This is new in Smap3D Plant Design 2021 for Solid Edge

Version 2021 of Smap3D Plant Design contains many enhancements and improvements in functionality and application.

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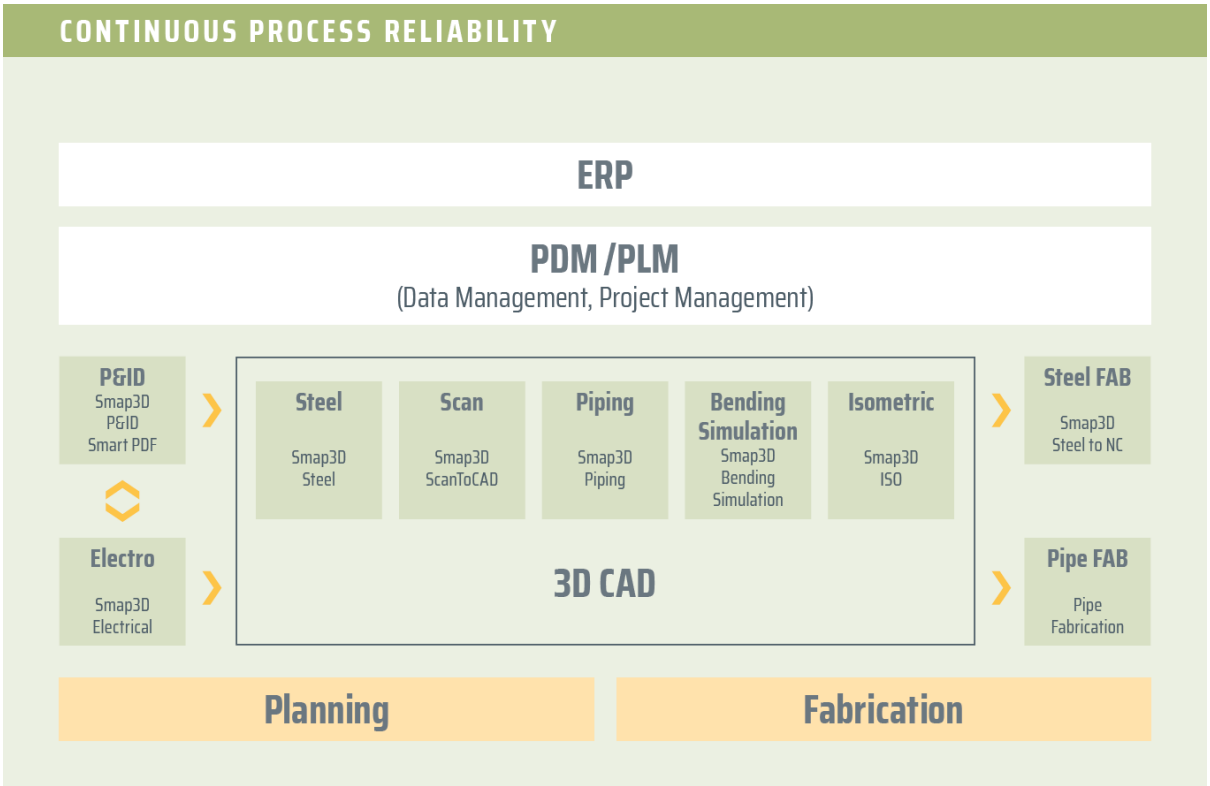
- General news
- ScanToCAD
- P&ID
- Coordinate System Wizard
- 3D Piping
- Pipe Specification Editor
- P&ID To-Do List
- Isometric



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General improvements

Supported CAD versions

Solid Edge

The release version of Smap3D Plant Design 2021 supports:

- **Solid Edge 2020 with Maintenance Pack 11 or later**
- **Solid Edge 2021 with Maintenance Pack 0 or later**

Licensing in version 2021 required

With Smap3D Plant Design 2021 the CAD-Partner Licensing Software (License Server Manager and License Client Manager) must be installed in version 2021.

For existing license server installations of CAD-Partner Licensing 2020 (when using floating and node-locked licenses), must be updated/upgraded to version 2021 (for details see **installation guide chapter > Upgrade**).
This upgrades an existing installation of License Server Manager 2020 to version 2021.
The updated version number of License Server Manager 2021 is then **6.21.100.5**.
The corresponding FlexNet Publisher version is then **11.17.1.0**.

All local license client managers 2020 must be uninstalled along with the Plant Design 2020 applications, before the new installation of Plant Design 2021 for the client computers can run.

Product licenses from version 2020 still valid

The Plant Design 2021 applications can also work with licenses issued for Plant Design version 2020. For this reason, customers with a valid maintenance contract will receive a mailing with the download link for the new 2021 version, **but no new license files**.

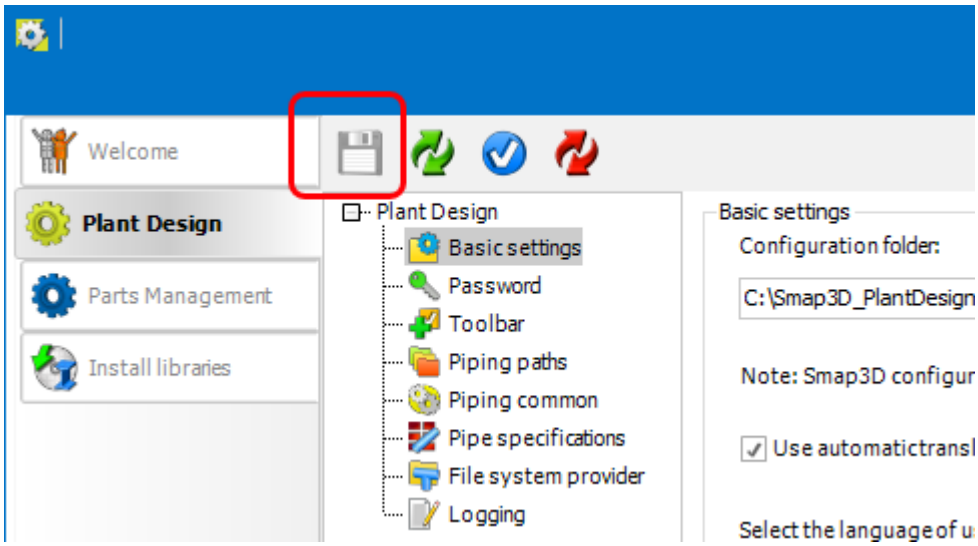
Installation of the applications tor C:\Program files

All Smap3D Plant Design applications from version 2021 onwards are now installed by default in the path **C:\Program files\CAD-Partner**, which is intended for 64-bit programs.
Up to version 2020, the default installation path was always C:\Program Files (x86), which was intended for 32-bit programs.

Save button only active after changes

After opening the administrator programs **Plant Design Administrator** and/or **Parts Management Administrator**, the **Save** button was always active immediately.
Users could not always clearly see whether they had made unintentional changes to the settings or not.

To prevent such uncertainty for users, the **Save** button now only reacts to changes in the settings. If you open these programs just to look something up without changing any settings, the button always remains inactive (grey).



New option, not delete redundant pipe files

In very special situations at customers, in interaction with certain PDM systems, it is not desired that redundant pipe files that are no longer used in assemblies are deleted from the file system. However, this has always been and still is the standard case with Smap3D Piping.

For all customers with this very special requirement, we have already built an invisible option into the software with Version 2020 service pack 2, in order to make this possible and to leave redundant and no longer used pipe files on the file system.

With version 2021 this invisible option is now prepared by default in the configuration file **PDGlobalSettings.pdsx** and **does not have to be written there individually**.

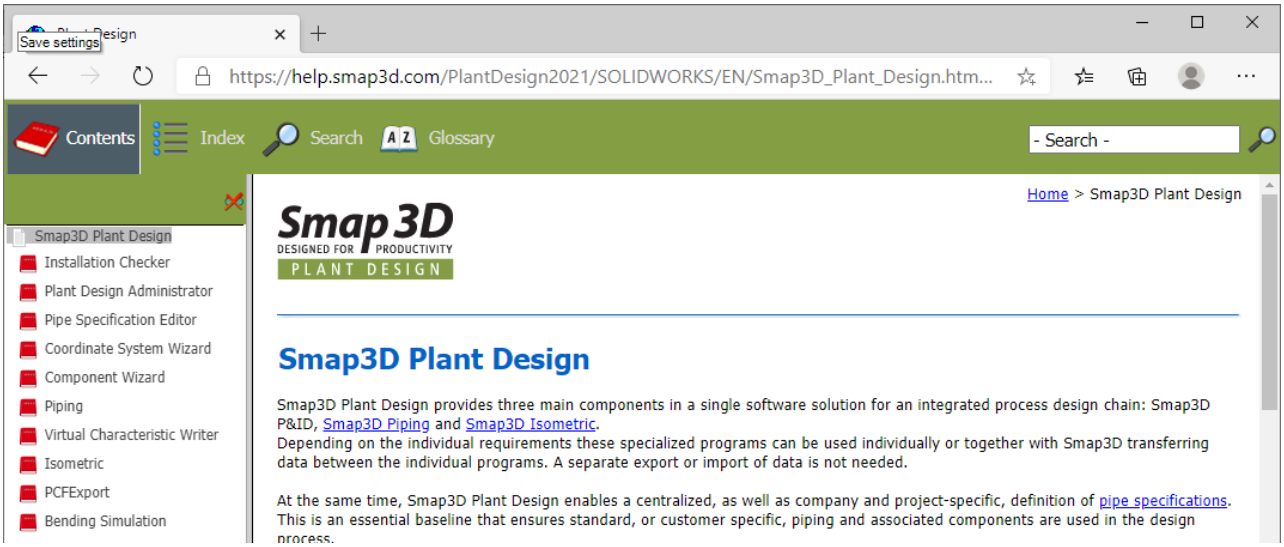
The configuration file **PDGlobalSettings.pds**x now also contains the entry:

<OtherSettings>
 <Guid>NonUISetting</Guid>
 <Key>**DeleteUnusedPipeFiles**</Key>
 <Value>**True**</Value>
</OtherSettings>

However, this has not changed the standard behavior of piping:
If option entry is not there: pipe files are deleted
Is option value **True** : Pipe files are deleted
Is option value **False** : pipe files are not deleted

New graphical layout of Smap3D online Help

After the introduction of online help with version 2020, for all **Smap3D Plant Design programs** (except P&ID, Electrical and ScanToCAD), with version 2021 we have renewed the graphic layout of the online help pages and adapted them to our company and website CI.



Reminder: Automatic log files during installation

Since the Hardware and software combinations of the computers on which Smap3D Plant Design is installed, are becoming more and more complex, from version 2020 onwards our installation routine will always automatically generate log files.

After an installation, the storage path of these log files is always in the directory **[ProgramData]\CAD_Partner_GmbH\SetupLogs**.

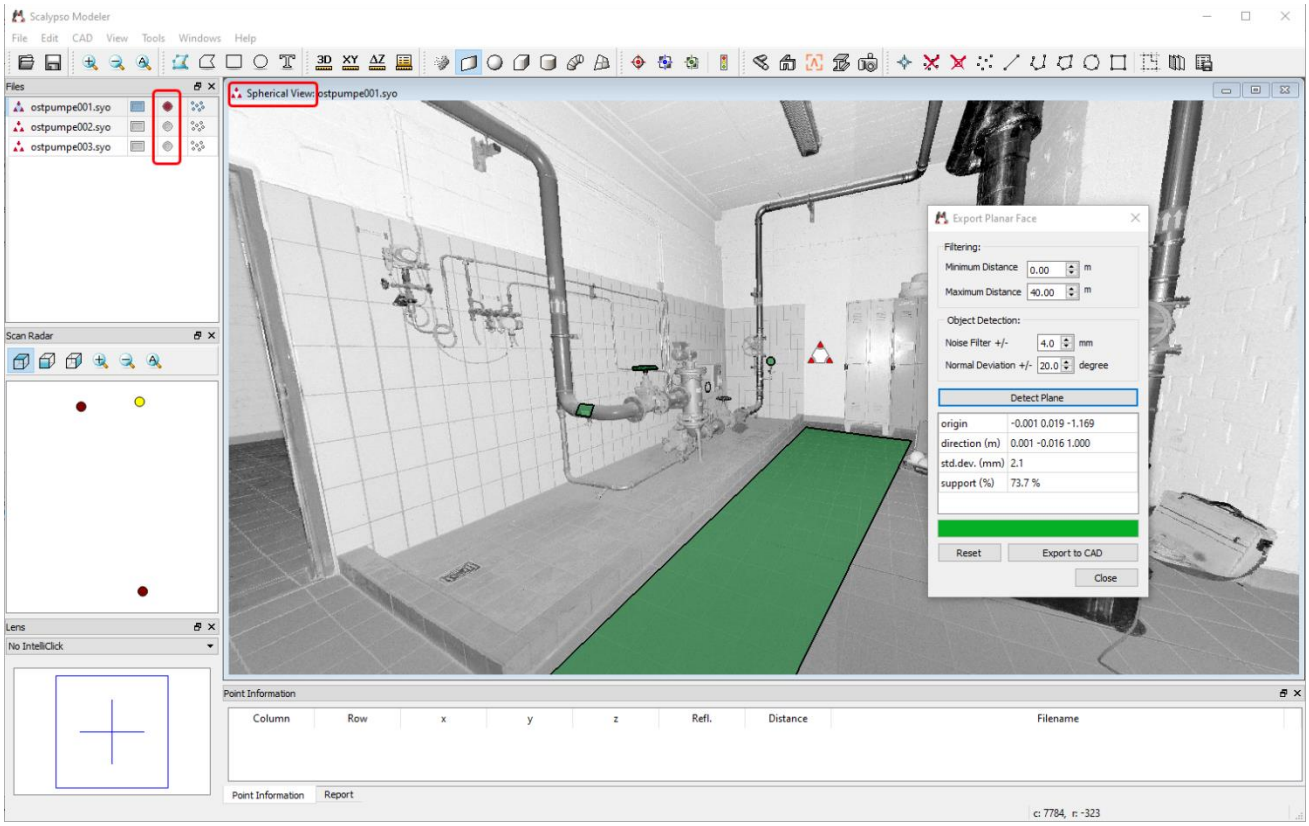
New in Smap3D ScanToCAD

Selection and CAD export from the spherical views

In previous versions of ScanToCAD, point and geometry selection for CAD export was only possible in the **planar view** (unfolded planar 360-degree view) of a scan.
The navigating and working in planar views took some users some getting used to or was even problematic.

To simplify this, the basic handling has now been considerably extended and all existing selection functions can now also be used in the **spherical views**.
This is intended to help experienced 3D CAD users to avoid having to deal with the planar view of Scan.

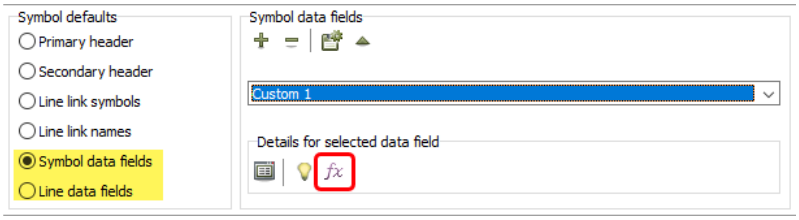
Selections and CAD exports can now also be performed in the 3D environment of a **Spherical view**.



New in Smap3D P&ID

Formula editor supports mathematical operations

With version 2019, the **data field formula** technology was introduced in Smap3D P&ID.
In the menu **Settings > Text/Symbol defaults** the button **Function for data field** is available for **line** and **symbol data fields**.

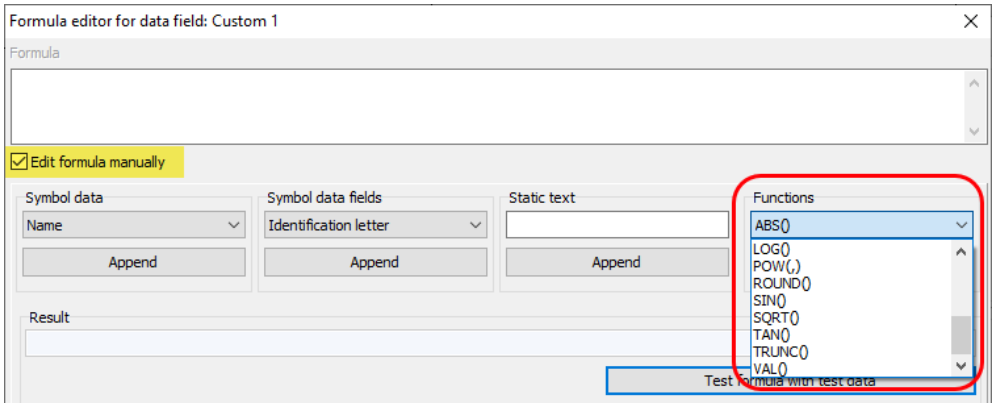


After selecting an existing data field, this button opens the **Formula editor** dialog.
With the previous functions it was mainly possible to create a string of single data field values, to a single text result and display it visibly in the diagram.

With the new functions you can now also perform **mathematical operations** on values from data fields and then display the calculation results visibly.
For this purpose, the dialog of the formula editor has been extended with the additional areas **Functions** and **Test Data**.

The new mathematical operations can be used in formulas, once the **Edit formula manually** option is enabled.
Only then the corresponding Append button for the **Functions** area becomes active.

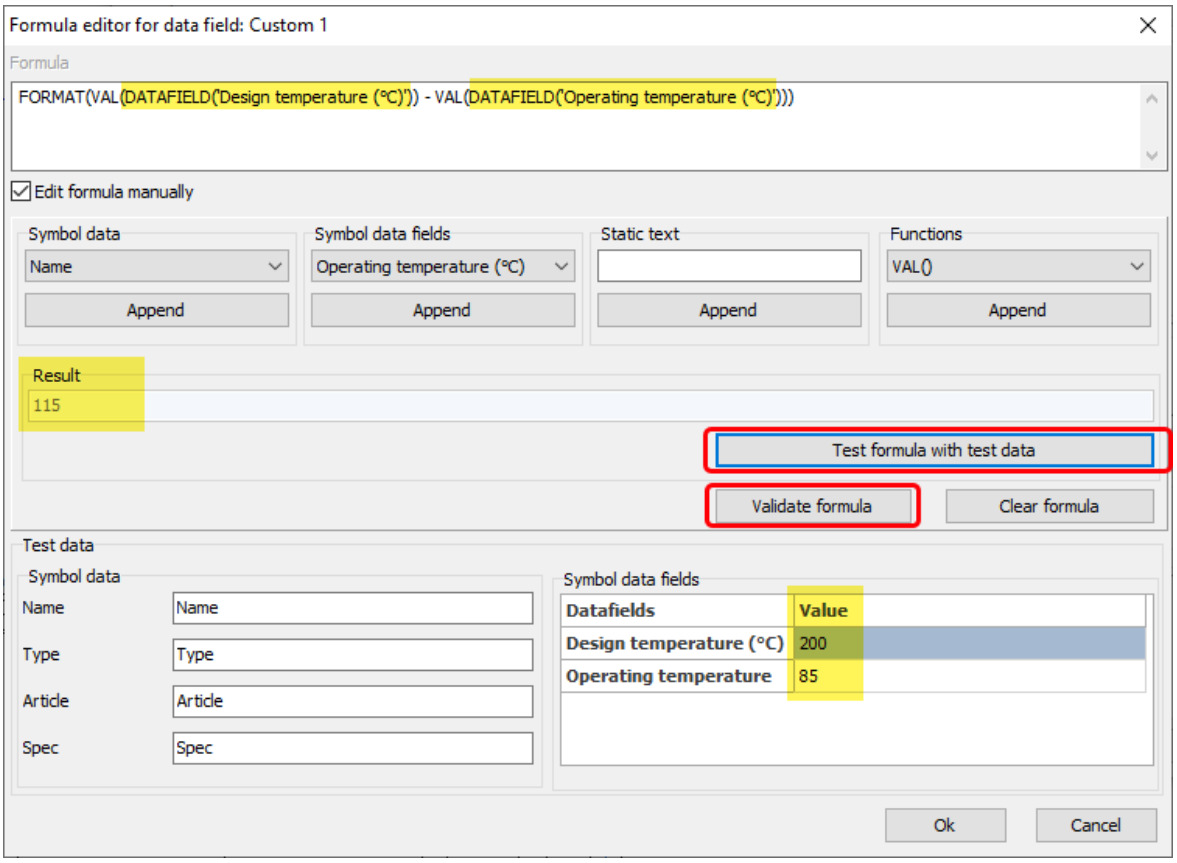
Supported are **arithmetic operators**, **boolean operators**, **comparison operators**, **equality operators** and **string operators**.
All available functions are available in the list field and can be selected from there.



Depending on the mathematical operation to be performed with a formula, it must contain all needed components to be a valid formula.
With the button **Validate formula** the string defined in the **Formula** field can be checked for its correct syntax.
For a formula to function correctly later, no error message may appear during validation. If you receive error messages, they must be corrected before leaving the formula editor, or the whole formula is deleted again with the **Clear formula** button.

If a formula has a correct syntax, you can use the **Test formula with test data** button to check it against test data and suitable sample values entered by the user for the data fields used.
In the **Result** field of the formula you can see whether the operation is performed correctly.

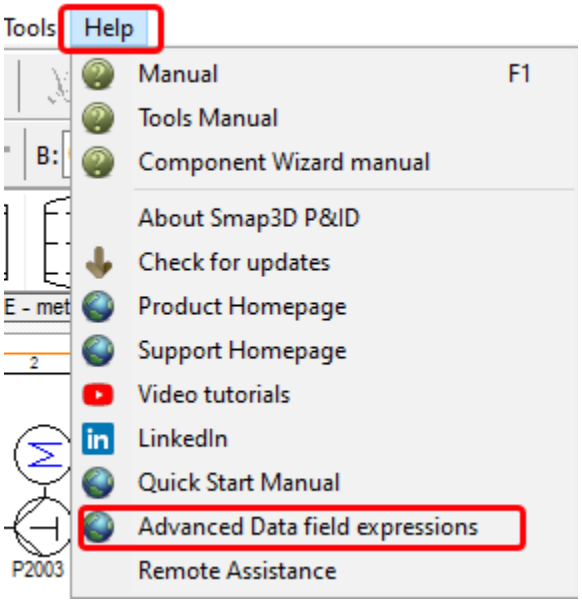
The following example shows a formula for a subtraction of the values of two different data fields.



If you close the formula editor with **OK**, from now on the created formula is applied to the selected data field.

If a user uses an object with this formula in a diagram and enters meaningful values in the underlying data fields, the P&ID software will automatically perform the calculation according to the formula and display the corresponding result in the data field with the formula.

Since the application possibilities, but also the rules behind these new mathematical operations are very extensive, we have created a separate documentation for them.
This documentation is included in every P&ID 2021 installation and can be accessed via the **Help menu**.



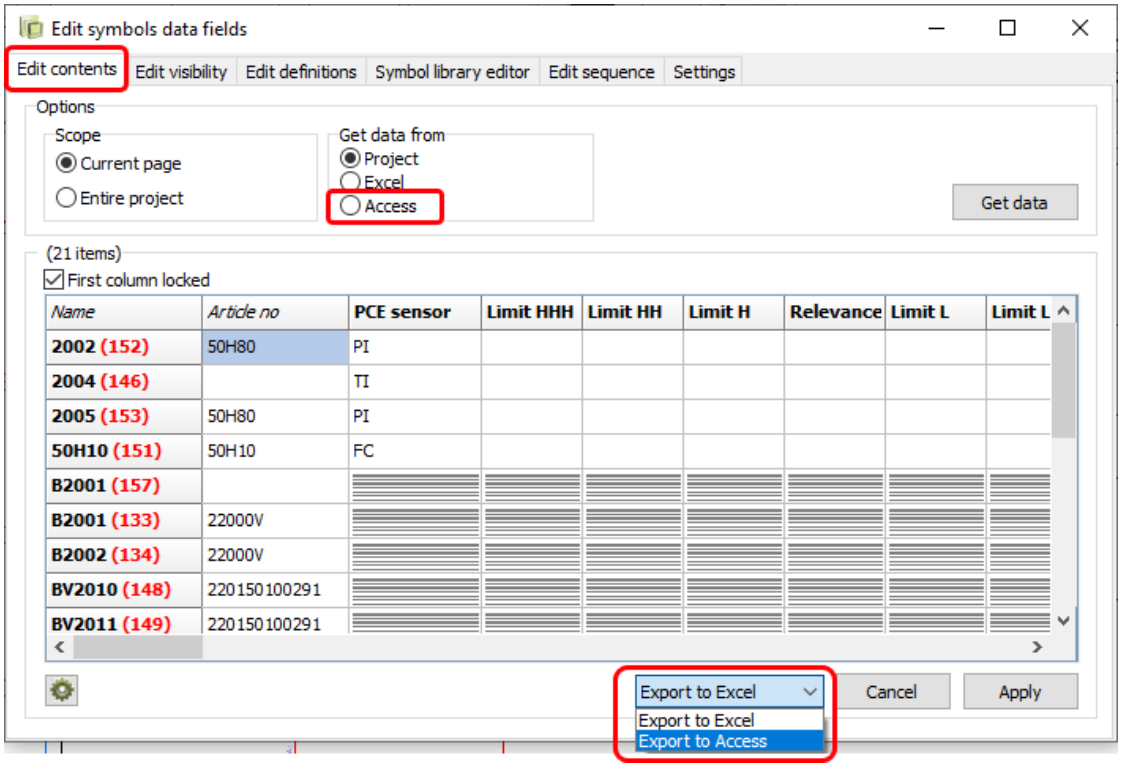
We would be happy to train you on these new possibilities within the framework of a specific training course.

Due to numerous customer requests, we have further expanded the options for exporting and importing symbol data field contents in the tool Edit symbols data fields.

Support of Microsoft Access MDB files

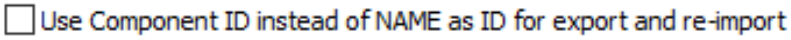
In the **Edit contents** tab, support for Microsoft Access MDB database files for **export** and **import** has been implemented in addition to the existing formats.

The function and procedure within Smap3D P&ID have not changed in this context.
The new MDB format only extends the possibility of data processing outside P&ID.

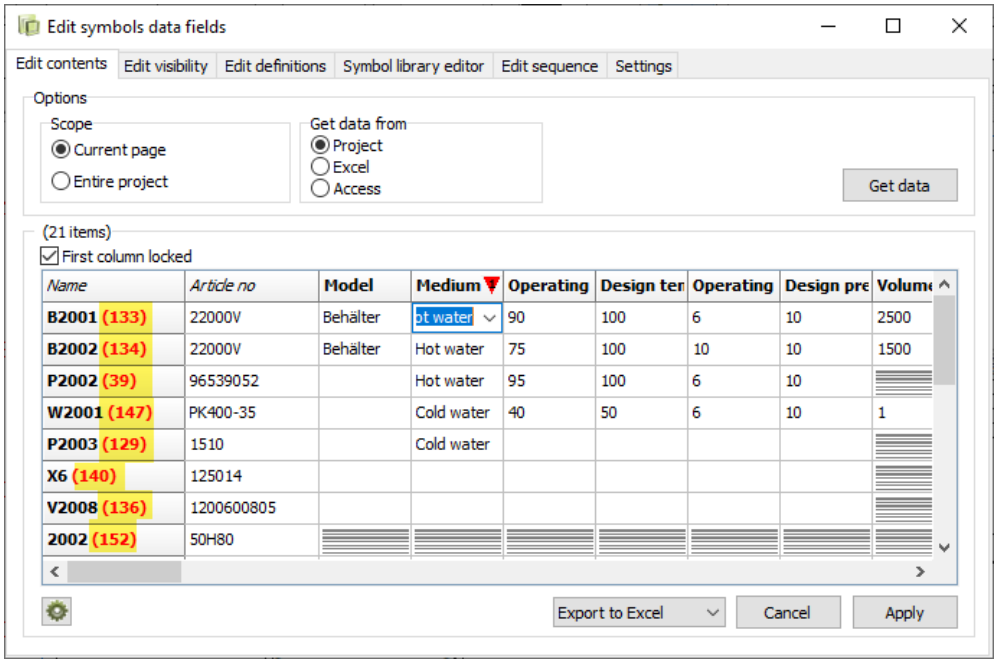


New option Component ID

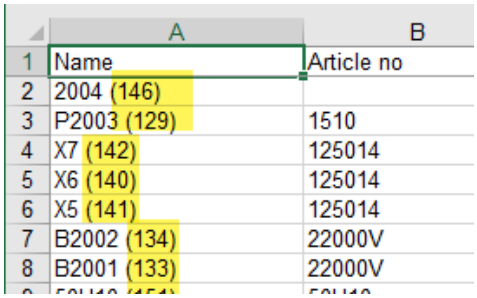
The new option **Use component ID instead of Name as ID for export and re-import** is located on the **Settings** tab of the Edit symbols data fields tool.
By default, this is not activated (no check mark).



This new option has been added to the **Edit contents** tab in P&ID 2021. In addition to the name value (TAG), an internal and unique **component ID** is now also evaluated in the **Name** column. This component ID is additionally displayed in **red and in brackets**.



This component ID is always exported even when exporting to Microsoft Excel or Microsoft Access (regardless of whether the new option is enabled or not).



	A	B
1	Name	Article no
2	2004 (146)	
3	P2003 (129)	1510
4	X7 (142)	125014
5	X6 (140)	125014
6	X5 (141)	125014
7	B2002 (134)	22000V
8	B2001 (133)	22000V

This option is only used when **re-importing** symbol field contents from Microsoft Excel or Access MDB files.

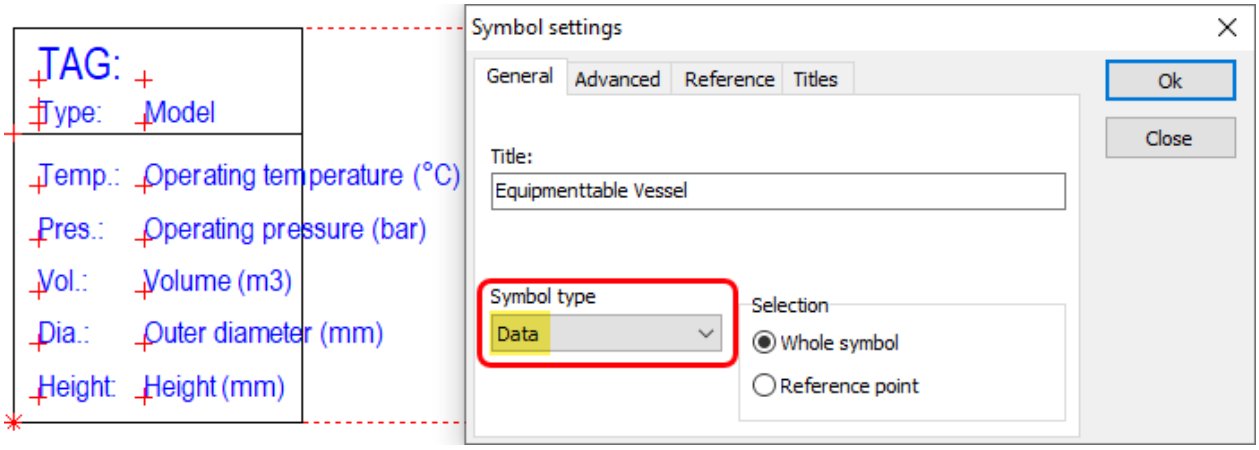
- If this option is **not activated** (default), only the name value in the **Name** column is used as unique ID for the assignment of data field contents to the diagram symbols during re-import. Here the values for Name (TAG) in the external file **must not be changed**, otherwise the necessary assignment between external file and the symbols in the diagram no longer exists.

- If this option is **activated**, it is also possible to change the name values/TAG in the Name column **(not the value in brackets)** in the external file (Excel or Access).
  The component ID (value in brackets) is then used for the re-import and thereby for the unique assignment between external file and diagram symbol.

Use of data symbols for equipment tables

The placement of equipment tables on diagram drawings is a common method of presenting additional information on equipment components in a structured form on P&ID drawings.

Table symbols of this type must be stored with symbol type **Data** in the symbol library to ensure that the information from a component symbol (tank, pump, etc.) is automatically transferred to a separate equipment table.



Only if this basic prerequisite is fulfilled, the placement of such a table symbol can establish associativity with the component symbol regarding the information contained therein.

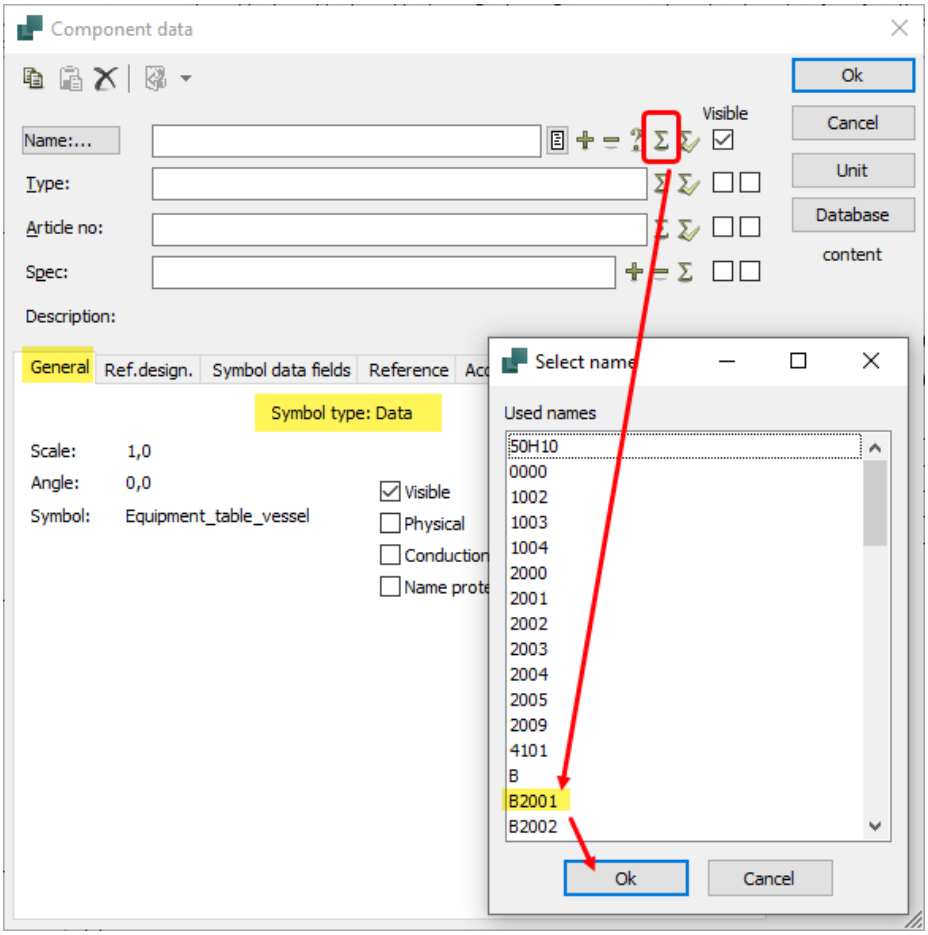
The associativity is then created when the table symbol is placed with the following workflow:

1. Instead of manually entering the **Name** of the desired component symbol (whose information is to be displayed in this table), this name must be selected using the **Used** function.

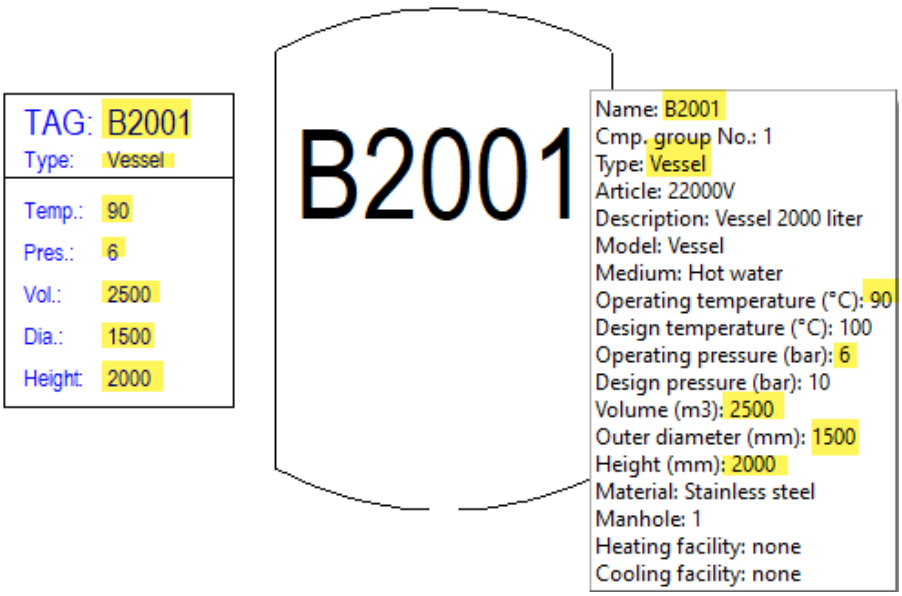
2. After clicking on the button Σ, the corresponding **Select name** dialog opens.

3. From the list of used (existing) names, you should select the desired name of the component and press **OK**.

4. Close the **Component Data** dialog with **OK**.

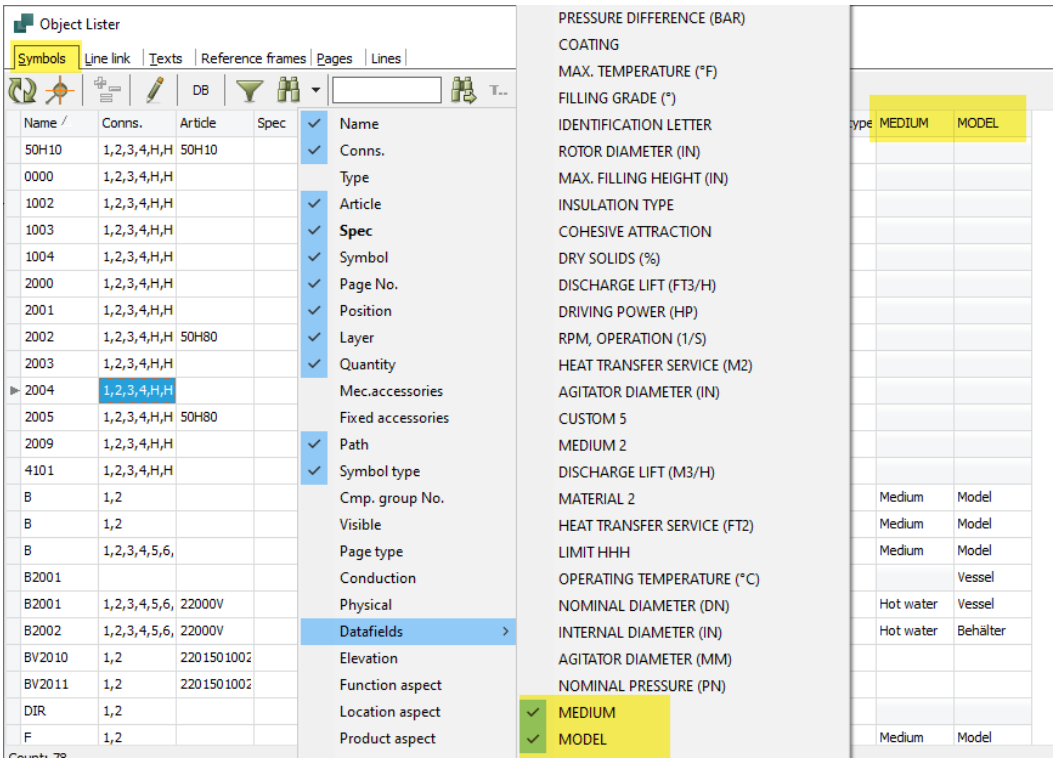


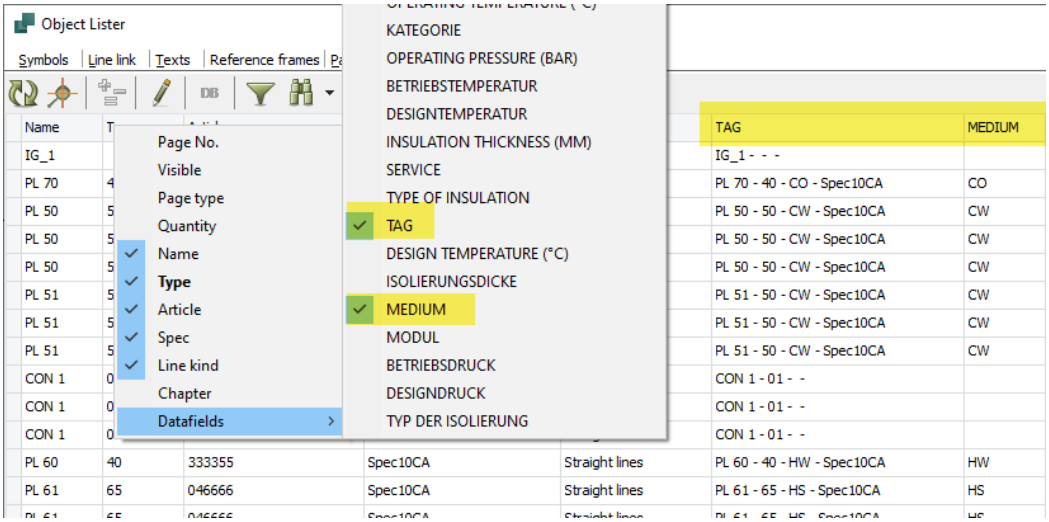
The result is an automatically filled out table symbol with the information from the component symbol.
The associativity created in this way, also updates information changes in the component symbol, automatically in the table symbol.



Object lister – individual data fields display

The available settings for selecting columns in the Object Lister, now allow for the individual display of individual data fields for symbols (on the Symbols tab) and lines (on the Lines tab).





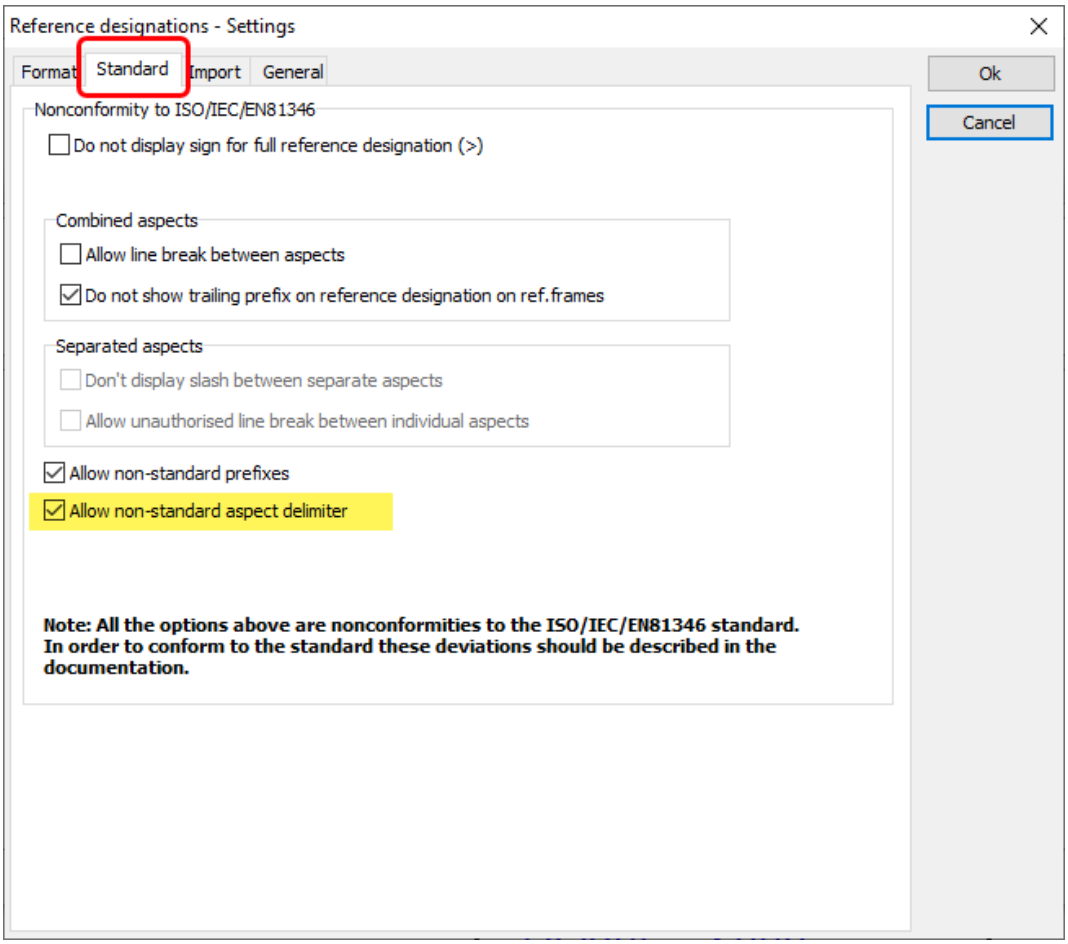
New NON-standard conform aspect delimiter

To obtain a specific representation of symbol names/TAG numbers, when using reference designations and their aspects, in the P&ID diagram and on lists, we have introduced **two new non-standard aspect delimiters**.

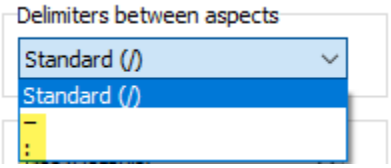
Until now, only the aspect delimiter **/** (slash) allowed by **ISO/IEC/EN 81346** was available.

The **Reference designation - Settings** have been changed to allow for the activation and adjustment of **non-standard aspect delimiters**.

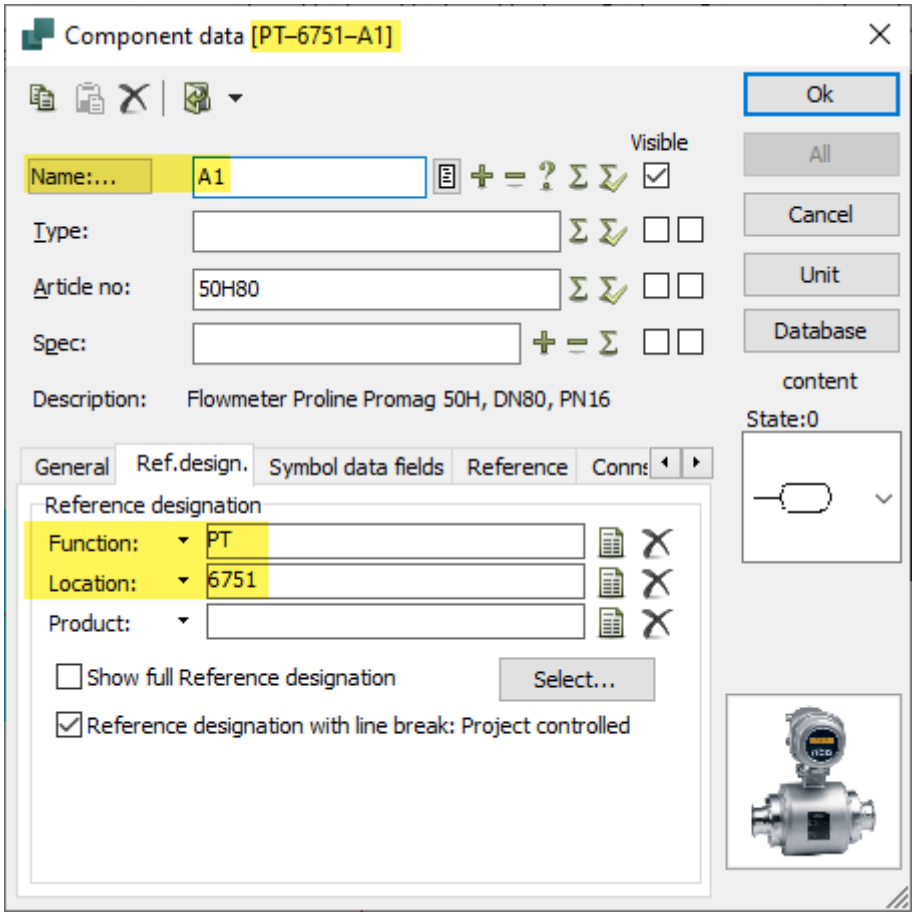
Their use must be activated in the Standard tab.



In the **Format** tab, a non-standard selection can then be made.
In addition to the Standard (**/**) selection there are now a **-** (hyphen) and **:** (colon).



When displayed in diagrams or lists, the aspects assigned to a symbol are then displayed with the corresponding separator.

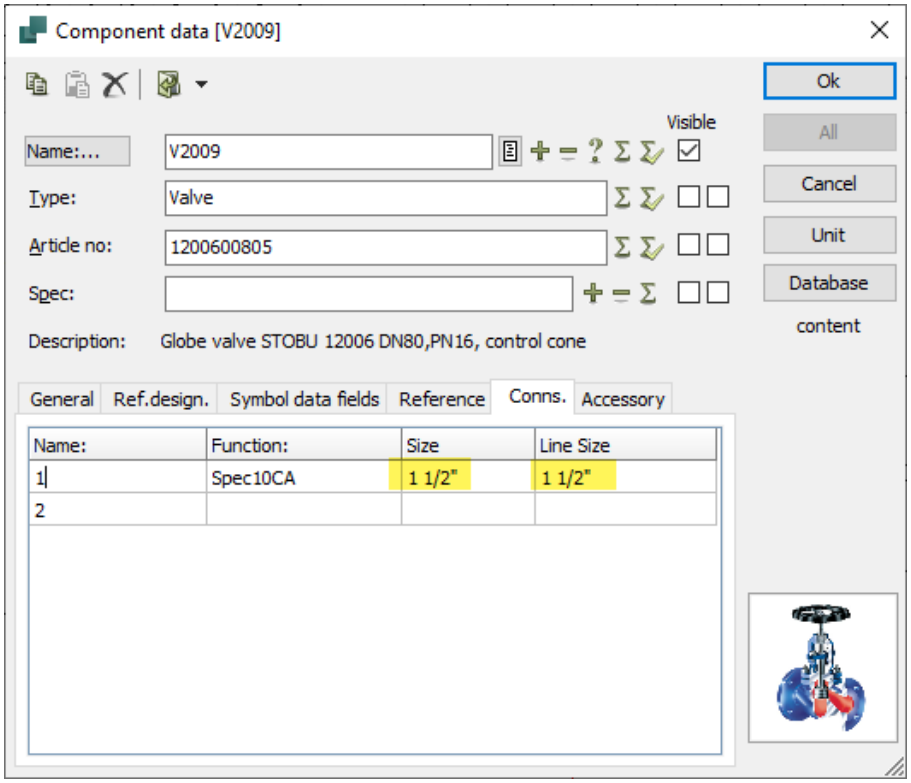


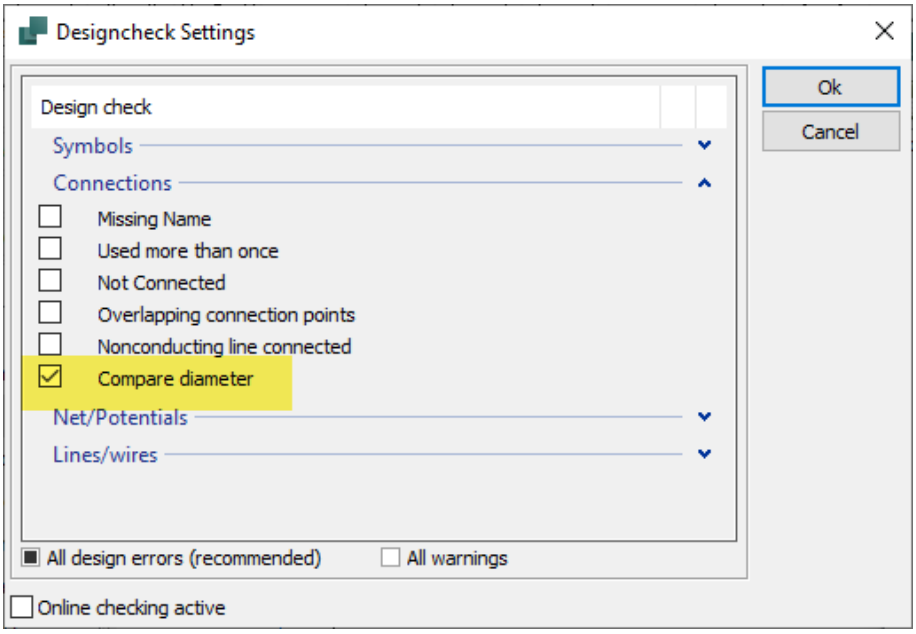
Row	Comp. Name
1	PT–6751–A1

Compare diameter supports fractional numbers

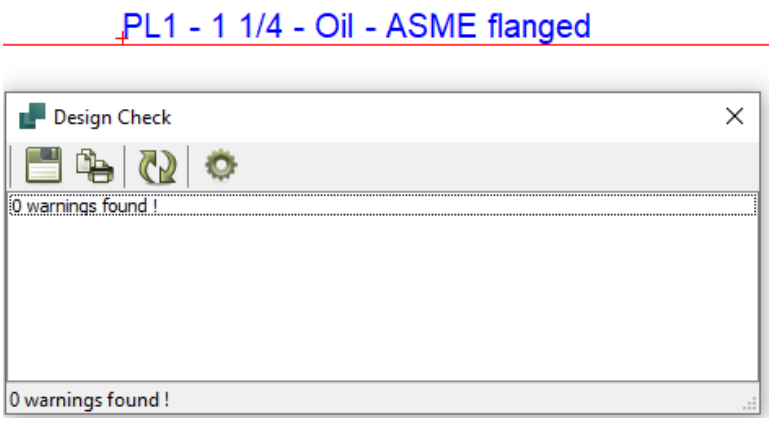
Up to P&ID 2020, the design check **Compare diameter** always displayed existing inch values and the **"** (inch character) used for size and line sizes (e.g., 3/4" or 1 1/2") with the message **Size and Line size must be a number**.

As of P&ID 2021, we also fully support fractions and the **"** (inch character) as components of the connection and line size in this design check.



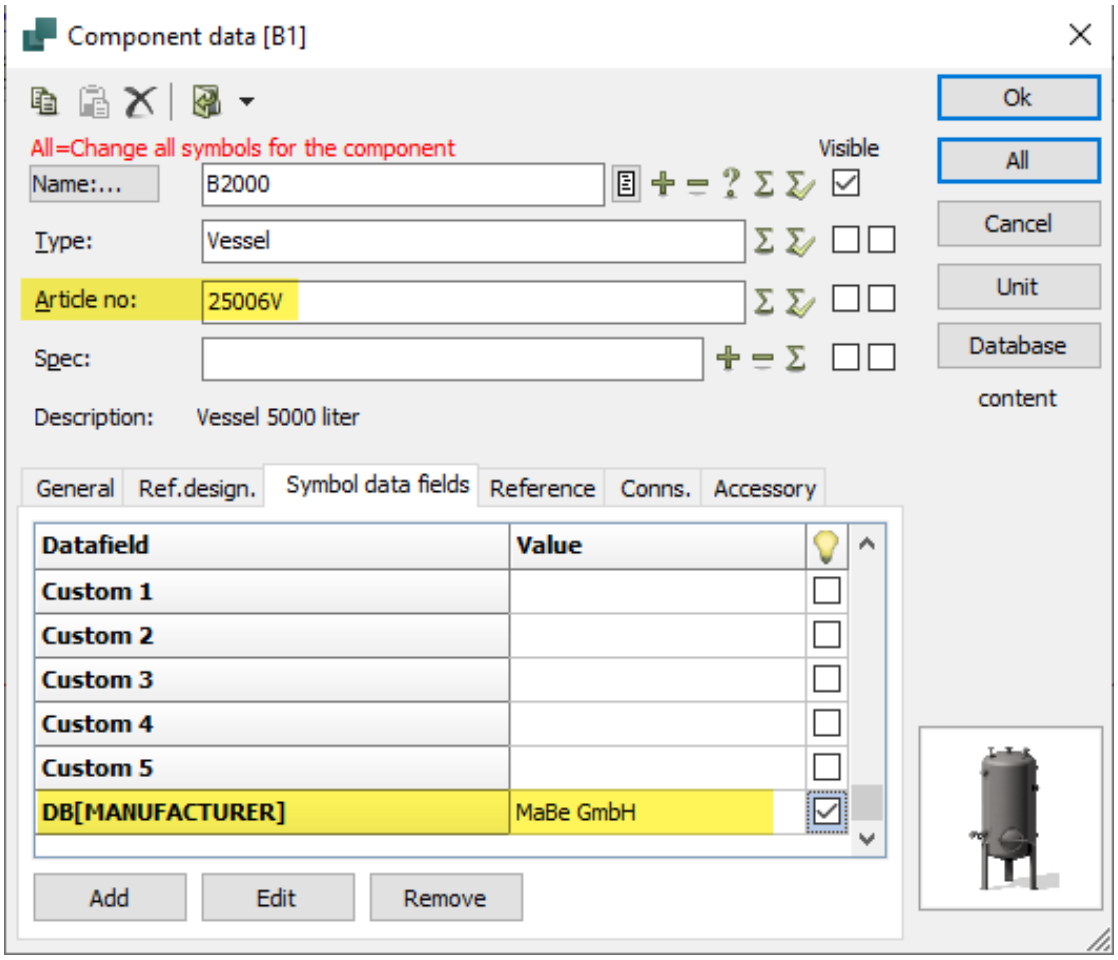


There will now only be messages if there are errors in the value match.



New option to allow writing to symbol data fields from DB

P&ID 2020 introduced write protection for symbol data fields (**DB[xxx]**), which are linked to the database and are only filled from the database.

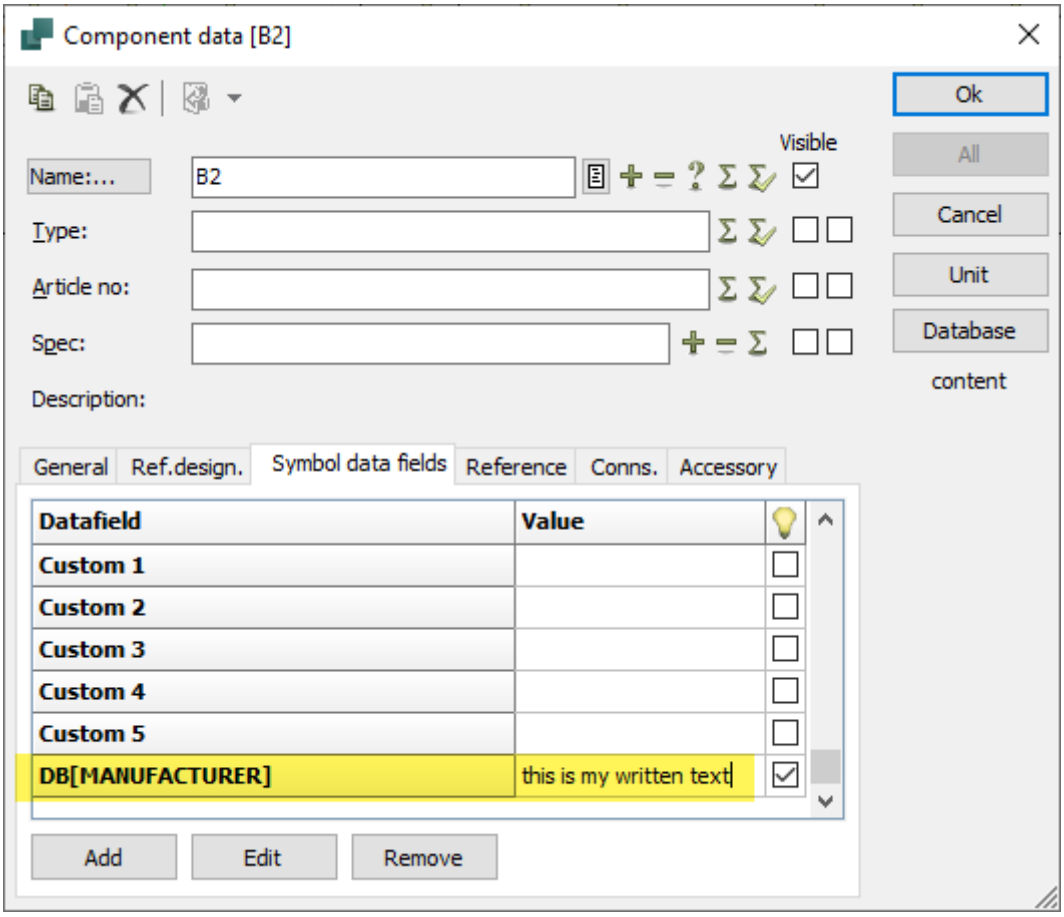


Due to customer requests, who must enter values into these fields manually before assigning article data from the database, we have created a special option to remove the default write protection on these fields.

All customers who have this special requirement can add an entry in the PCSCAD.INI used (in the Workstation folder while Smap3D P&ID is closed) from P&ID 2021 onwards to control the behavior.

In PCSCAD.INI in the area **[SystemData]** must be inserted:
DBDatafieldsReadOnly=No

If this entry is present and the value **No**, a user can (after restarting Smap3D P&ID) enter values into such data field **DB[xxx]**.



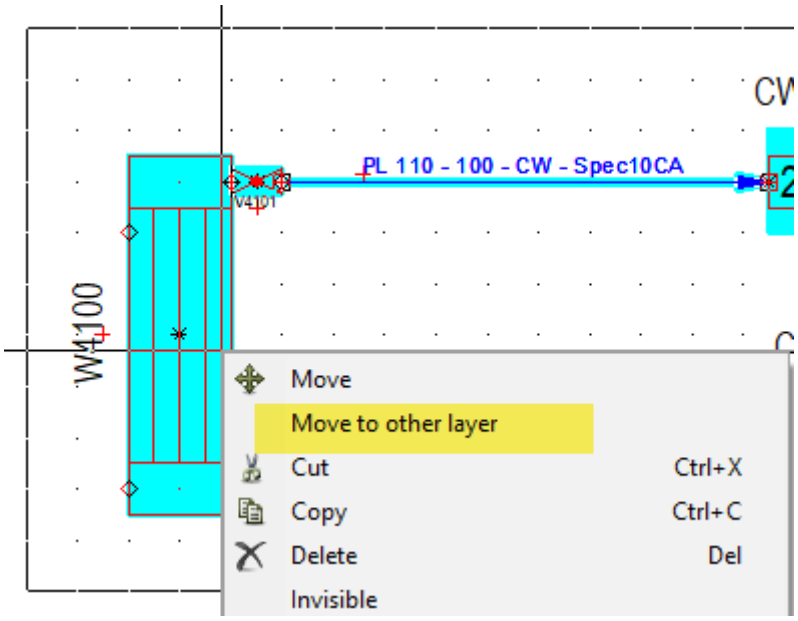
However, the basic rule for these DB[xxx] fields still applies that manually entered values are overwritten with those from the database as soon as an article assignment is made from the database.

New function "Move to other layer"

This new function now offers the user a much faster and easier way to move selected P&ID objects between layers.
Previously, such tasks always had to be performed using copy/cut and paste via the Windows clipboard.

As before, the desired objects must be selected before moving.
Basically, these can be symbols, lines or a combination of both, selected using the Area or Object Groups function.

The **Move to other layer** function is then active in the context menu.

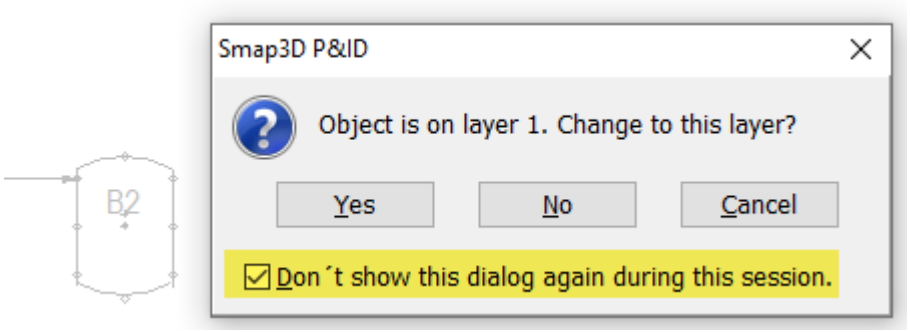


After clicking on it, the **Layer** dialog opens with the known functions.

After double-clicking on the desired other layer, it is automatically active, and the selected P&ID objects are attached to the mouse cursor. By clicking on the desired position in the diagram, the moving process is completed.

New option when switching layer with ALT key

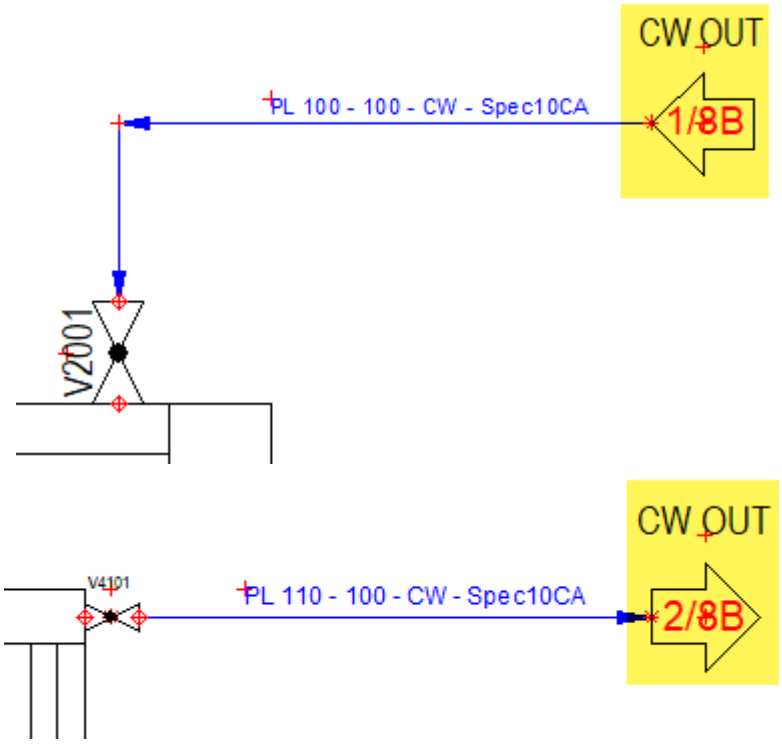
Since many versions it is already possible to switch between layers (if P&ID objects are placed on different layers) with the shortcut **ALT + click** on object.
Until now, each time you did this, a dialog was displayed in which you had to select the desired action (Yes, No or Cancel) and click.



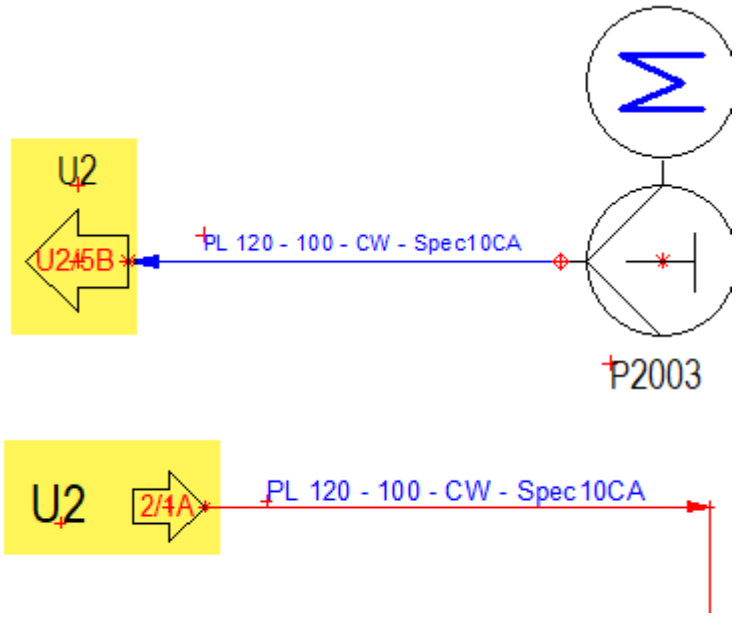
With the new option **Don't show this dialog again during this session**, you can prevent this dialog from being shown until the next restart of Smap3D P&ID.
Each time the keyboard shortcut is used, the selected action will then be performed automatically.

Line link reference to UNIT pages possible

Until now, line links (linked line/media inputs and outputs) could only be created between lines/lines on normal diagram pages.



Starting with P&ID 2021, line links can now also be created from lines on diagram pages to lines on **unit pages**.



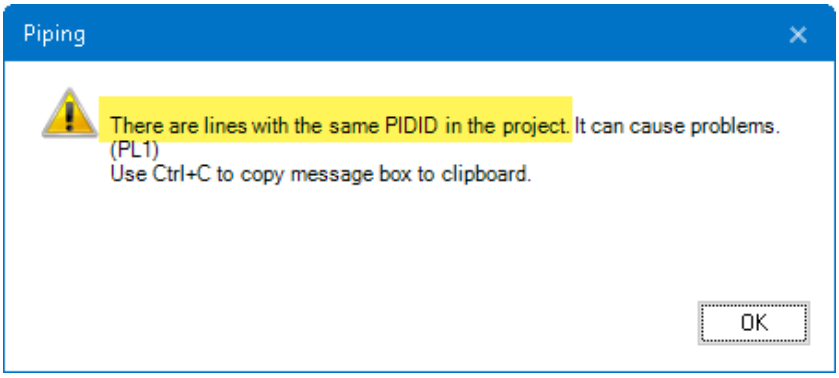
New special function Check for unique P&ID ID´s

For Smap3D P&ID to work correctly with Smap3D Piping **via the P&ID To-Do list**, there are certain requirements and rules that must be met in a P&ID project in order to load and use such a PRO file with the P&ID To-Do list.

One of these basic rules, are unique internal line identification numbers (internal PIDID) for the lines contained in the diagrams.

In previous versions of Smap3D P&ID, in some use cases and situations of P&ID creation it could happen that several lines received the same ID, i.e. **NO unique PIDID's existed**.

Users who use such PRO files in combination with Smap3D Piping, the P&ID To-Do list receives the following message when loading the PRO file **There are lines with the same PID ID's in the project**.



To solve this problem easily in the affected PRO file, we have developed the special function **Check for unique PidID's**.
This can be found in the menu **Functions > Special Functions > Check for unique PidID's**

If the P&ID To-Do list reports this problem when loading a PRO file, a user can execute this function. For all problematic lines (with non-unique ID's) the ID's are then automatically corrected.

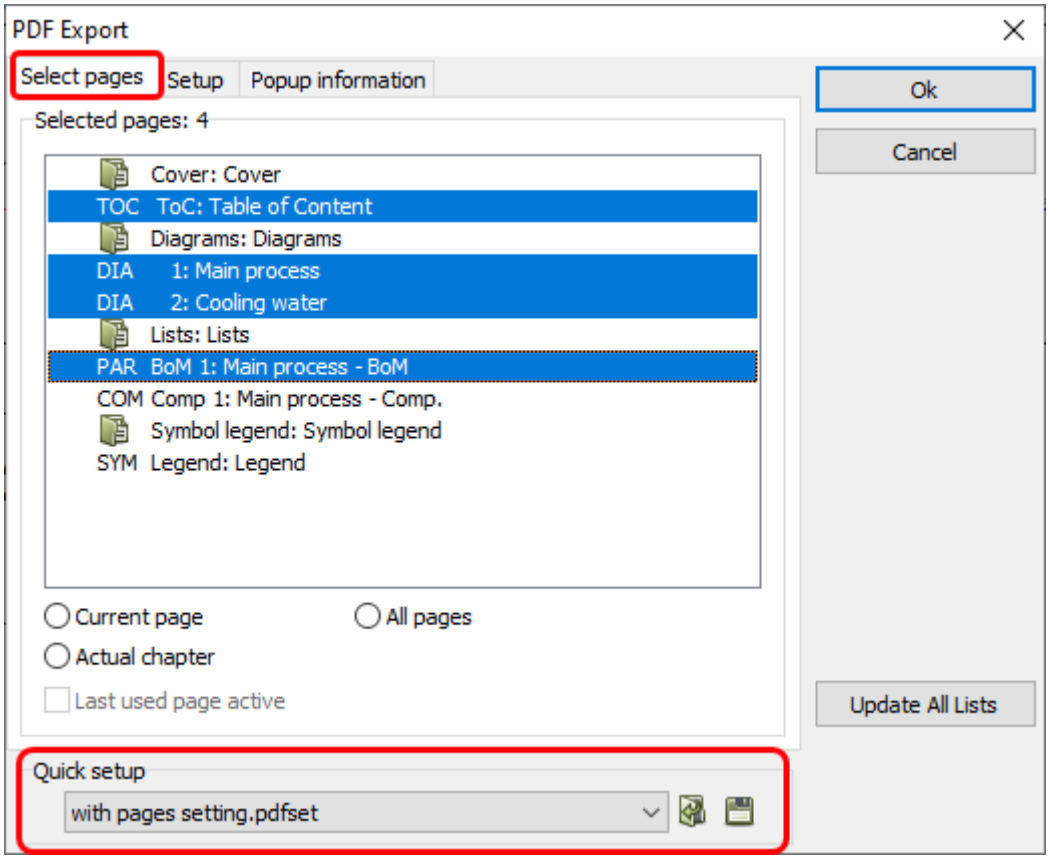
After the correction, the corresponding message appears for the user.

When subsequently reloading the corrected and saved PRO file, this error is no longer reported by the P&ID To-Do list.

Settings for Select pages are now stored in PDFSET

For the function **Export to PDF** there is since several versions the possibility of storing the associated settings for a simple re-use in configuration files with the file extension ***.pdfset**. So far in these configuration files only the attributes from the option tabs **Setup** and **Popup information** were stored.

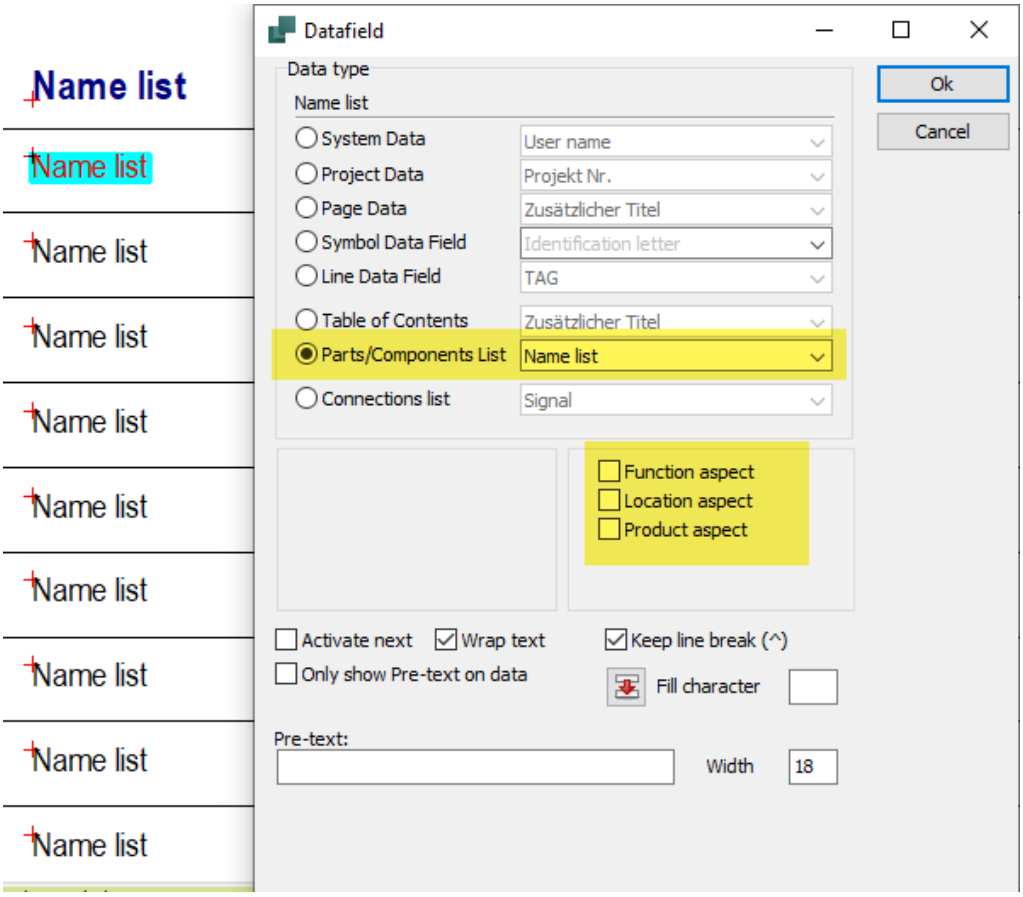
Now you can also save the settings from the **Select pages** tab in a ***.pdfset**.



Name list in Parts lists supports aspects

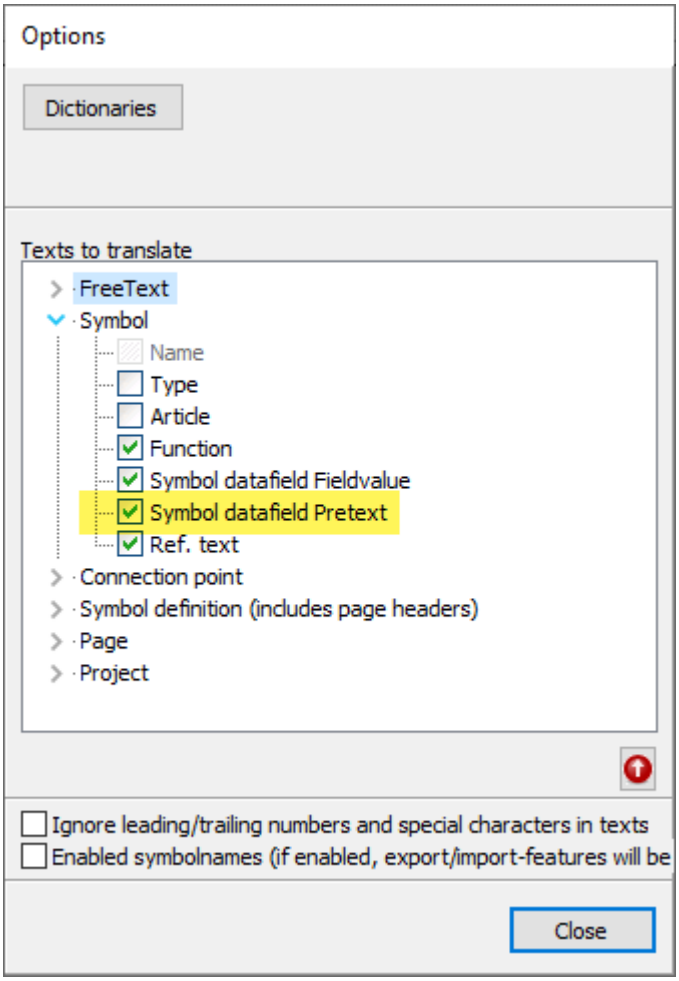
For the display of the names/TAG numbers in a parts list (evaluation according to quantity per article number), the data field **Name list** has been available for some time.

Starting with P&ID 2021, the use and display of aspects for reference designations can be configured individually and as required for this field.



Text translator supports Pretexts

In the Text translator tool, prefix texts of symbol data fields can now also be selected for translation.



Description in all list settings

The **Description** data field, newly introduced with P&ID 2020, is now available in all list settings, including the **Sort by** and **Replaces** tab.

New language Italian

With P&ID 2021, the software can also be operated with Italian language for the user interface.
In the **Select language** setting, the entry **IT** stands for **Italian**.

New language Chinese Traditional/Taiwanese

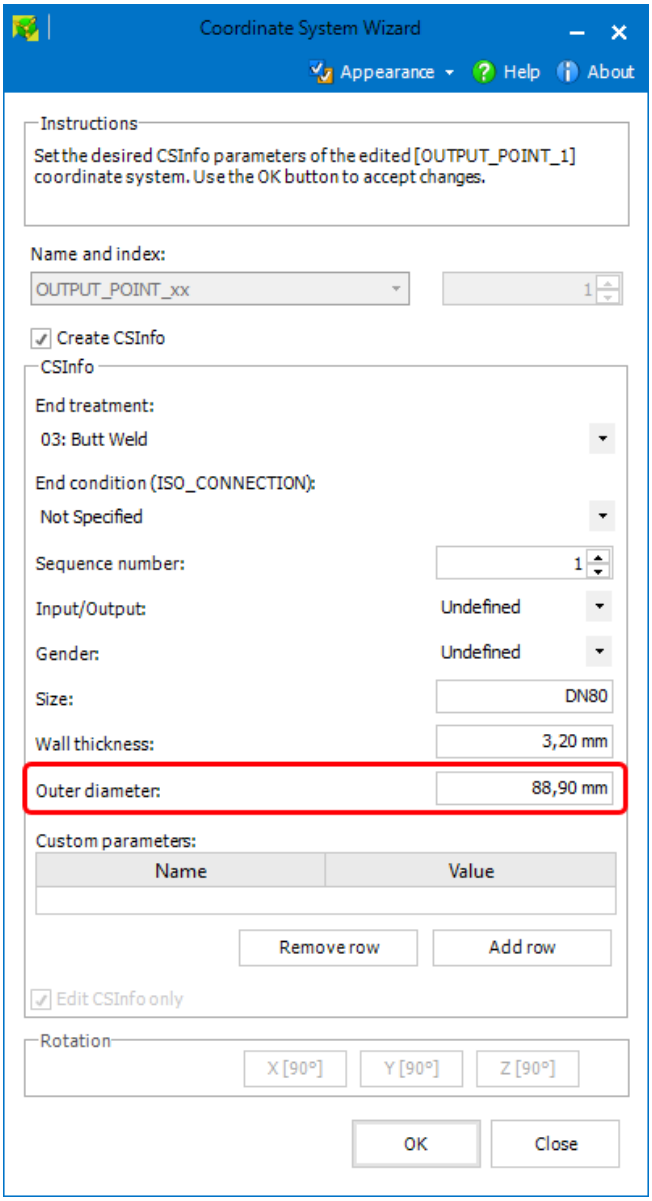
With P&ID 2021, the software can also be operated with Chinese Traditional/Taiwanese language for the user interface.
In the **Select language** setting, the entry **TW** stands for **Taiwanese** equivalent to **Chinese traditional**.

New in Coordinate System Wizard

New input field for outside diameter

In order to be able to carry out the necessary basic definitions of standard parts, for new functions and automations, the new input field for **Outer diameter** was added to the Coordinate System Wizard.

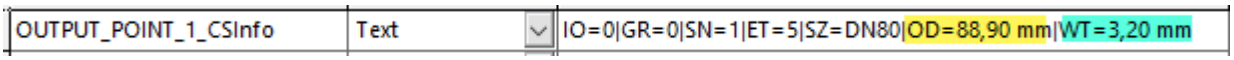
The values to be entered here for **Outer diameter** and **Wall thickness** should correspond to the dimensions of a pipe, which fits to this connection point of the standard part.



The values entered here in the CS Wizard for Outer diameter (and Wall thickness) are automatically written to the CSInfo property for the respective connection point if they are accepted correctly.

Parameter **OD** = Outer Diameter

Parameter **WT** = Wall Thickness



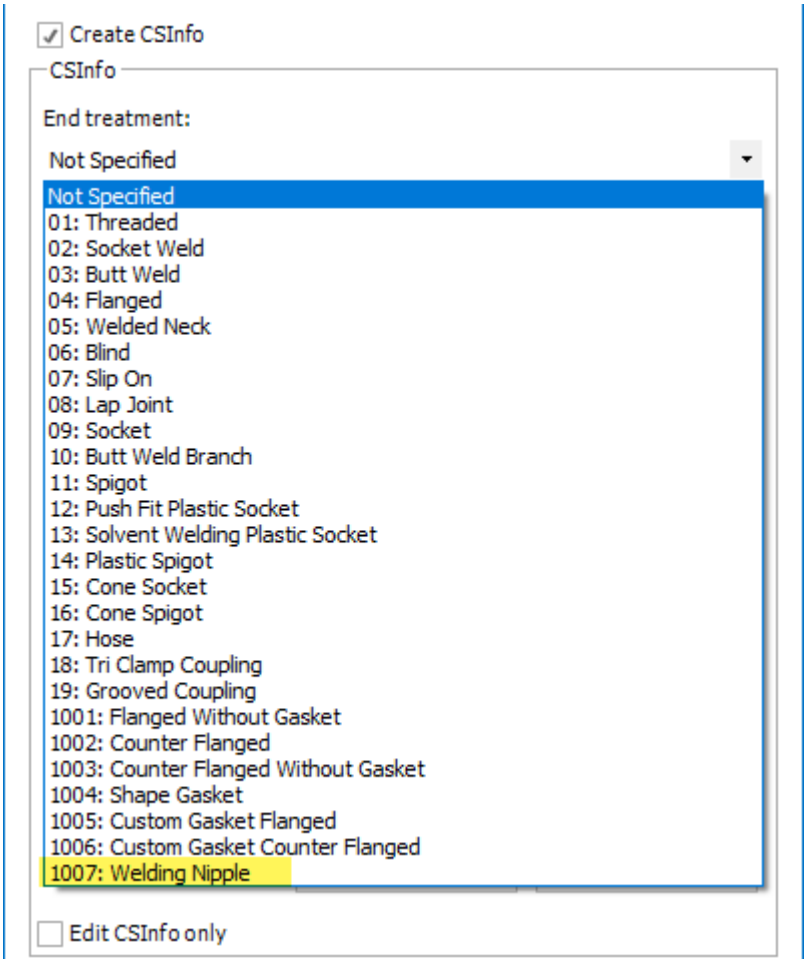
These parameters (if available) are now used for the following functions:

- In Smap3D Piping, the **Design Checks** function can now also use the outer diameter parameter in a **CSInfo check.**
- In Smap3D Isometric, available values in the parameters OD and WT are used to calculate welds between fittings.

New end treatment Welding nipple

For the new QuickPlace method Welding nipple, this new end treatment must be defined at the corresponding connection points (in the CSInfo property).

This definition **ET=1007** must be available at the branch coordinate system in the requirement component (branch dummy or similar), but also at the welding nipple component to be placed (which is defined in the pipe specification with the key name Welding nipple) necessarily at **OUTPUT_POINT_1**.



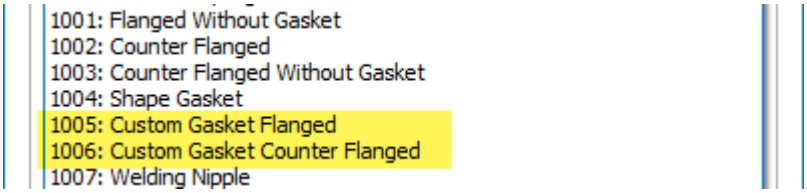
New end treatment Custom Gasket

For the new QuickPlace method Custom Gasket, one of these two new end treatments must be defined at the corresponding **connection points of a requesting component** (in the CSInfo property).

The gasket component to be placed must not have this end treatment.

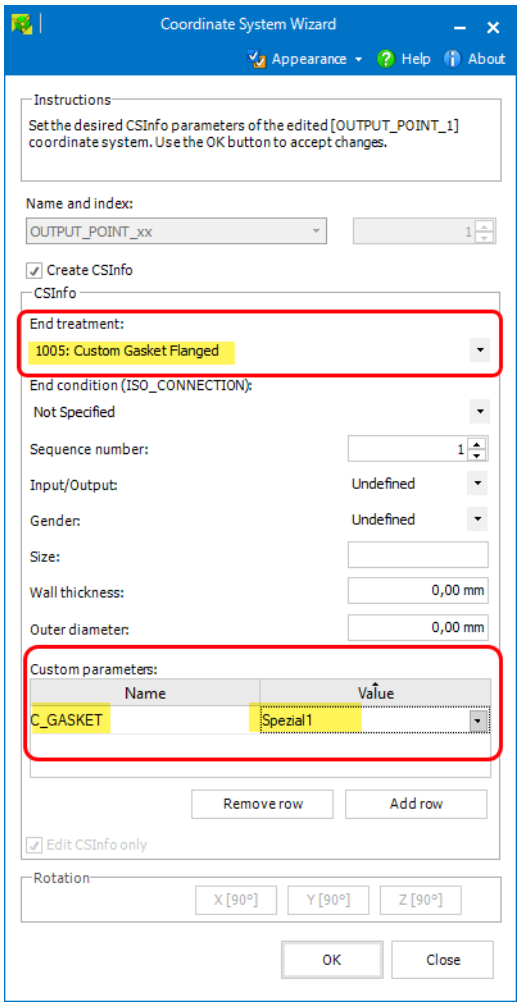
If the component defined as a **flange** from the pipe specification is to be placed together with a **custom gasket**, the end treatment **Custom Gasket Flanged (ET=1005)** must be used.

However, if the component defined as the **counter flange** is to be placed together with a **custom gasket** from the pipe spec, the end treatment **Custom Gasket Counter Flanged (ET=1006)** must be used.



For each of these two end treatments, the definition of the key name from the pipe specification to be used by the QuickPlace method must also be made.
This is done in the **Custom parameters** area with the entry **C_GASKET** in the **Name** column. The entry **C_GASKET** must be written exactly as shown below!

The desired designation (**key name**) must then be written in the **Value** column, from which the desired gasket component for this connection point is to be provided in the pipe specification editor and automatically placed by the QuickPlace method **Custom Gasket**.



These entries are then stored in the CSInfo property belonging to the selected connection point.

OUTPUT_POINT_2_CSInfo	IO=2|GR=0|SN=2|ET=1005|EC=FL|SZ=50|OD=0,00 mm|WT=0,00 mm|C_GASKET=Spezial1

The value (**key name**) entered for the parameter **C_GASKET** can be freely defined.
But **ONLY letters and numbers** may be used, **NO special characters**!!!

This value must then be **written exactly the same** as the key name in the pipe specification editor!

New in Smap3D Piping

New QuickPlace method Custom Gasket

For the automated use of different gaskets with Smap3D Piping, we have specially developed the new QuickPlace method (placement automation) **Custom Gasket**.

Until now, only **one gasket** per diameter could be defined for piping in a pipe specification for simultaneous automatic use (favorite).
In practice, however, there are situations where special components (valves, instruments, etc.) within a pipe specification and diameter require a different gasket for assembly and correct functioning than that required for other fittings of the same pipe spec and same diameter.

The existing **QuickPlace method Gasket has not changed** in this context.

This new method is additional and should make it possible to automate extended customer requirements relating to gaskets in pipeline planning.
However, since we do not know all the practical variations on gaskets for all customers around the globe and for all available fitting standards and industries, the concept and implementation of this QuickPlace Method **Custom Gasket** has been implemented as **flexible** and **adaptable** as possible.

The reasons for using one or more custom gaskets in a pipe specification (per diameter) can be:

- the use of **different 3D gasket models** (**with different file properties**) is necessary for different requesting components, if necessary, additionally still with **different virtual characteristic definitions** per line and 3D model

or,

- for different requesting components the multiple use of the **same 3D gasket model** (identical 3D file with identical file properties), but with **different virtual characteristic definitions** in several lines is necessary.

For Smap3D Piping to use this new QuickPlace method for the functions **Create route**, **Edit route**, **Generate pipeline**, **Recalculate pipeline** and **Edit pipeline** and to deliver correct results, the following basic requirements must be met:

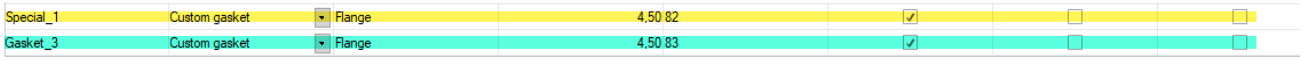
- In a pipeline to be generated/processed, 3D components must be present, which contain the definitions for the requesting of a custom gasket at their connection points (CSInfo property).

- In the pipe specification and diameter used, gasket components must be defined under the specific key names specified in the respective CSInfo property of a requesting component.

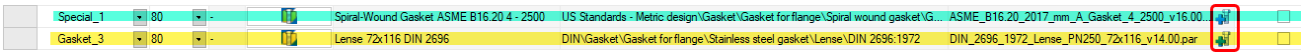
Example

The following is an example based on 2 different custom gasket definitions on one component (butterfly valve) and in one pipe specification for same diameter.

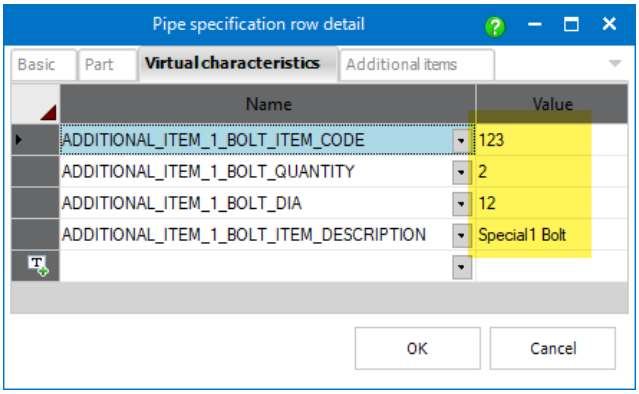
In the **common settings of the pipe specification editor** the necessary key names for the QuickPlace method Custom gasket are defined, which have been previously defined at connection points of a requesting component.

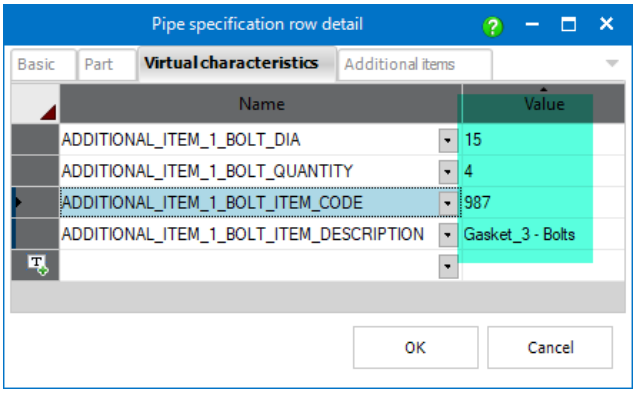


In the pipe specification, two different gasket models were defined for the same diameter for the previously defined key names.



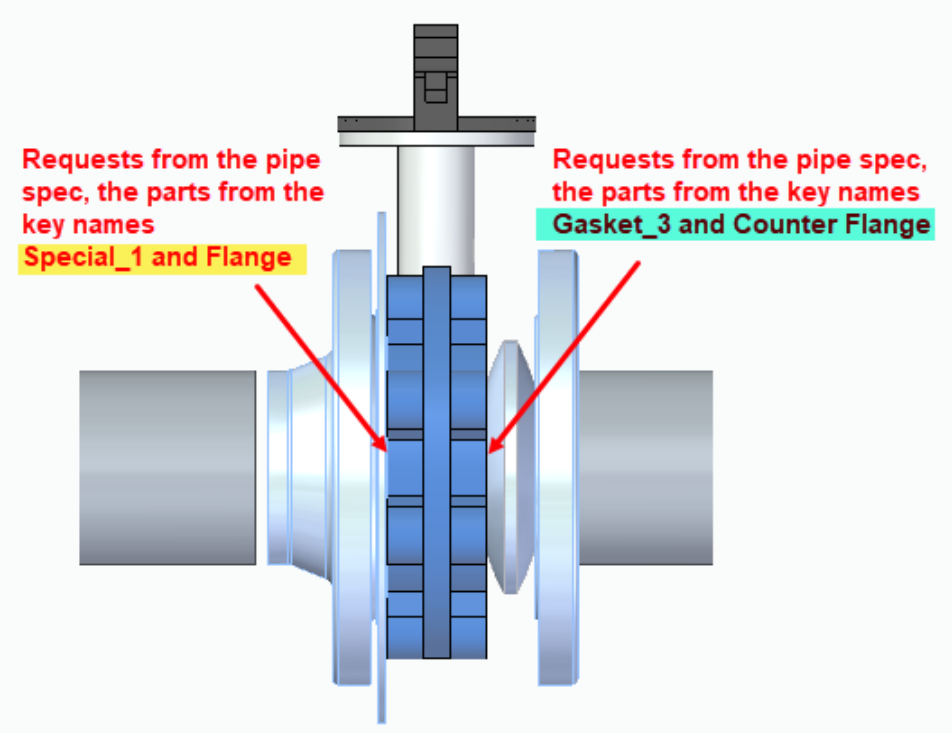
In addition, an individual set of **virtual characteristics** for **Additional Items** (additional non-graphical components) was given to each row/3D model.



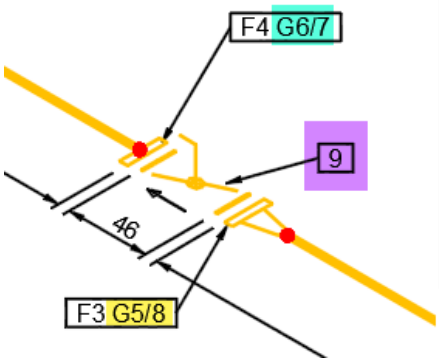


A requesting component (butterfly valve) with **custom gasket** requests, was placed on a 3D pipeline path.

When executing one of the piping functions **Create route**, **Edit route**, **Generate pipeline**, **Recalculate pipeline** and **Edit pipeline**, this request is recognized and automatically reacted to with the contents of the pipe specification.



The components (physical and virtual) contained in a 3D pipeline are recognized by Isometric and are processed and displayed accordingly during the generation of drawings (depending on the used style).



MATERIAL LIST - FAB

PT.NO	SIZE	ITEM CODE	
1	80	Rohr 88,9 x 3,2 - EN	
2	80	Bend - EN 10253-2 -	
3	80	Flange - EN 1092-1/1	
4	80	Flange DIN EN 1092-1	
5	100	Spiral-Wound Gasket	
6	80	Lense 72x116 DIN 269	
7	15	987	Gasket_3 - Bolts
8	12	123	Special1 Bolt
9	80	Butterfly Valve DN 8	

New quick place method Welding nipple

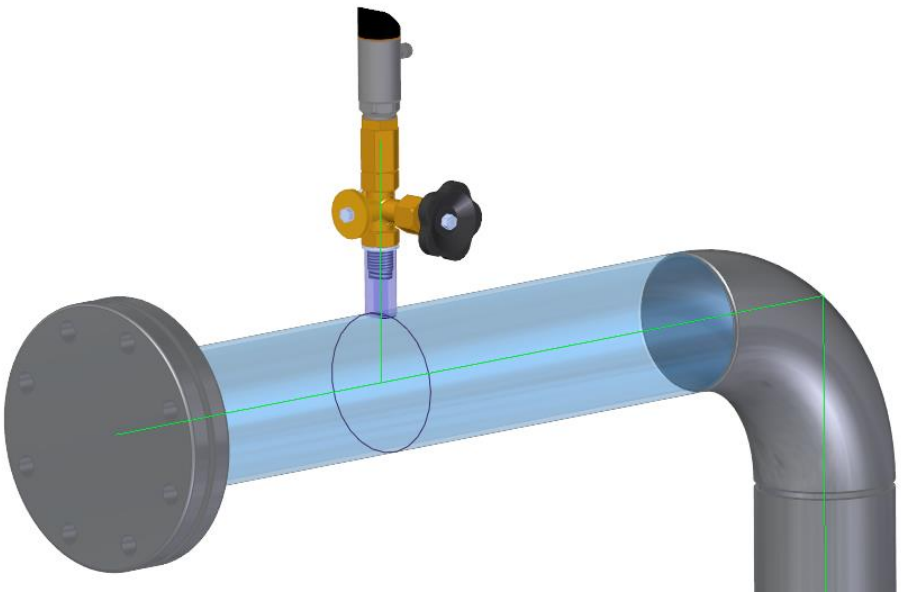
For the automated use of welding nipple components with Smap3D Piping, we have developed and added the new QuickPlace method (placement automation) **Welding nipple**.
This new automation is specially designed for all branches on pipelines where standard components such as nipples, olets, etc. are to be placed, but where **NO further pipeline generation is desired** along the drawn branch path.

Therefore, if Smap3D Piping detects a component with the end treatment ET=1007 on a pipeline path, then only a welding nipple component from the pipe specification will be placed at that point. After the corresponding connection point, however, NO further pipeline generation is carried out.

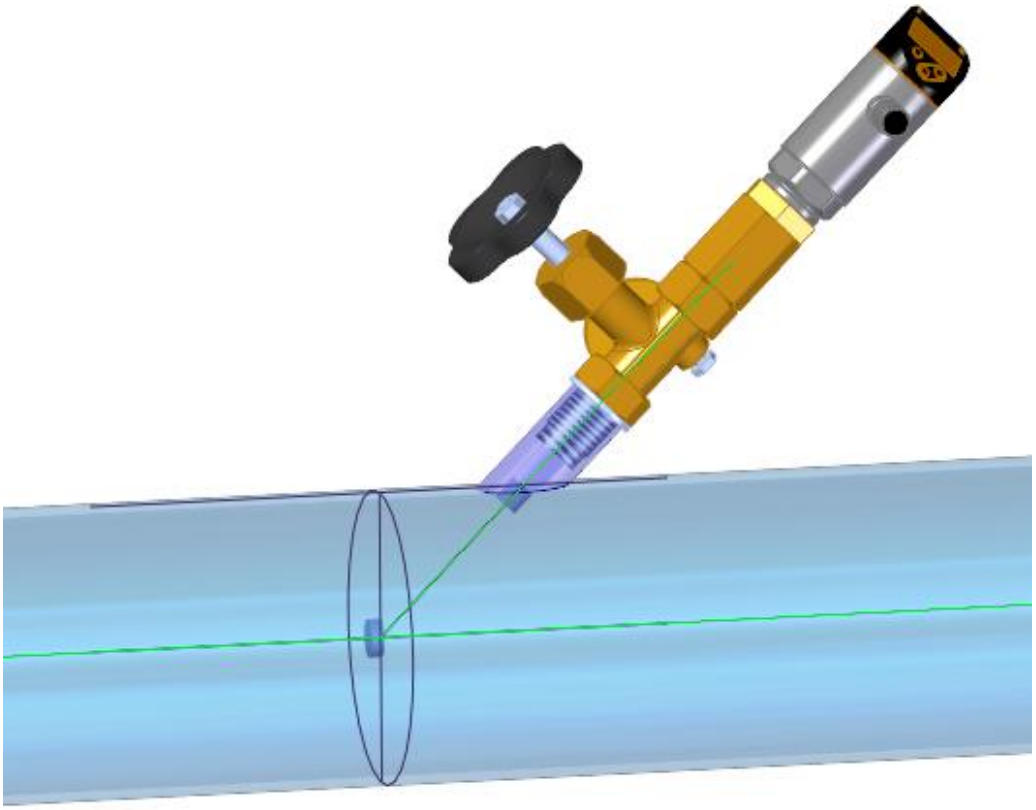
In practice this is often used for standard branches for measuring instruments, probes, drains, etc. In these cases, a standardized nipple component is usually welded onto a pipe, on which a specific component is then built directly, without the need to place any other pipes in between.

The welding nipple automation does not replace or change the Branch automation (for Collar - Weldolet parts)**.**

Rather, it is a specific extension that can be based on the QuickPlace method Branch, to additionally place standardized nipple components on branch dummies without their own part geometry, which were previously placed automatically as branch part on a branch situation.



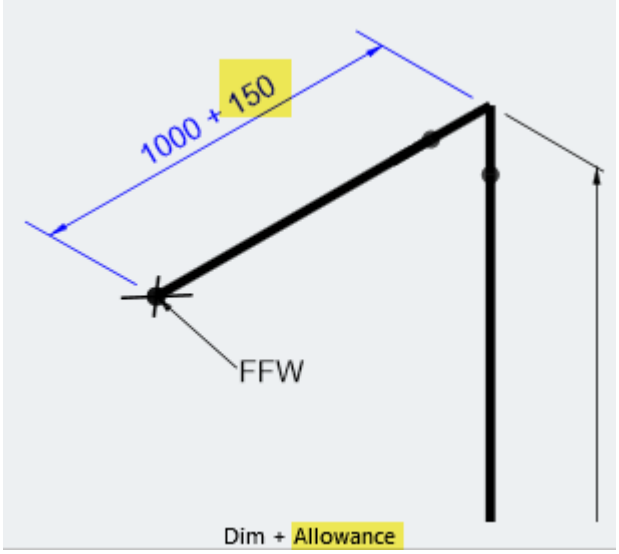
Of course, the **welding nipple** automation can also be requested from manually placed branch fittings or branch dummies, such as when working with angled branches, if these parts are prepared accordingly on the branch coordinate system.



New function Define Cut piece allowance

Basically, **CUT_PIECE_ALLOWANCE** is a parameter for Isogen© to define a value for an additional length to a pipe end of a pipeline.
This value is then displayed accordingly on a production isometric.

Smap3D Isometric and Isogen have always been able to work with this parameter and produce the corresponding results on a drawing.

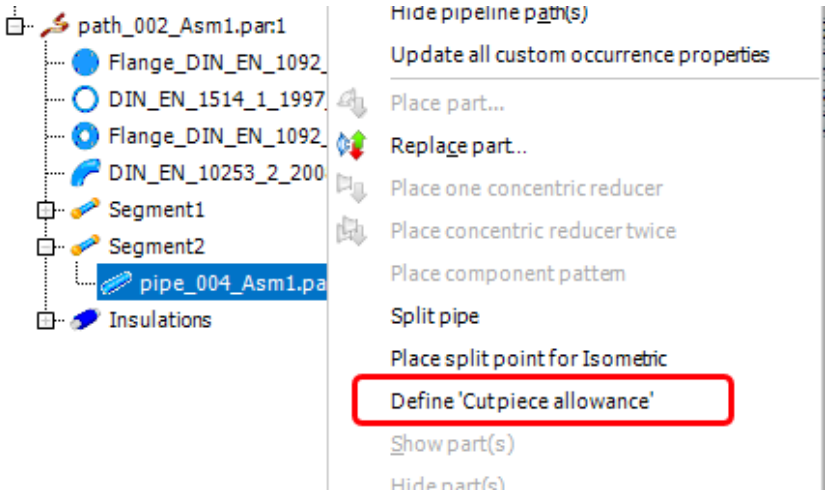


With the previous versions of Smap3D Plant Design, a definition for the **CUT_PIECE_ALLOWANCE** parameters were either written to a 3D pipe with the **VC Writer** or specified as text attributes for a pipe in Smap3D **Isometric Step 2** during the generation of a drawing.

In both cases the correct use of the necessary **pipe end number** (1 or 2) was the greatest challenge. The end number of a pipe (1 or 2), could previously only be found out by the user by reading and interpreting other pipe attributes.

The new Smap3D Piping function **Define Cut piece allowance** has now been developed to make this definition for a specific pipe end easier and more user-friendly.
The end selection on a 3D pipe is graphically displayed, and the writing of the correct parameter with its length value to the selected pipe end and its end number is automated.

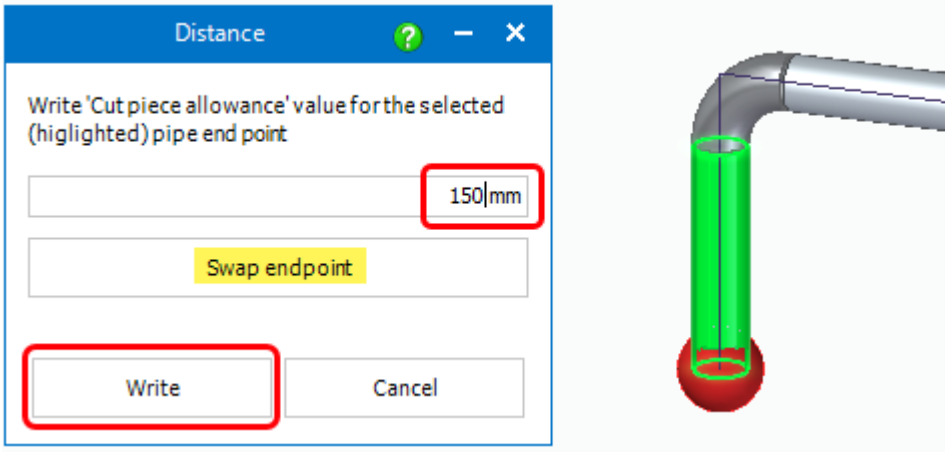
The new function is started by selecting a pipe in the Piping structure tree and clicking on the **Define Cut piece allowance** entry in the context menu.



The **Distance** dialog opens, and a red ball is displayed at one end of the selected pipe.

Use the button **Swap end point** to toggle the selection of the pipe end.
In the line **Write ...** the desired value for the **CUT_PIECE_ALLOWANCE** parameter at the selected pipe end must be entered.

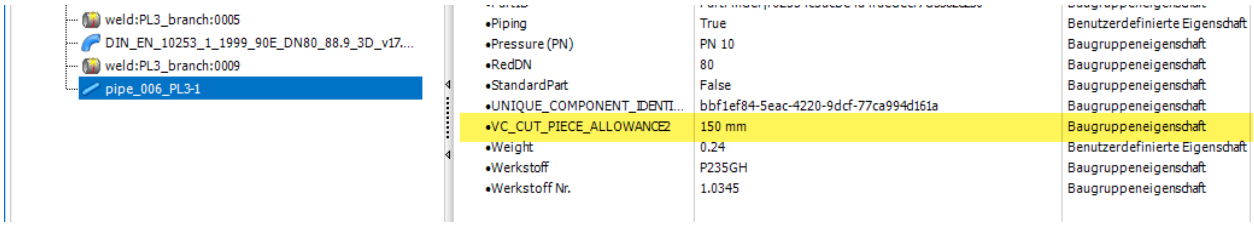
With the **Write** button, the parameter with value is written to the pipe/pipe end as a virtual characteristic.



Depending on the pipe end for which this information was written, the saved virtual characteristic is called **VC_CUT_PIECE_ALLOWANCE1** or **VC_CUT_PIECE_ALLOWANCE2**.

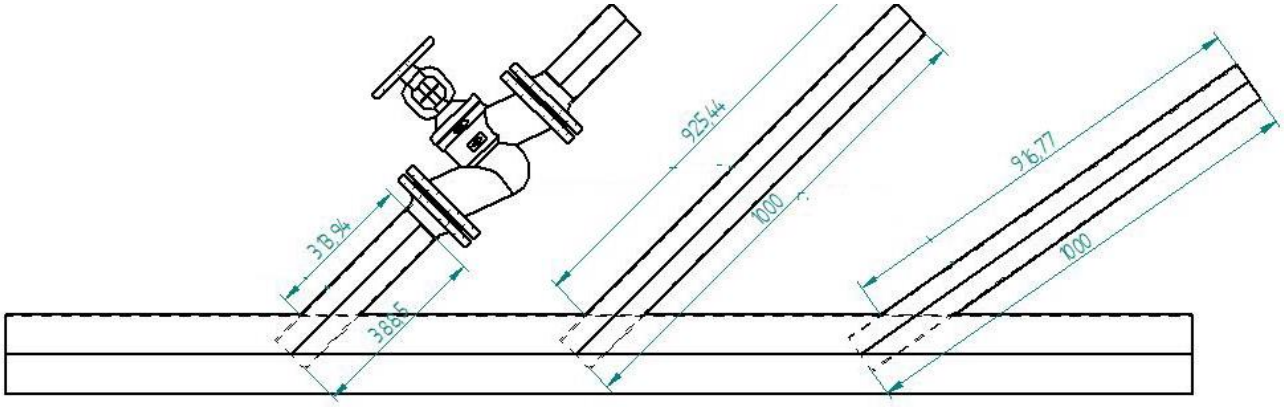
The stored value for this characteristic can be edited either by calling the **Define Cut Piece Allowance** function again for the same pipe,
or by using the **VC Writer** if the appropriate characteristic names CUT_PIECE_ALLOWANCE1 and CUT_PIECE_ALLOWANCE2 are defined in its configuration file.

In Isometric step 2, this stored information for a pipe is then displayed as follows.



Working with angled branch dummy parts

We received support requests from more and more customers asking us to recommend a procedure for the best possible realization of angled pipe branches (pipe on pipe without branch fittings) with Smap3D Piping.
The special challenge here was that every branching 3D pipe (regardless of branch angle and diameter) should be shortened on the circumference of the main pipe in order to obtain realistic cut lengths for prefabrication.



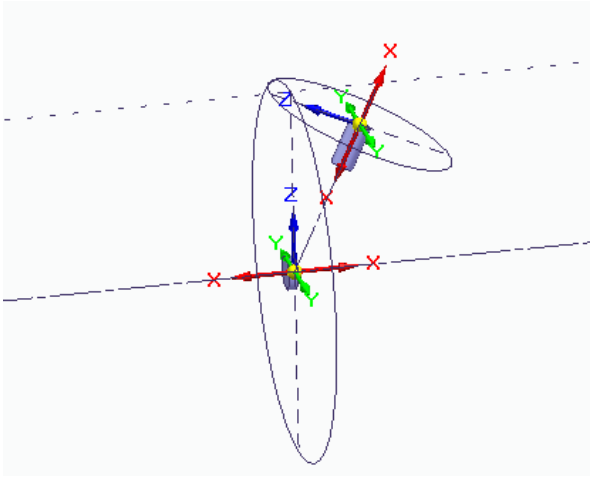
Since in such pipe-on-pipe branch situations no regular fittings can be used for dimensional control, the only way to give the piping software the necessary information for correct pipe length generation is to use placeholders (dummies).
For most customer requests, the branch situations should also be very variable with respect to the branch angle, so that it is not possible to react only to a few fixed standard angles.

Based on all these requirements, we developed a **variable, angled branch placeholder component**. We offer this CAD file free of charge to every customer with these requirements. A request to the Plant Design support is enough.

Our CAD file contains only sketch and construction geometry and no physical volume.

The following parameters are variables created by us and can be controlled by the variable table:

- Diameter of the main pipe
- Angle of the branch
- Branch pipe diameter

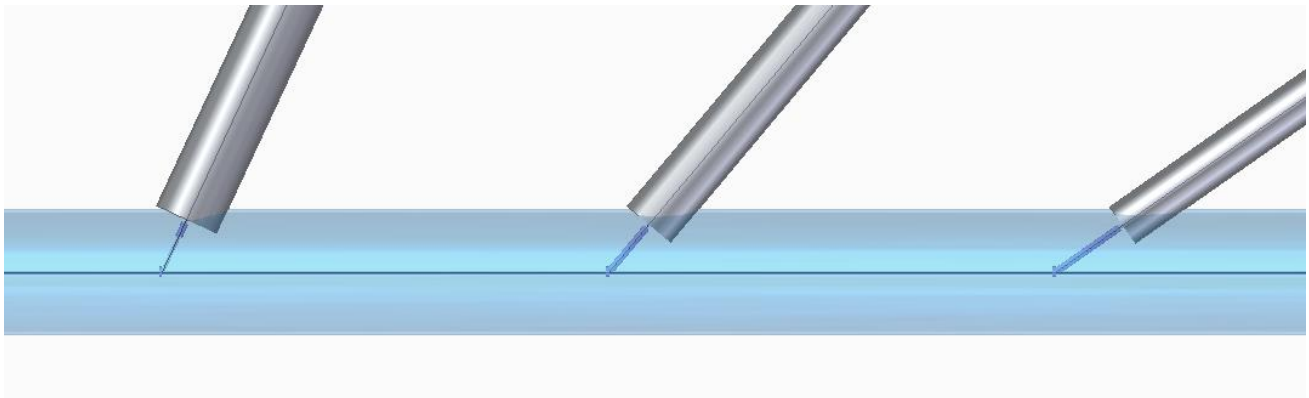


The most important contents for the Smap3D Piping software are the contained coordinate systems and their positions.
For the main pipe there is a pair of coordinate systems with information about the correct handling of a branch pipe (end treatment is ET=10), and for the branch pipe there is a pair of coordinate systems with information about where the branching pipe must end.

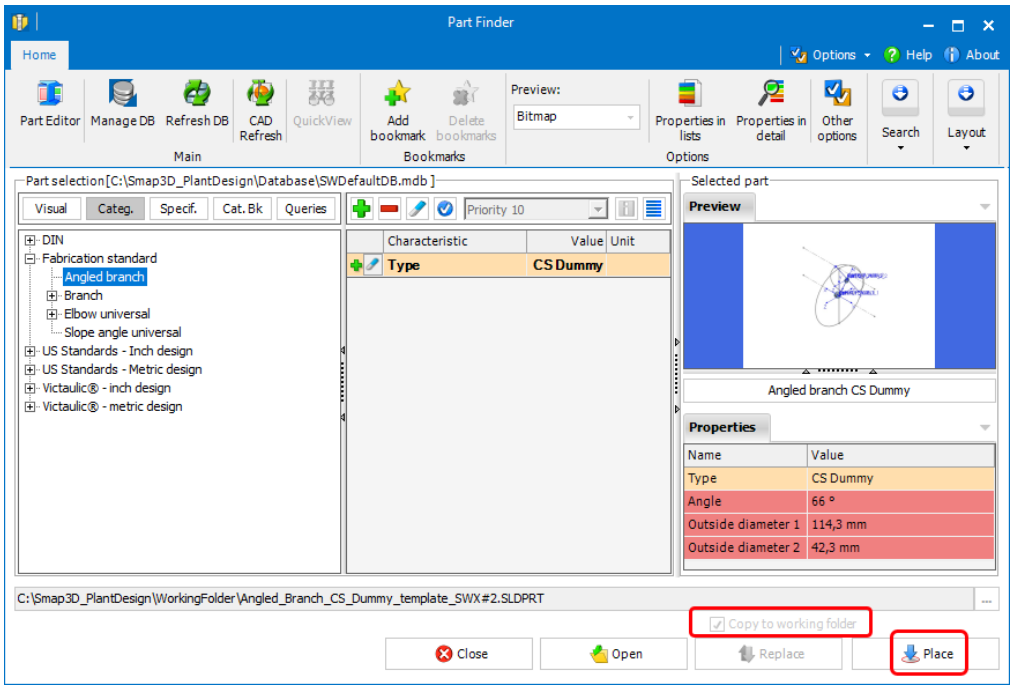
Based on our variable placeholder/dummy part, you as a customer can create any number of file copies, which you can then control according to your needs using the prepared variables for each different situation (main diameter, angle and branch diameter) in your 3D pipelines.

Smap3D Piping does **NOT offer any automation** to automatically add such a placeholder/dummy part to an angled branch situation.
A placeholder/dummy part adapted to a geometric situation must be placed **manually by the user on the pipeline paths using native CAD constrains**.

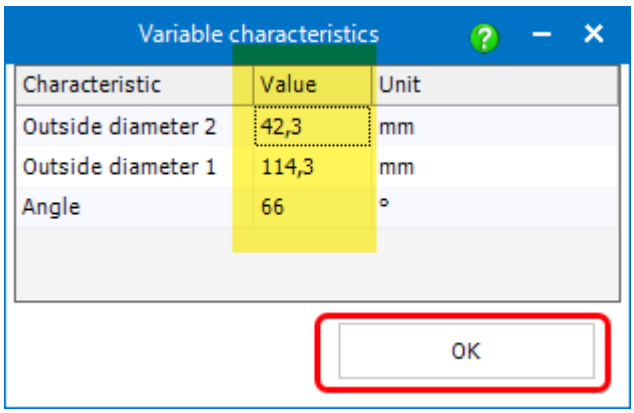
If a suitable placeholder/dummy part is placed, Smap3D Piping can automatically solve the pipe shortening at the branch.



All customers who request our placeholder part and then register it in the Part Finder database (is prepared by us accordingly), can have the necessary individual files for different geometric branching situations conveniently created by Part Finder.



With the variable characteristics prepared for the Part Finder, an independent file is created from the Part Finder during each placement operation and saved to the specified working folder.
After clicking the **Place** button, the input dialog for the desired dimensions of the new file appears.



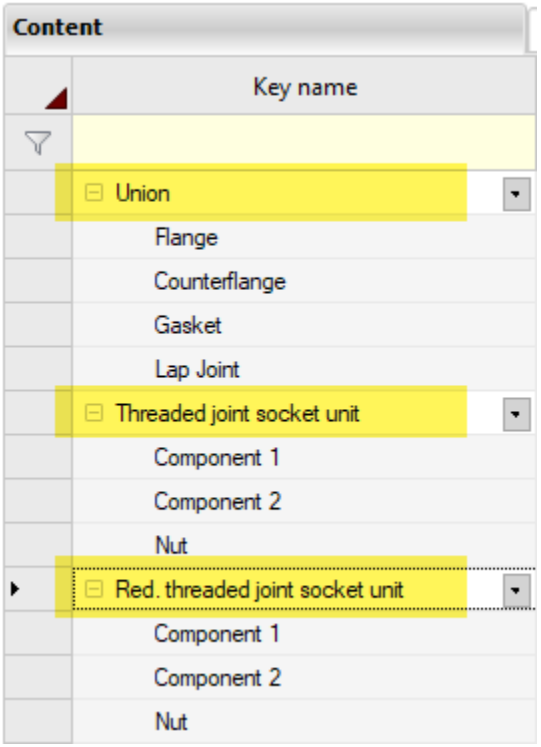
If you enter the required values for each line and click **OK**, the new placeholder file with these values is then automatically generated, saved and placed in the active assembly.
From there, the user must manually place it with native **CAD constraints** to the desired position on the pipeline paths.

Replace part function supports component packages

With the introduction of the automation for tube fittings in version 2020, there are currently three key names for which several components (component packages) can be defined in a pipe specification.
These are automatically placed on a pipeline path by the corresponding QuickPlace methods when a request is made.

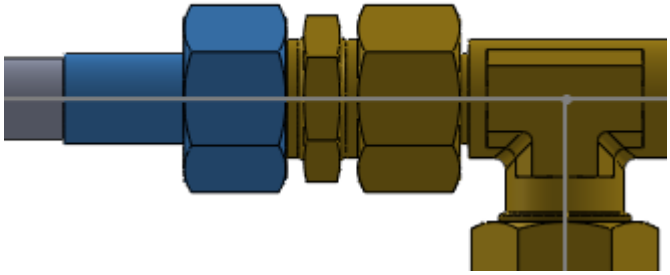
These are the key names **Union**, **Threaded joint socket unit** and **Red. threaded joint socket unit**. Up to 4 individual components can be defined for these in one pipe specification.

With the automations of the functions **Create route**, **Edit route**, **Generate pipeline**, **Recalculate pipeline** and **Edit pipeline**, the individual components defined in this way in a pipe specification are handled and processed as component packages.



The **Replace Part** function could not detect such component packages until now and could not replace them as packages.
This limitation has now been removed in version 2021.

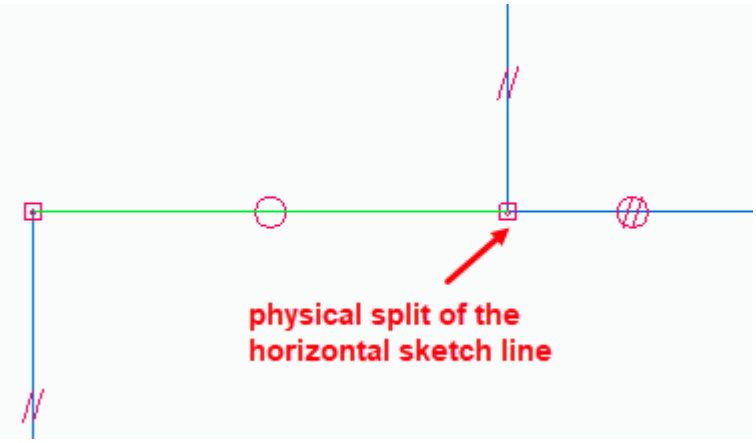
If such component packages have been placed on a pipeline path using one of the above mentioned automations and **only one part of such a component package is selected by a user** for the **Replace Part** function, Piping will automatically recognize and select the other parts of the placed component package when the **Replace Part** function is started.



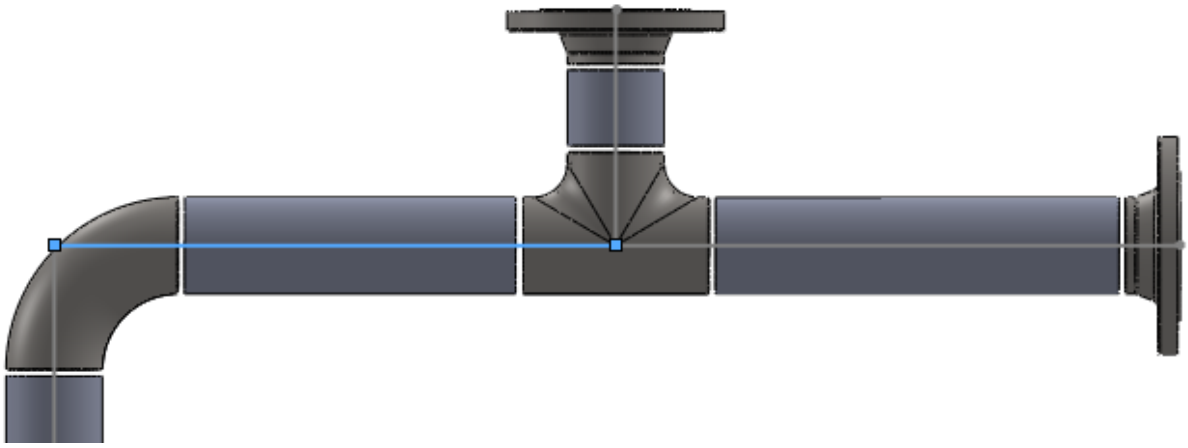
Depending on the selection made by the user, the **selected parts of the component package** are replaced with a single part or several parts of another component package.
This enhancement greatly simplifies and accelerates the replacement of component packages.

New option to automatically split a sketch line

With the introduction of the **Remove/Adapt pipe** function in Piping 2019, the requirement for a pipeline sketch to split a baseline using the **Split Line** function before drawing branching lines in the 3D sketch arose.



Only if a pipeline sketch at a branch is split, and there is a separate line element for each resulting 3D pipe (to the left and right of the branch) after the pipeline has been generated, the **Remove/Adapt pipe** function can work correctly and provide meaningful results for a 3D pipe.

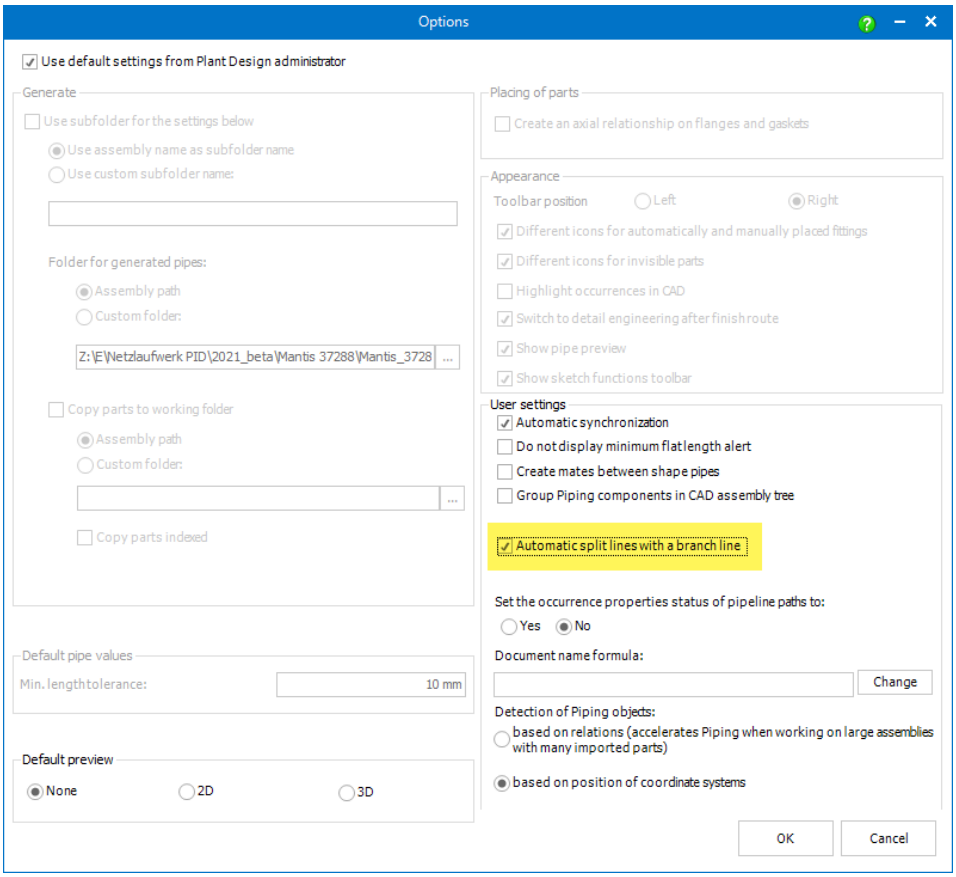


Unfortunately, this basic requirement was not always considered, and users have used the **Remove/Adapt pipe** function on their pipelines with non-split sketch lines for such situations. This led either to error messages (if the pipeline sketch was fully defined by dimensions) or, if there were enough degrees of freedom in the 3D sketch (according to the programmed rules), to completely undesired changes in the 3D piping.

To better support users in meeting this basic requirement when creating sketches, the new Piping option **Automatic split lines with a branch line** has been developed.

This option is in the Piping **Options** dialog, in the **User settings** area.

By default, this option is **not active** (no check mark) but can be activated by any user as required.



If this option is active, it works as follows:

- the 3D sketch environment for a **new 3D sketch** is started by the **Create Route** function,

so, when closing / exiting the 3D sketch environment, all lines in the active 3D sketch feature are checked to ensure that **they are only connected at line end points**.
If this is not the case and there are line elements also connected somewhere along a continuous line (relation endpoint to line), the baseline is automatically split at this point (as with **split line**), and then only endpoint to endpoint connections are made between the individual physical lines.

This automation works in Solid Edge only with the above-mentioned **function of Smap3D Piping**.

If the 3D sketch environment is activated with native Solid Edge functions, this automation is NOT executed!

Material from template is used for pipeline path

For the generation of pipeline path files, a template file can be set in the Plant Design Administrator.
Previously, this template was only used as a supplier of **specific file properties** for the generated pipeline path files.

Starting with version 2021 it is now also possible to **define a material** in the template file (using the Solid Edge material library).
When generating new pipeline path files, not only the prepared file properties are then transferred to the individual pipeline path, but also the existing material definitions (density, face coloring, etc.).

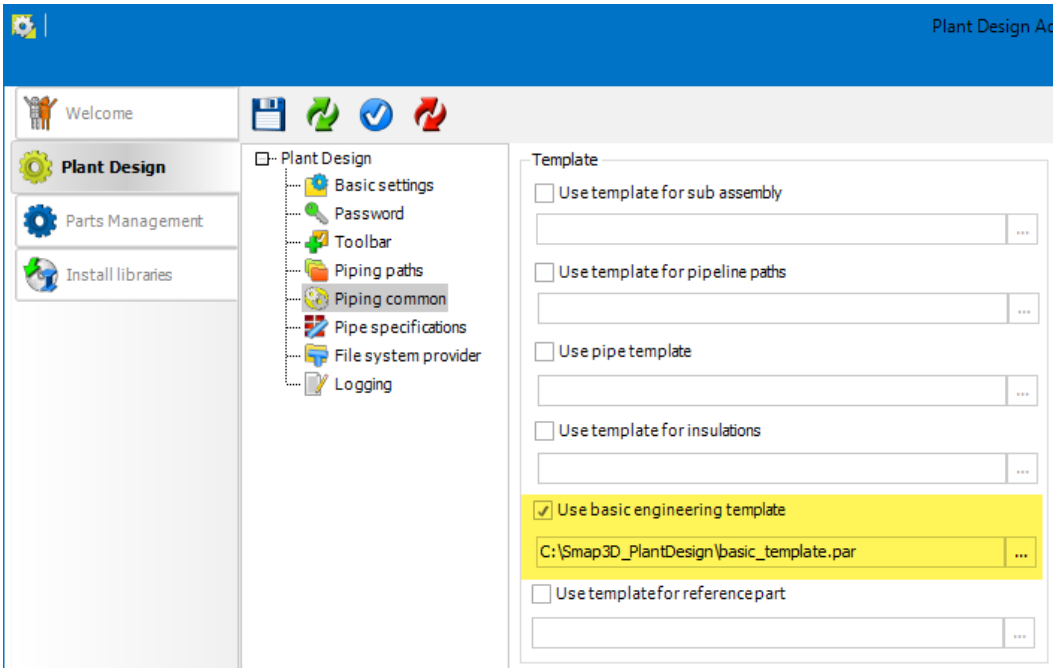
This new option is useful for all customers who use the pipeline path files after generating a pipeline and after additional manual adjustments, for example to determine the internal volume of a pipeline.

File properties and material from template is used for basic engineering tubes

Since the introduction of the Basic Engineering editing mode (version 2019), more and more customers have been using this mode in the design and pre-planning of their piping systems, thus taking advantage of the corresponding performance benefits.
Only at a later point in time (days or weeks) are these pipelines then switched over to detailed pipeline generation.

In this context, several expansion requests arose in order to be able to provide more information to the basic engineering pipes than just the coloring (via pipe specification) as was previously possible.

For this purpose, the possibility of using a **template** for **basic engineering pipes** is implemented. This template, like the other templates, can now be set in the **Plant Design Administrator > Piping common** in the **Template** section.



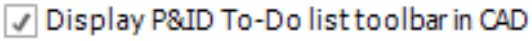
If you have certain file properties and material definitions that should always be written to the individual pipe files when generating the basic engineering pipes, you can now define them in the set template.

New option to hide the toolbar of P&ID To-Do list

Since the **full integration** of the programs **Piping** and **P&ID To-Do list** into the user interfaces of Solid Edge, the toolbar of the P&ID To-Do list and its application windows have always been displayed automatically with the toolbar installation of Piping in the Solid Edge interface. However, customers who do not use the P&ID application did not want this toolbar and window to appear in the Solid Edge interface.

To make this easy and comfortable, the option **Display P&ID To-Do list toolbar in CAD** has been added to the **Toolbar** tab in the **Plant Design Administrator**.

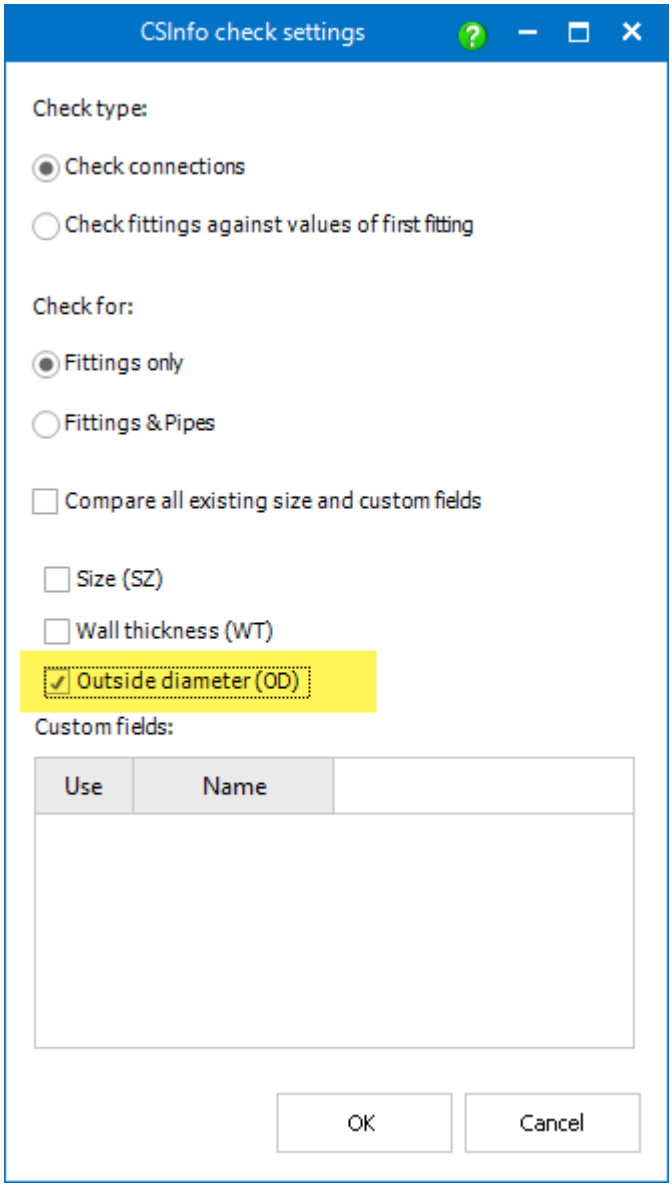
By default, this is activated (with a check mark) and thus has the same behavior as before.



To remove the P&ID To-Do List toolbar and associated window from the Solid Edge user interface, this option must be disabled (no check mark) and saved in the Plant Design Administrator.
Solid Edge must be closed.

Piping Design check allows checking for parameter OD

The parameter Outside Diameter (OD), newly available in the Coordinate System Wizard, was introduced primarily for the extended evaluation of weld seam attributes in Isometric.
However, to ensure a uniform working method, this new parameter OD was also made available in the Piping Design check **CSInfo**.



New option to set occurrence property Higher Level to No

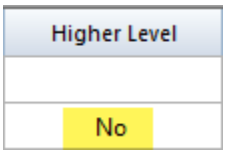
In addition to the settings that have long been available in the Piping Options to control the **Solid Edge occurrence properties** (**Reports/Parts Lists**, **Drawing Views**, **Physical Properties** and **Interference Analysis**) for pipeline path files, there was a special customer requirement to also be able to control the definition of the **Higher Level** occurrence property through Piping.

To enable this customer requirement, but not to overload (complicate) the piping user interface and its setting options, the necessary option was developed as **not visible in the user interface**.

All customers for whom the Solid Edge occurrence property **Higher Level** should automatically be set to **No** for newly generated pipeline path files, the following entries must be added to the **PDGlobalSettings.pdsx** configuration file (with closed applications and with an external editor program), absolutely exactly as written here:

```
<OtherSettings>
 <Guid>dcc4481e-92c2-4001-b251-b2ea9e80fc32</Guid>
 <Key>VisiblePipelinepathInHigherLevel</Key>
  <Value>False</Value>
</OtherSettings>
```

If after a restart of the Solid Edge and Piping application this entry is present in the configuration file, the **Higher Level** occurrence property is automatically set to **No** for all **newly generated pipeline path files**.



New in Pipe specification editor

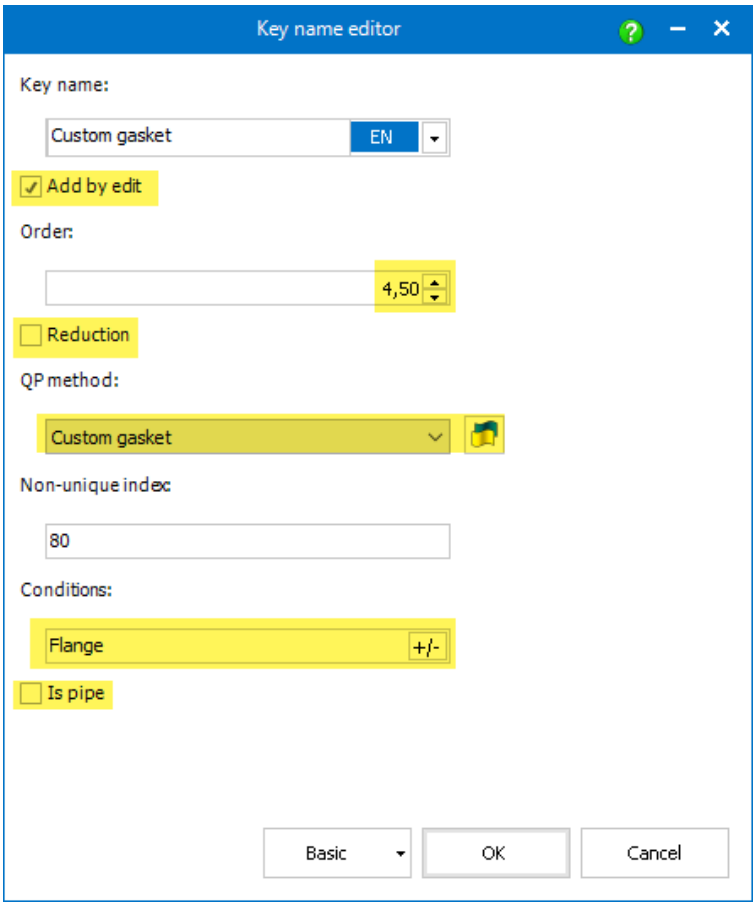
New key name Custom Gasket

For the automated use of individual gaskets on specific components with Smap3D Piping, we have specially developed and added the new QuickPlace method **Custom gasket**.

With version 2021, the new key name **Custom gasket** with corresponding QuickPlace method Custom gasket is available in the pipe specification editor.
This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade (from PD 2020 to PD 2021 - Installation option Existing data).

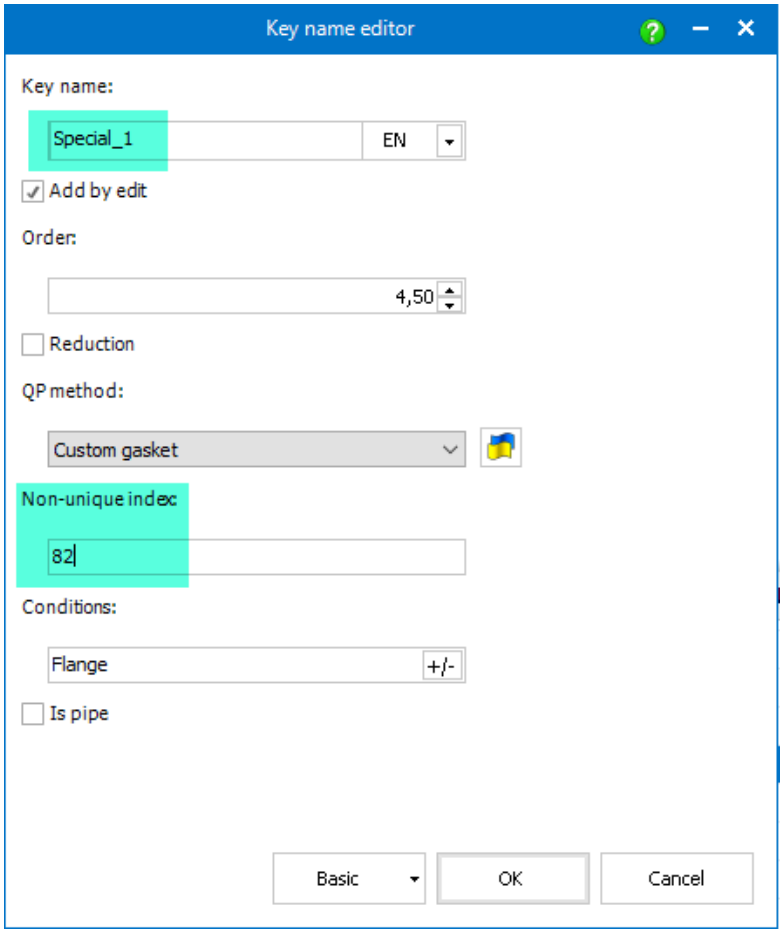
Unlike all other key names we provide, this key name **Custom gasket** is intended only as an example of a custom key name to be created, according to the user-specific definitions in the CSInfo property.

The direct use of Custom gasket in a pipe specification is not intended and will not lead to meaningful results as there are no components with these connection point definitions.



Rather, the settings defined by us should serve as a template for the creation of customer-specific key names (special for QP Custom gasket) in the **Pipe Specification Editor - Common settings** dialog.

According to the QuickPlace method example described, the new key name definition should look like this:



Only the definition of the **Key name** and the **Non-unique index** are different from the example **Custom gasket** prepared by us.

If Smap3D Piping is to use several different key names at the same time for the same automation, the **non-unique index** must be **different** for each key name.
If this were not the case (different key names would have the same non-unique index), the pipe specification editor would ask for a favorite when saving. This would negate the advantage of this new QuickPlace method.

New key name Welding nipple

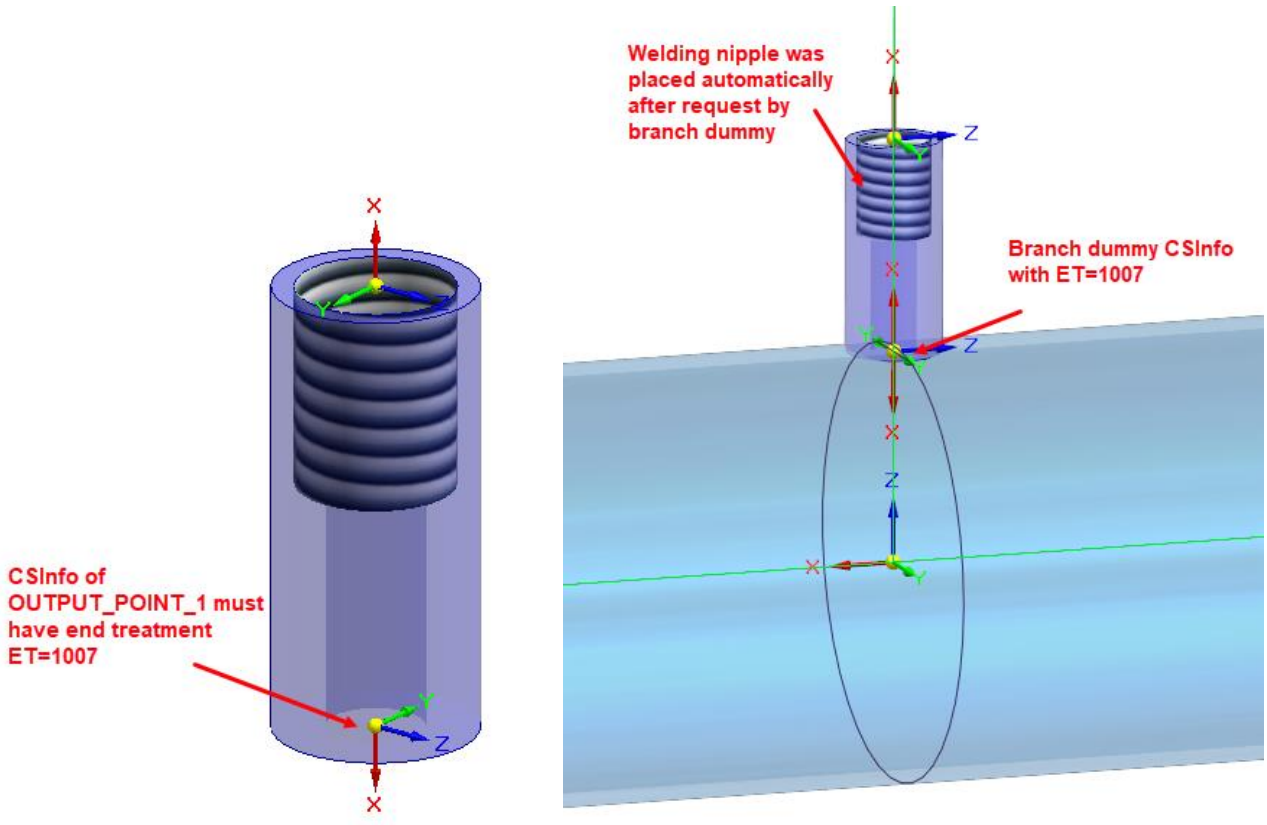
For the automated use of welding nipples on branch pipes with Smap3D Piping, we have developed and added the new QuickPlace method **Welding nipple**.

With version 2021, the new key name **Welding nipple** with corresponding QuickPlace method **Welding nipple** is available in the pipe specification editor.
This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade (from PD 2020 to PD 2021 - Installation option Existing data).

A 3D component, which is defined in a pipe specification with the key name **Welding nipple**, is always placed on an existing branch dummy or branch fitting with **ET=1007** end treatment when generating a pipeline with Smap3D Piping.

The basic requirements for a 3D component defined as a **welding nipple** in a pipe spec are the same as for a flow-through component. (For details see the software help in the chapter **Piping > Components for Piping**).

In contrast to a normal flow-through component it is necessary that both, in the requesting component (in branch dummy or branch fitting) and in the weld nipple component the end treatment Welding Nipple (ET=1007) must be defined in the CSInfo of the corresponding **OUTPUT_POINT** coordinate system.



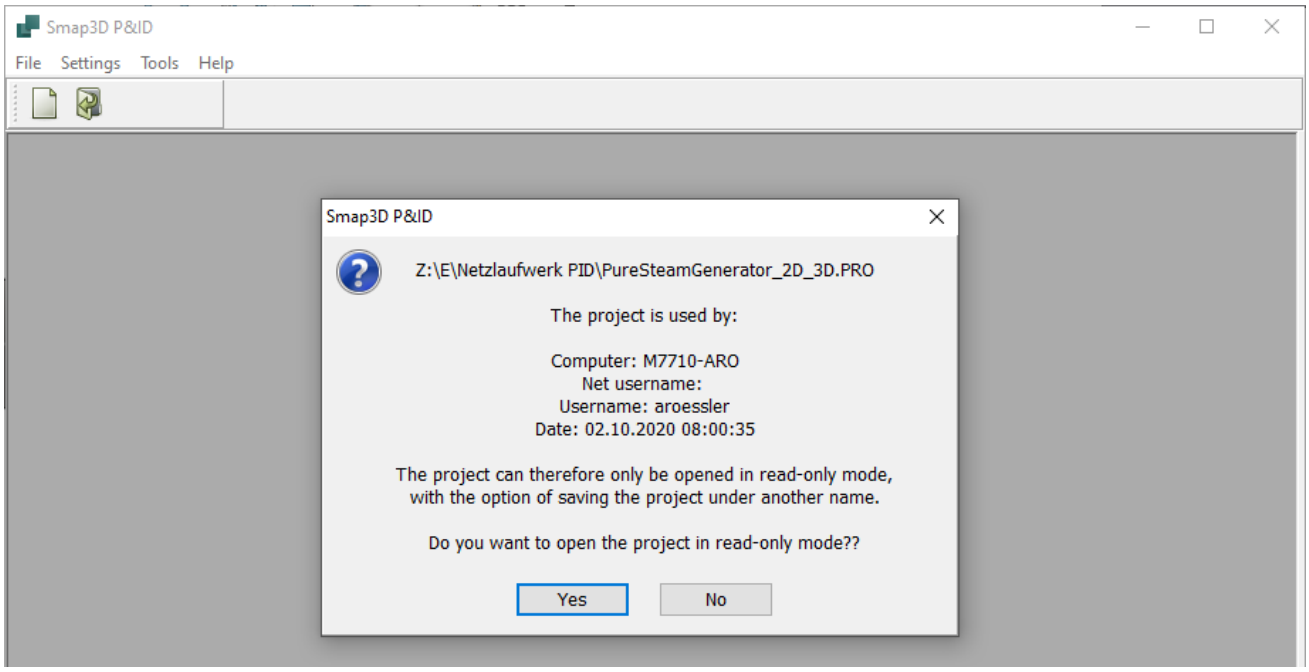
New in the P&ID To-Do list

Clear feedback if PRO file is already opened and in use

When opening a PRO file through the To-Do list, version 2021 and later will display a unique message if a PRO file is already open and in use on another computer at the same time.

Previously, a PRO file started from the To-Do list was automatically opened read-only if it was already open elsewhere, but without telling a user of the To-Do list that the PRO file was already open elsewhere.

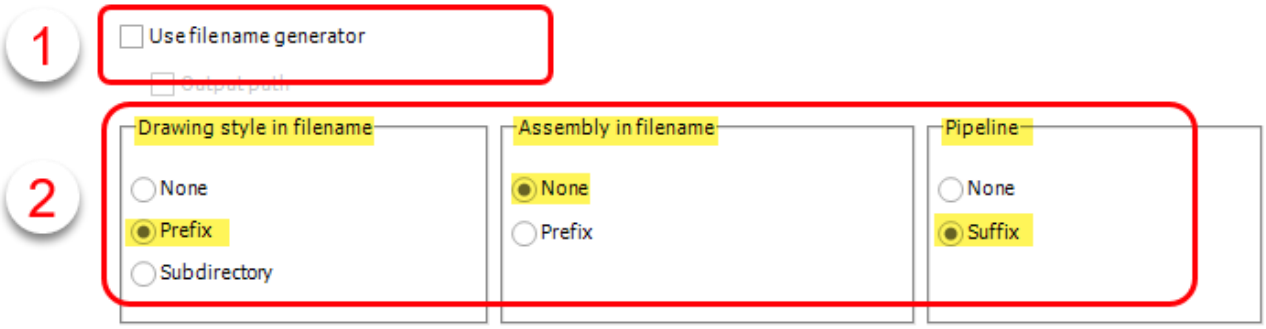
In order to prevent resulting ambiguities and data loss (in case of unauthorized changes) for a user of the To-Do list, this clear feedback was implemented.



New in Smap3D Isometric

Use of the Dynamic file naming of Isogen©

In all previous versions of Smap3D Isometric, only the methods (1 and 2) specially developed by Smap3D for naming Isometric output files were available to users.



Especially with the increased demand for project and/or piping specific information in the file names of the resulting isometric drawings and reports, the previous possibilities (2) were limited.

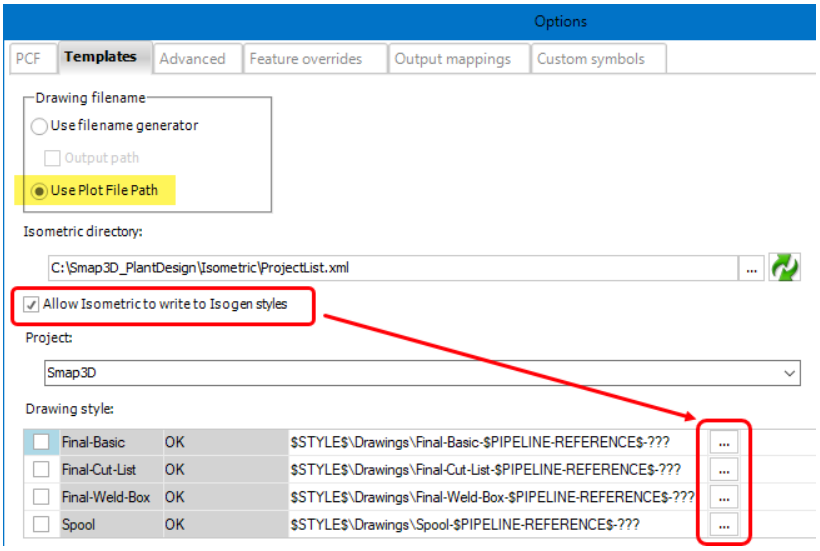
In order to offer our customers the highest possible flexibility as well as the maximum available range of functions when working with the market-leading Isogen© kernel, we have completely switched to the use of the **dynamic file naming** of Isogen© instead of the functions (2).

Dynamic file naming is a method of **naming files output by Isogen** using dynamic attribute values. This applies to all output files, such as the drawings and reports. This method can also be used to create folders.

The new option for using this type of file naming in Smap3D Isometric is called **Use Plot File Path**. This must be activated in the Isometric **Options** on the **Templates** tab to use dynamic file naming.

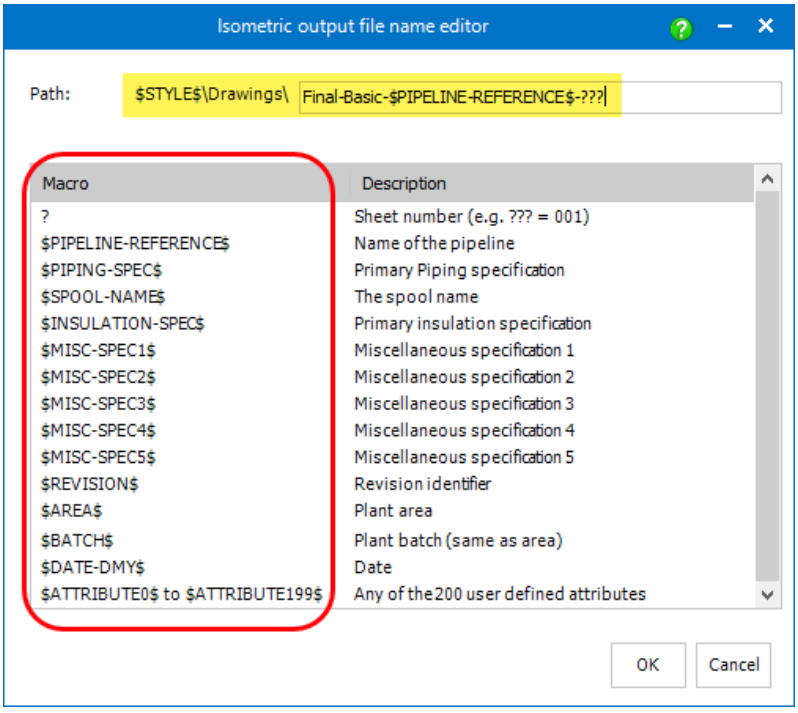
For all installations of Smap3D Isometric 2021 using the **standard configuration and data** provided by us, the provided styles have already been changed to the new definition method.
For each style, this corresponds to the previous default setting for file naming **Drawing style name in filename** as **Prefix**, together with **Pipeline** name as **Suffix**.

When upgrading from previous versions, existing styles **must** also **be changed** to this new definition method before generating drawings with Isometric 2021.
Detailed instructions can be found in the 2021 installation guide included in the software package in the chapters Upgrade and FAQ.



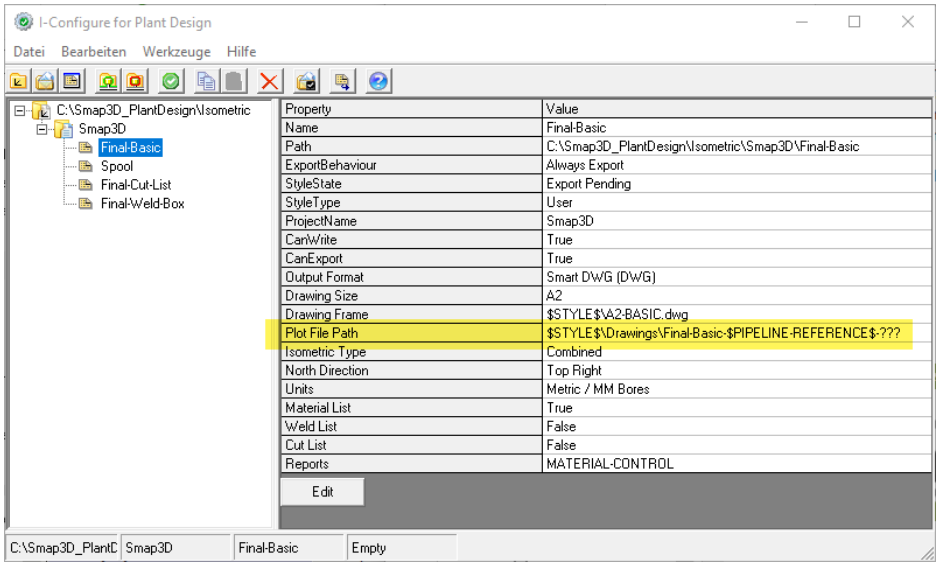
To change/adapt existing dynamic file naming definitions (per style), the new option **Allow Isometric to write to Isogen styles** must be activated. This will activate a new button behind each of the existing styles.

After a click on this button, the dialog of the **Isometric output file name editor** opens for the corresponding line.

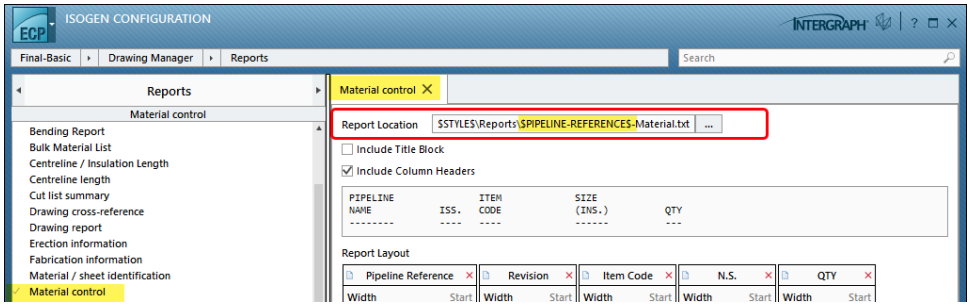


This **Isometric output file name editor** contains all available macros (variables) with which you can define a syntax for future drawing file names based on the new Isogen© Dynamic File Naming method.
Each macro listed here can be combined individually or together with others to a desired file name syntax (in the **Path:** field). It is also possible to use static texts (strings) or only single letters or characters.

If the status of each available style in Smap3D Isometric is set to **OK**, then the file name definitions performed in Isometric will also be correctly written to the **Plot File Path** setting of the corresponding Isogen styles (visible in I-Configure).
These definitions, made in Smap3D Isometric, are then used for the file names of the drawings that will be generated with this style in the future.



If, in addition to the file naming for drawings, the file names for reports should also be based on the Isogen© Dynamic File Naming, this must be done in the **I-Configure** tool for the respective style and its reports.



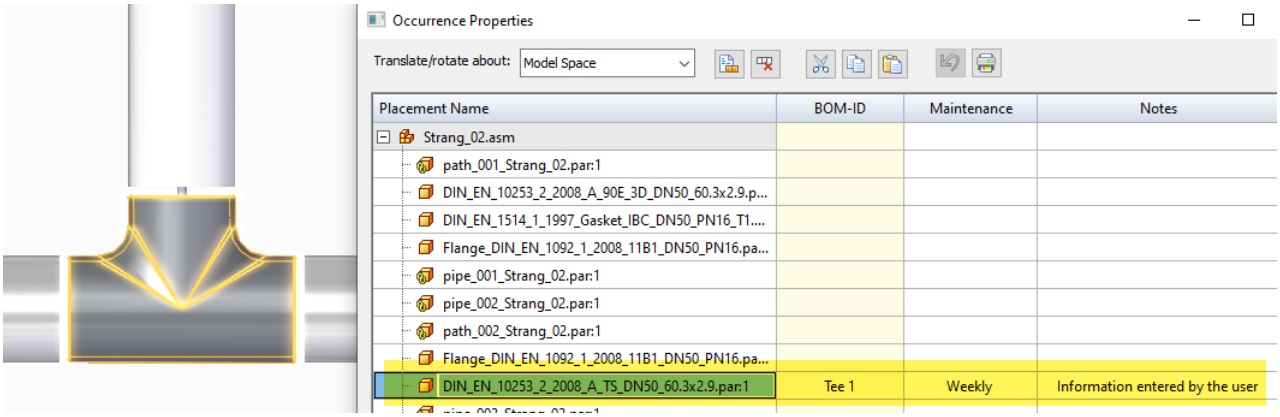
We are happy to train you on these Isogen© Dynamic file naming methods and the new resulting possibilities for your drawing and report output, in a well-founded administrator training.

Using the Solid Edge occurrence properties

Since version 2017, Smap3D applications **Piping**, **P&ID To-Do List** and **VCWriter** support the transfer of Smap3D specific virtual properties as information into the native Solid Edge function **Occurrence Properties**.

The goal at that time was, to make the virtual properties of Smap3D programs, previously invisible to a user and to Solid Edge, visible and available with native Solid Edge functions (e.g. parts list).

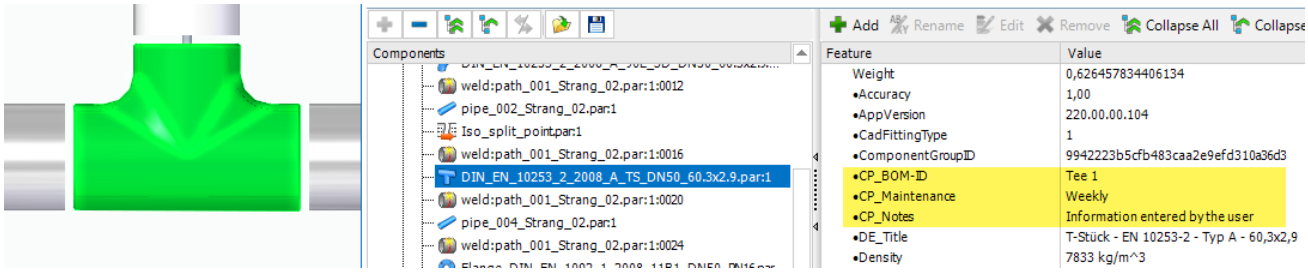
Now more and more customers have demanded the opposite for Smap3D Isometric.
Isometric is intended to read out existing information on parts in assemblies that have been entered by users in the Solid Edge function **Occurrence Properties** and use it for isometric generation.



All necessary data analysis enhancements were built into Isometric 2021 (from step 1 to step 2).

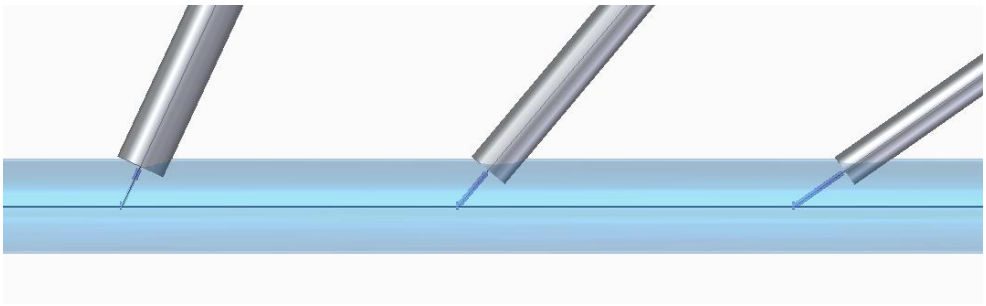
All relevant information from the Solid Edge assemblies is now read out and made available.
To distinguish this information from other, possibly identically named properties, Isometric adds the prefix **CP_** in front of the original property names.

For creating rules or mapping such a CAD property to a PCF property in Isometric, **the exact property name displayed in step 2 must be used**, not the one displayed in Solid Edge!



Support of angled branch placeholder parts

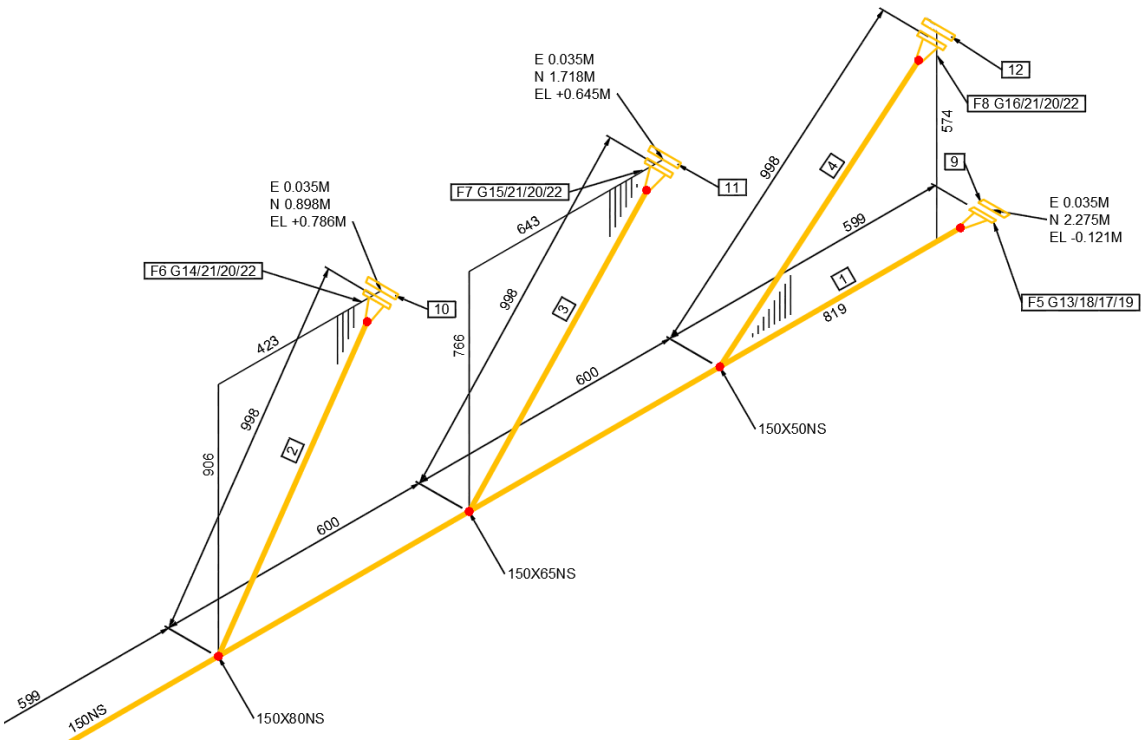
For the correct realization of angled pipe branches (pipe on pipe without branch fittings), Smap3D Piping has introduced the use of angled branch placeholder parts (Angled Branch CS Dummy).



Isometric 2021 also supports such placeholder components if they are included in a 3D piping assembly during isometric generation.

However, the basic requirement is that such placeholder parts contain the following information as user-defined CAD file properties for Isometric:

- **ComponentType** with the value **TEE-SET-ON**
- **SKEY** with the value **TESO** (for planar branches on the same axis plane as the main pipe)
- **SKEY** with the value **TOSO** (for non-tangential **offset** branches, with offset axis planes between branch pipe and main pipe)



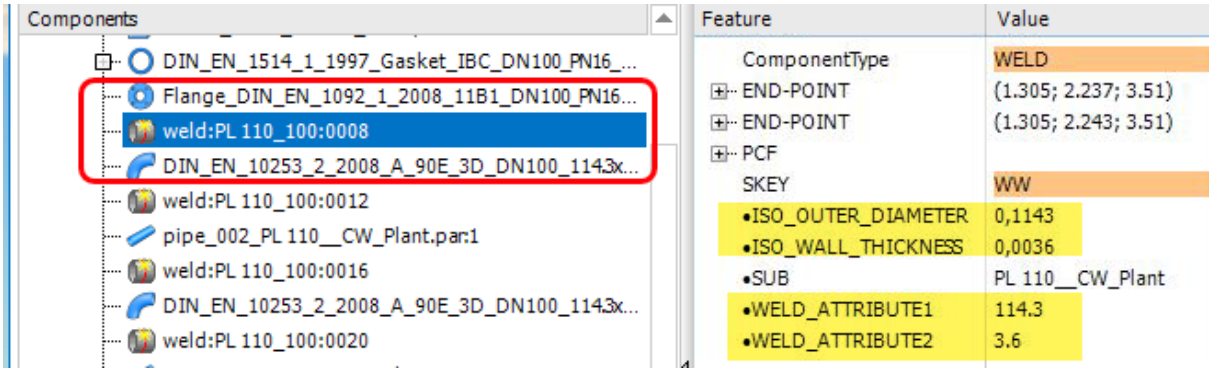
Take over weld seam attributes from adjacent fittings

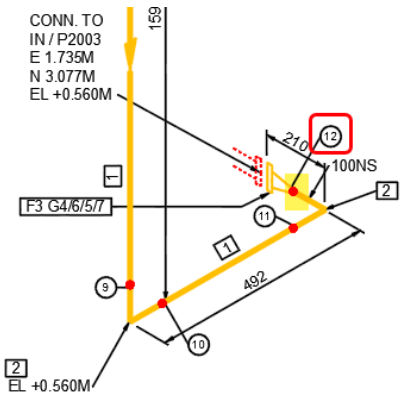
Previously, when generating isometrics, the required weld seam parameters (if they were to be evaluated and displayed on an isometric drawing or list) were by default only read from the connected pipes and used accordingly.

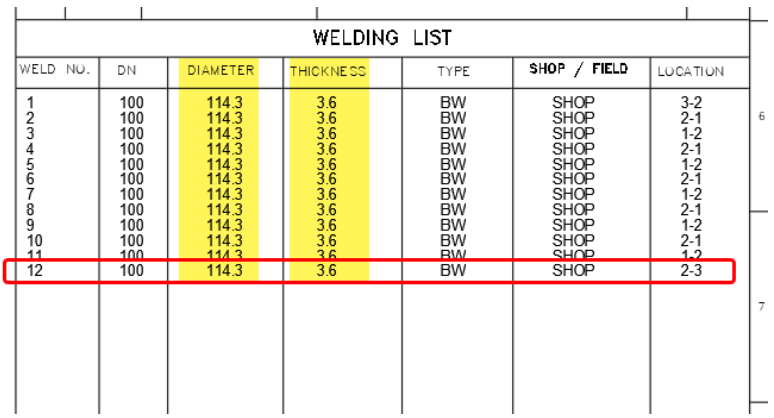
This previous standard behavior led to limitations in welding seams that were required directly between fittings (when there is no pipe in between).
In these cases, the parameter values (outside diameter and wall thickness) for the weld could not be read and displayed.

With Isometric 2021, this limitation has been removed, and corresponding parameter values for welds are now also read from the fittings/standard parts if there is a weld in between.
However, the necessary values must be defined in the standard parts, in the respective CSInfo (per connection point).

To support the input of these required parameter values by the user, a function extension was also implemented in the Coordinate System Wizard.





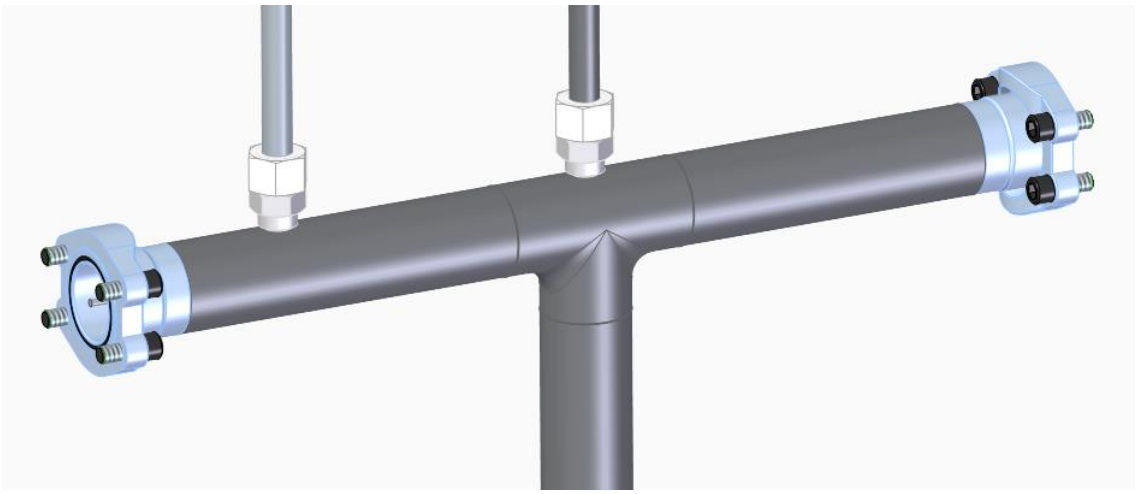


For the Smap3D standard parts supplied by us, Isometric 2021 can read and use the necessary parameters from the **piping stream** invisible to users.

In the case of customer-specific and specially created standard and supplier parts, these parameters must be set by the user within the scope of the CSInfo definition per connection point.

Support of additional olet branches on tee-pieces

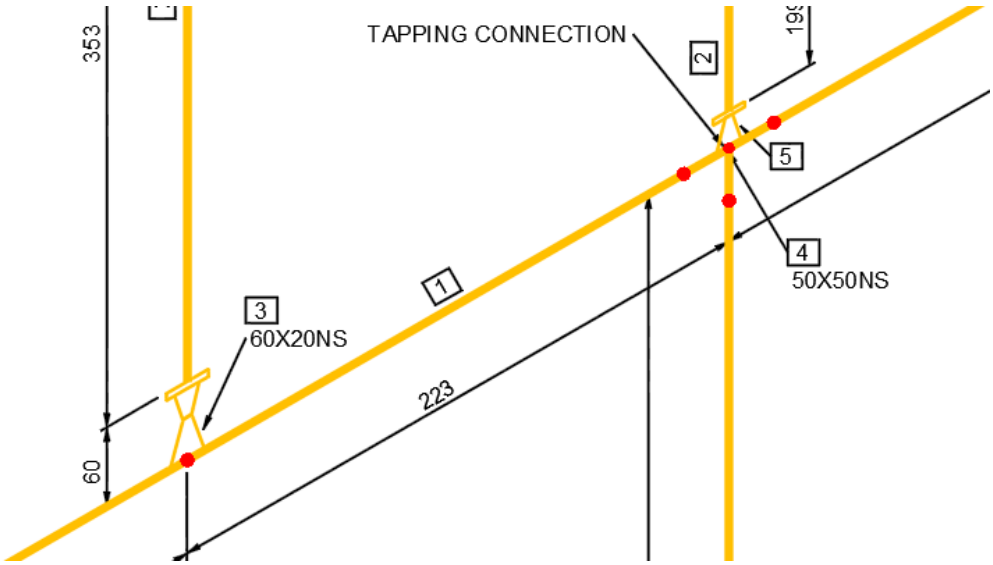
In specific applications in 3D piping design it is necessary to realize additional branches by means of olets, nipples, or similar.



From the point of view of Smap3D Isometric and Isogen, the difference so far has been whether such a branch is placed on a pipe or a tee-piece fitting.
Olet branches on pipes have long been supported in Isometric and Isogen and output in drawings.

Up to now there was a limitation for applications where such an olet branch was placed directly at a tee-piece. However, Smap3D Isometric and Isogen could not output a correct drawing until now.

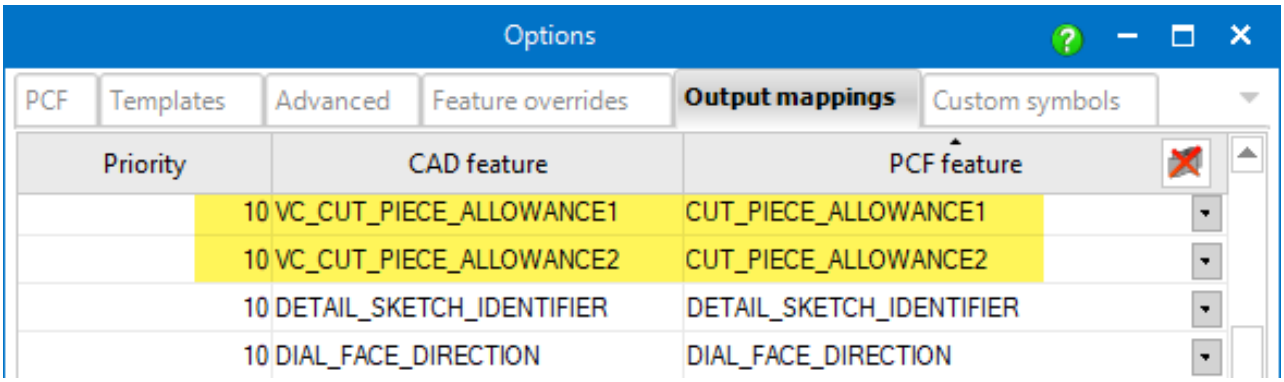
With version 2021 this limitation has also been removed and an isometric derivation at such **TAPPING CONNECTIONS** is now correctly created.



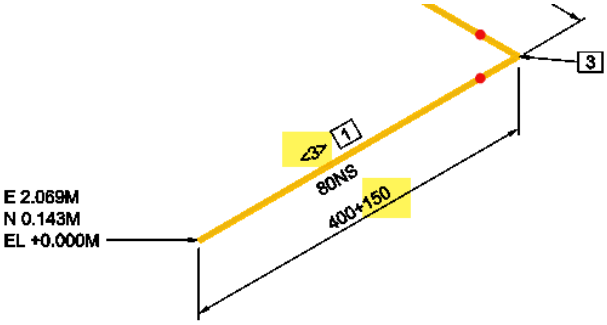
Further standard definitions for Output mappings

With the new Piping function Define Cut piece allowance, the same virtual feature names (CAD feature) are always used for pipes for use as **CUT_PIECE_ALLOWANCE** parameters (PCF feature).

To ensure that a user has no additional configuration effort in the Isometric when using this piping function, the corresponding property assignment is available as standard from version 2021.



An isometric drawing then automatically contains the corresponding information for the pipe end selected in Piping.



CUTTING LIST

CUT PIECE	SIZE	LENGTH	END PREP-1	END-PREP-2
1	80	588	BEVEL	BEVEL
2	80	688	BEVEL	BEVEL
3	80	430	BEVEL	SQ.CUT
4	80	288	BEVEL	BEVEL

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