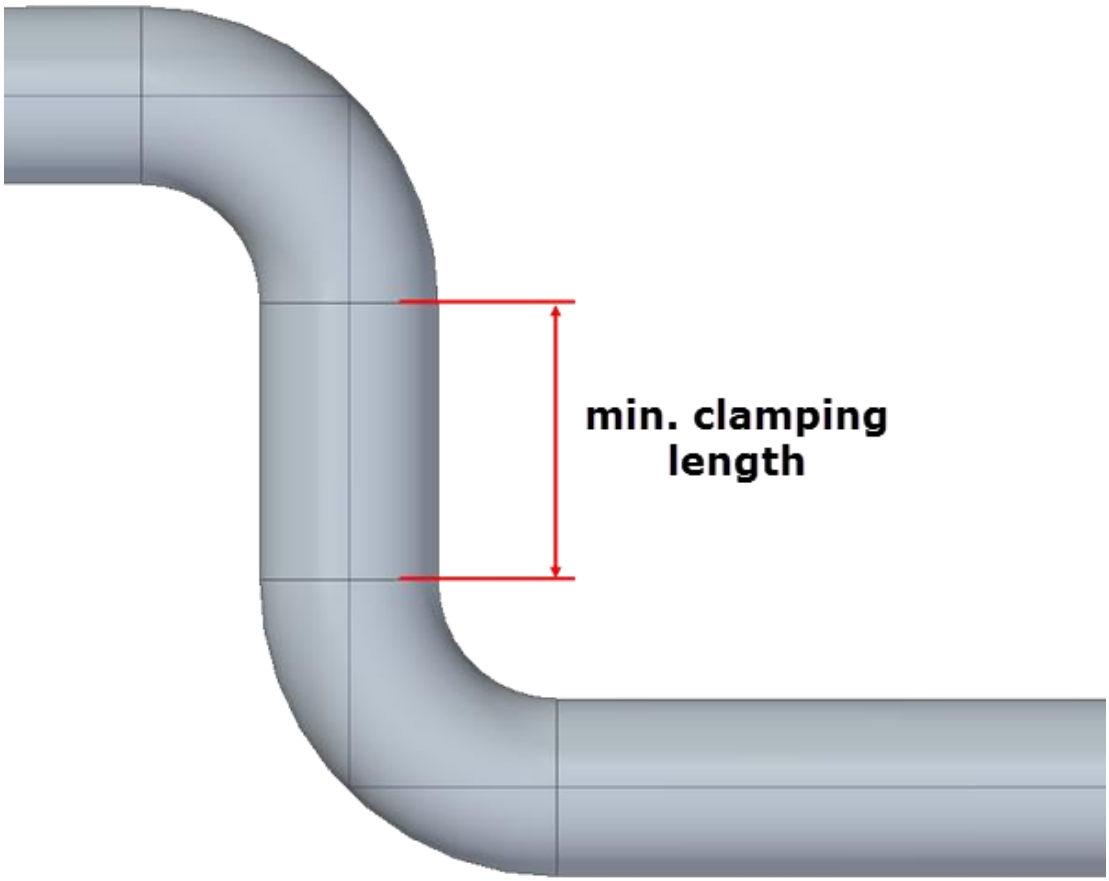


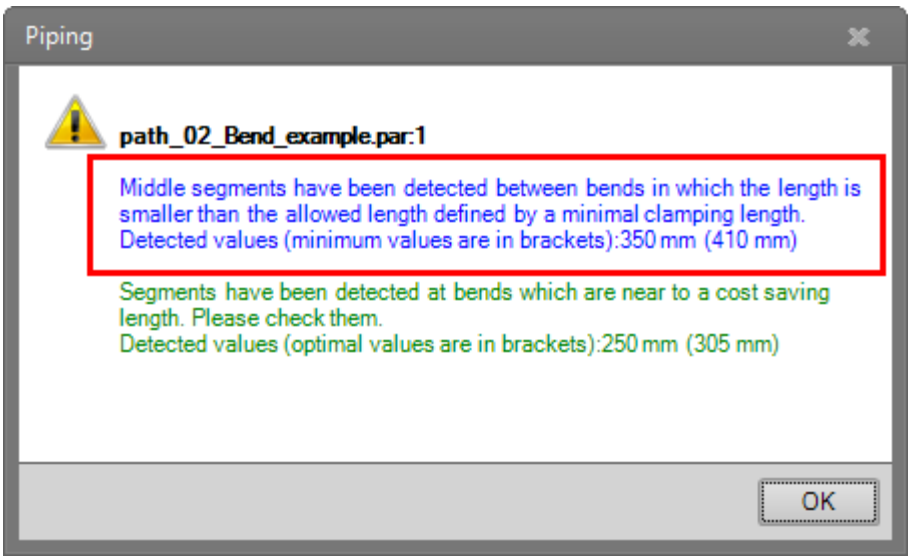

A **180° bend** requires a minimal **technical tolerance of exactly 0.01 mm.**
Therefore, the length of the line segment must be **2x bend radius + 0.01 mm**.



## Minimal clamping length

The minimal clamping length is used to ensure the ability to manufacture **bent pipes in bending machines**.
This setting only applies to the **two-line segments of a pipe bend**.



# Further news in the Pipe Specification Editor

## New options for controlling the results at pipeline ends

Since Smap3D Piping V11, in conjunction with SOLIDWORKS, components with the key name **Flange** are automatically placed on line endings of pipeline paths based on connected line elements from SOLIDWORKS 3D-sketches.

Further options have been built in for users for whom the previous standard properties did not yield the desired result.
Now pipe specifications can be defined and adjusted more flexibly according to specific requirements

### Place selected flange on connected ends

The option **Place selected flange on connected ends** can be activated for a pipe specification as needed, if the standard properties for this pipe specification do not deliver the desired result.

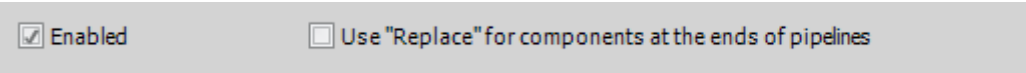
Depending on the current pipe specification content, this can be used to determine whether the components with the **key names Flange** or **Counter-flange** will be placed on a connected line end.

- This option can be defined **Global** (all diameters) or individually for each **diameter**.

**Use "Replace" for components at the ends of pipelines**

The option **Use "Replace" for components at the ends of pipelines** controls the behavior of Smap3D Piping for components at connected line ends when using the **Edit pipeline** function.

- This option can only be defined **Global** (all diameters) for a pipe specification.




If this option is **not active (no checkmark)**, all automatically placed components are deleted from the pipeline when carrying out the **Edit Pipeline** function, so that the pipeline can be completely regenerated with the new definitions.
In this case the current situation at a line end is also re-analyzed and re-implemented accordingly.

**Example:**
If the line ends were connected when generating a pipeline, Piping automatically placed a defined Flange/Counter flange there.
If the connections at the line ends are removed by the user after generating the pipeline (this process is required for some customers when using various PDM systems), a different situation results at the line ends.
If the same pipeline is then changed with the **Edit pipeline** function, the flanges placed on the ends are deleted and end components defined in the pipe specification are placed.
Basically, this pipeline will now be distorted or incorrectly changed by the **Edit pipeline** function.

To prevent this, the option **Use “Replace” for components at the ends of pipelines** was developed.

If this option is **active (checkmark present)**, all components at a pipeline end are viewed as manually placed components when carrying out the **Edit Pipeline** function.

This way they are **not deleted**, but replaced based on the key name used when the pipeline was generated.
If a **flange** was placed when generating the pipeline (in a connected state), it is **replaced in Edit pipeline** (in a non-connected state) and switched with the flange component from the **new pipe specification**/diameter.

## Priority for key names of end components was changed to 1.50

In connection with the new key name Spool Flange and the associated QuickPlace method Spool Flange, it was necessary to change the placement priority for the standard key names **End component**, **Blind flange** and **Cap** from 1.00 to **1.50**.
This ensures that once spool flange components are defined in a pipe specification, their placement on non-connected line ends is prioritized.

***This key name change is also automatically applied to existing pipe specification configurations (Common settings - PipeSpecificationCommon.pssx) with an update (from Plant Design 2016 to 2017 - option existing data).***

# New in Piping

## New QuickPlace method Spool Flange

In the previous versions of Piping, either end components (blind flange, cap, etc.) were placed on the **non-connected pipeline ends** or a pipe was generated, depending on the content of the pipe specification used.
In both cases the user had to perform manual follow-up work to produce a typical spool pipeline (flange to flange).

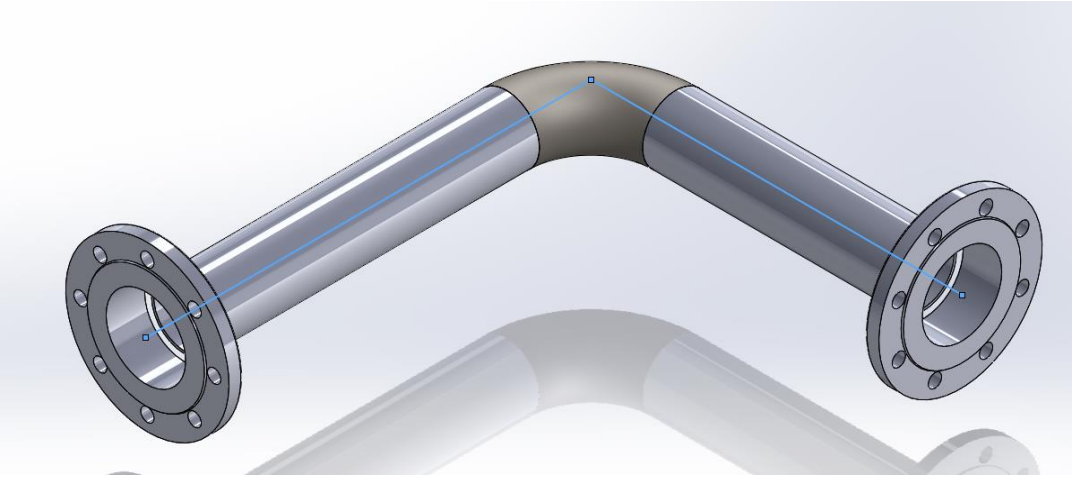
The new QuickPlace method **Spool Flange** offers the option of automatically generating pipelines as **spools** with Smap3D Piping.

The key name Spool Flange required for this is included in the new installation (standard data) and is automatically added to an existing configuration during an update (existing data).

To obtain a spool pipeline, the desired pipeline paths are drawn **not-connected to the pipeline ends**.
Then Smap3D Piping is used to generate the pipeline path and pipeline with a corresponding pipe specification.

The components defined as spool flanges in the pipe specification used are automatically placed on the pipeline ends. The sealing surface of the flange is congruently positioned at the end of the pipe path as an alignment.

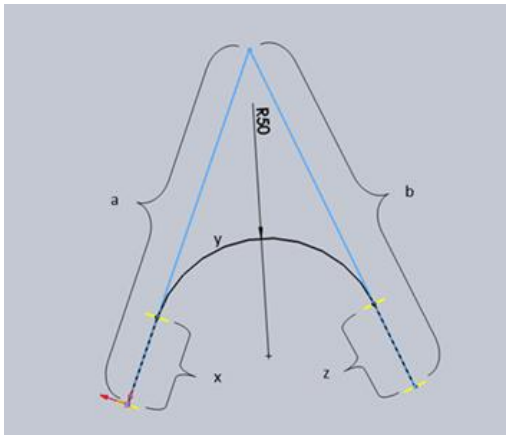
With the new QuickPlace Method Spool Flange, users can now create a spool pipeline directly and without manual follow-up work.



If end components are also defined in the same pipe specification, these are not used when generating the pipeline, since the QuickPlace method Spool Flange now has the higher priority by default.

## Re-Design of algorithms for calculating the property Length

Along with the additions to the minimal cutting and clamping length, the calculation algorithms for the extended length (user-defined property **Length**) of bent pipes were completely reprogrammed. In the previous versions of Piping, the value for the **user-defined property Length** was only calculated based on the drawn pipeline paths (a + b) as well as the connection point of a bend fitting placed on it.
This was always sufficient for straight pipes when using bend/angle fittings at the corners.




As more and more Smap3D customers switch to the production of bent pipes to reduce production costs and Smap3D Plant Design penetrates wider market areas where mainly bent pipes are used (such as hydraulics), redevelopment of the corresponding calculation algorithms was urgently needed.

For the extended length of a pipe, the new calculation algorithm now uses the actual bend radius (l = x+y+z) for each bend.

Thus, the value of the user-defined property **Length**, which is written by Smap3D Piping into every pipe generated, is now correct when bent pipes are involved. This improved algorithm is also used by the maximum length function as well as the optional placement of separators/connectors.

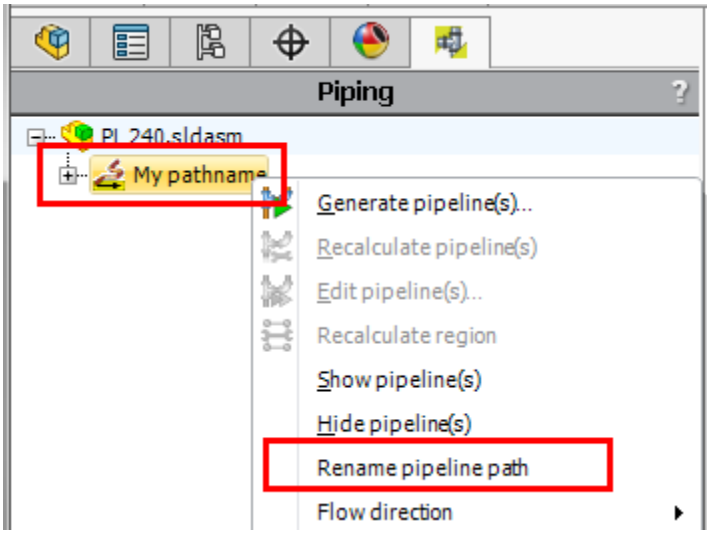
# Further news in 3D Piping

## New Rename Pipeline path function in the context menu

Up to the last 2016 version, the displayed name of a pipeline path (generated automatically with the settings from Plant Design Administrator) could only be changed and individually adapted when working with the P&ID To-Do List.

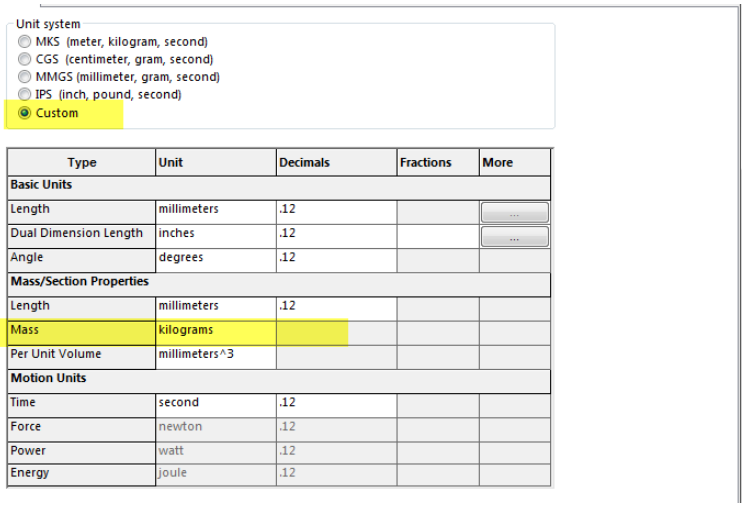
In Plant Design 2017, a new function was built into the Smap3D Piping context menu.
When an automatically generated pipeline path is marked, the function **Rename pipeline path** is available in the context menu.
A user can now enter and display a desired path name.

If a pipeline path was already changed/renamed with the P&ID To-Do List, the Piping function **Rename pipeline path is deactivated**.
The function in the context menu has a low priority and may not change any pipeline path names from the P&ID To-Do List.

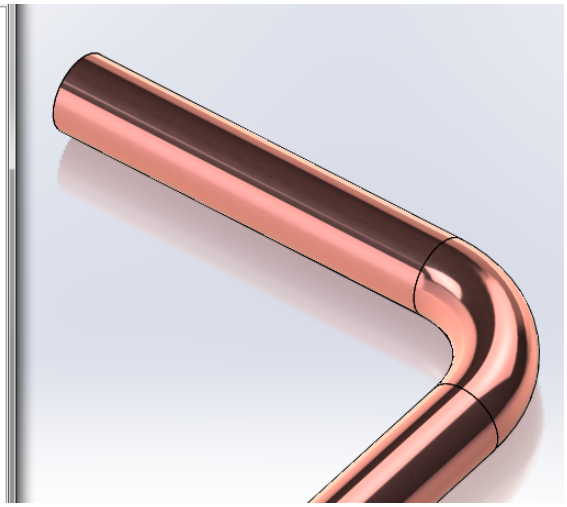


## Changed template for generating pipes in SOLIDWORKS

The SOLIDWORKS template used by Smap3D Piping for generating pipes was changed.
Now, after generating the pipes, the unit system in the files is automatically set to **Custom** and the unit for mass to **kilograms**.
In previous versions, this was always on MMGS and grams and hat to be changed manually.




| Type | Unit | Decimals | Fractions | More |
|---|---|---|---|---|
| **Basic Units** | | | | |
| Length | millimeters | .12 | | ... |
| Dual Dimension Length | inches | .12 | | ... |
| Angle | degrees | .12 | | |
| **Mass/Section Properties** | | | | |
| Length | millimeters | .12 | | |
| Mass | kilograms | | | |
| Per Unit Volume | millimeters^3 | | | |
| **Motion Units** | | | | |
| Time | second | .12 | | |
| Force | newton | .12 | | |
| Power | watt | .12 | | |
| Energy | joule | .12 | | |



## Expanded detection of component units (component groups) in Edit pipeline function

With Piping 2017, the detection of **component units** when carrying out the **Edit pipeline** function was improved.

Component units are **3D components built together,** which are partly built with manual functions (e.g., place part, replace part) and additionally partially built with automatic functions (e.g., recalculate pipeline, recalculate region) on a pipeline path.
**It is important that mates between the parts with the coordinate systems (connection points) were created.**

Up to Piping 2016, such component units could not be detected and processed together in the Edit pipeline function.
Therefore, after the completion of the Edit pipeline function, there was often an erroneous mate between the manual and the automatically placed components.

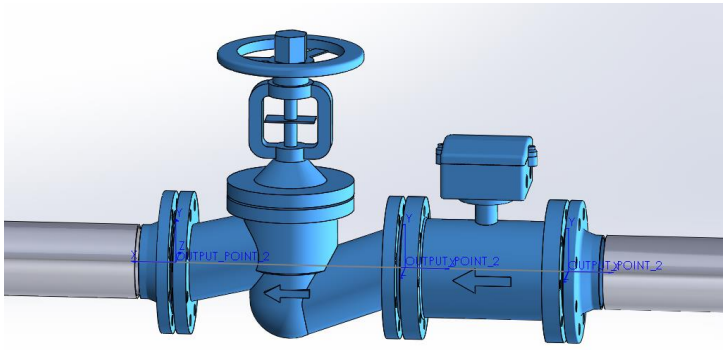
As of Piping 2017, component units are now also recognized as a unit in the Edit pipeline function and completely replaced if the **mate between the contained components were created with the coordinate systems of the connection points**.

If other mates between surfaces and other component geometries were used, this can still lead to erroneous mates after editing the pipeline.

**Example for creating such a component unit correctly:**

- With **Place Part,** place the 1st component (e.g. shut-off valve) on the pipeline path in the axial direction.
- With **Recalculate pipeline** (or Recalculate region), allow the automatic components (seals and flanges) to be placed.
- Mark the flange at the desired position and use the **Replace part** function to place/replace the 2nd component (e.g. flow meter).
- With **Recalculate pipeline** (or Recalculate region), allow the remaining automatic components (seals and flanges) to be placed.

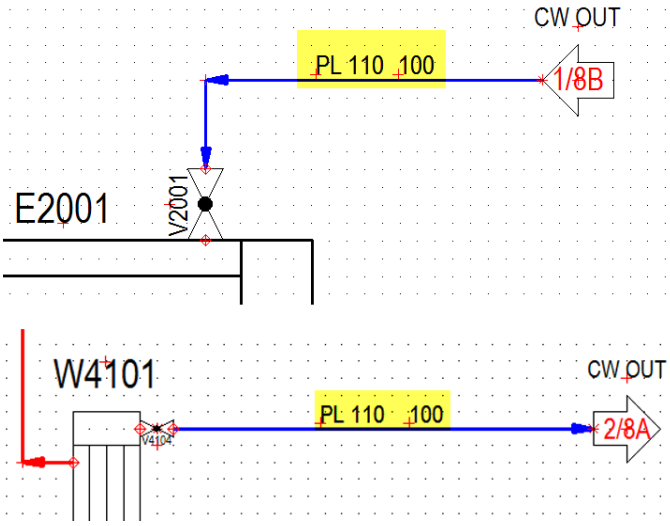
With this approach, the coordinate systems of the connection points are always used correctly for the connections between all components.



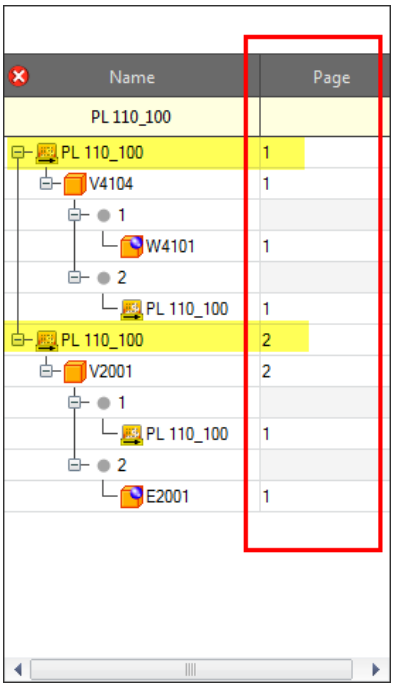
# New in the P&ID To-Do List

## Improvement for lines across multiple diagram pages

The To-Do List now supports the analysis of lines across multiple P&ID diagram pages, without requiring the display criterion **Page**.

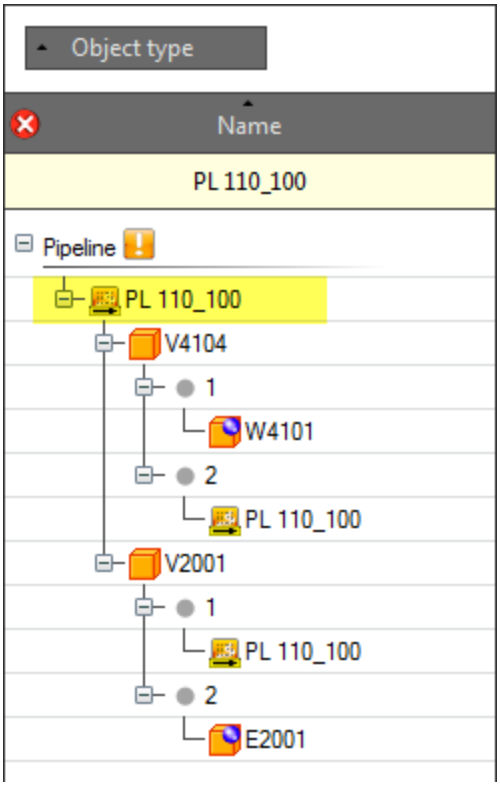



Up to and including version 2016, the property **Page** had to be used as a display criterion for analysis in the To-Do List.
The lines on the individual diagram pages were thereby displayed several times as individual entries in the structure.
The entries were handled separately, so that when the pipeline number was assigned to a 3D pipeline, an entry always remained in the To-Do list without assignment (orange).



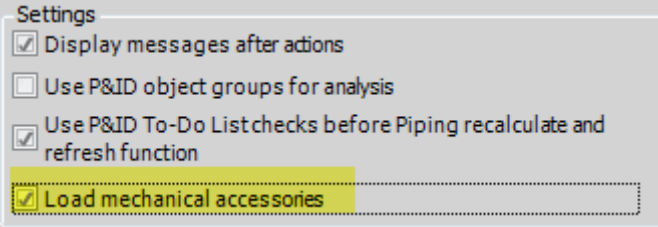
As of version 2017, the use of the **Page** display criterion is **no longer required**.
The To-Do list can also display all relevant information and structures for such **a cross-page line** from the P&ID correctly **without a page**.

The unique assignment of the pipeline from the To-Do list to a pipeline path in Piping also leads to the correct and complete assignment from 2D to 3D.
If the display of the page is still used in the To-Do list (2 paths are displayed), the unique allocation to a 3D pipeline path is sufficient to assign both entries from the To-Do list at the same time (green).

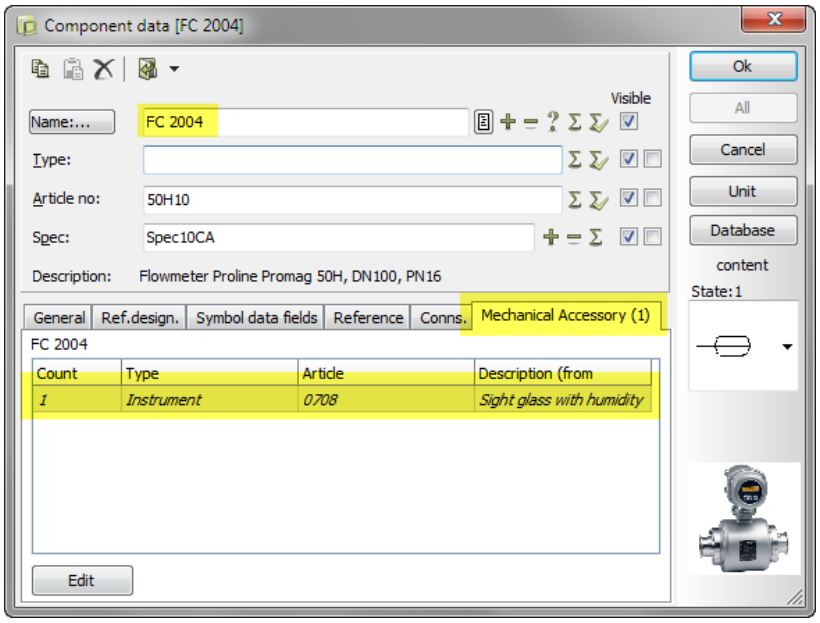


## Support of mechanical accessories from P&ID

The new option **Load mechanical accessories** is available in the **P&ID To-Do List Options**.
If this option is activated, the mechanical accessories from P&ID are also analyzed and displayed in the To-Do List.




Mechanical accessories in P&ID are defined as non-graphical information about components that is also required in addition to the existing graphical components (symbols) and whose definition contains at least one **article numbe**r.

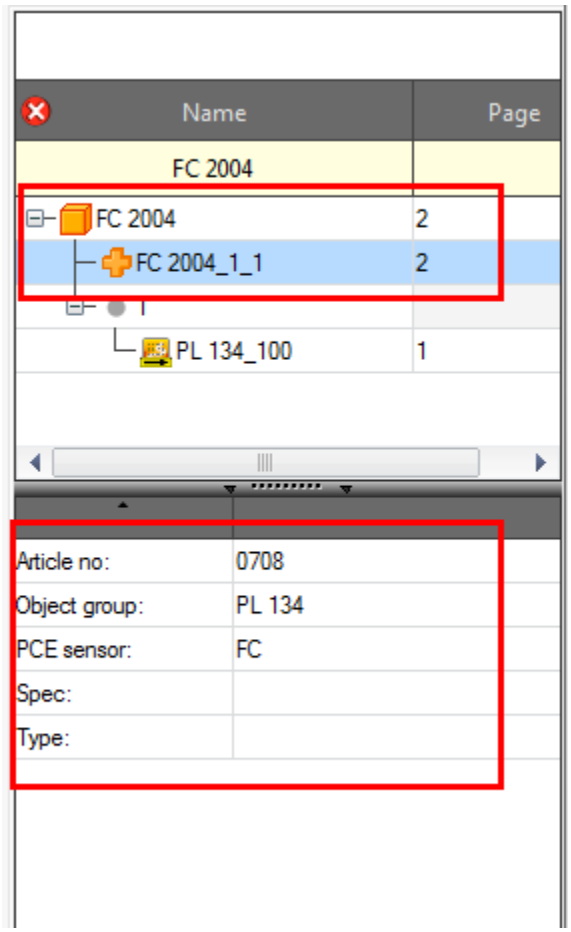



Mechanical accessories can be added to a component in different ways in P&ID:

- to an article/data record in the database, so that the predefined accessory is automatically used when assigning the article to a symbol.
- to a symbol in a diagram via the **Component data** dialog (Mechanical Accessory tab).
    - On the one hand, a data record from the database can be selected as an accessory,
    - Or, a one-time accessory definition can be created manually by entering the relevant data from users.

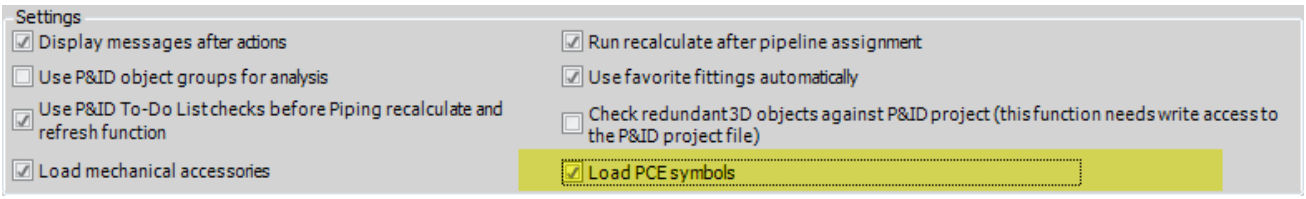
Mechanical accessories from a P&ID diagram are displayed in the structure under the component for which it was defined.
A new icon (colored plus sign) in the To-Do list identifies such accessory components.
If such a component is marked, the available information for it is displayed in the lower section of the To-Do list.

The option should only be activated if you do not want to assign non-graphical accessories from P&ID to existing graphical components models in the 3D assembly.
The To-Do list functions for assigning components or updating information work identically for normal components as well as accessory components.

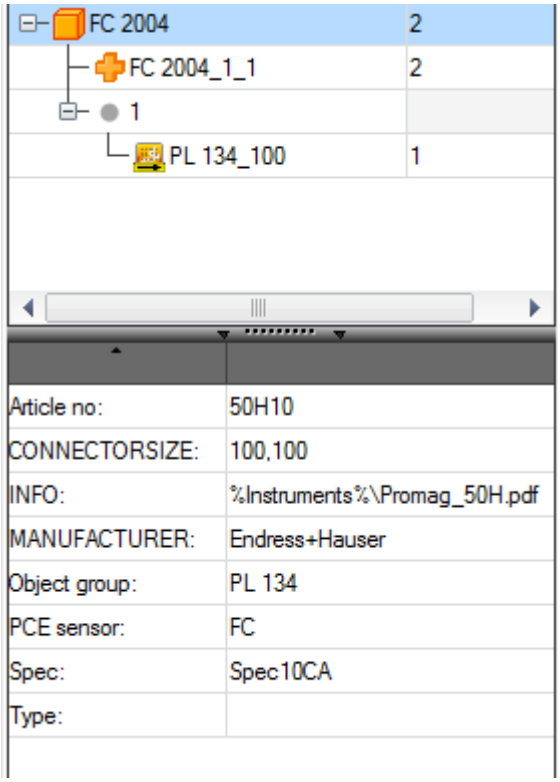


## Support of the new PCE symbol type

The new **Load PCE symbols** option is available in the **P&ID To-Do List Options**.
Is this option activated, symbols with the symbol type PCE from a P&ID diagram are also analyzed and displayed.




PCE symbols are then displayed in the To-Do List as normal components and with all available information




This option should **not be activated** if users do not want to use PCE symbols (measuring points) in the To-Do list.
These are then filtered out and ignored when the P&ID diagram is loaded.

## Improved automations

### Update info in CAD differentiates between definition changes and property changes

When using the To-Do List function **Update information in CAD** for a pipe, a distinction is made between a definition change (parameter for pipe specification, diameter, pressure, etc.) or a change of a property (text value).

Previously the function **Update information in CAD** has always recalculated the complete pipeline, even if only the value of the information/text property had changed.

In Plant Design 2017, this improves performance when working with the To-Do list, since pipelines are recalculated only when basic parameters have been changed.
Potential waiting times are significantly reduced for the user.

# New in Isometric & PCF Export

## Subassemblies are analyzed as pipeline components

In the previous versions, Smap3D Isometric & PCF Export had to contain all components which should be analyzed and displayed in the isometry or PCF file, in the same assembly plane as the piping path of Piping.

With Plant Design 2017 we made this requirement significantly more flexible.

It is now possible to **prepare** relevant pipeline components (or whole pipelines) as **subassemblies**, in order to build them as prefabricated units into or on the 3D pipeline.

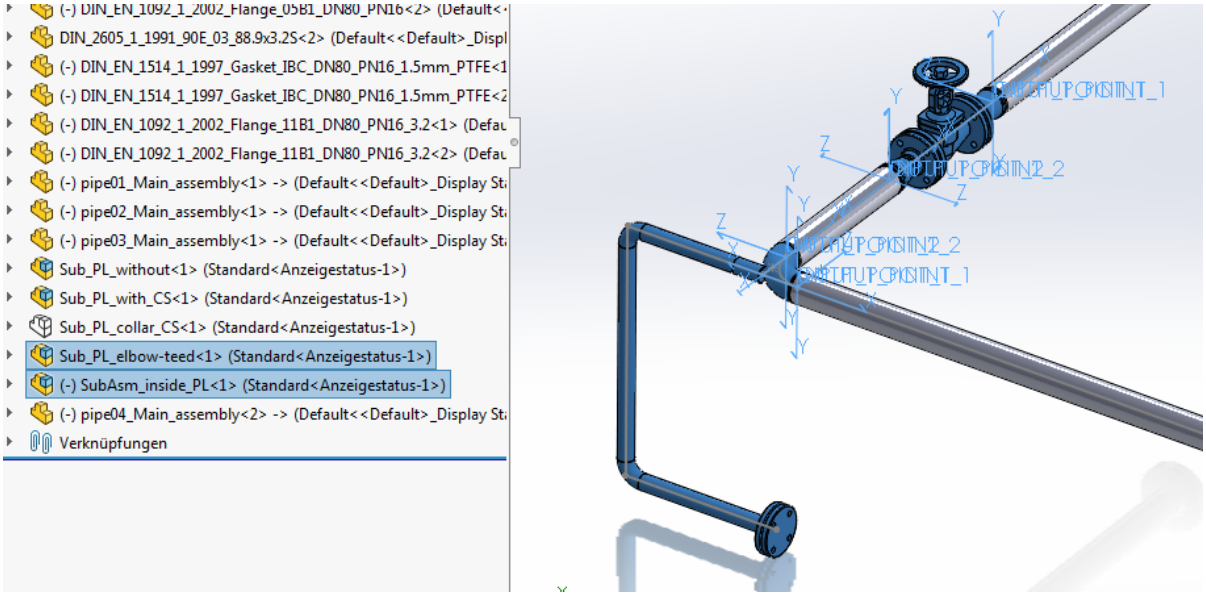
For **Smap3D Piping**, the corresponding **connection points for this unit must be placed as coordinate systems** in such subassemblies.
This allows Piping to recognize these assemblies as components and process the 3D pipeline correctly.

However, the **P&ID To-Do list** can still recognize the contents of the subassemblies as individual objects, and components can be assigned to individual content.

For **Smap3D Isometric & PCF Export**, the custom property **ISO_IsAssembly** with the value **TRUE** must be contained in such subgroups.
As a result, Isometric & PCF Export recognize that this module is to be used with its individual contents for the isometric drawing/PCF output file.
In this case, the individual components of such a sub-assembly must, of course, be prepared with the **correct information for SKEY and ComponentType**.

If the P&ID To-Do List was used to assign information from P&ID to a component, this is also analyzed and displayed individually in Isometric & PCF Export.

For users, this new function simplifies the handling of recurring standard content in pipelines, analysis in a PCF output file and the production of isometrics.


(-) DIN_EN_1092_1_2002_Flange_05B1_DN80_PN16<2> (Default<<
DIN_2605_1_1991_90E_03_88.9x3.2S<2> (Default<<Default>_Displ
(-) DIN_EN_1514_1_1997_Gasket_IBC_DN80_PN16_1.5mm_PTFE<1
(-) DIN_EN_1514_1_1997_Gasket_IBC_DN80_PN16_1.5mm_PTFE<2
(-) DIN_EN_1092_1_2002_Flange_11B1_DN80_PN16_3.2<1> (Defau
(-) DIN_EN_1092_1_2002_Flange_11B1_DN80_PN16_3.2<2> (Defau
(-) pipe01_Main_assembly<1> -> (Default<<Default>_Display St
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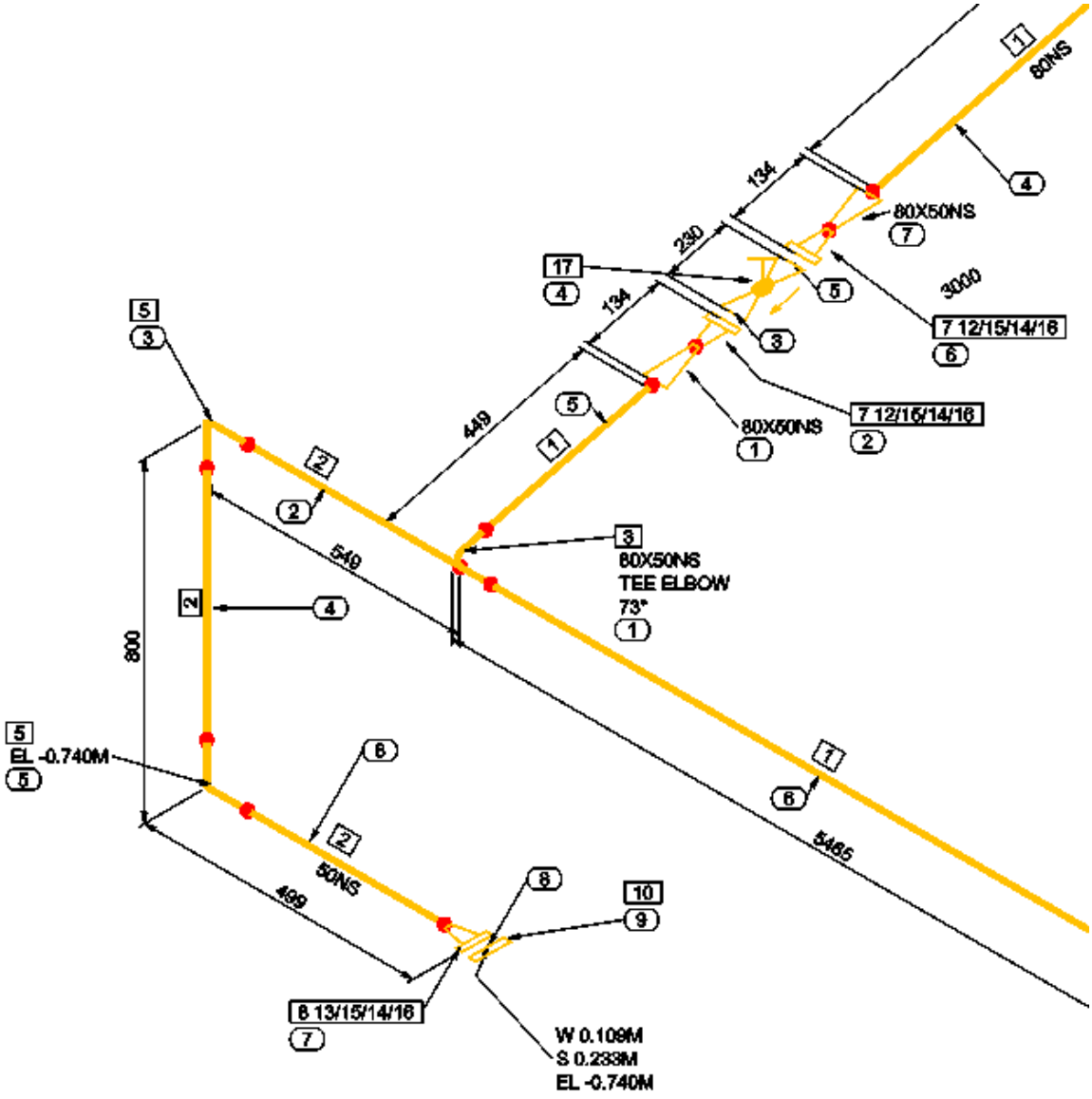
Sub_PL_with_CS<1> (Standard<Anzeigestatus-1>)
Sub_PL_collar_CS<1> (Standard<Anzeigestatus-1>)
Sub_PL_elbow-teed<1> (Standard<Anzeigestatus-1>)
(-) SubAsm_inside_PL<1> (Standard<Anzeigestatus-1>)
(-) pipe04_Main_assembly<2> -> (Default<<Default>_Display St
Verknüpfungen


80NS
80X50NS
7 12/15/14/16
80X50NS
7 12/15/14/16
80X50NS
TEE ELBOW
73°
50NS
8 13/15/14/16
EL -0.740M
W 0.109M
S 0.233M
EL -0.740M
134
230
134
3000
449
549
800
499
5485

## Further news

### PCF property PIPELINE_REFERENCE available in assignment of properties

The PCF property **PIPELINE_REFERENCE** can now be used in the mapping of Smap3D Isometric & PCF Export in order to use/display user-specific requested information as **PIPELINE_REFERENCE**. Until now, this PCF property was not available in mapping, but was permanently linked to the pipeline path name.

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