

This is new in Plant Design 2023

Once again, the latest version of Smapp3D Plant Design contains many enhancements and improvements in the areas of functionality and application.

The most important ones are described below.

The different chapters in this document are:

[General news](#)

[New in P&ID](#)

[New in Parts Management](#)

[New in 3D Piping](#)

[New in the P&ID To-Do List](#)

[New in Isometric](#)

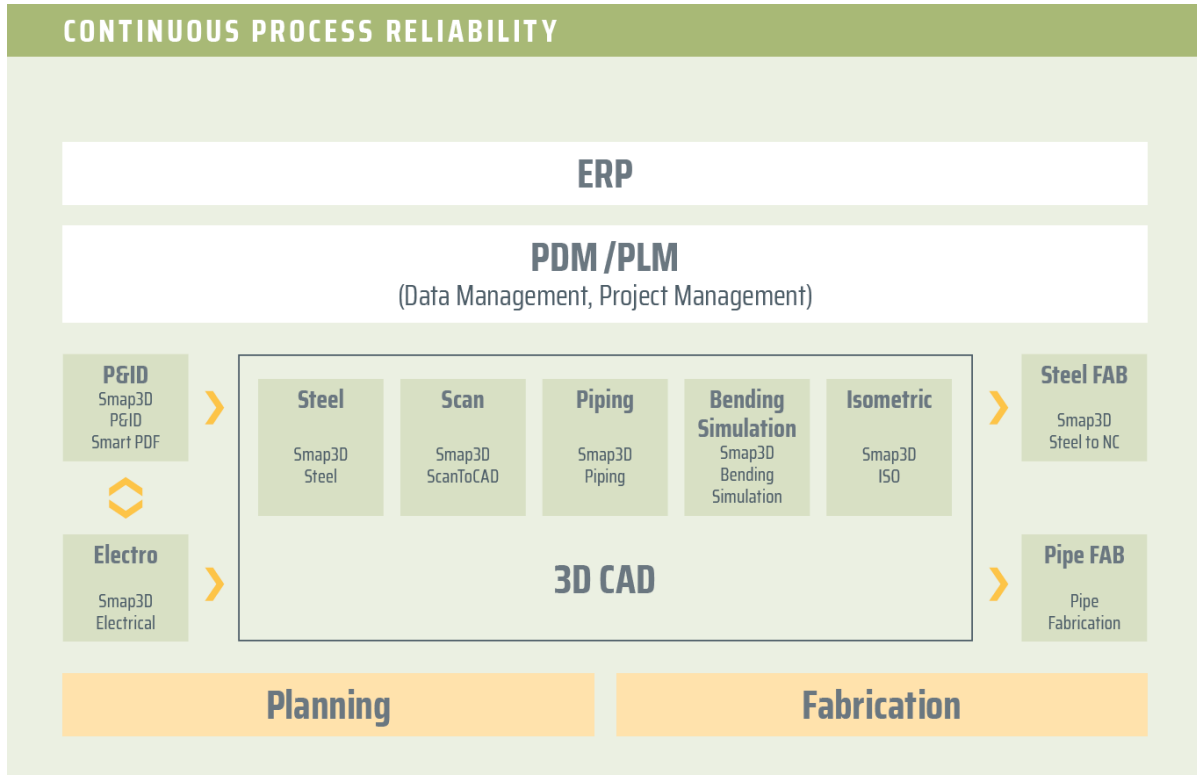


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General improvements

Supported CAD versions

Autodesk Inventor

The release version of Smap3D Plant Design 2023 supports:

- **Autodesk Inventor 2023 with Update 2023.2 or later**

Licensing required in version 2023

With Smap3D Plant Design 2023, the CAD-Partner Licensing software (License Server Manager and License Client Manager) must also be installed in version 2023.

Existing **License server installations** of CAD-Partner Licensing (when using floating and/or multiple node-locked licenses) must **be updated to version 2023**. For details, refer to the **Installation Guide (in chapter > Upgrade)**, included in the installation package of version 2023.

This **will upgrade** an existing installation of License Server Manager 2022 **to version 2023**.

The updated version number of License Server Manager 2023 will then be **8.23.0.35**

The name of the associated service is then **CAD Partner License Server 2023**.

The associated **FlexNetPublisher** version will be **11.19.1.0**.

All local License Client Managers, along with the Smap3D applications, must be uninstalled before an installation of Plant Design 2023 for client computer can be performed.

No new product licenses required for version 2023

The Plant Design 2023 applications can also work with the licenses issued for version 2022. Therefore, customers with valid maintenance contract will also **NOT receive a new license file**.

Support of Microsoft Windows 11

With the release of Smap3D Plant Design 2023 / Smap3D Parts Management 2023, the Microsoft Windows 11 operating system for client/workstation installations is officially supported.

Minimum required is an edition of **Windows 11 Enterprise** or **Professional** (64-bit only) version **21H2** or later/higher.

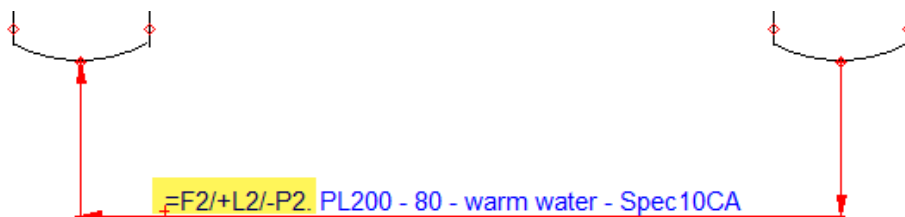
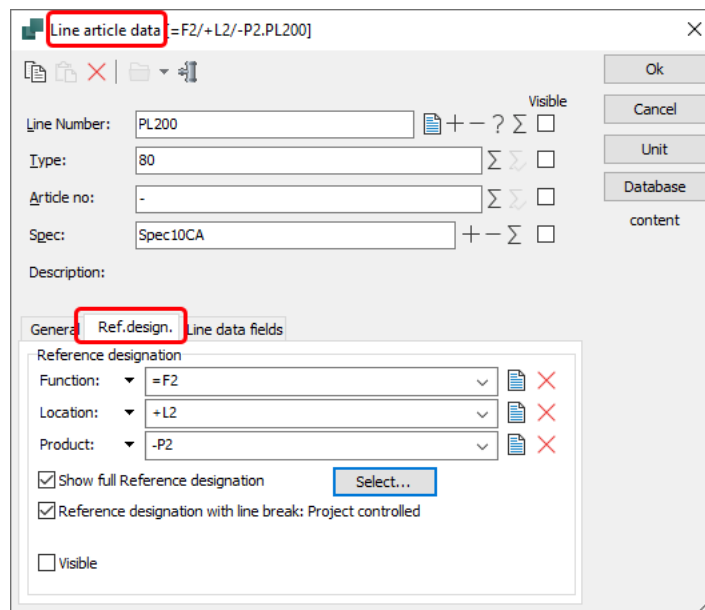
For more details on system requirements, see the **Installation guide** included with the 2023 version installation package in the **Documents** folder.

New in Smapp3D P&ID

Reference designations can now also be used for lines

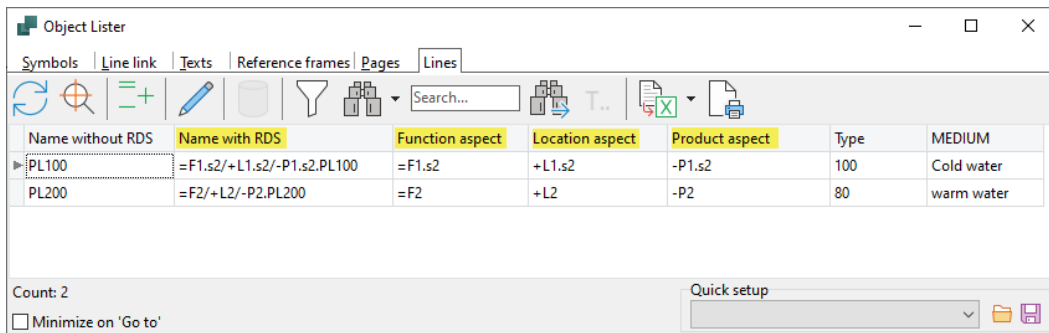
One of the big wishes of many customers and users of Smapp3D P&ID was, to be able to use the existing functions for reference designations not only with symbols/components, but also for lines. We have now realized this possibility with version 2023.

In the new **Ref. design.** tab of the **Line article data** dialog, the existing functions for assigning and using reference designations are now also available for lines.

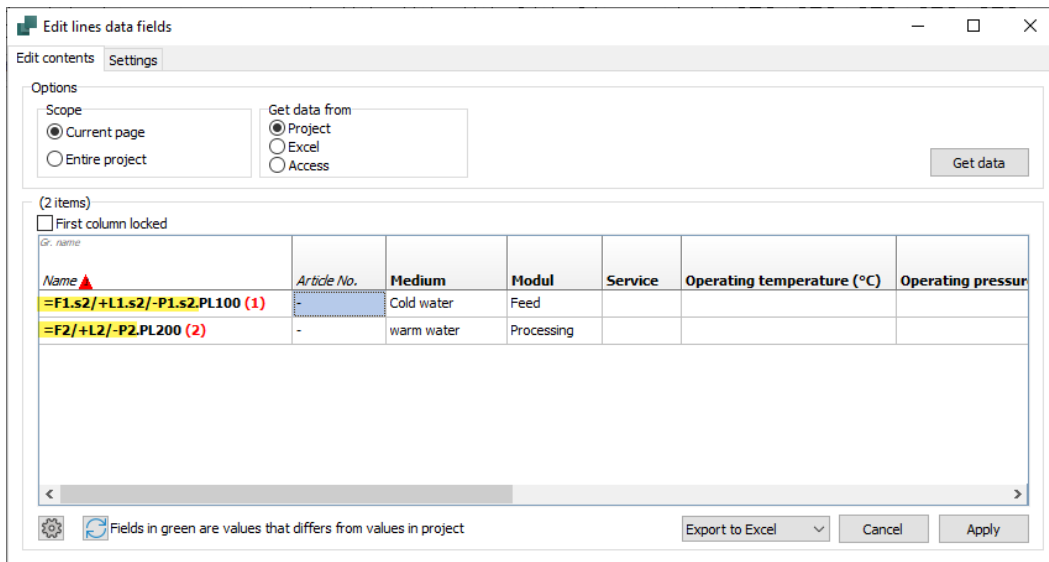


Of course, reference designations on lines are also supported and used by all further functions and tools in the P&ID application. The necessary improvement has been made.

In the Object Lister



In the Edit lines data fields module



For list evaluations with lines

Row	Comp. Name	Article no.	Type	Manufacturer	Description	Medium
1	=F1/+L2/-P1.B1	22000V	Vessel	MaBe GmbH	Vessel 2000 liter	Cold water
2	=F2/+L2/-P2.B2	25006V	Vessel	MaBe GmbH	Vessel 5000 liter	Hot water
3	=F1.s2/+L1.s2/-P1.s2.PL100		100			Kaltwasser
4	=F2/+L2/-P2.PL200		80			Warmwasser

Enhancements in the Object Lister

Selection for symbol data fields sorted alphanumerically

If in the Object Lister (Symbols tab), symbol data fields are selected for display, the corresponding selection list with available symbol data fields is now displayed sorted alphanumerically.

The screenshot shows the Object Lister window with the 'Symbols' tab selected. The main table lists various objects with columns for Name with RDS, Conns., Article, Spec, Symbol, Page No., and Syml. A dropdown menu is open over the 'Syml' column, showing a list of available symbol data fields. The 'Datatypes' option is highlighted in blue. The list of fields includes: (All), (None), ACTUATOR, ACTUATOR TYPE, AGITATOR DIAMETER (IN), AGITATOR DIAMETER (MM), BULK DENSITY (KG/M3), BULK DENSITY (LB/FT3), COATING, COHESIVE ATTRACTION, COMMENTS, CONNECTION, CONSTRUCTION, CONVEYOR GOODS (T), CONVEYOR HEIGHT (FT), CONVEYOR HEIGHT (M), CONVEYOR LENGTH (FT), CONVEYOR LENGTH (M), CONVEYOR SPEED (FT/S), CONVEYOR SPEED (M/S), COOLING FACILITY, CRUSHER POWER (T/H), CUSTOM 1, CUSTOM 2, CUSTOM 3, CUSTOM 4, CUSTOM 5, DESIGN PRESSURE (BAR), DESIGN PRESSURE (PSI), DESIGN PRESSURE 2 (BAR), DESIGN PRESSURE 2 (PSI), DESIGN TEMPERATURE (*C), DESIGN TEMPERATURE (*F), DESIGN TEMPERATURE 2 (*C), DESIGN TEMPERATURE 2 (*F), and DESIGN TEMPERATURE 2 (*F).

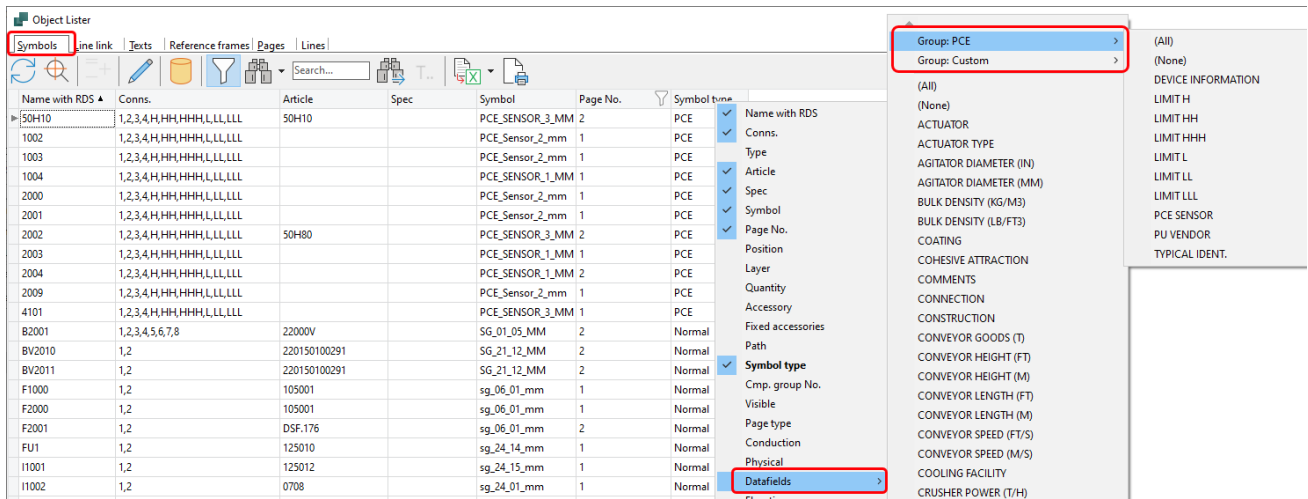
Name with RDS	Conns.	Article	Spec	Symbol	Page No.	Syml
50H10	1,2,3,4,H,HH,HHH,L,LL,LLL	50H10		PCE_SENSOR_3_MM	2	PCE
1002	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_Sensor_2_mm	1	PCE
1003	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_Sensor_2_mm	1	PCE
1004	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_SENSOR_1_MM	1	PCE
2000	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_Sensor_2_mm	1	PCE
2001	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_Sensor_2_mm	1	PCE
2002	1,2,3,4,H,HH,HHH,L,LL,LLL	50H80		PCE_SENSOR_3_MM	2	PCE
2003	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_SENSOR_1_MM	1	PCE
2004	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_SENSOR_1_MM	2	PCE
2009	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_Sensor_2_mm	1	PCE
4101	1,2,3,4,H,HH,HHH,L,LL,LLL			PCE_SENSOR_3_MM	1	PCE
B2001	1,2,3,4,5,6,7,8	22000V		SG_01_05_MM	2	Norr
BV2010	1,2	220150100291		SG_21_12_MM	2	Norr
BV2011	1,2	220150100291		SG_21_12_MM	2	Norr
F1000	1,2	105001		sg_06_01_mm	1	Norr
F2000	1,2	105001		sg_06_01_mm	1	Norr
F2001	1,2	DSF.176		sg_06_01_mm	2	Norr
FU1	1,2	125010		sg_24_14_mm	1	Norr
I1001	1,2	125012		sg_24_15_mm	1	Norr
I1002	1,2	0708		sg_24_01_mm	1	Norr
K1001	1,2,3,4,5,6,7,8	102005		sg_02_06_mm	1	Norr
P2000	1,2	114001		sg_15_01_mm	1	Flow
P2001	1,2	96539052		sg_15_02_mm	2	Flow
P2002	1,2	96539052		sg_15_02_mm	2	Flow
P2003	1,2	1510		15-10_mm	2	Flow
RV2000	1,2	122001		SG_22_02_MM	1	Flow
SV1002	1,2	0504		sg_23_02a_mm	1	Flow
V1002	1,2,3	0605		SG_21_04_ACTUATOR	1	Norr
V1003	1,2,3	0615		SG_21_04_ACTUATOR	1	Norr

Support of symbol data field groups for display selection

All users working with symbol data fields groups (created in the **Edit symbols datafields** module), can now also use these group definitions for display selection in the Object Lister.

Existing symbol data field groups are displayed at the top of the list during **data field** selection. Via new/additional drop-down windows, the data fields belonging to a group can be selected for deselected individually, or quickly with **All** or **None**.

This new option makes the search/selection of symbol data fields for display in the Object Lister faster and more convenient.



Save and load column display configurations

Since some versions it is possible to individually configure and set the columns of the displayed **attributes/data fields** in all tabs of the Object Lister.

Some customers use these individual display combinations very intensively, but missed the possibilities to save, switch and load them.

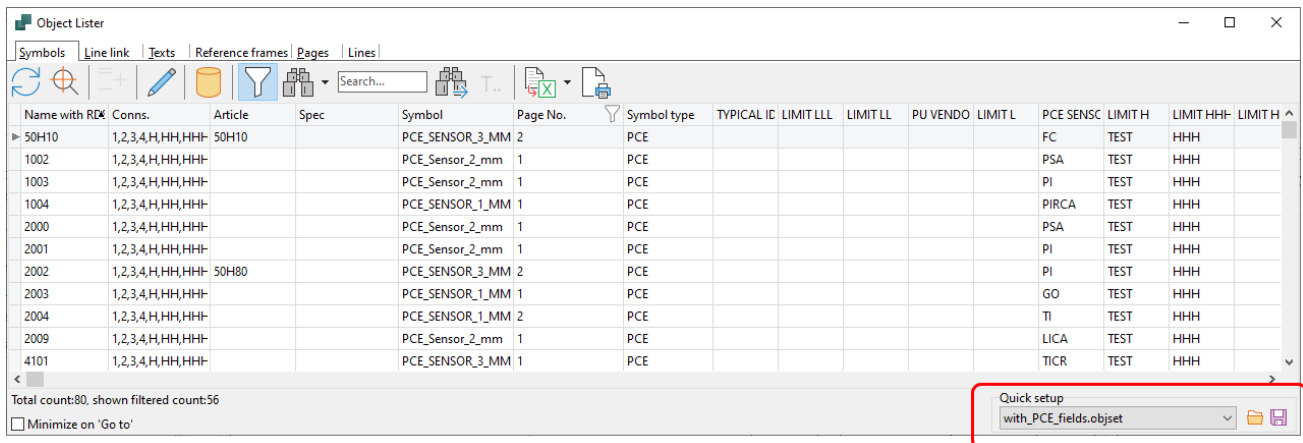
The necessary functions have now been implemented in P&ID 2023.

In the lower right corner of the Object Lister window (no matter which tab is active), you will now find the **Save** and **Open/Load** buttons.

If a user has created an individual combination of a column display in the Object Lister (no matter which tab), this can be saved to a file with the extension **.objset** via the Save button.

If you have created several such **.objset** files with different combinations, you can load them with the **Open** button and thus switch them as needed.

The column display is immediately changed to match the combination saved in the loaded **.objset** file.



Enhancements to the Edit symbols data fields module

Content changes of data field values are highlighted after import

In the tab **Edit contents**, there is since several version the possibility to export the collected data field values (from a project or a single diagram page), to an external Excel file.

There the data field values can be changed/extended (e.g., by persons without P&ID application).

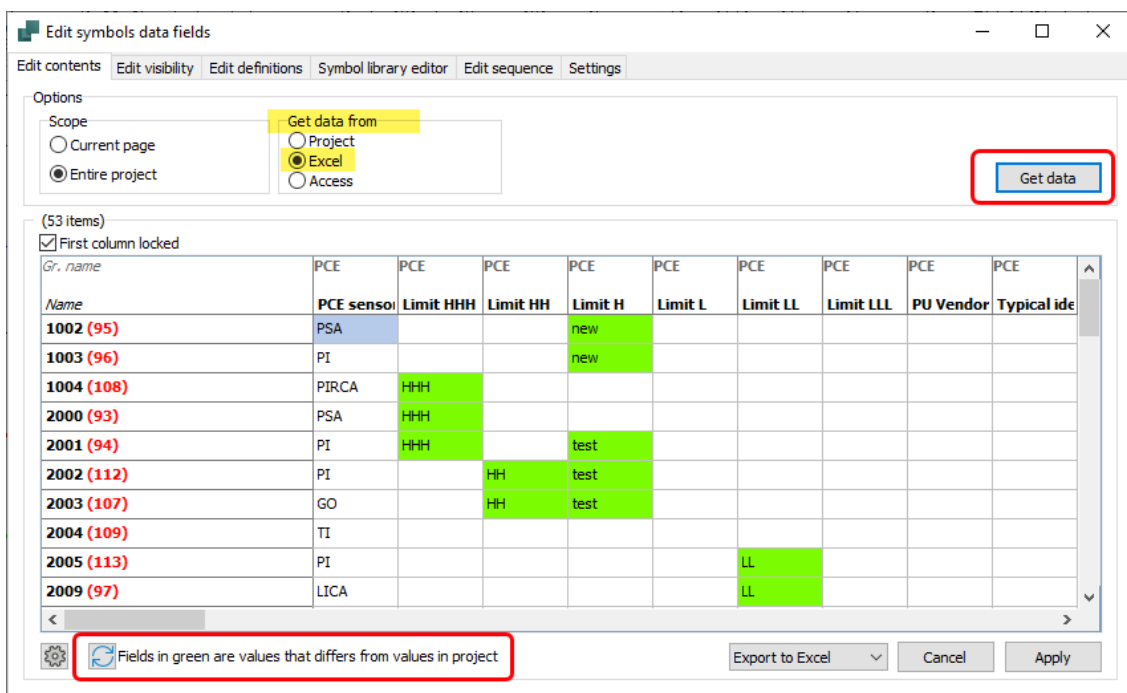
Also in the **Edit contents** tab, such Excel files can then be imported again (function **Get data from Excel**) to write them back into the project (into the symbols) with the **Apply** button.

After applying, you will get the information whether and how many data field values have been changed.

Before clicking on Apply, however, it was previously not apparent to a user how many and which contents would change because of the imported Excel file.

To make this better and more transparent, from version 2023 onwards the current values of the data fields in the project are compared with those of the imported Excel file and then highlighted in color (green) in the **Edit contents** tab.

This give a user an overview of the changed before they are applied.



However, this new check/display **ONLY** works for Excel tables that have been exported exclusively with the **Edit symbols data fields** too, in the **Edit contents** tab, from the currently open and active project. This check is based on system internal symbol IDs and can therefore only work correct under these conditions.

For Excel tables which have been created and filled with data field contents in other ways, or which have been exported from other PRO files, this new validation and display will NOT work.

Enhancements in the symbol editor

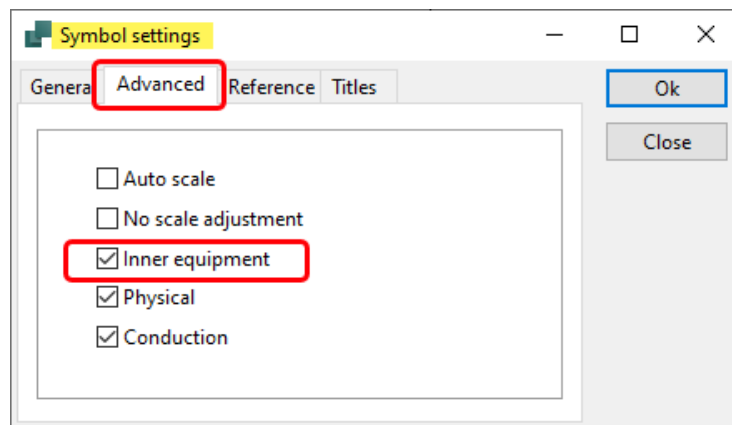
Predefinition for Inner Equipment setting on symbol level

Until now, the Inner Equipment setting was only available at P&ID project/diagram level (with default value active/checked) and had to be adjusted by a user for each inserted symbol as needed.

Now there is the new **Inner Equipment** option when creating or editing a symbol, in the symbol editor (Symbol menu).

This now allows a predefinition of the **Inner Equipment** setting already on symbol level in the symbol library.

The new option is in the **Symbol settings**, in the **Advanced** tab.



Predefinition of values for symbol data fields per symbol

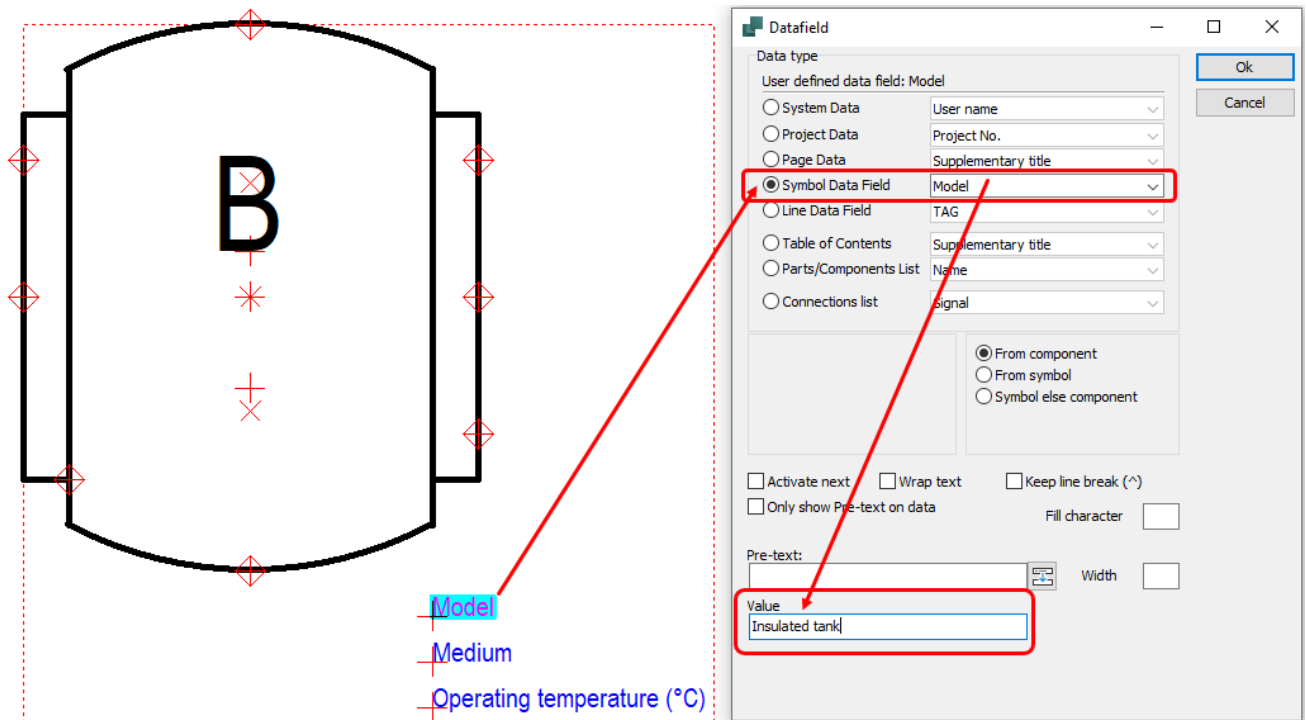
Previously, predefinitions for symbol data field values could only be made at the P&ID application level (with value lists).

These value lists are then already valid for the respective symbol data field, regardless of which symbol (e.g., in tank, pump or valve) this symbol data field is then used in.

To provide our customers with better object-oriented default options for this, we have created a new option to predefine a default value for a symbol data field in a symbol.

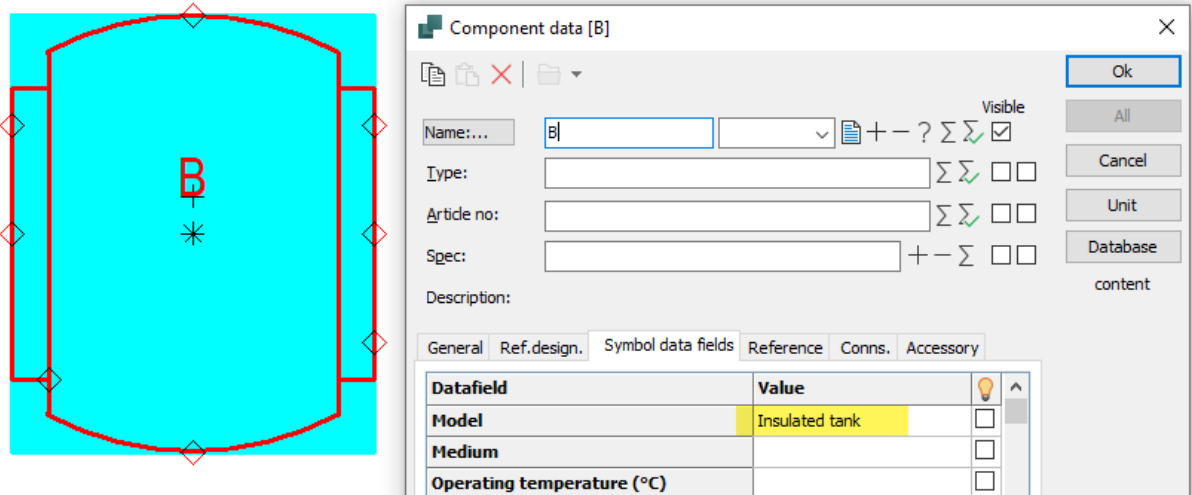
This new definition is only available in the symbol editor, with the **Insert Datafield** function or with the symbol data fields already inserted in a symbol.

Thus, for the same symbol data field, in different symbols, in each case a suitable value can be predefined.



When placing such a symbol in a diagram, the value predefined in the symbol has the highest priority and is automatically used/filled in the symbol's component data.

The use of the value lists on system level is of course still possible. However, their values then have the lower priority and are not automatically used in this case.



New module "Pipeline list"

Up to now, only the component list and/or connection list were available for evaluation of lines. However, both evaluations always consider all existing, individual line elements. But the results that could be achieved did not match the process engineering expectations of all customers.

The **Pipeline list** module now gives users a new evaluation option of pipelines, according to various process engineering requirements.

Basis is the evaluation of a list of pipelines which reacts on a configurable **pipeline identifier** (i.e., either the line name or an existing line data field e.g., TAG).

This gives users the ability to subdivide pipeline networks into logical sections based on line attributes (line data fields) e.g., a cooling water line **L100** which consists of a **suction** and a **feed** line (to and from a pump), so that it gets two entries in the pipeline list.

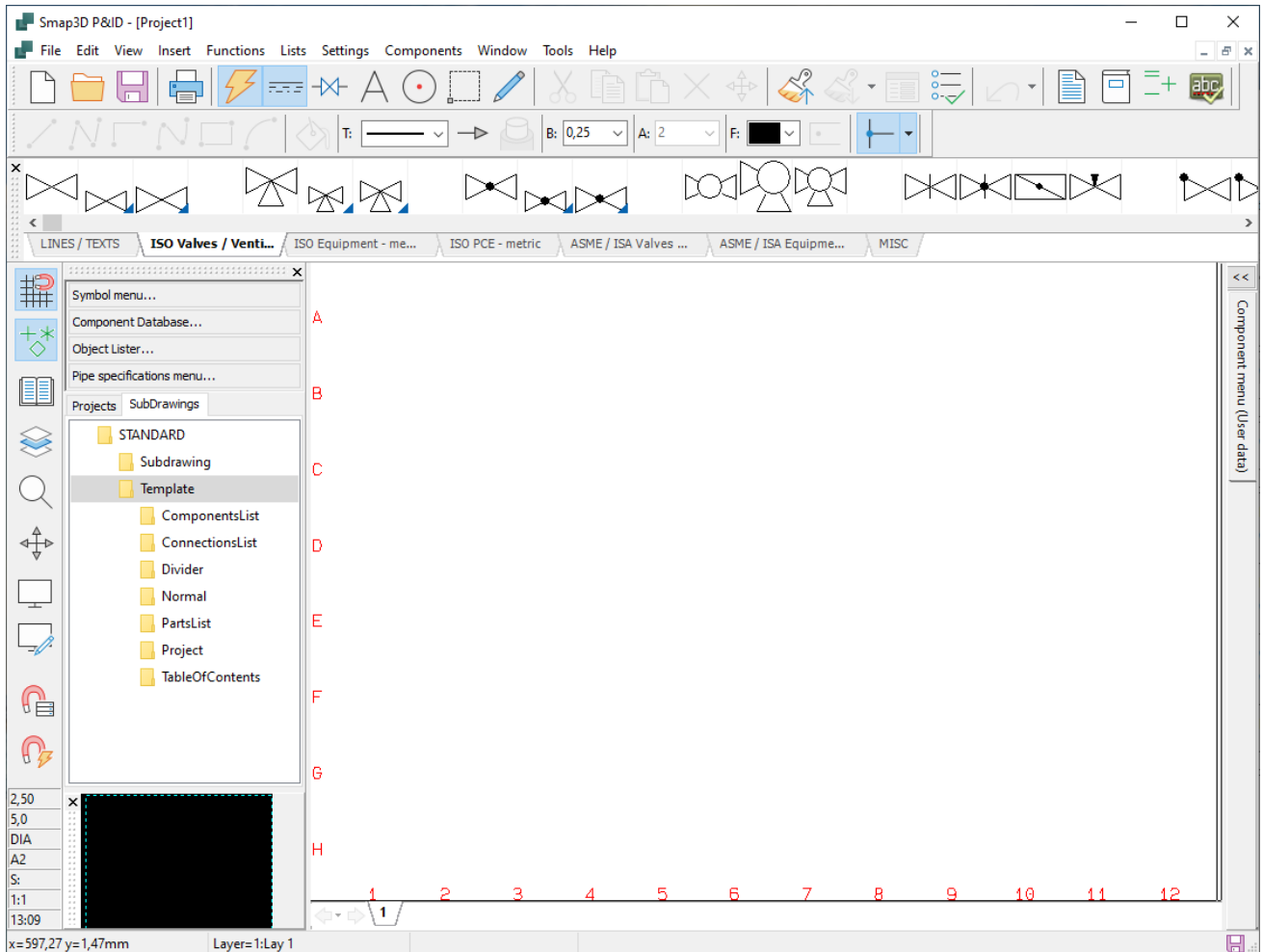
Furthermore, the flow direction (drawing order with flow arrow on drawn lines) is evaluated so that in the pipeline list a **From / To** information to the connected plant equipment can be displayed.

The **Export to Excel** button can be used to export the evaluated pipelines with associated data to Excel.

Line name	From	To	TAG	Medium
PL 131	P2001:2	B2001:1	PL 131 - 100 - HW - Spec10CA	HW
PL 132	B2001:2	P2002:1	PL 132 - 80 - HW - Spec10CA	HW
PL 133	P2002:2	F2001:1	PL 133 - 100 - HW - Spec10CA	HW
PL 134	F2001:2	W2001:2	PL 134 - 100 - HW - Spec10CA	HW

New user interface color scheme

A new color scheme will be used for the windows and buttons in the P&ID application starting with version 2023.



Enhancement in the formula editor for symbol and line data fields

Additional system data are now available for creating formulas for data fields (symbol and line data fields).

As of P&ID 2023, the complete reference designation (RDS) or only the individual aspects Function, Location or Product (RDSfunc, RDSloc and RDSprod) can now be used in formulas.

For symbol data fields it is additionally possible to use the data field Subname.

Formula editor for data field: Custom 1

Formula

Edit formula manually

Symbol data: Name, Type, Article, Spec, **Subname**, RDS, RDSfunc, RDSloc, RDSprod

Symbol data fields: Actuator

Static text: []

Functions: ABS()

Buttons: Append, Test formula with test data, Validate formula, Clear formula

Test data:

Symbol data	Symbol data fields
Name: [Name]	Datafields Value
Type: [Type]	
Article: [Article]	
Spec: [Spec]	

Buttons: Ok, Cancel

Formula editor for data field: TAG

Formula

Line+ '-' +Size+ '-' +DATAFIELD('Medium')+ '-' + Spec

Edit formula manually

Line data: Line, **Line**, Size, Article, Spec, RDS, RDSfunc, RDSloc, RDSprod

Line data fields: Medium

Static text: []

Functions: ABS()

Buttons: Append, Test formula with test data, Validate formula, Clear formula

Test data:

Line data	Line data fields
Line: [Line]	Datafields Value
Size: [Size]	Medium Medium
Article: [Article]	
Spec: [Spec]	

Buttons: Ok, Cancel

New file .PROlck as lock file for PRO files

In the previous versions, for a PRO file opened in the P&ID application, a **hidden lock file** with syntax **~\$Filename.PRO** was always created in the same folder as the opened PRO file.

A lock file contains the information of the computer name and the active user login which was loaded/opened the PRO.

If another workstation then tries to open this PRO file, it receives the message that it is already in use by computer XXX and user XXX-XXX.

Unfortunately, the use of a hidden lock file caused increasing problems for customers working with central Microsoft OneDrive drives.

With P&ID 2023, the file extension for such lock files is therefore changed to **Filename.PROlck**. Furthermore, these are now no longer hidden but always visible after creation, in the same folder as the PRO file.

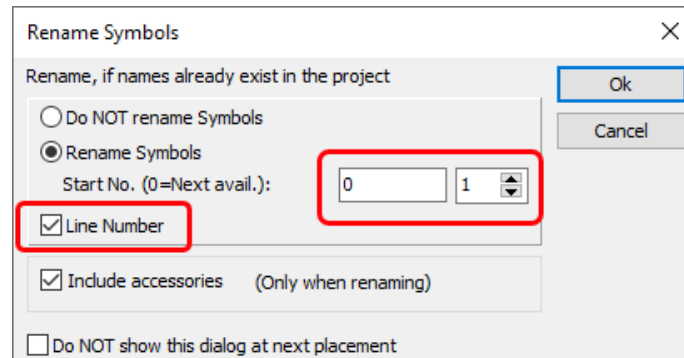
These .PROlck files are now no longer automatically deleted after closing a PRO file.

This means that one **.PROlck** per project is always present after its initial creation. Only depending on whether a PRO file is opened or not, either information is written in it (computer name and user login), or the file is without content (empty).

"Count value" option now also for line number

With P&ID 2022, the **Count value** option has been introduced for the **Rename Symbols** function. However, this has only been used for symbols so far.

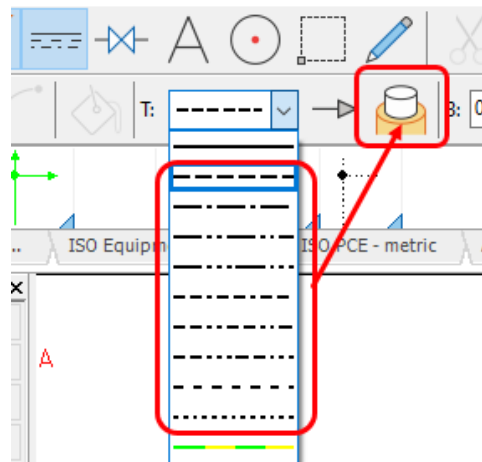
As of P&ID 2023, a set count value is now also applied to lines if this is activated with a check mark for **Line number**.



Option "Insulation" applicable for dashed standard lines

Previously, the **Insulation** option could only be used for the continuous lines with predefined flow arrow.

We have now enhanced this so that the Insulation option can now also be used for the various dashed/dotted standard lines.

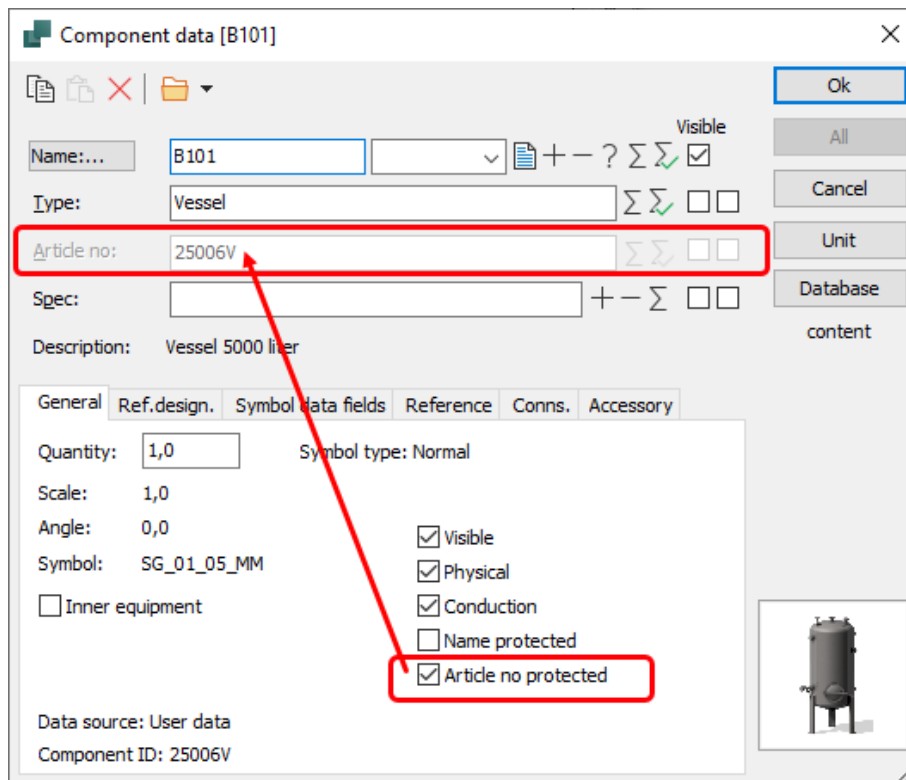


New option "Article no protected"

In the Component data dialog, the new **Article no protected** option is available in the lower area. This is intended to prevent accidental and unintentional manual entries in the **Article no** data field.

If this option is activated (checked), the data field Article no with the related buttons is deactivated (grayed out) and locked to do **manual input**.

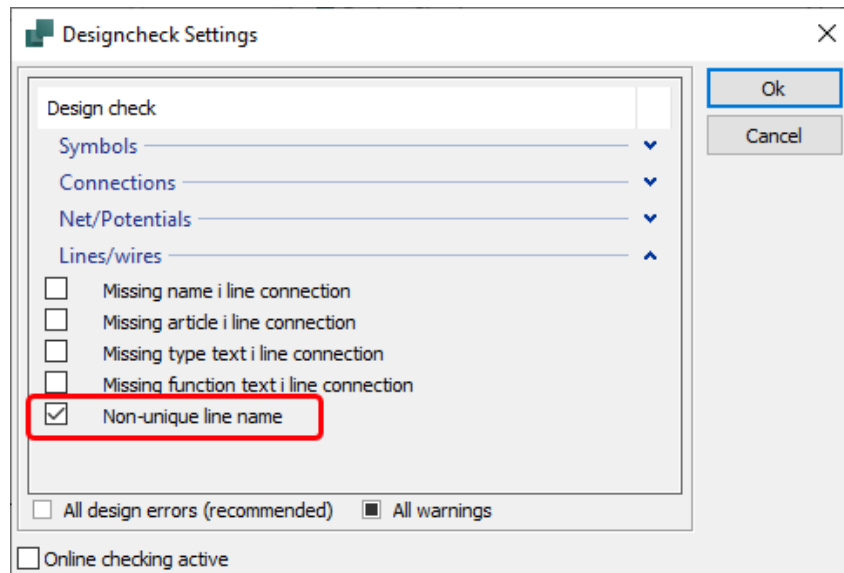
However, the use of **Database** button to select and assign an article from the database is still possible.



New design check "Non-unique line name"

The new **Non-unique line name** option can be used to check whether there are multiple lines with same value in the **Line Number** data field in the active project.

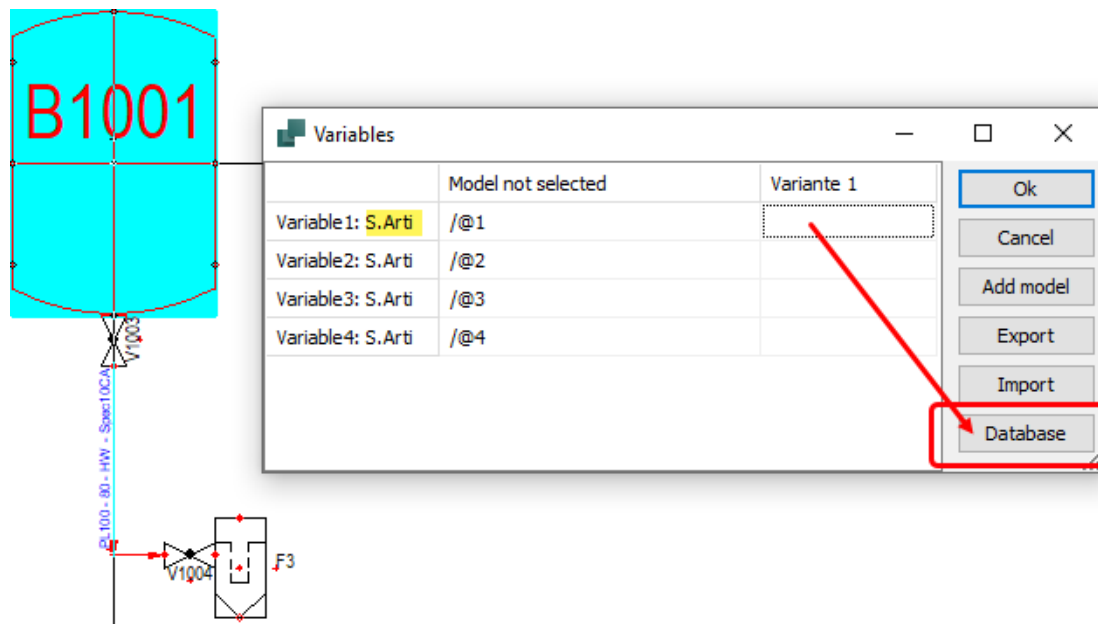
If this is the case, the corresponding message appears in the results of the design check.



Subdrawings - new button Database for article no with variables

For all users who work with subdrawings and thereby control **Article no** field with variables for different models, the **Database** button is now available for defining the models. Previously, all required values had to be written/typed in manually when creating the models.

With the new button Database, you can select the required values from the database for the field **Article no** and have them transferred.



"Symbol ID" data field available for list exports

A few versions ago, the **Symbol ID** data field was introduced in the P&ID application. This was previously only used in the **Edit symbols datafields** module in **Edit contents** tab, to have a unique identifier instead of the symbol name for exports from this module and subsequent re-imports.

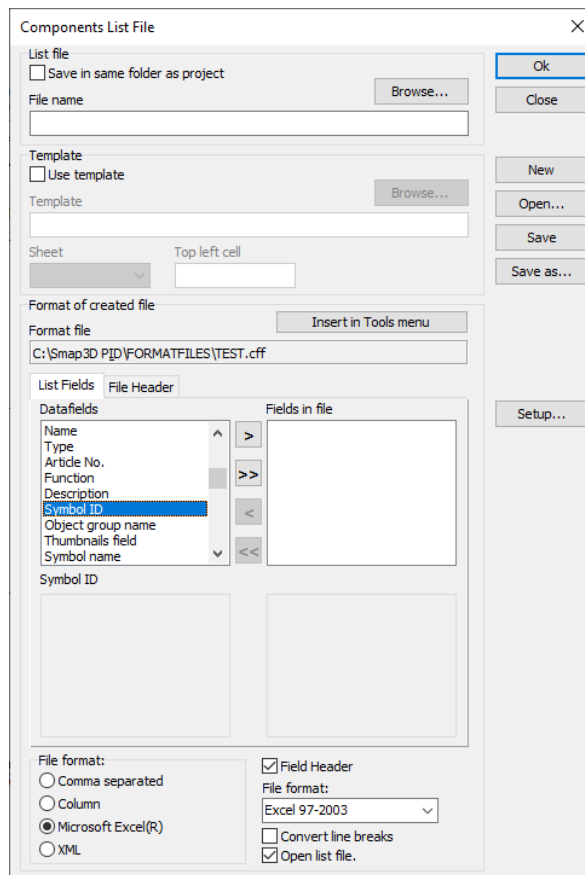
(4 items)

First column locked

Gr. name		
Name	Article No.	M
B1001 (1)		
F3 (4)		
V1003 (2)		
V1004 (3)		

As of P&ID 2023, this data field is now also available for other list exports to file. Customers can use this data field as content of their list exports to have a unique identifier for various purpose.

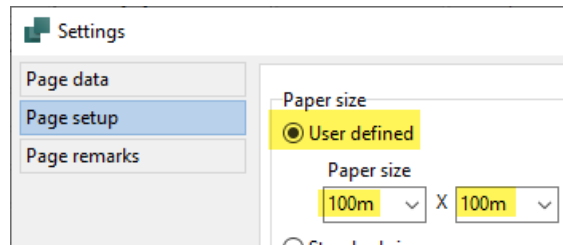
However, it is NOT possible to use such **Lists to file with Symbol ID** in the **Edit symbols data field** module in the **Edit contents** tab, because the Name and the Symbol ID must be combined in one cell for this. But when exporting lists to file, the Symbol ID is written to a separate column.



User defined paper size up to 100m x 100m possible

For specific purposes, it is now possible to set a user defined paper size for a diagram page of 100m x 100 m (100000mm x 100000mm).

Previously, this was limited to 10x10m.



Administrative option in "PCSCAD.INI" to disable automatic SQL query when entering part numbers

For specific use cases it is now possible in the PCSCAD.INI with a new parameter, to disable the automatic SQL query to the active database, when typing values into the field Article no.

For this purpose, you must add/enter in **PCSCAD.INI**, in the section

[SystemData]

the parameter **IgnoreUrlUpdate=Yes**

as displayed with the value **Yes**.

If this parameter is not present PCSCAD.INI or does not have the value **Yes**, the behavior is same as before.

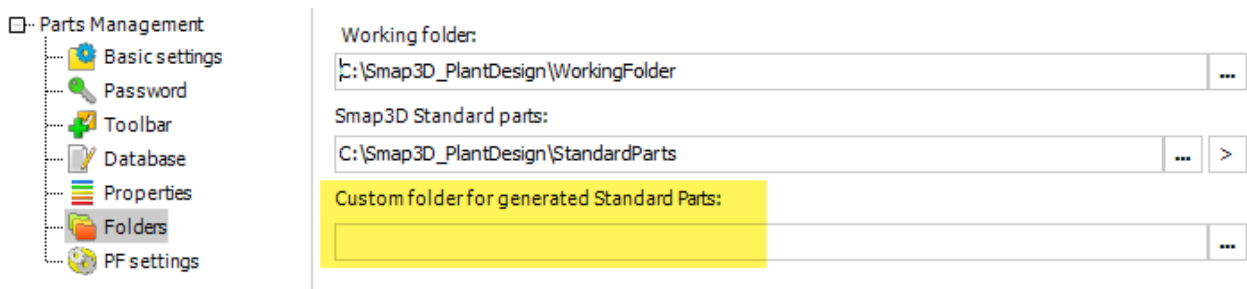
Changes to PCSCAD.INI (in the installation directory and the workstation folder) may **ONLY be made by an experienced administrator** with the P&ID application closed.

IMPROPER CHANGES TO THE PCSCAD.INI MAY RESULT IN THE P&ID APPLICATION NO LONGER BEING ABLE TO BE STARTED OR NO LONGER FUNCTIONING CORRECTLY!

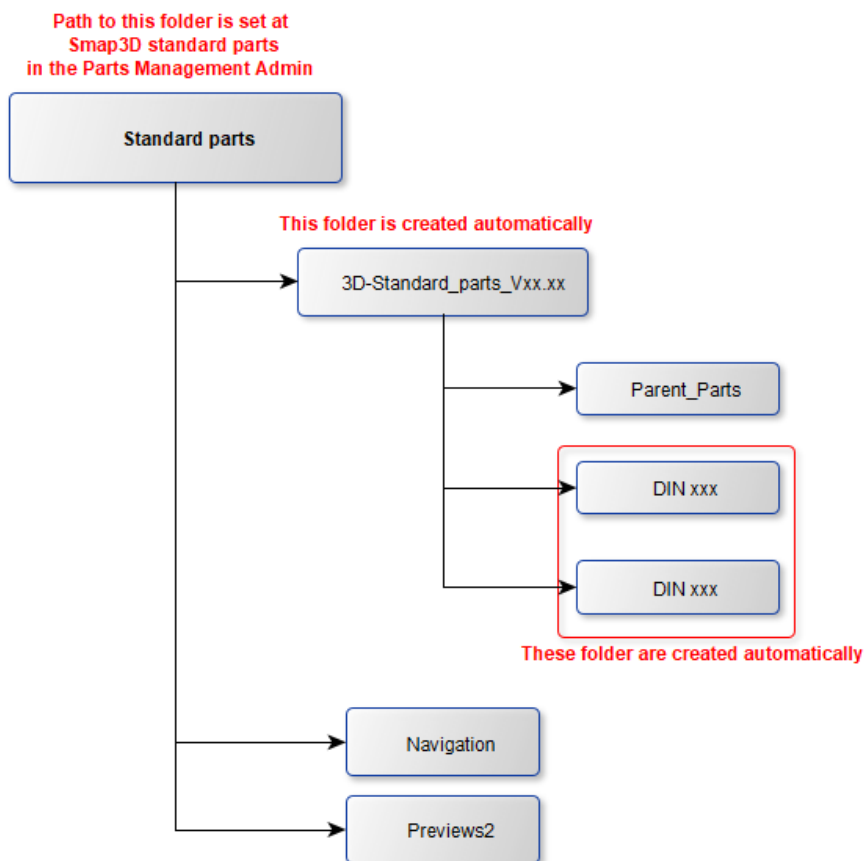
New in Smapp3D Parts Management

Parts Management Administrator - Custom folder for generated Standard Parts

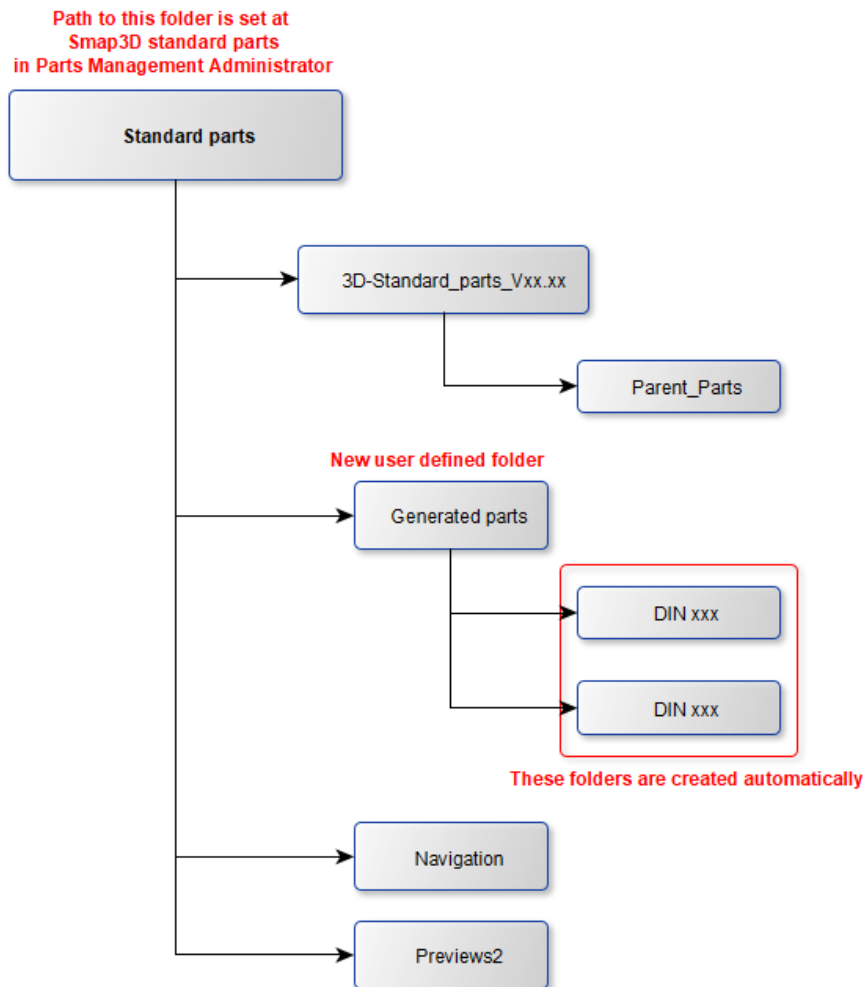
In the Parts Management Administrator, the new option **Custom folder for generated Standard Parts** has been implemented.



Until now, generated standard parts were always stored in folders (with the name of the respective standard) in parallel next to the **Parent_Parts** folder, in a fixed and non-influenceable manner according to the following scheme.



With the new setting (by creating and selecting the required user-defined path), it is possible to influence the storage path of the folders with the generated standard parts files.



New "PMAgent.exe" application

For all Smapi3D Parts Management workstation or client computers where the users do not have permissions to run the Parts Management Administrator, the new **PMAgent.exe** application can be used to load the saved settings of the central configuration file (PMGlobalSettings.pmsx) to the respective client computer.

This central configuration file also always contains the settings for the Parts Management toolbars in the CAD system.

Thus, with this new application, you can perform the initial setup of the Parts Management toolbars in CAD after a new installation, as well as reset the previous Parts Management toolbars to the default settings.

The equivalent application PipingAgent.exe for the Plant Design toolbars already exist since some version.

New in Smap3D Piping

New QuickPlace method "Elbow fixed angle"

In previous versions of Piping, the placement of elbow fittings for specific bend angles could only be automated, if we developed and provided a specific QuickPlace method to do so.

During the versions, some partly very customer specific QuickPlace methods for different bend angles came together here (e.g., **Bend 87, 88, 92, 93 and 95**).

To enable our customers to further automate elbow fittings, also with very customer-specific bend angles, but to avoid having repeatedly develop new/independent QuickPlace methods in our development, we are introducing the much more flexible QuickPlace method **Elbow fixed angle** with Piping 2023.

Nothing has changed in the previous QuickPlace methods for **elbow/bend fittings** in this regard.

The difference to the previous QuickPlace methods for elbow fittings is, that you can use the **Elbow fixed angle** method with associated [key name](#) multiple times in the same pipe spec and diameter for different elbow parts with different fixed bend angle.

The information for which exact bend angle on a pipeline path the respective elbow component is to be placed, is thus no longer determined by the QuickPlace method (and its key names) but the elbow part and a property defined therein.

Elbow fixed angle		80	-		
Elbow fixed angle	80	-		DIN EN 10253 Elbow fixed - 55 - DN80 - 88.9 x 3.2	
Elbow fixed angle	80	-		DIN EN 10253 Elbow fixed - 66 - DN80 - 88.9 x 3.2	
Elbow fixed angle	80	-		DIN EN 10253 Elbow fixed - 77 - DN80 - 88.9 x 3.2	

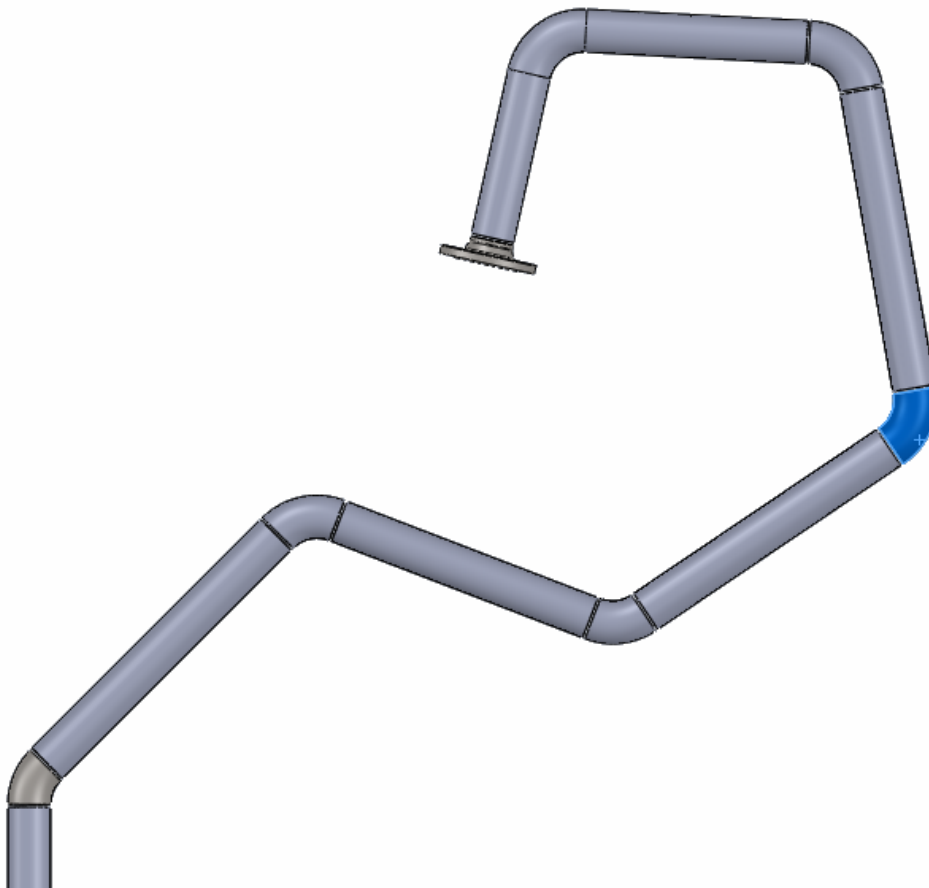
The requirements for this new QuickPlace method to work so flexibly, are as follows:

- for each diameter, wall thickness and specific bend angle, there must be a 3D model/CAD file with the corresponding geometries.
- the corresponding coordinate system pairs (e.g., created with CSWizard or ComponentWizard) must be placed at the two connection points. The CSInfo property must contain the corresponding parameters per connection point. Details see [here in the Plant Design online help](#).
- each of these CAD files must contain a custom property named **Angle**. This property name is language independent and must be written exactly like this, no matter in which user interface language you use the Smap3D software.
- the fixed bending angle of the respective 3D model must be specified as the value of the user-defined **Angle** property. This value must be written with the valid unit character for the angle. For example, **66°**

- each of these CAD files must be available in the active standard parts provider (or in the additional standard parts provider).

Property Name	Type	Value / Text Expression
Angle	Text	66°

In combination with other **elbows with fixed angle components**, you can now flexibly create individual pipelines with different bending angles without having to use the QuickPlace method universal elbow.



New QuickPlace method "Tee fixed angle"

As above described for the new QuickPlace method [Elbow fixed angle](#), tees with specific branch angles could only be automated if we developed and provided specific QuickPlace methods for this purpose.



Also, for this we have created some very customer specific QuickPlace methods for branch angles so far (e.g., **Tee 85** and **87**).

To enable our customers to further automate tee fittings with customer-specific branch angles, but to avoid having to repeatedly develop new QuickPlace methods for this purpose in our development department, we are introducing the much more flexible QuickPlace method **Tee fixed angle** with Piping 2023.

Nothing has changed in the previous QuickPlace methods for tee fittings in this regard.

The difference to the previous QuickPlace methods for tees is, that you can use the **Tee fixed angle** method with associated [key name](#) multiple time in the same pipe spec and diameter for different tee parts with different branch angle.

The information for which exact branch angle on two pipeline paths respective tee component is to be placed, is thus no longer determined by the QuickPlace method (and its key names) but the tee part and a property defined therein.

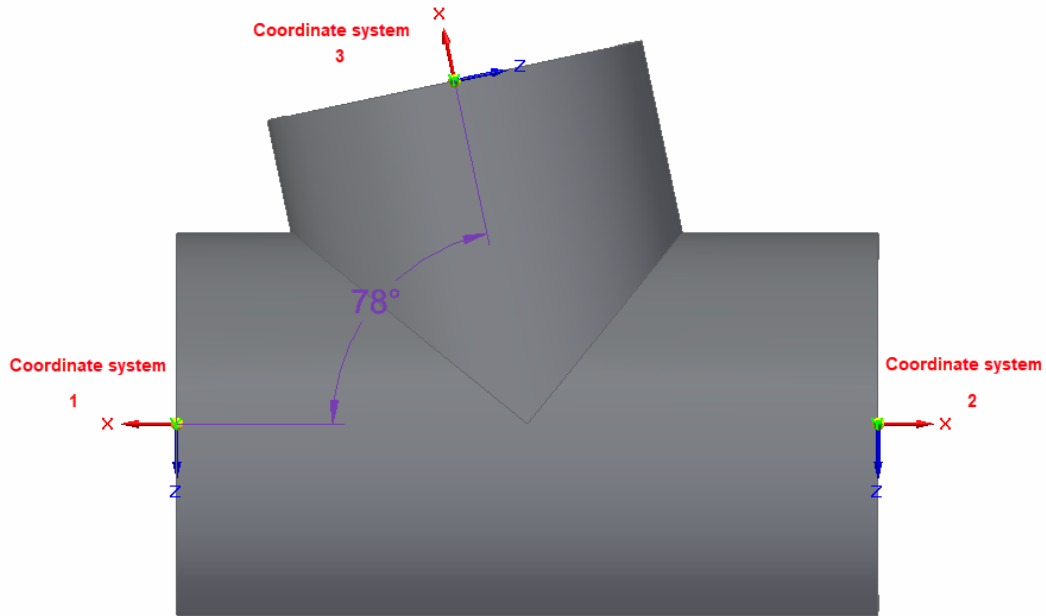
Method	Angle	Unit	Icon	Model Name
Tee fixed angle	80	-		custom angle Tee - 78 - 88,9 x 3,2
Tee fixed angle	80	-		custom angle Tee - 82 - 88,9 x 3,2

The requirements for this new QuickPlace method to work so flexibly, are as follows:

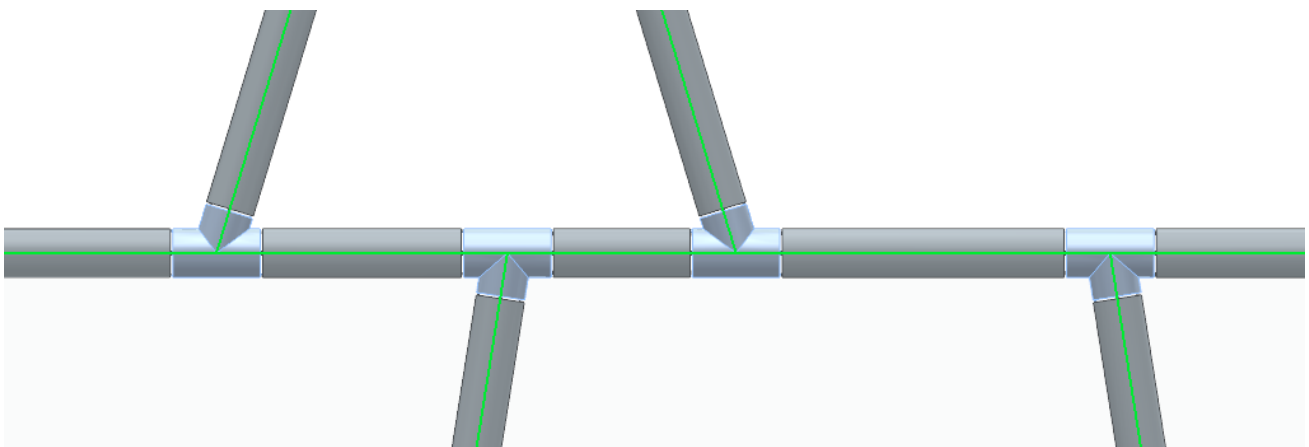
- for each diameter, wall thickness and specific branch angle, there must be a 3D model/CAD file with the corresponding geometries.
- the corresponding coordinate system pairs (e.g., created with CSWizard or ComponentWizard) must be placed at the three connection points. The CSInfo property must contain the corresponding parameters per connection point. Details see [here in the Plant Design online help](#).
- the **branch angle must be defined/measured between the first and the third connection point**.
- each of these CAD files must contain a custom property named **Angle**. This property name is language independent and must be written exactly like this, no matter in which user interface language you use the Smapp3D software.
- the fixed branch angle of the respective 3D model must be specified as the value of the user-defined **Angle** property. This value must be written with the valid unit character for the angle. For example, **78°**

- each of these CAD files must be available in the active standard parts provider (or in the additional standard parts provider).

Angle	Text	78°
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If all requirements are fulfilled, it is now possible to create very flexible and automatic individual pipelines with different tee branch angles without having to place the special tees manually at the branches.



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 two 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.

Special_1	Custom gasket	Flange	4,50.82	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Gasket_3	Custom gasket	Flange	4,50.83	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

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

Special_1	80	-	Spiral-Wound Gasket ASME B16.20 4 - 2500	US Standards - Metric design\Gasket\Gasket for flange\Spiral wound gasket\G...	ASME_B16.20_2017_mm_A_Gasket_4_2500_v16.00...	<input checked="" type="checkbox"/>
Gasket_3	80	-	Lense 72x116 DIN 2696	DIN\Gasket\Gasket for flange\Stainless steel gasket\Lense\DIN 2696:1972	DIN_2696_1972_Lense_PN250_72x116_v14.00.par	<input type="checkbox"/>

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

Pipe specification row detail

Basic | Part | **Virtual characteristics** | Additional items

Name	Value
ADDITIONAL_ITEM_1_BOLT_ITEM_CODE	123
ADDITIONAL_ITEM_1_BOLT_QUANTITY	2
ADDITIONAL_ITEM_1_BOLT_DIA	12
ADDITIONAL_ITEM_1_BOLT_ITEM_DESCRIPTION	Special1 Bolt

OK Cancel

Pipe specification row detail

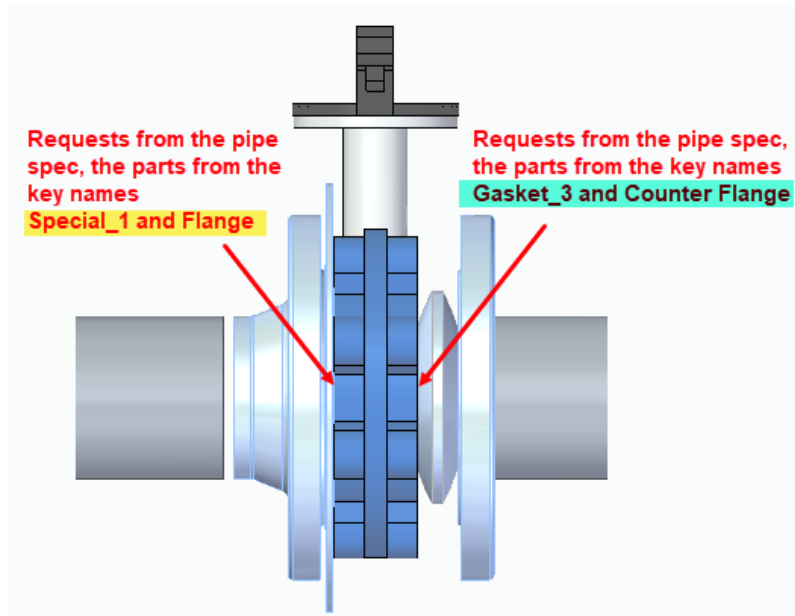
Basic | Part | **Virtual characteristics** | Additional items

Name	Value
ADDITIONAL_ITEM_1_BOLT_DIA	15
ADDITIONAL_ITEM_1_BOLT_QUANTITY	4
ADDITIONAL_ITEM_1_BOLT_ITEM_CODE	987
ADDITIONAL_ITEM_1_BOLT_ITEM_DESCRIPTION	Gasket_3 - Bolts

OK Cancel

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, are processed, and displayed accordingly during the generation of drawings (depending on the used style).

MATERIAL LIST - FAE			
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 Smapp3D Piping, we have developed and added the new QuickPlace method (placement automation) **Welding nipple**.

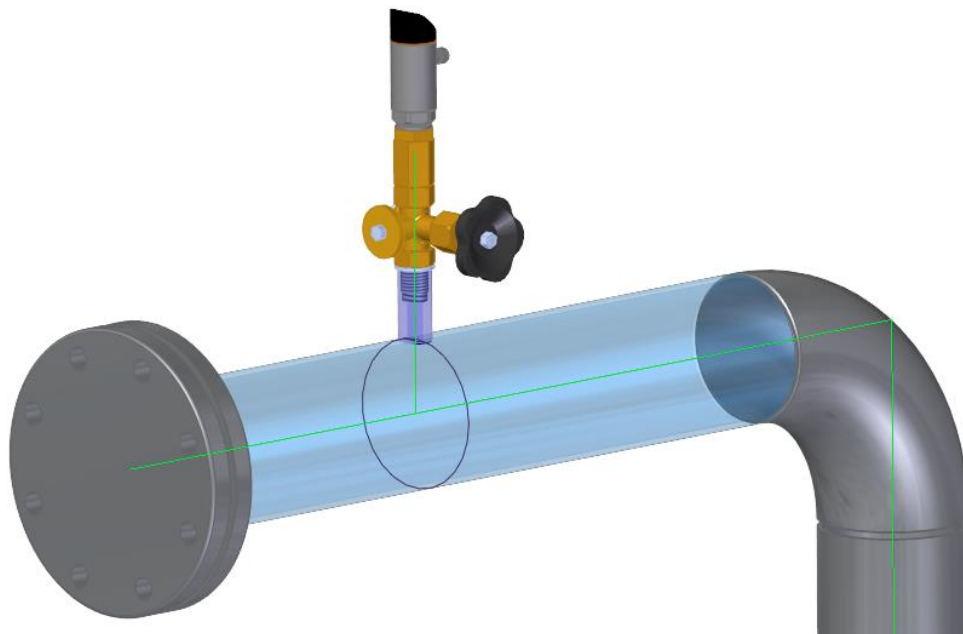
This new automation is specially designed for all branches on pipelines where standard components such as nipples, olets, etc. are to be placed, but where **NO further pipeline generation is desired along the drawn branch path**.

Therefore, if Smapp3D 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 executed**.

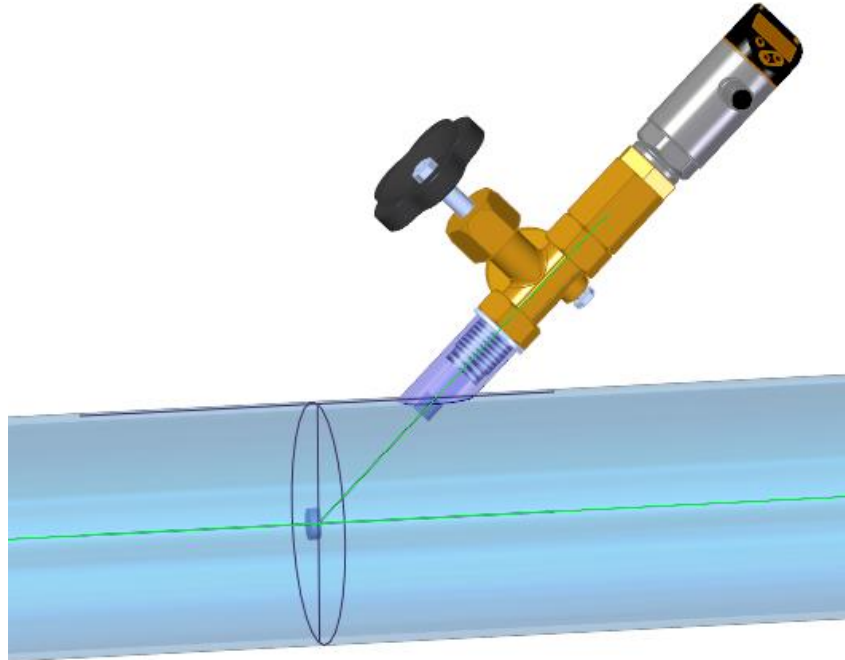
In practice this is often used for standard branches for measuring instruments, probes, drains, etc. In these cases, a standardized nipple component is usually welded onto a pipe, on which a specific component is then built directly, without the need to place any other pipes in between.

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 QuickPlace methods for tube fittings

For the automated use of tube fitting components with Smap3D Piping, we have developed and added several new QuickPlace methods (placement automations).

These new automations are intended for placement of tube fitting components to **EN ISO 8434-1** through **-3**, as well as manufacturer-specific tube fitting systems as Parker®, Ermeto®, Swagelok® and others.

Typical for such tube fitting components are the different **connection variants at the component ends**.

The main distinction is between a **Spigot end** (or tapered spigot end) and a **Socket end** (or tapered socket end).

Since the development of these new methods was mainly based on the **EN ISO 8434-1 to -3** standard, our name for the QuickPlace methods is selected accordingly.

For the first time, we have **named these QuickPlace methods not after the part** that is automatically placed, **but after the part connection (cone spigot ends, cone socket ends, and red. cone spigot ends and red. cone socket ends)** to which the add-on components are automatically placed.

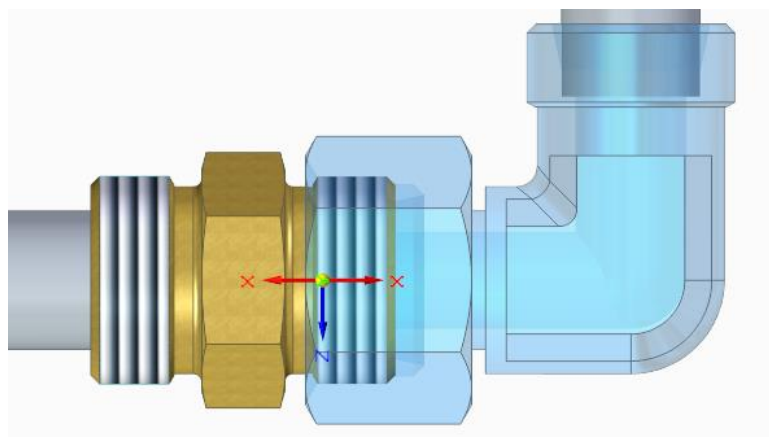
Our goal is to make the use of automations available to our customers as universally as possible.

QuickPlace method for "cone spigot ends"

The new QuickPlace method **Cone spigot ends**, places a component defined in a pipe specification under the Key name **Threaded joint nipple** to existing components (e.g., bends, tees, etc.), which contain the definition of **Cone spigot ends** (with **ET16** end-treatment) at a connection end.

For the use cases known to us, the automatic placement of a single tube fittings component (e.g., nipple) at such connection ends is required.

Here in the picture, the defined screwed fitting nipple (brass) was automatically placed at the cone spigot end of the bend (blue transparent).

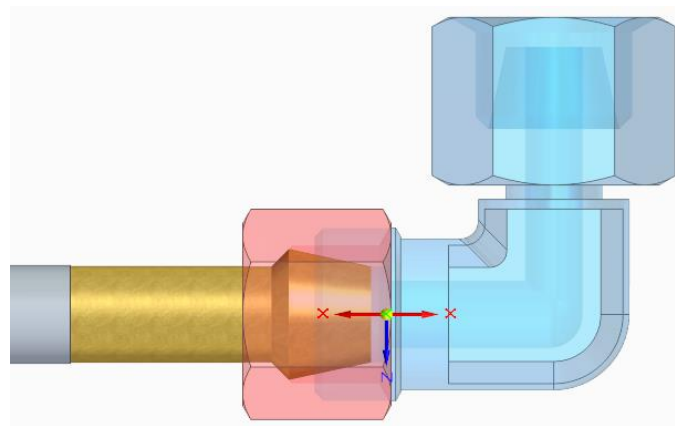


QuickPlace method for "cone socket ends"

The new QuickPlace method **Cone socket ends**, places the components defined in a pipe specification under the Key name **Threaded joint socket unit** to existing components (e.g., bends, tees, etc.), which contain the definition of **Cone socket ends** (with **ET15** end-treatment) at a connection end.

For the use cases known to us, the automatic placement of up to three individual components (e.g., welding nipple and nut) at such connection ends is required.

Here in the picture, the defined welding nipple (brass) plus nut (red transparent) was automatically placed at the cone socket end of the bend (blue transparent).

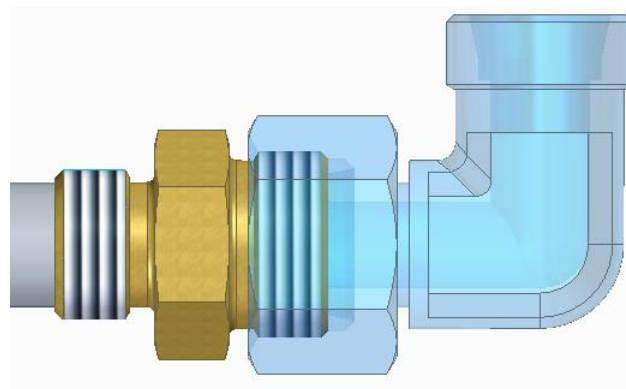


QuickPlace method for "reduced cone spigot ends"

The new QuickPlace method **Red. cone spigot ends**, places one component defined in a pipe specification under the Key name **Red. threaded joint nipple**, when generating a reduced pipeline with Smap3D Piping (e.g., at reduced tee situations) always to an existing component which contains the definition **reduced cone spigot end** (with **ET16** end-treatment) at a connection end but is placed and associated at the larger pipe diameter.

In most use cases, the automatic placement of a single tube fitting (e.g., reduced nipple) at this connection end is required.

Here in the picture the defined reduced nipple (brass) was automatically placed at the cone spigot end of the bend (blue transparent).



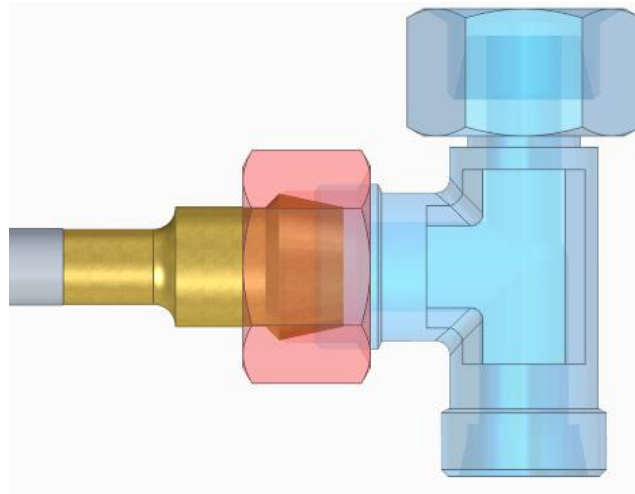
QuickPlace method for "reduced cone socket ends"

The new QuickPlace method **Red. cone socket ends**, places the components (up to 3 individual components) defined in a pipe specification under the Key name **Red. threaded joint socket unit**, when generating a reduced pipeline with Smap3D Piping (e.g., at reduced tee situations) always to an existing component which contains the definition **reduced cone socket ends** (with **ET15** end-treatment) at a connection end, but is placed and associated at the larger pipe diameter.

Most use cases require the automatic placement of up to three individual components (e.g., reduced welding nipple and nut) at this connection end.

Usually, only **component 1** is reduced. If a component 2 or a nut is necessary, it has the normal diameter as the "large" pipeline.

Here in the picture, the defined reduced welding nipple (brass) plus nut (red transparent) was automatically placed at the cone socket end of the tee (blue transparent) in a reduced tee situation.



New QuickPlace methods for Shape pipes and Shape components

For the automated use of non-round pipes/ducts and fittings/components with Smapi3D Piping, we have developed and added several new QuickPlace methods (placement automations).

QuickPlace method "Shape pipe"

Unlike pipes (round), the generation of shaped pipes uses the actual 3D geometry of the defined shape pipe parts.

Shape pipe parts must be registered as a standard part in the standard parts management.

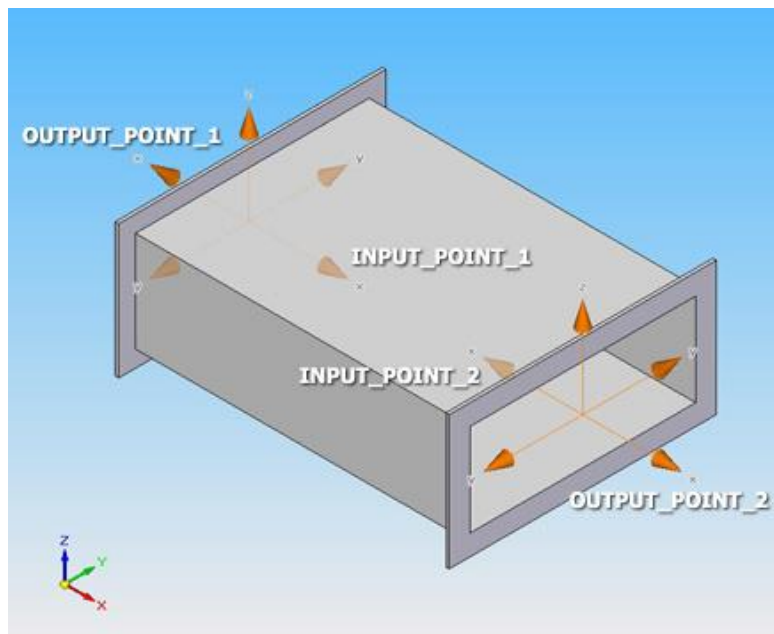
Furthermore, they must contain a variable characteristic for length in the standard parts management.

The 3D geometry of shape pipes must contain a pair of coordinate systems (INPUT_POINT and OUTPUT_POINT) for each pipe end.

The X - axes of the coordinate systems must always point in the INPUT and OUTPUT direction.

The orientation of the coordinate systems also determines the geometrical orientation of the generated shape pipes.

Shape pipes and shape components are connected in such a way, that the INPUT_POINT of a shape pipes matches the OUTPUT_POINT of a shape component and vice versa.



QuickPlace methods for shape components

The new QuickPlace methods (and key names) for shape components are as follows:

- Shape Bend 90 (left, right, up, and down)
- Shape Bend 45 (left, right, up, and down)
- Shape Tee (left, right, up, and down)
- Shape Tee 45 (left, right, up, and down)

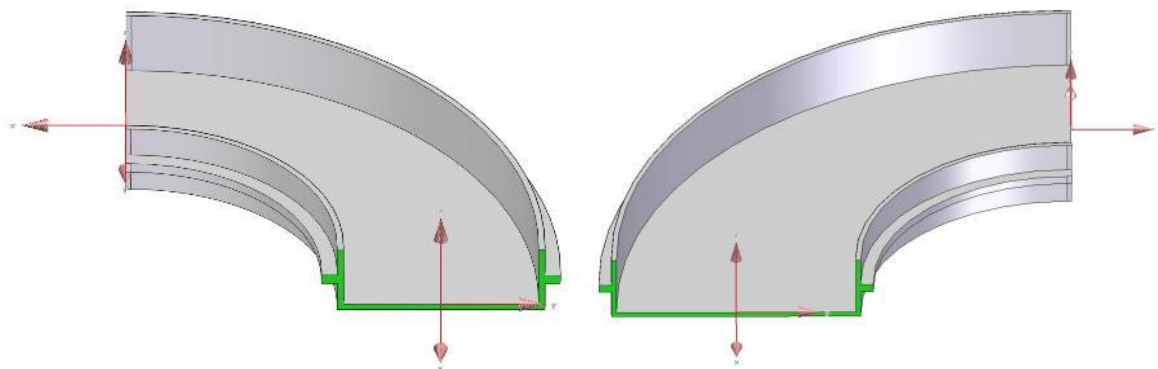
The use of shape components demands specific requirements in comparison to the use of normal (round) fittings.

Shape components also require coordinate systems, and these are necessary for assembling shape pipes for creating axial relationships/mates with the pipeline path.

The coordinate system OUTPUT_POINT_1 has a very special importance for shape components. It is used as a referenced for determining the component alignment.

In the picture you can see examples of a left and right cable duct elbow.

The coordinate system displayed in the foreground (in both cases) is the OUTPUT_POINT_1 coordinate system.



For Smap3D Piping to work correctly, each shape component must be registered as a standard part in the standard parts management.

QuickPlace method "Shape gasket"

The following requirements apply to shape gaskets, i.e., the components which are inserted between shape pipes and shape components which have defined Shape Component with Gasket End Treatment (ET=1004).

For shape gasket, there is just a single **CO_ORDS** coordinate system required.

Orientation and position of the coordinate system must correspond to the **OUTPUT_POINT** coordinate system of the shape pipe/component end to which the gasket should be attached to.

Shape gaskets are inserted so that the **CO_ORDS** coordinate system orientation and position matches the orientation and position of the **OUTPUT_POINT** (with ET=1004) coordinate system of the adjacent shape pipe/component.

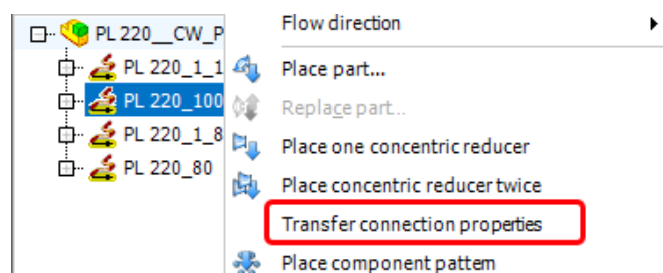
To get proper results, the origin of the coordinate system should be positioned at half the thickness of the gasket.

New function "Transfer connection properties"

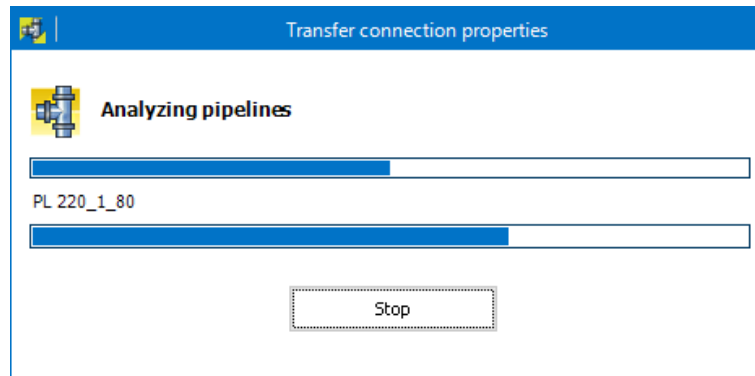
Smapp3D Piping and Smapp3D Isometric can only evaluate and use information that are stored in an active piping assembly and its components. Automatic evaluation of information from parallel or higher-level assemblies (from the point of view of an active piping assembly) is not possible.

To provide customers with information about "external" assemblies (equipment) to which piping assemblies are connected, and to use/map them e.g., in Isometric drawings, we have developed the **Transfer connection properties** function for Smapp3D Piping.

This is available in the Piping context menu, when one or more pipeline paths are selected.



A window with a progress bar displays the steps taking place there.



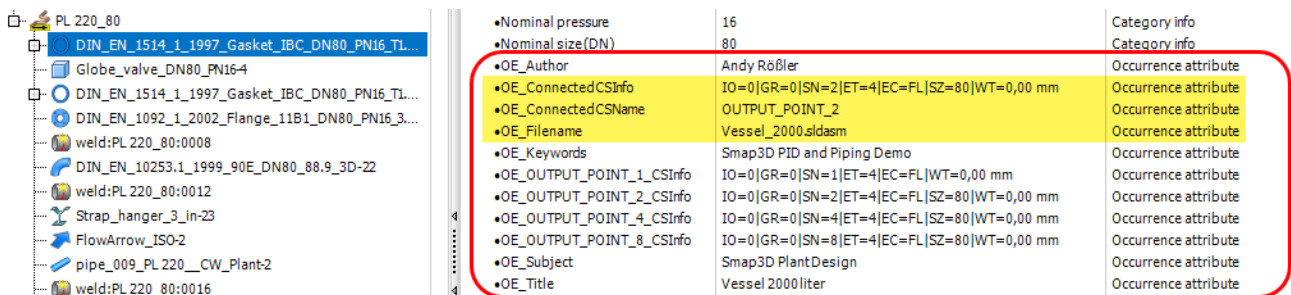
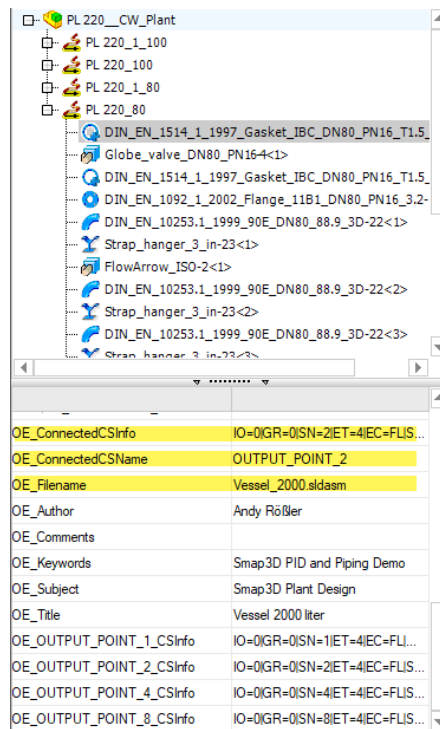
- In the first step, any existing old connection information is deleted from the paths/parts within the active assembly.
- Then the function searches for the selected pipeline paths (via the CAD API) for the "external" assemblies/components to which the 3D sketch lines of the paths are connected with relationships/mates.
- If a found external component meets the basic requirement (i.e., it is correct coordinate system with corresponding CSInfo property at the connected/attached point of a 3D sketch line in an external assembly/component) then all available file properties are read from it.

- In the active pipeline assembly, the read information (file properties with values) per connected line/**path end** is written to the "last" component placed there (e.g., gasket or flange or also pipe) as virtual characteristic with the prefix **OE_**.
- The most important of the transferred information are **OE_ConnectedCSName** (corresponds to the name/number of the connector on external component to which this path end is connected), **OE_ConnectedCSInfo** (corresponds to the complete CSInfo information for this connector) and **OE_Filename** (corresponds to the file name of the external component, to which the path end is connected).

These steps are performed for each selected pipeline path and its path ends, where the 3D sketch lines are connected to external components.

The transferred connection properties are then displayed as virtual characteristics in the Piping window and in Isometric structure tree (step 2).

Through appropriate Isometric mappings and rules, this information can be brought to ISOGEN and onto the Isometric drawing.



New function "Import path sketch lines from PCF"

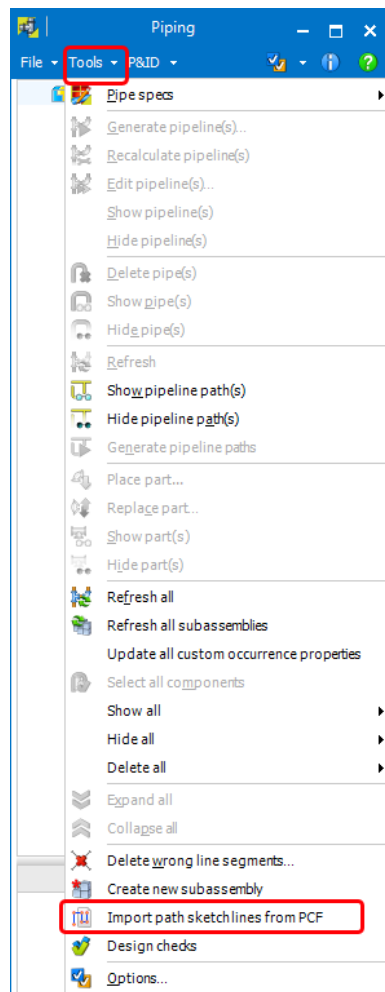
For all users/customers who want/need to use existing piping isometrics in the form of PCF files for 3D pipeline creation with Smapp3D Piping, the **Import path sketch lines for PCF** function is now available.

This function is intended for users who must reconstruct pipelines from existing plants (brown field engineering) based on existing PCF files in their 3D CAD system or migrate them from other systems.

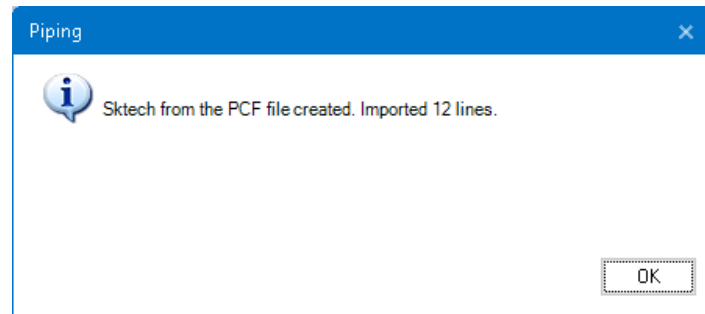
For this reason, this function **can NOT be used with the automatic route functions**, but the "old" basic way for 3D pipeline creation must be used.

The related procedure is as follows:

- A user activates or opens an existing subassembly (native CAD function **Edit assembly** or **Open assembly**) or creates the required piping subassembly in which the path is to be imported using the Piping function **Create subassembly**.
- The **Import path sketch lines from PCF** function is started from the **Tools** menu of the Piping window.



- The **PCF file select** window opens. The required PCF file for the import must be selected there.
- Depending on the content of the selected PCF file, the user then receives information on how many line segments have been imported into the automatically created 3D sketch that is now active in the CAD working window (Edit mode).



- Depending on the user requirements, the active 3D sketch can be extended with dimensions and/or sketch relation/mates (with native CAD functions. If the 3D sketch meets the requirements, then the **3D sketch editing mode must be closed/exited**.
- From the Piping toolbar, the **Generate pipeline paths** function must be executed.
- After selecting the required pipeline paths, the 3D pipelines are then generated using the Piping function **Generate pipelines**.

New function "Create pipe summary files"

The new **Create pipe summary files**, is specifically intended for customers whose CAD system is integrated with Smmap3D Piping in an environment managed by an ERP/PLM system.

In such environments, there is often the requirement that the pipe length summation for the pipe bill of materials must take place with native CAD BoM function in the ERP/PLM system, but these can only react to the file names of the pipe files.

In such cases, there was previously the problem that the individual 3D pipe files generated by Piping (each with an individual file name) could not be summarized and evaluated with the native CAD list functions in the ERP/PLM system as desired by the customer.

To solve this problem, we now provide this new function to perform a required pipe length summation of identical pipes in the 3D pipelines already with Smmap3D Piping. The native CAD BoM functions then only need to transfer one **pipe summary file** with the summed information to the ERP/PLM system, and not many individual 3D pipe files as before.

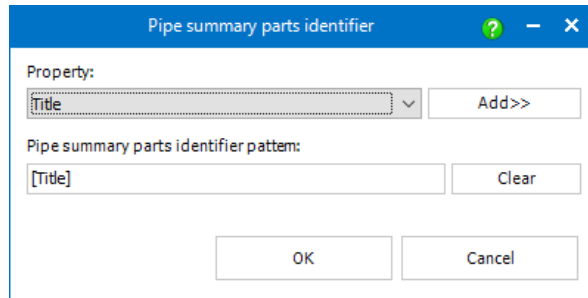
Since this is a very specific function, it must be explicitly activated and configured in the Plant Design Administrator (Piping common).

Only if **Create pipe summary files** is checked and saved, the configuration area is activated, and the function becomes available in the Piping application.

The screenshot displays the configuration interface for Smmap3D Piping. It is divided into several sections:

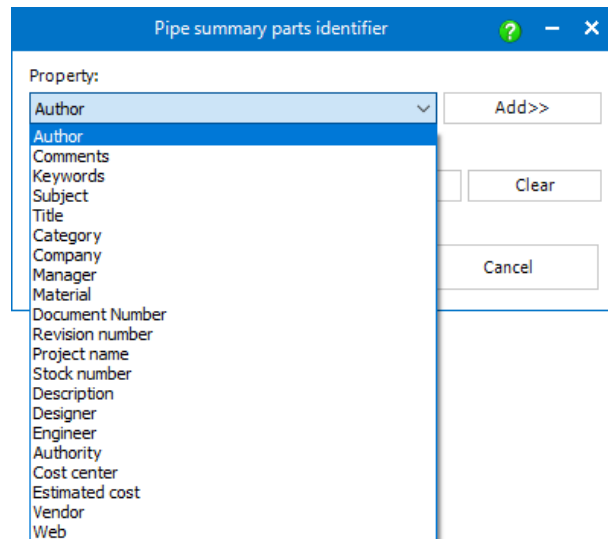
- Placing of parts:** Contains three checkboxes:
 - Create an axial relationship on flanges and gaskets
 - Don't write properties from Bend to Bend universal
 - Create pipe summary files (highlighted with a red box)
- Characteristics file:** Shows a text field with the path "C:\Smmap3D_PlantDesign\MultiLang.pip" and an "Edit" button.
- VC Writer settings:** Shows a text field with the path "C:\Smmap3D_PlantDesign\VirtualCharsWriterNames.txt" and an "Edit" button.
- Min. lengthtolerance:** Input field with value "10,00 mm".
- Maximum flat length:** Input field with value "6000,00 mm".
- Flat length at bends:** Input field with value "0,010 mm".
- Bend angle tolerance (0-10°):** Input field with value "0,0 °".
- Properties:**
 - Precision of property "Length": Spinners set to 0 and 1.
 - Precision of property "Bend": Spinners set to 1 and 0.1.
 - Write units to separate properties
- Pipe summary parts:** (highlighted with a red box)
 - Pipe summary parts identifier pattern: Input field containing "[Title]" and an "Edit" button.

In the configuration area (in Plant Design Administrator), a single CAD property or a combination of several must then be defined as the identification pattern for the summation. Clicking **Edit** opens the following dialog.



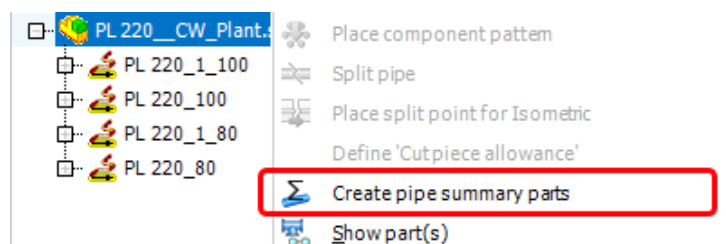
In the list box of the **Property** row, the native CAD standard properties are available for selection as identifier.

At least one property must be selected and set as an identification pattern using the **Add** button.



Based on the identification pattern defined here, when the function is executed in Smapp3D Piping, the 3D pipe files present in the piping assembly are analyzed, summed and then the resulting number of pipe summary files in the assembly are generated.

This function is started in the Smapp3D Piping context menu (without preselection of paths).



As result of this function, you will receive at least one pipe summary file in your CAD structure tree (NOT in the Piping structure tree), with a file name according to the associated file name generator setting in Plant Design Administrator.

Pattern for pipe summary parts ▲

summary_?????,summary_00000	Default
-----------------------------	---------

Use assembly name as a prefix for the filename of pipe summary parts

Use assembly name as a suffix for the filename of pipe summary parts

A pipe summary file does NOT contain 3D geometry (but only file properties) and is NOT placed on a pipeline path (therefore NOT available in the Piping structure tree) but placed at the origin of the active CAD assembly.

In the properties transferred by our function from the individual pipes, the **Length** property contains the length summation of all the individual lengths of the 3D pipe files in the assembly, what have the identical value of the identifier.

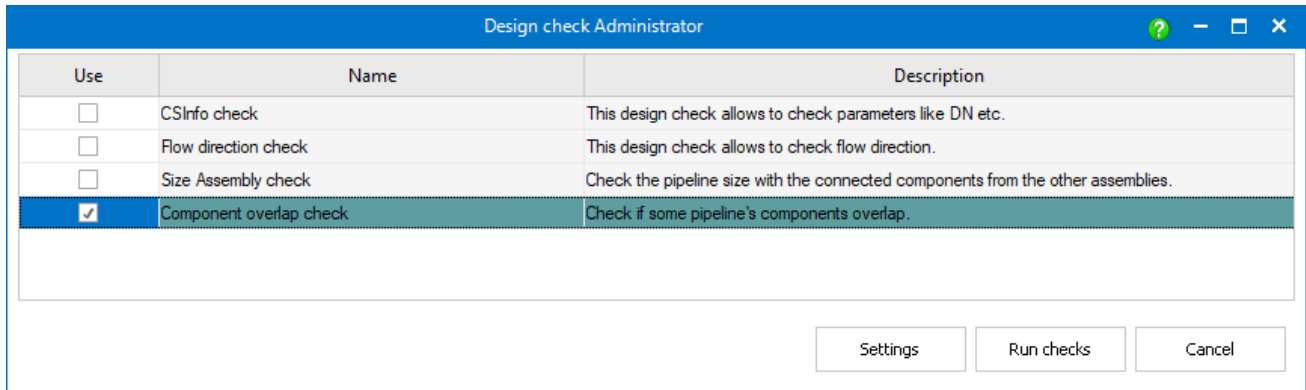
PipeSpec	Text	Spec10CA
SummaryID	Text	Tube 114,3 x 3,6 - EN 10220
Length_Unit	Text	mm
Length	Text	2608
Piping	Text	True

These pipe summary file(s) are then used in the CAD BoM with ERP/PLM integration, instead of the individual pipe files generated by Smap3D Piping.

New "Component overlap check"

The new design check **Component overlap check** now allows to check a piping assembly for component overlaps already in CAD.

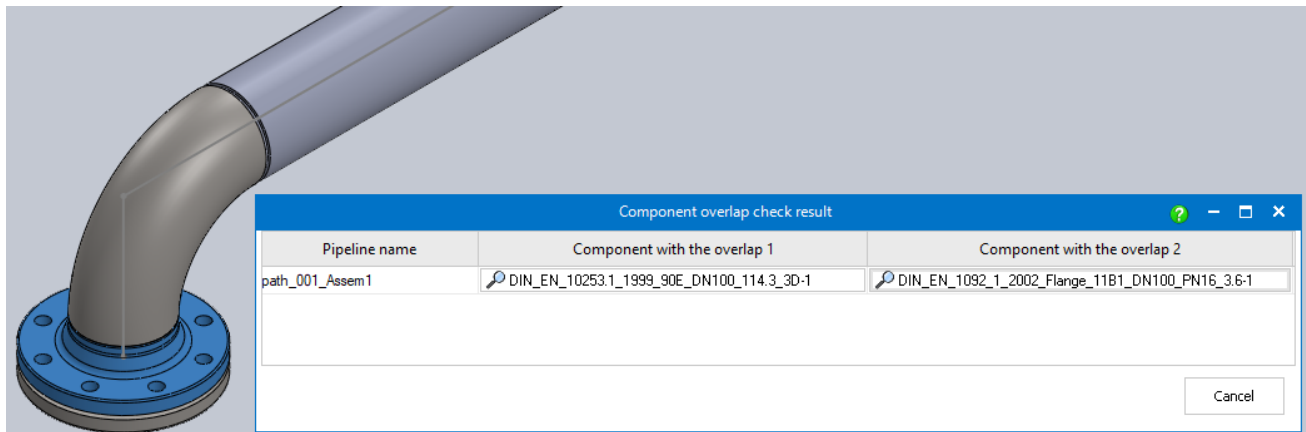
Until now, a user could only use the native CAD function for collision analysis to find possible volume overlaps of fittings in a pipeline.



When running this design check, the coordinate systems (INPUT/OUTPUT) contained in the fittings are checked for overlap, not their solid model.

If overlaps are found, the affected components are displayed individually in a result dialog.

By clicking on the respective entry, the component is highlighted in the CAD work window.



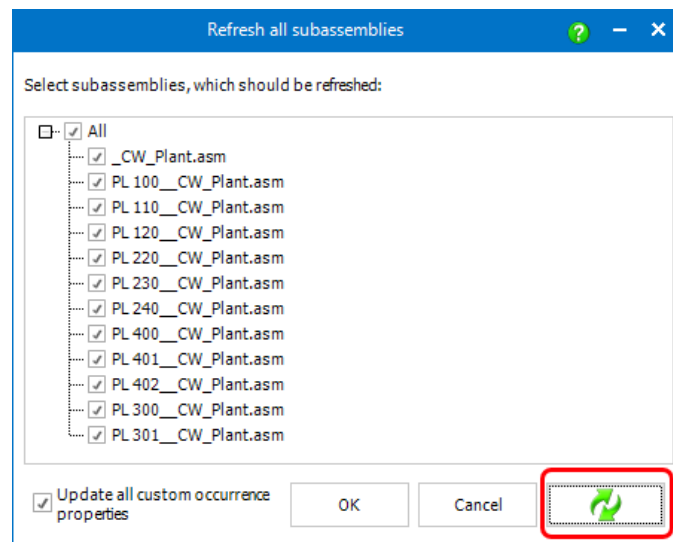
New button/function "Refresh list of all piping subassemblies"

In the dialog of the function **Refresh all subassemblies** there is a new button **Refresh list of all piping subassemblies**.

This allows a user to have the displayed list of subassemblies to be edited updated.

After executing data management procedures (e.g., renaming or duplicating a pipeline assembly, or similar), it may happen that the respective pipeline assembly is not displayed/listed correctly in the dialog of these functions.

This new function has been added to obtain the status of the list of piping assemblies before executing the **Refresh all subassemblies** function.



New function "Place split point for Isometric"

For later use in Smap3D Isometric it is possible to define isometric split points in the 3D pipeline at which the isometric drawings are then split (except for spools).

The **Place split point for Isometric** function automatically places the **Iso_split_point.ipt** component which contains only the required coordinate systems and the user defined ComponentType property.

In the Smap3D Piping structure tree, select the pipe or component to which you want to place an isometric split point and start the **Place split point for Isometric** function from the context menu.

If a fitting has been selected, a dialog box appears in which you must select the desired coordinate system of the fitting, using the **arrow button**.

If a pipe has been selected, the **Split Definition for Isometric** dialog appears. Enter the desired distance for the split point (the point on the pipe where the isometric drawing is split).

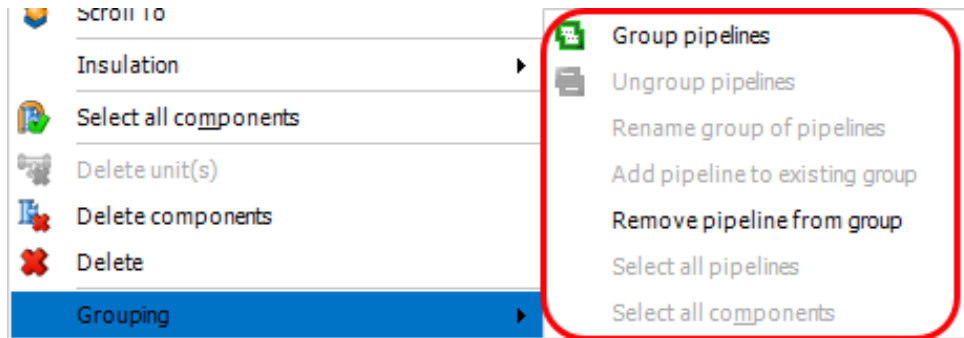
After confirming the respective dialog with **OK**, the **Iso_split_point.ipt** file is automatically placed and displayed in the Piping structure tree.

New functions for grouping in the structure tree

The **Group pipelines** function (and its sub-functions) allows the user to arrange the structure of pipelines (pipeline paths) in the Piping window in a clearer way. The groups can be composed according to different criteria.

However, a pipeline (pipeline path) cannot be assigned to several groups at the same time!

To group pipelines, select the desired pipeline paths in the Piping window and use the **Grouping** menu commands in the context menu as needed.

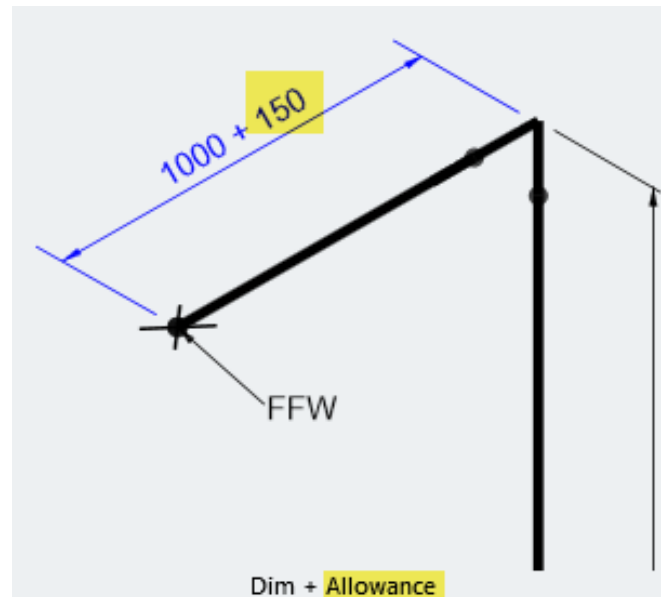


New function "Define Cut piece allowance"

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.

Smapp3D Isometric and Isogen© have always been able to work with this parameter and produce the corresponding results on a drawing.



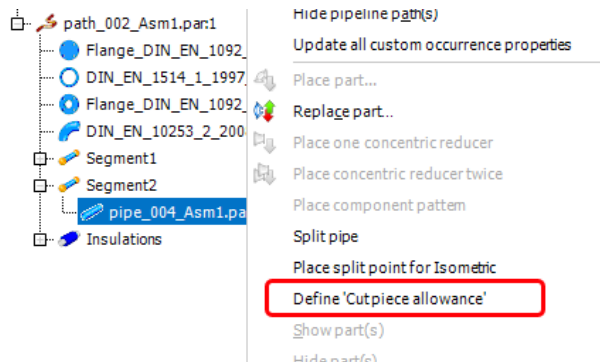
With the previous versions of Smapp3D Plant Design, a definition for the **CUT_PIECE_ALLOWANCE** parameters were either written to a 3D pipe with the **VC Writer** or specified as text attributes for a pipe in Smapp3D **Isometric Step 2** during the generation of a drawing.

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 Smapp3D 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.

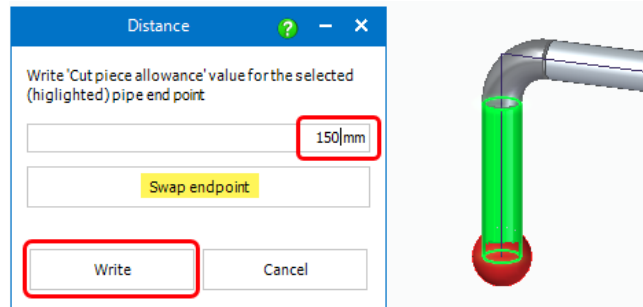
The new function is started by selecting a pipe in the Piping structure tree and clicking on the **Define Cut piece allowance** entry in the context menu.



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.

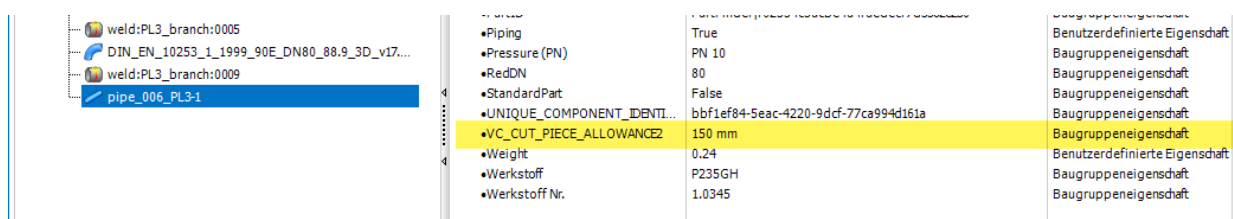
With the **Write** button, the parameter with value is written to the pipe/pipe end as a virtual characteristic.



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.



New function "Delete unit(s)"

The **Delete unit(s)** function is available in the context menu when a fitting is selected. This function deletes fittings that are related to the selected fitting with assembly relationships (e.g., a flange with gasket to fittings and fitting to gasket with counter flange).

After calling the function, the related fittings are highlighted, and the user must confirm the deletion process.

New in Pipe specification editor

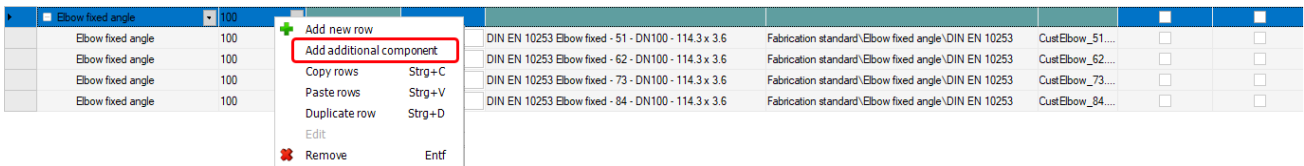
New key name "Elbow fixed angle"

For the automated use of individual elbow components for custom bend angles with Smap3D Piping, we have developed and added the new QuickPlace method Elbow fixed angle.

With version 2023, the corresponding new key name **Elbow fixed angle** is available in the pipe specification editor.

***i** This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade to Plant Design 2023 (Installation option - Existing data and configuration).*

Using the **Add additional component** function (in the context menu), you can define several rows (different CAD files with different bending angles) for the same diameter in a pipe specification for the key name.



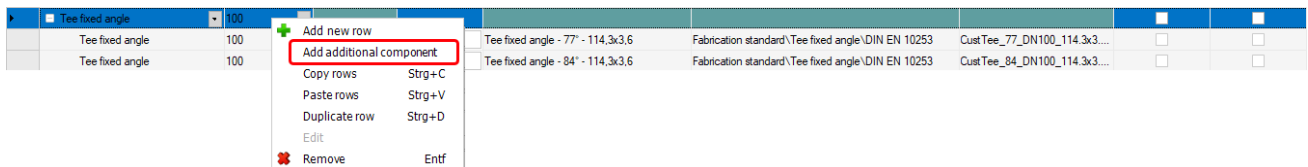
New key name "Tee fixed angle"

For the automated use of individual tees for custom branch angles with Smap3D Piping, we have developed and added the new [QuickPlace method Tee fixed angle](#).

With version 2023, the corresponding new key name **Tee fixed angle** is available in the pipe specification editor.

i *This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade to Plant Design 2023 (Installation option - Existing data and configuration).*

Using the **Add additional component** function (in the context menu), you can define several rows (different CAD files with different branch angles) for the same diameter in a pipe specification for the key name.



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**.

The new key name **Custom gasket** with corresponding QuickPlace method Custom gasket is available in the pipe specification editor.

i This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade to Plant Design 2023 (Installation option - Existing data and configuration).

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 key name **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:

The screenshot shows a 'Key name editor' dialog box with the following fields and values:

- Key name: Special_1 (highlighted in green), EN
- Add by edit
- Order: 4,50
- Reduction
- QP method: Custom gasket
- Non-unique index: 82 (highlighted in green)
- Conditions: Flange
- Is pipe

At the bottom, there is a 'Basic' dropdown menu, 'OK', and 'Cancel' buttons.

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

If Smapp3D 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 Smapp3D Piping, we have developed and added the new QuickPlace method **Welding nipple**.

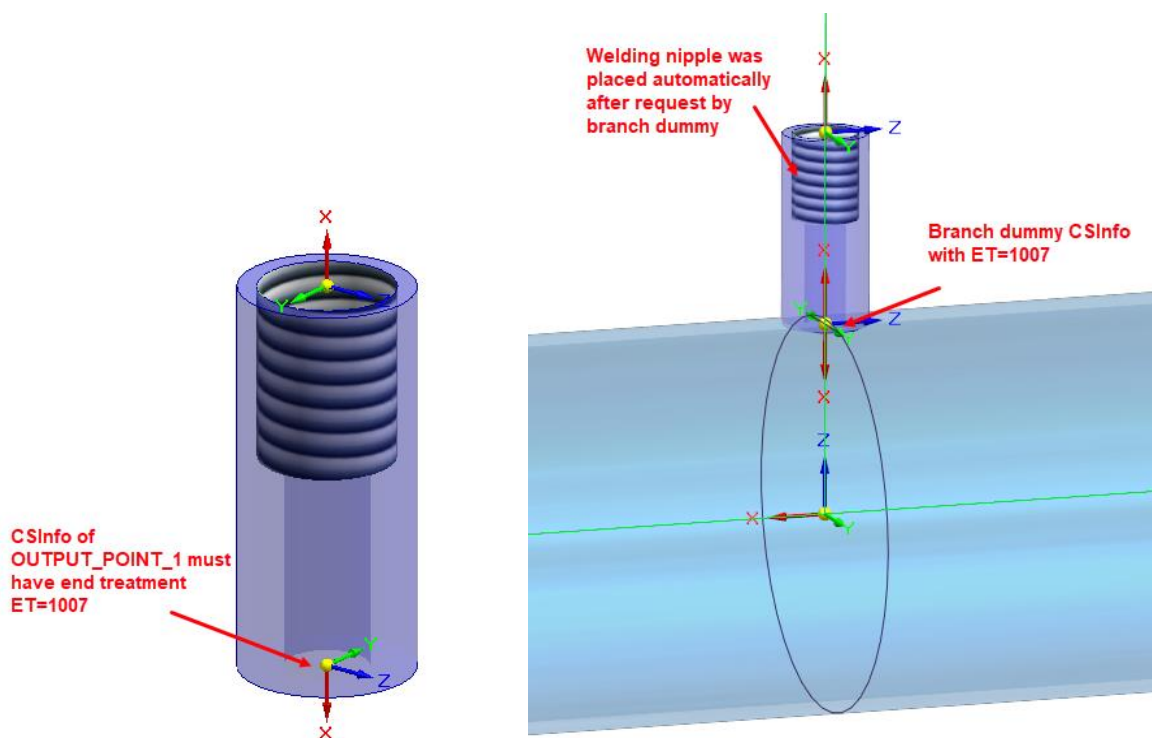
The new key name **Welding nipple** with corresponding QuickPlace method **Welding nipple** is available in the pipe specification editor.

i This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade to Plant Design 2023 (Installation option - Existing data and configuration).

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 Smapp3D 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 key names for tube fittings

The new key names **Threaded joint nipple**, **Threaded joint socket unit**, **Red. threaded joint nipple** and **Red. threaded joint socket unit** with corresponding QuickPlace methods for tube fittings are available in the pipe specification editor.

i *This new key name is automatically added to existing pipe specification configuration (Common settings - PipeSpecificationCommon.pssx) during an upgrade to Plant Design 2023 (Installation option - Existing data and configuration).*

Key name "Threaded joint nipple"

A 3D component, which is defined in a pipe spec with the key name **Threaded joint nipple**, is always placed on an existing component/fitting with **Cone spigot ends (ET16)** when generating a pipeline with Smapp3D Piping.

The basic requirements for a 3D component to be defined as a **Threaded joint nipple** in a pipe specification are the same as for **flow-through components** (details in the software help in chapter **Piping > Components for Piping**).

In addition to these basic requirements, in the component for **Threaded joint nipple** one of the OUTPUT_POINT coordinate systems in the CSInfo must be defined with the end-treatment **Cone spigot ends (ET15)**.

Key name "Threaded joint socket unit"

With the key name **Threaded joint socket unit** up to three different 3D components can be defined in a pipe specification.

When generating a pipeline with Smapp3D Piping, these are always placed on an existing component/fitting with **Cone socket ends (ET15)** end-treatment.

Component	Size	End Treatment	Material	Description
Threaded joint socket unit	20	-		
Component 1	20	-		Connector ISO 8434-1 - CR - S20 - ...
Component 2	20	-		
Nut	20	-		Connector ISO 8434-1 - N - S20 - br...

The basic requirements for the respective 3D components depend on the position of the **Threaded socket unit** at which this component is to be defined in a pipe spec.

- For a **Component 1**, the end-treatment **Cone spigot ends (ET16)** must always be defined for one of the OUTPUT_POINT coordinate systems and its CSInfo.
- For a **Component 2**, it depends on the relation in which this component is to be placed to component 1.
- A **Nut** component must then contain the basic definitions of Lapjoint components (details in the software help in chapter **Piping > Components for Piping**).

Due to the generally held implementation of this automation, all (or at least most) of the applications knows to us from practice in the field of tube fittings should be automated with it.

Key name "Red. threaded joint nipple"

A 3D component, which is defined in a pipe specification with the key name **Red. threaded joint nipple**, is always placed to an existing component/fitting with end-treatment **Cone spigot end (ET16)** when generating a reduced pipeline with Smap3D Piping (e.g., at reduced tee situations), starting from the larger pipeline diameter.

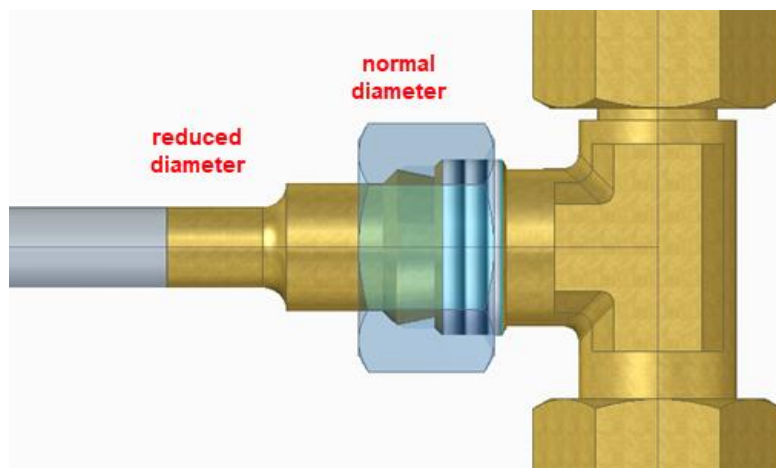
The basic requirements are the same as for the components for the key name **Threaded joint nipple**, where the 2nd connection point has a reduced diameter compared to the 1st connection point.

Key name "Red. threaded joint socket unit"

With the key name **Red. threaded joint socket unit**, up to three different 3D components can be defined in a pipe spec, just as with threaded joint socket unit.

When generating a reduced pipeline with Smap3D Piping (e.g., at reduced tee situations), these are always placed on an existing component/fitting with **Cone socket end (ET15)** end-treatment, starting from the larger pipe diameter.

As a typical use case, **Component 1** is reduced (e.g., reduced welding nipple as shown). Component 2, which may be necessary, and a nut have the normal diameter like the large pipeline.



New in the P&ID to 3D connection (P&ID To-Do list)

New symbol data field allows exclusion of symbols from P&ID To-Do list

Up to now, symbols/components in P&ID diagrams could only be excluded from an evaluation in the P&ID To-Do list, if you either removed the **Physical** check mark or switched it to **Invisible** in the diagram.

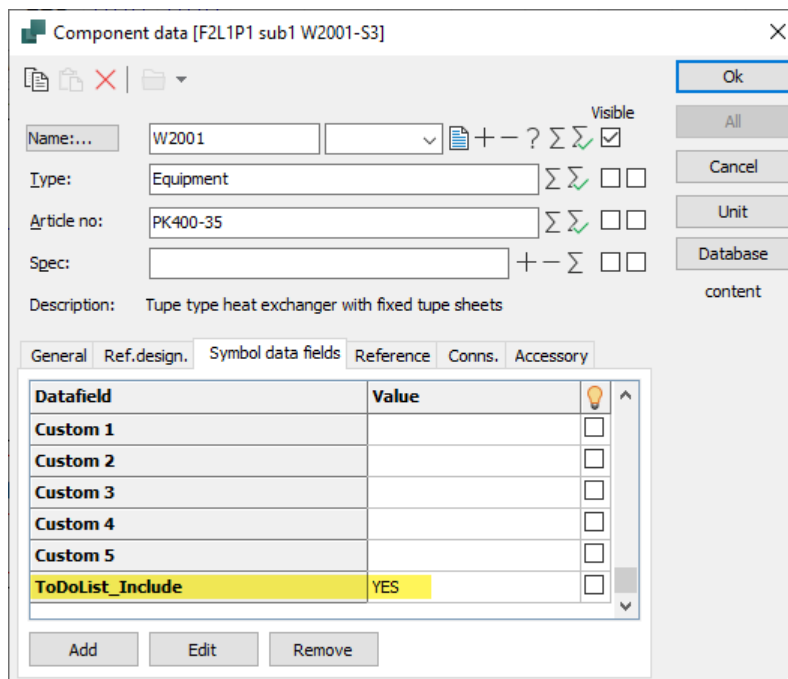
However, both ways then also have effects on the display and evaluation in the P&ID diagram and the P&ID application itself.

To offer users even more flexibility in this regard, we have introduced a new symbol data field with which, depending on the value entered (**Yes** or **No**), you can control for the P&ID To-Do list whether the respective symbol is to be evaluated or excluded from the To-Do list.

This new symbol data field has no effect on the handling of a symbol in the P&ID diagram and the P&ID application.

The name of this symbol data field must be **ToDoList_Include** and only **Yes** or **No** values are allowed.

The name and allowed values must be spelled correctly, but we do NOT check for upper and/or lower case.



The insertion of this symbol data field can be done already in the symbol library (by an administrator or Key User) for all library symbols, so that the P&ID user only must enter or change the required value **Yes** or **No** when using a symbol.

To avoid this administrative effort in the symbol library, it is also possible to insert the symbol data field into a diagram symbol only when needed (via the **Add** button).

Generally, the symbol data field **ToDoList_Include** must have been created in the P&ID application beforehand for both use cases.

If this is not the case; it cannot be used when editing symbols in the library or individually in a diagram.

The rules for the P&ID To-Do list response to this symbol data field are as follows:

- if the value is **Yes**, then the symbol is **evaluated** and **displayed** in the To-Do list
- if the value is **No**, the symbol will **not be evaluated** and will **not be displayed** in the To-Do list
- if the symbol data field does not exist or is not written/spelled correctly or exists but does not have a valid value (**Yes** or **No**), then it behaves as in the **Yes** use case.

New in Smap3D Isometric

Spool Analyzer

Der Spool Analyzer is delivered for the first time with version 2023 of Smap3D Plant Design and is a **special operating mode** of the **Smap3D Isometric** application.

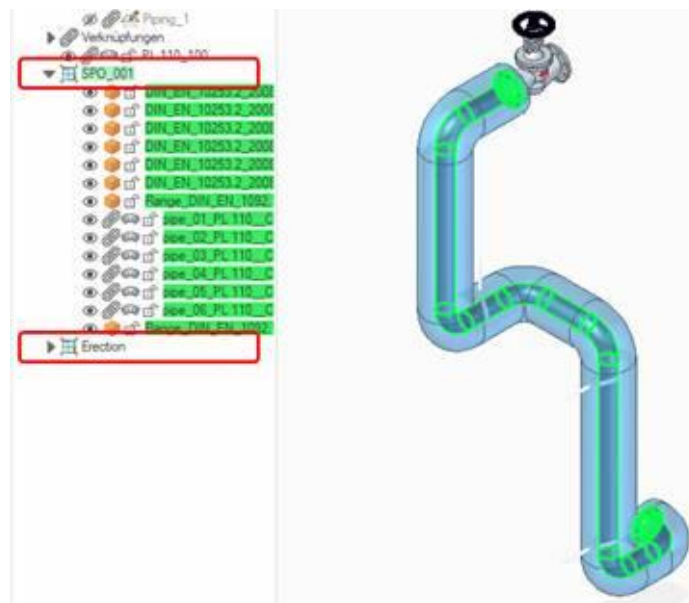
This means that the Spool Analyzer application can only be used with a working Smap3D Isometric installation and a valid PipingISO license.

Just like Smap3D Isometric, the Spool Analyzer can only work in 3D assemblies that contain pipelines created with Smap3D Piping.

The Spool Analyzer analyzes the active pipeline assembly (based on the settings of Smap3D Isometric) and transmits this information via PCF-file to the ISOGEN© Kernel.

Based on a specific **Spool Analyzer spool style**, ISOGEN© evaluates them and then returns the **spool allocation/spool compilation** information back to the Spool Analyzer.

The Spool Analyzer then creates corresponding **display groups** in the CAD structure tree based on this information.



As a result, a user will at least get one display group for a spool (with the defined spool numbering from the Spool Analyzer spool style), as well as a group with the name **Erection**.

The Spool Analyzer is a function that automatically transfers information about spools from ISOGEN© back into the CAD environment to make them visible to the user as display groups in the CAD structure tree.

Whether, and how these display groups are then further processed for other specific purpose, is up to the users/customers.



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