This is new in Plant Design 2023

Once again, the latest version of Smap3D Plant Design contains many enhancements and improvements in the areas of functionality and application.

The most important ones are described below.

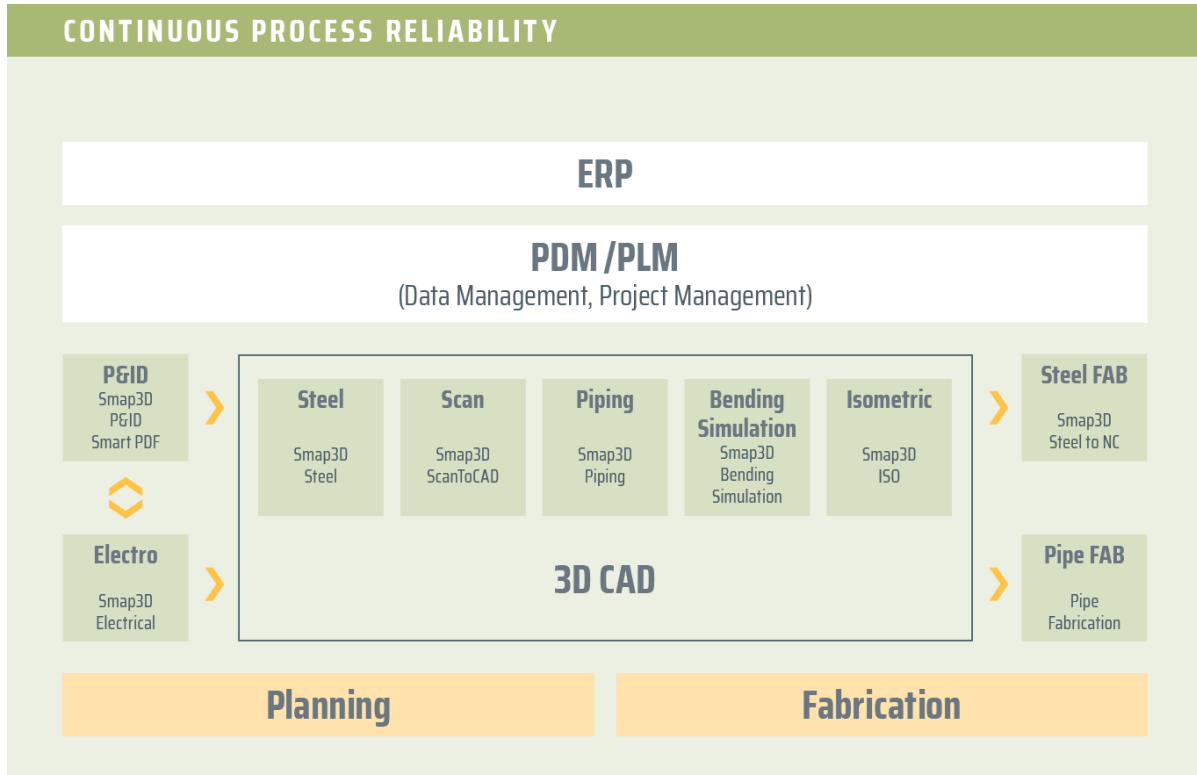## The different chapters in this document are:

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

Table of Contents

General improvements

Supported CAD versions

SOLIDWORKS

The release version of Smap3D Plant Design 2023 supports:

- **SOLIDWORKS 2022 with Service Pack 5.0 or later**
- **SOLIDWORKS 2023 with Service Pack 0.1 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 Smap3D P&ID

Reference designations can now also be used for lines

One of the big wishes of many customers and users of Smap3D 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.
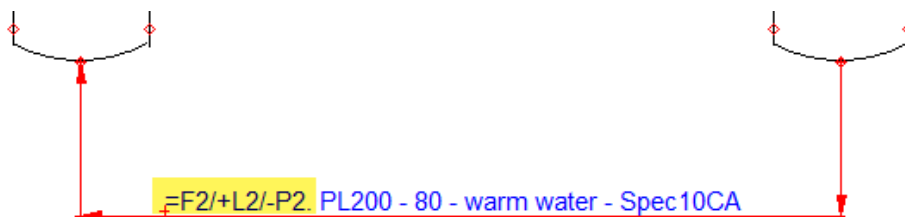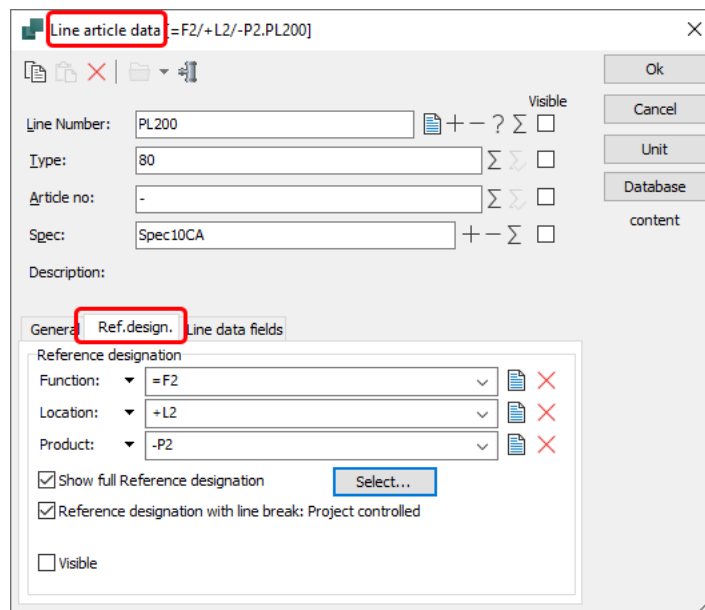
Of course, reference designations on lines are also supported and used by all further functions and tools in the P&ID application.
The necessary improvement has been made.

In the Object Lister
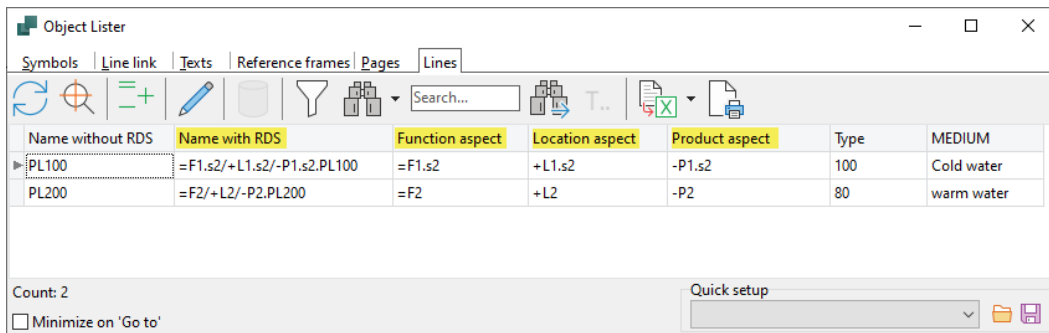
In the Edit lines data fields module
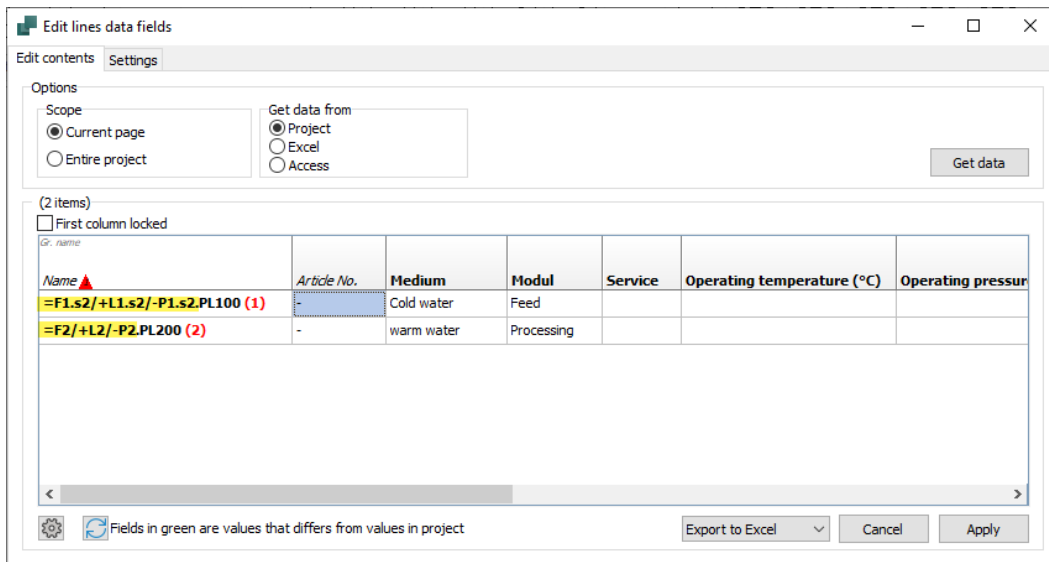
For list evaluations with lines

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.

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.
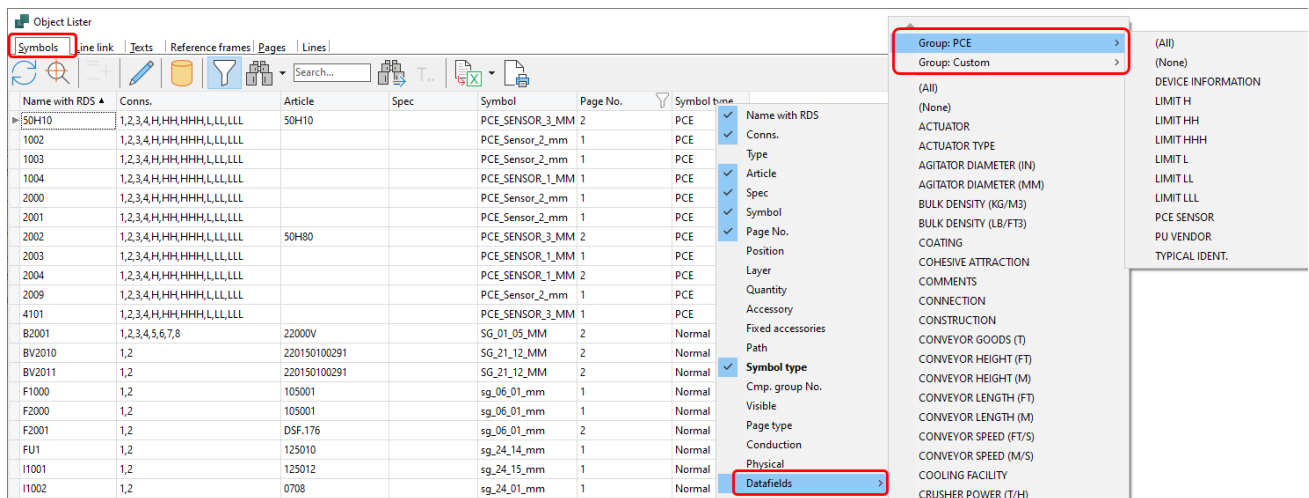
Save and load column display configurations

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.
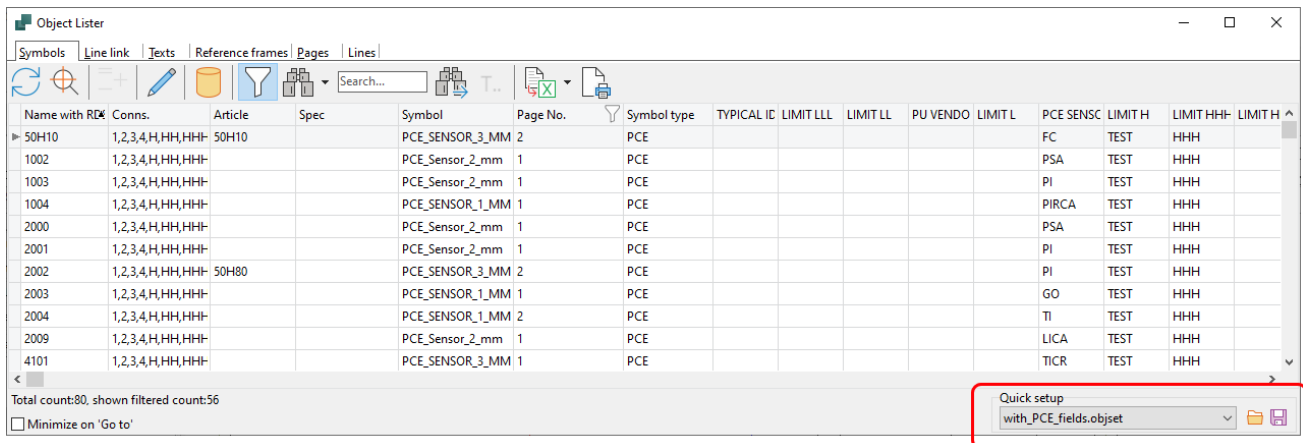
Enhancements to the Edit symbols data fields module

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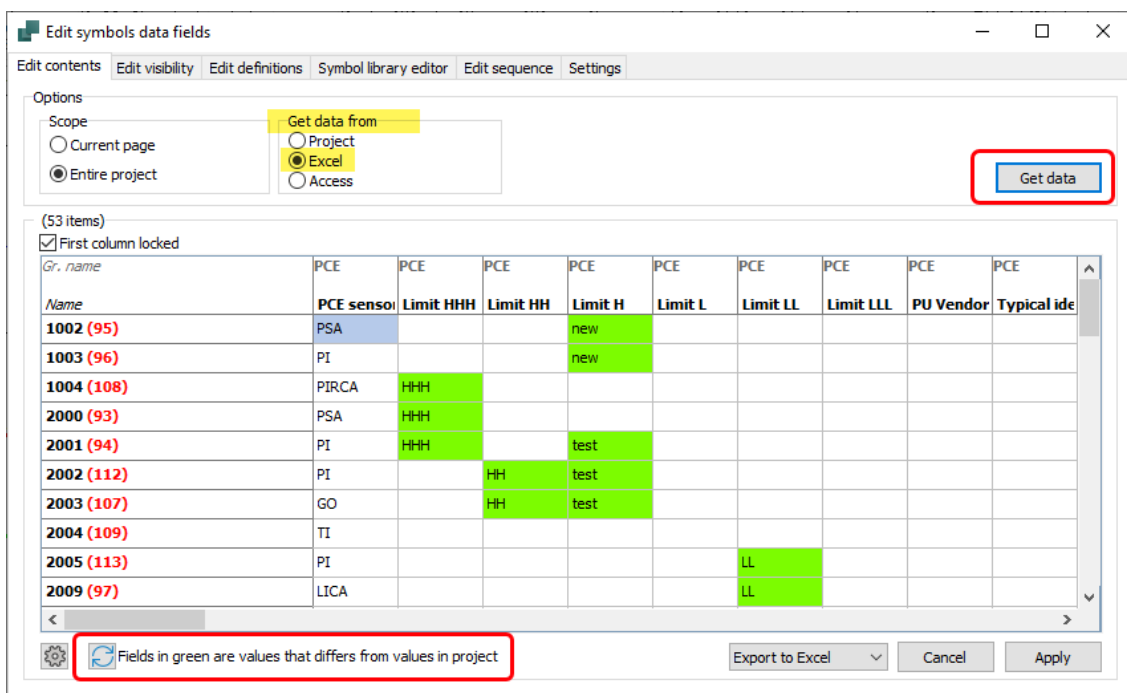
However, this new check/display **ONLY** works for Excel tables that have been exported exclusively with the **Edit symbols data fields** too, in the **Edit contents** tab, from the currently open and active project. This check is based on system internal symbol IDs and can therefore only work correct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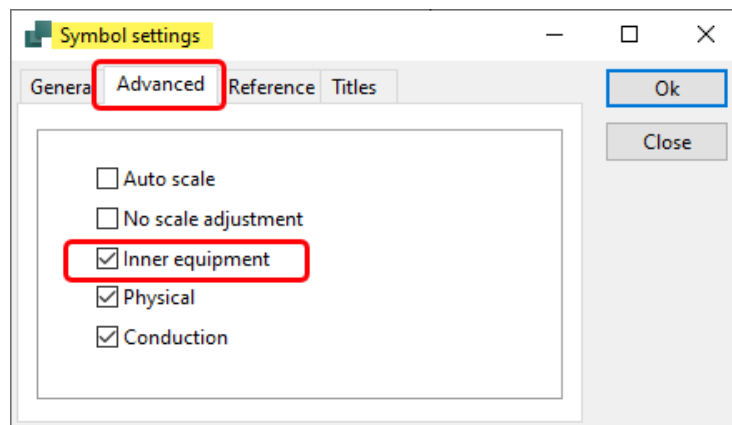
Predefinition of values for symbol data fields per symbol

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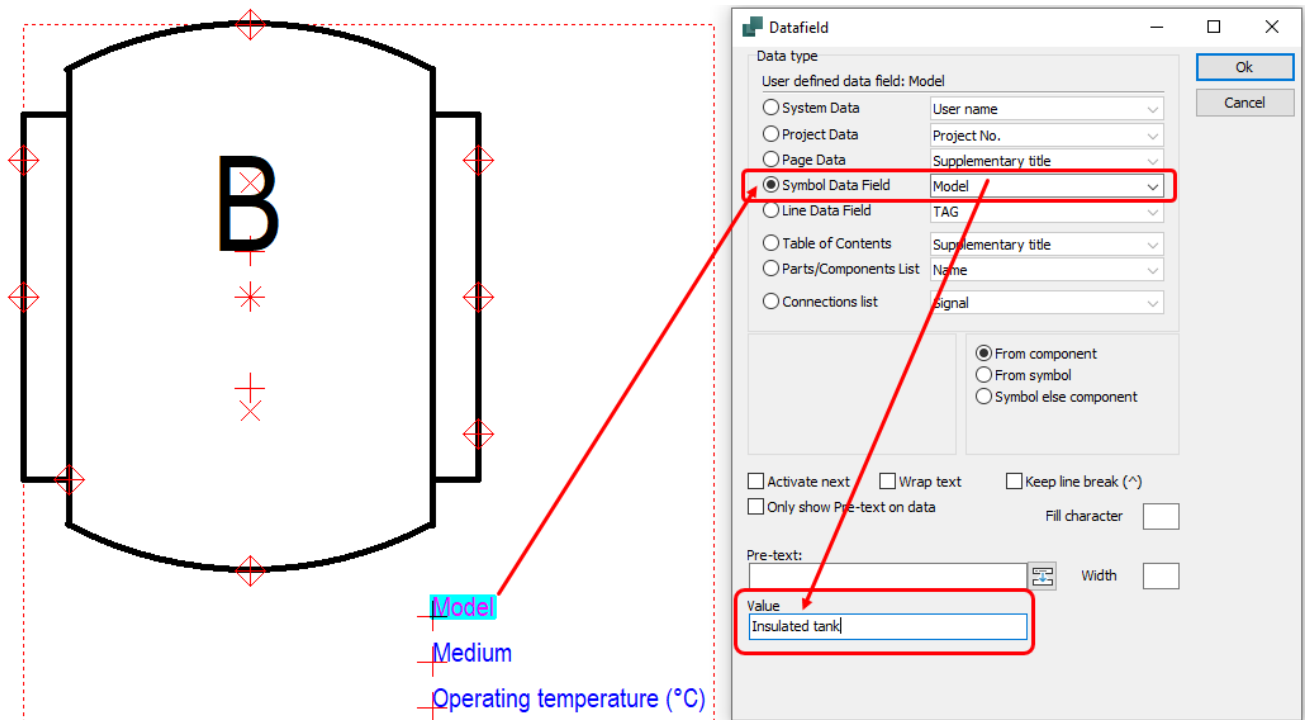
When placing such a symbol in a diagram, the value predefined in the symbol has the highest priority and is automatically used/filled in the symbol's component data.

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.
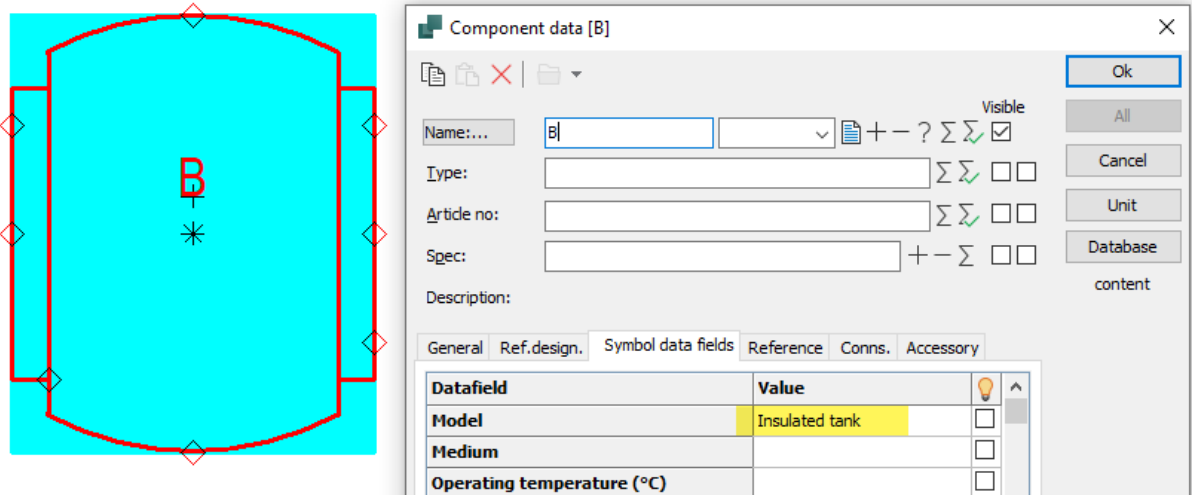
New module "Pipeline list"

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 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.

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.
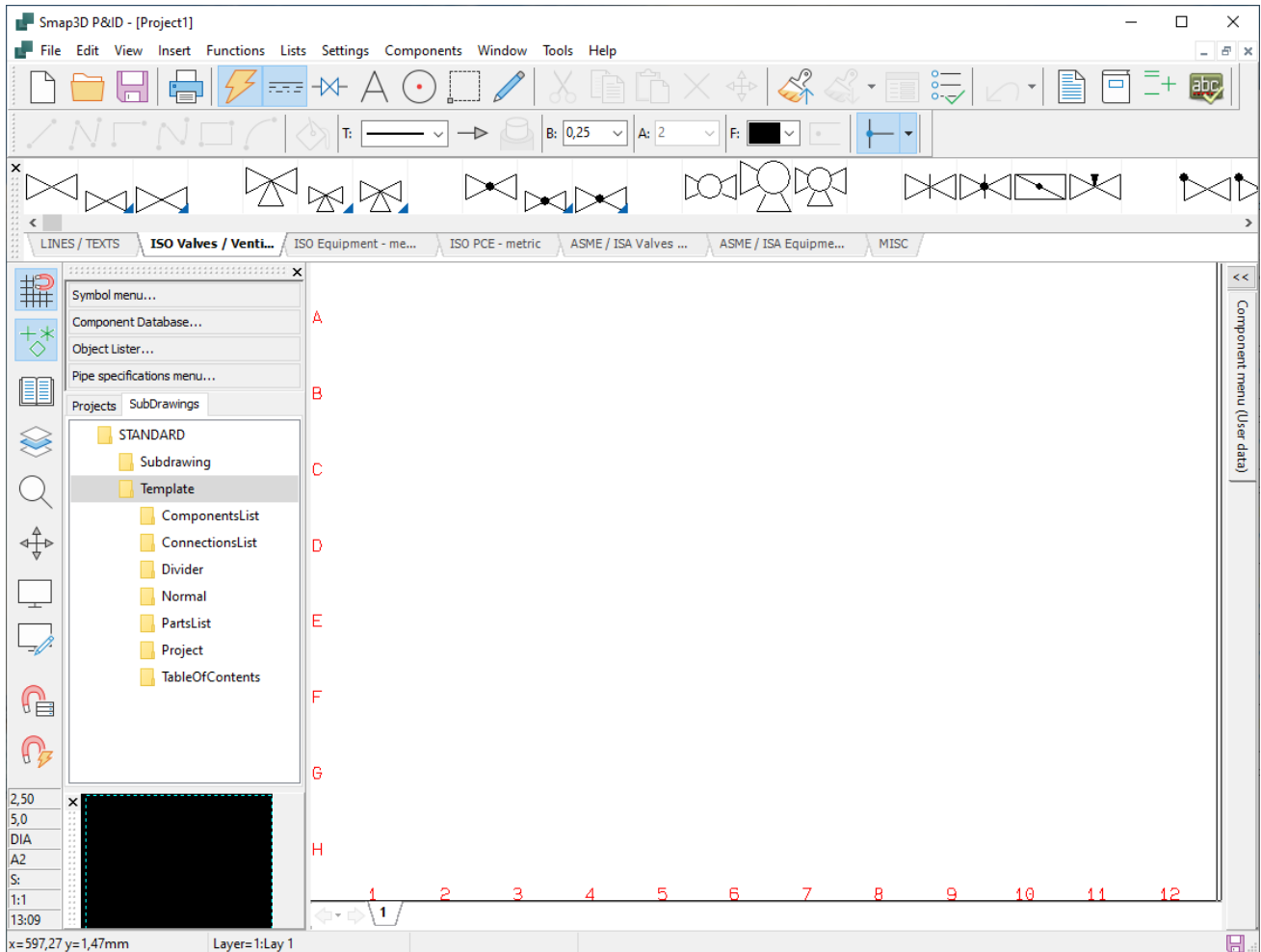
Enhancement in the formula editor for symbol and line data fields

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.

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 <mark>~$Filename**.PRO**</mark> 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 <mark>Filename**.PROlck**</mark>.
Furthermore, these are now no longer hidden but always visible after creation, in the same folder as the PRO file.
<mark>These **.PROlck** files are now **no longer automatically deleted** after closing a PRO file.</mark>
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**.
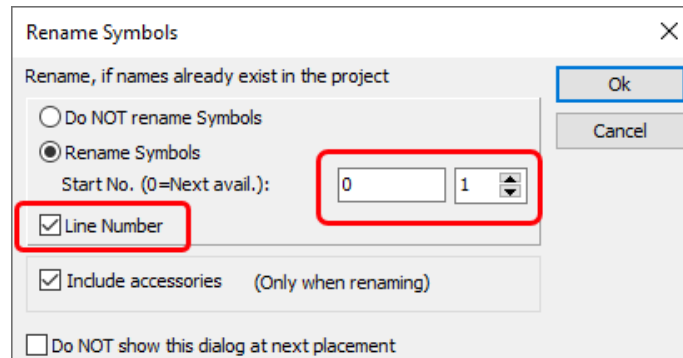
Option "Insulation" applicable for dashed standard lines

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.
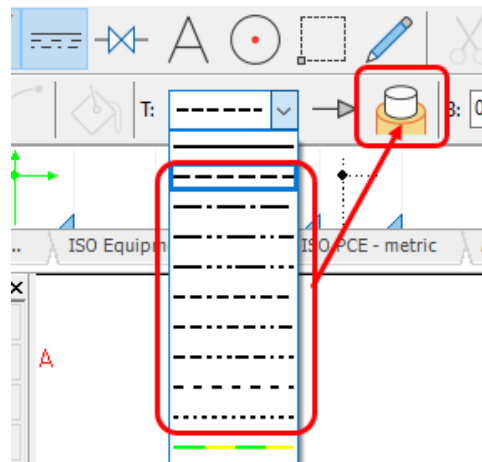
New option "Article no protected"

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.
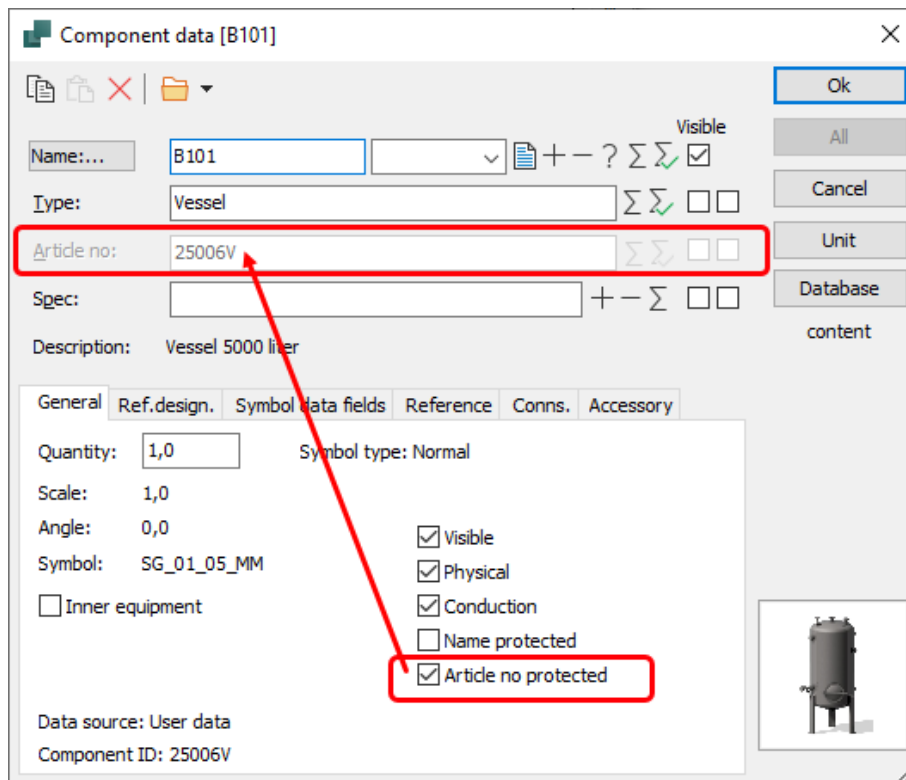
New design check "Non-unique line name"

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.
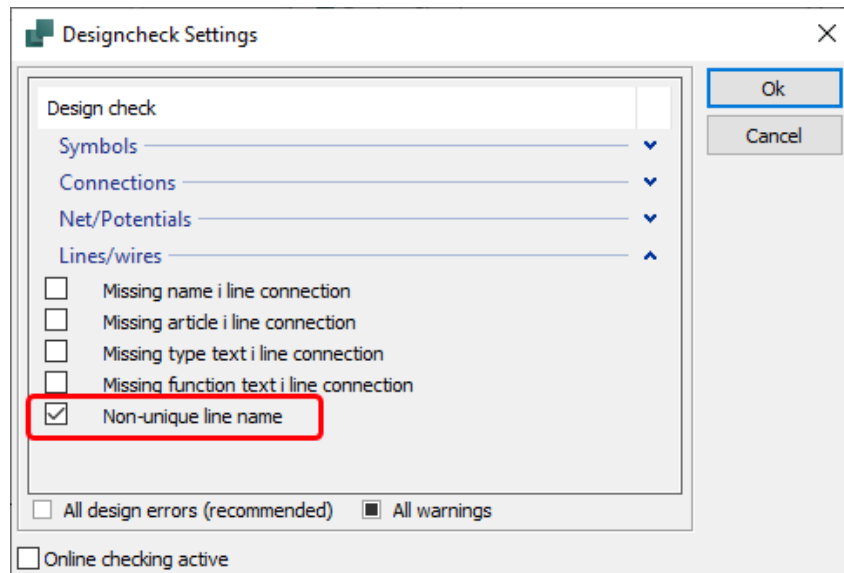
Subdrawings - new button Database for article no with variables

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.
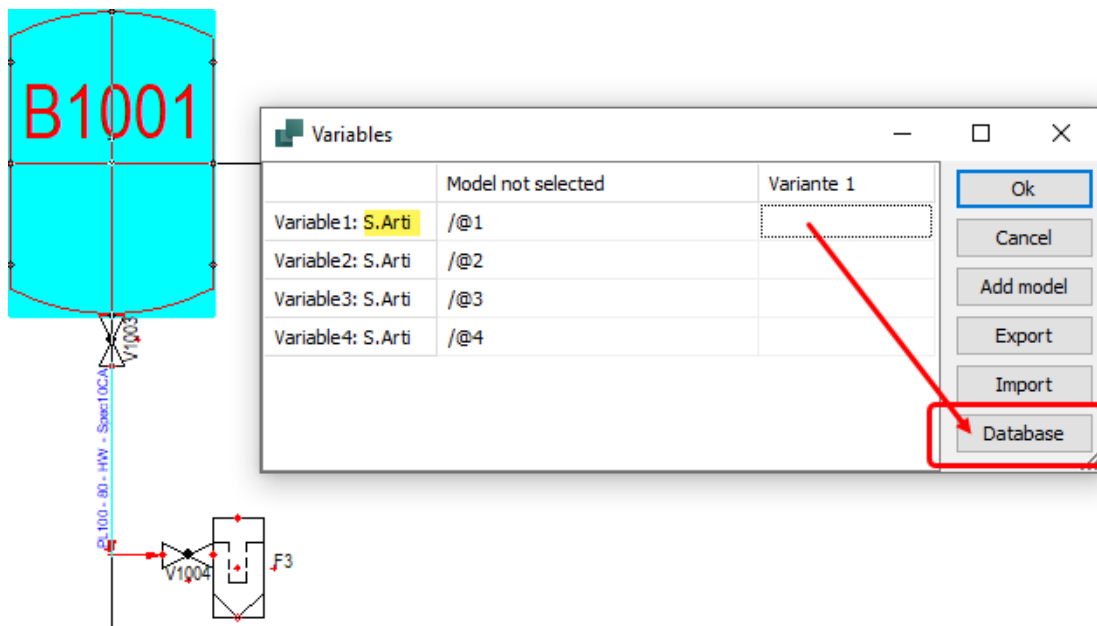
"Symbol ID" data field available for list exports

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.

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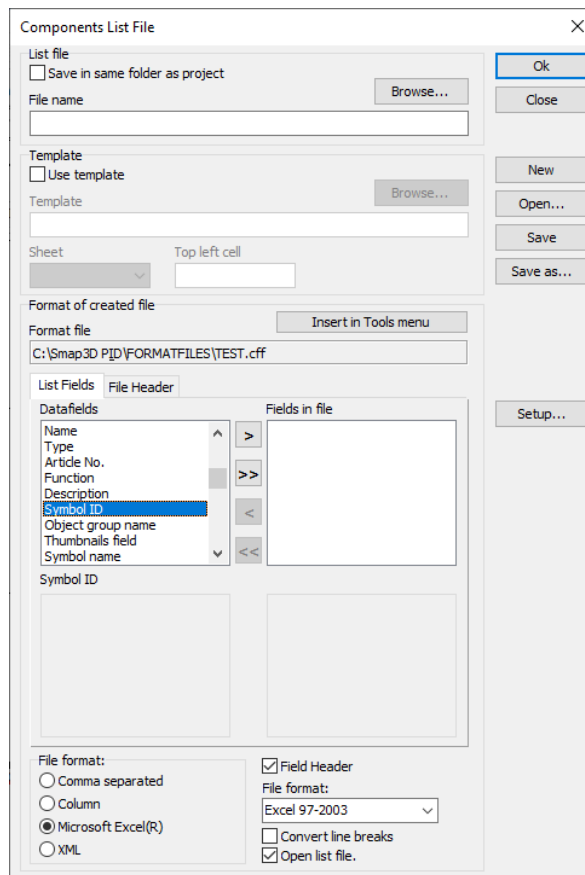
User defined paper size up to 100m x 100m possible

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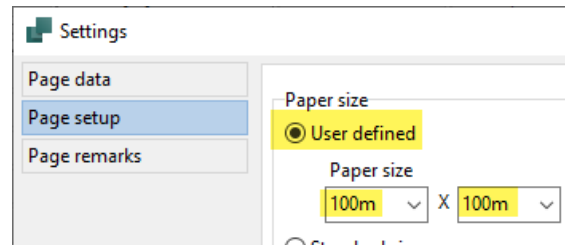
Administrative option in "PCSCAD.INI" to disable automatic SQL query when entering part numbers

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 Smap3D 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.
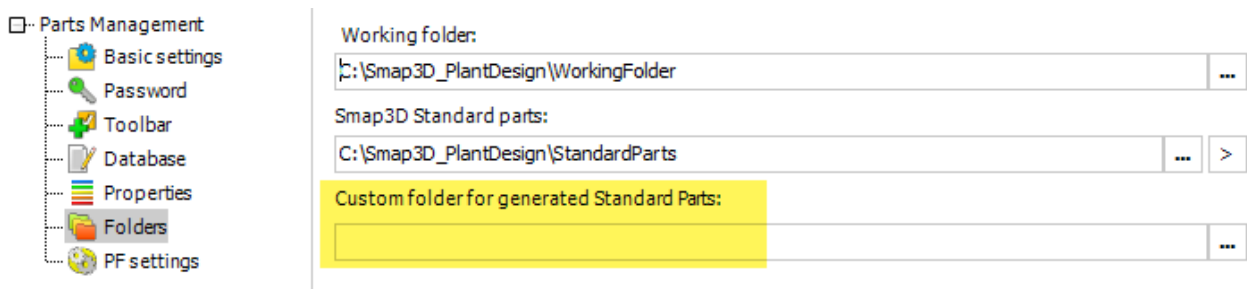
Until now, generated standard parts were always stored in folders (with the name of the respective standard) in parallel next to the **Parent_Parts** folder, in a fixed and non-influenceable manner according to the following scheme.
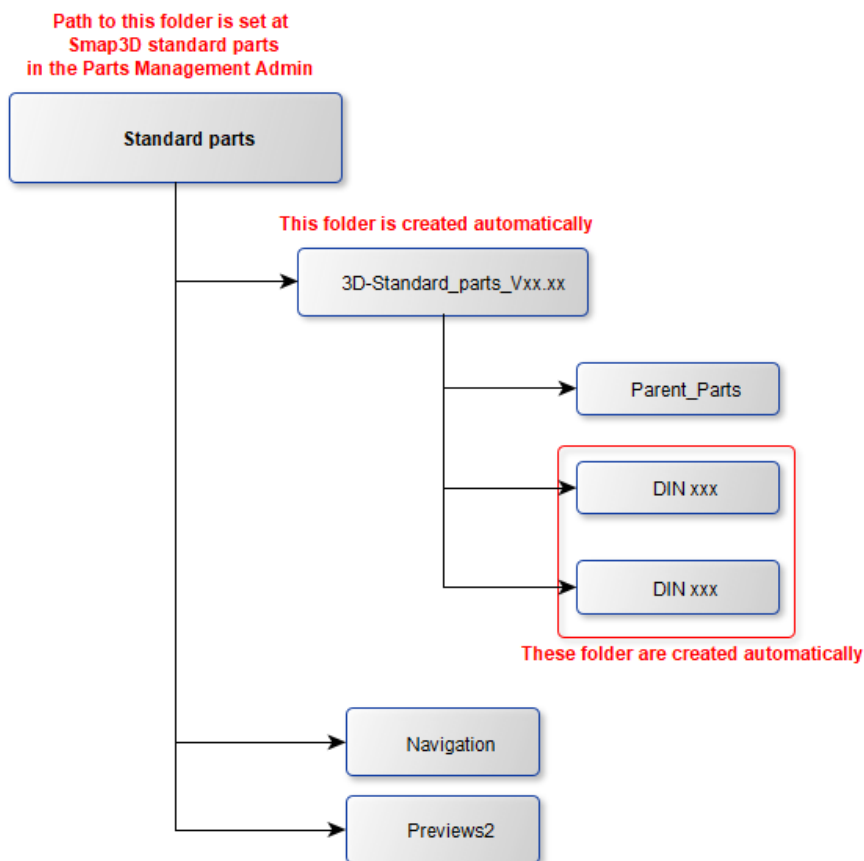
With the new setting (by creating and selecting the required user-defined path), it is possible to influence the storage path of the folders with the generated standard parts files.
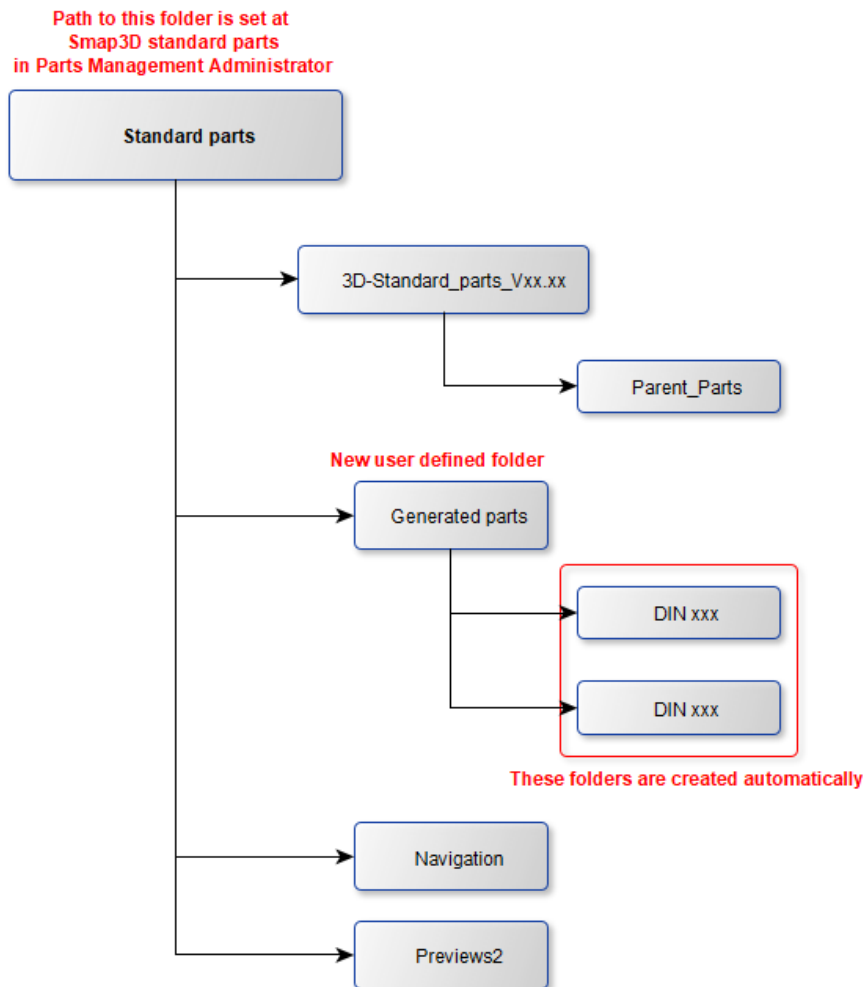
New "PMAgent.exe" application

For all Smap3D 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.
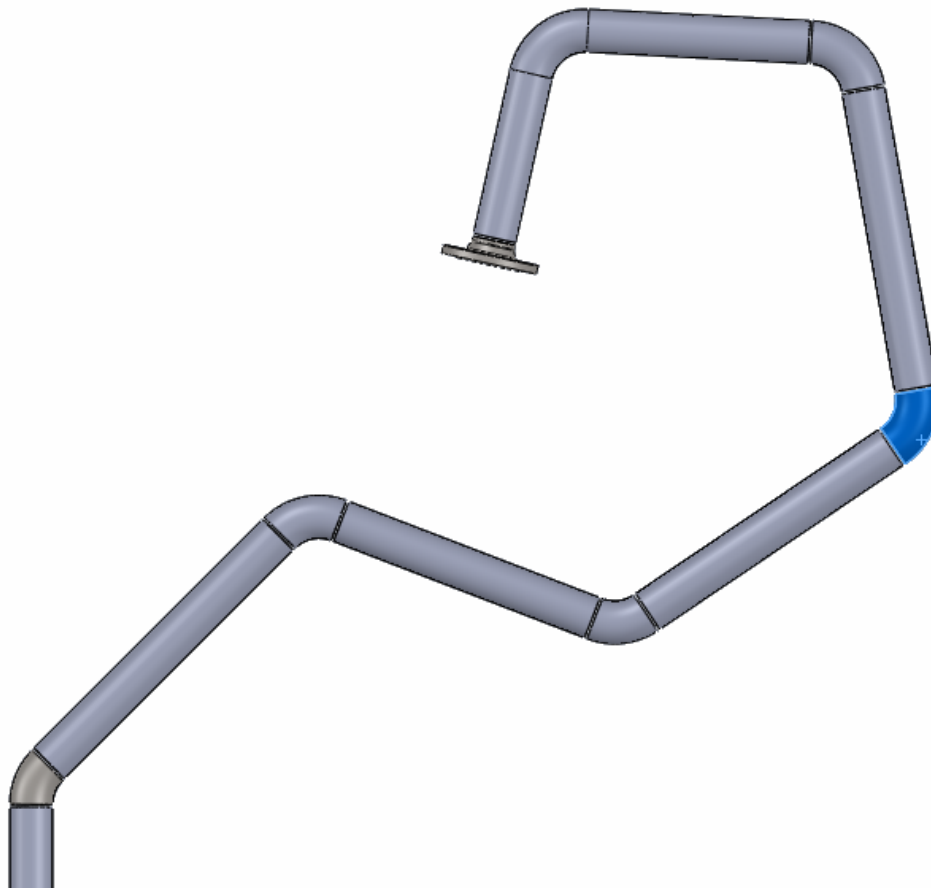
New QuickPlace method "Tee fixed angle"

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.

Tee fixed angle	80	-		
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 Smap3D 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).
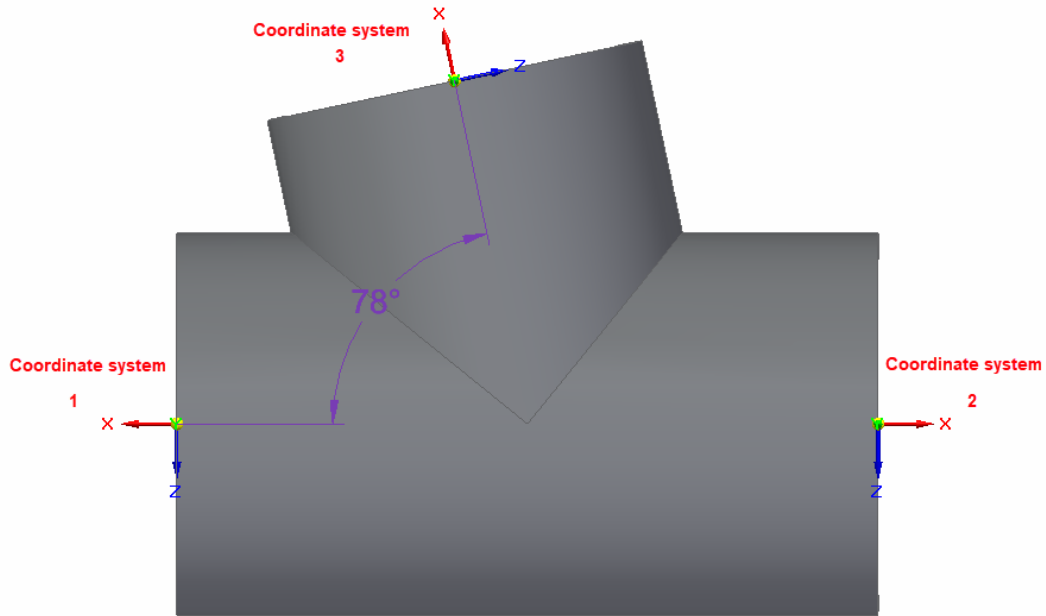
If all requirements are fulfilled, it is now possible to create very flexible and automatic individual pipelines with different tee branch angles without having to place the special tees manually at the branches.
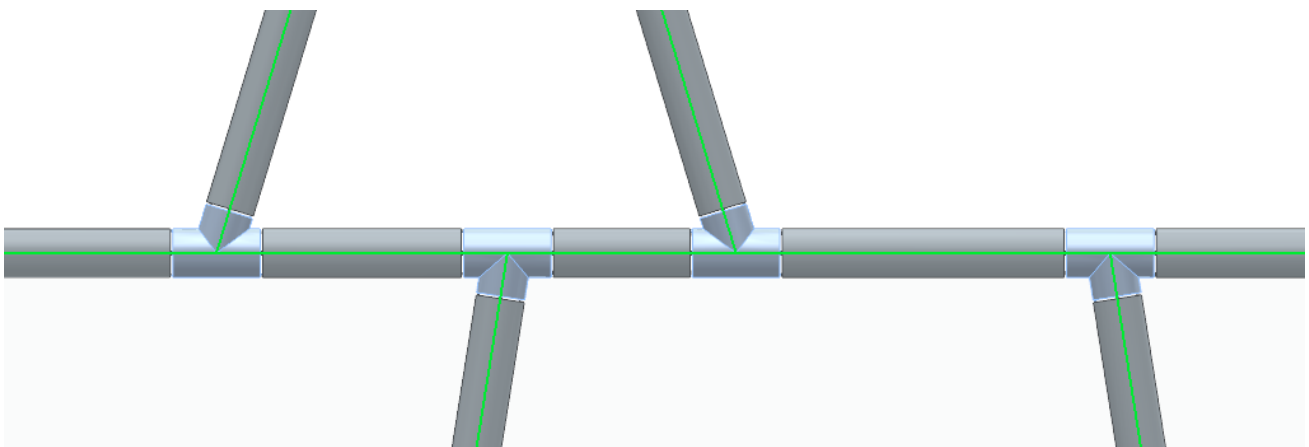
New function "Transfer connection properties"

Smap3D Piping and Smap3D 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 Smap3D Piping.

This is available in the Piping context menu, when one or more pipeline paths are selected.
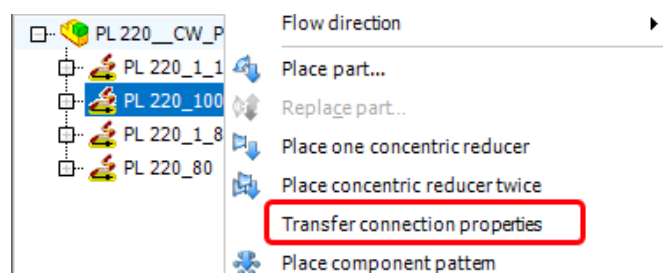
A window with a progress bar displays the steps taking place there.
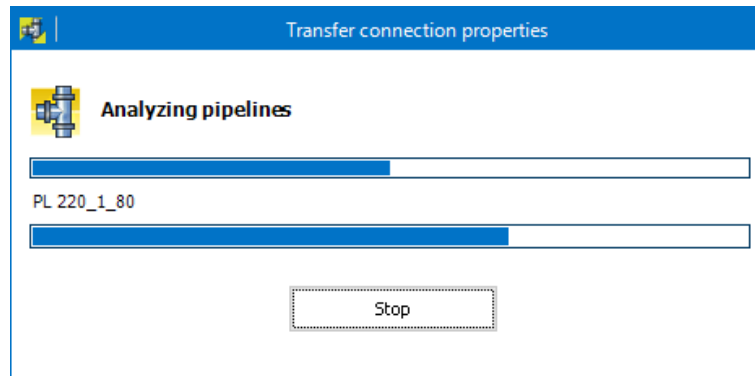
- In the first step, any existing old connection information is deleted from the paths/parts within the active assembly.

- 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.
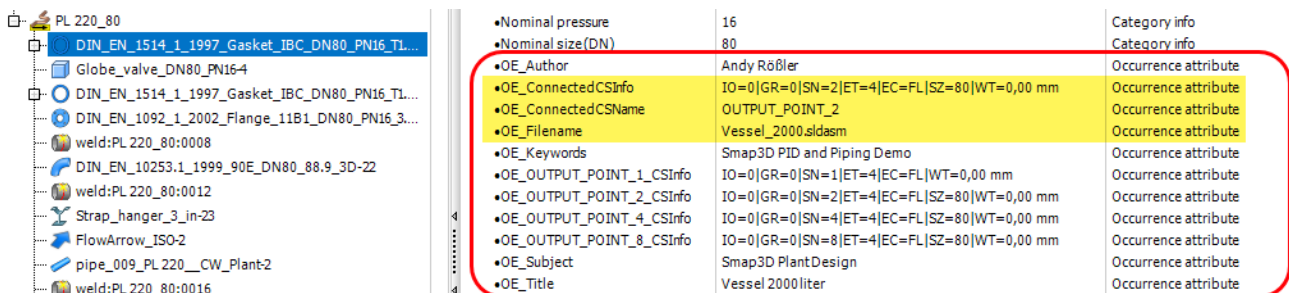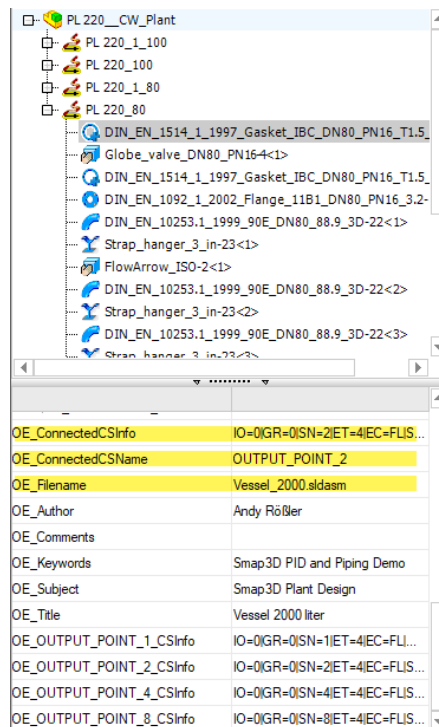
New function "Import path sketch lines from PCF"

For all users/customers who want/need to use existing piping isometrics in the form of PCF files for 3D pipeline creation with Smap3D 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 **Create** menu of the Piping toolbar.
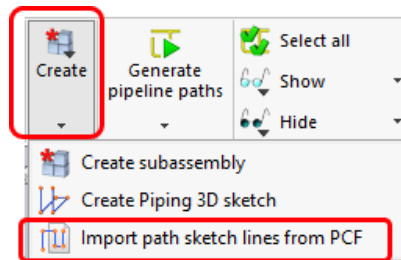
- The **PCF file select** window opens. The required PCF file for the import must be selected there.

- 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).
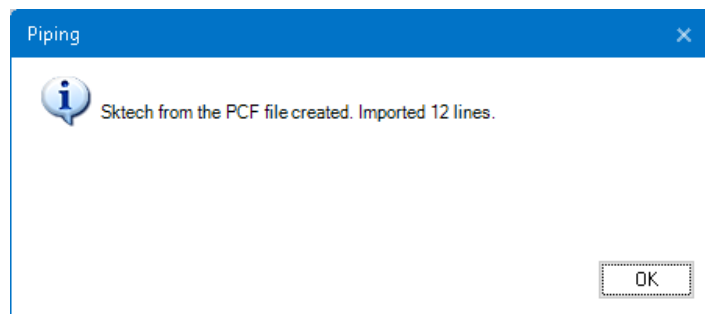
- Depending on the user requirements, the active 3D sketch can be extended with dimensions and/or sketch relation/mates (with native CAD functions. If the 3D sketch meets the requirements, then the **3D sketch editing mode must be closed/exited**.

- 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 Smap3D 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 Smap3D 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.

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.
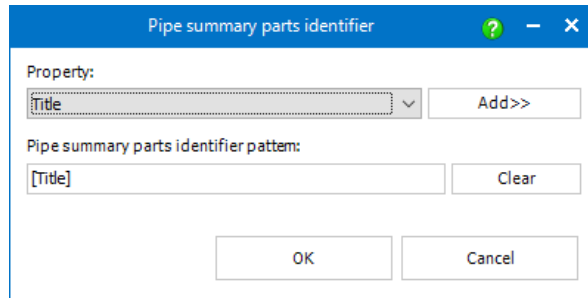
In the list box of the **Property** row, the native CAD standard properties are available for selection as identifier.
At least one property must be selected and set as an identification pattern using the **Add** button.
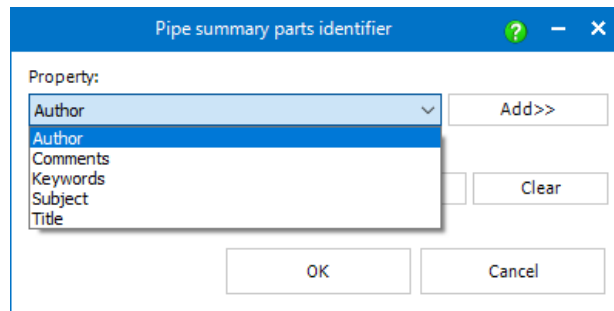
Based on the identification pattern defined here, when the function is executed in Smap3D Piping, the 3D pipe files present in the piping assembly are analyzed, summed and then the resulting number of pipe summary files in the assembly are generated.

This function is started in the Smap3D Piping context menu (without preselection of paths).
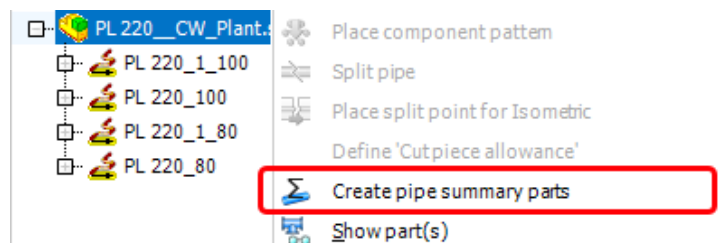
As result of this function, you will receive at least one pipe summary file in your CAD structure tree (NOT in the Piping structure tree), with a file name according to the associated file name generator setting in Plant Design Administrator.

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.
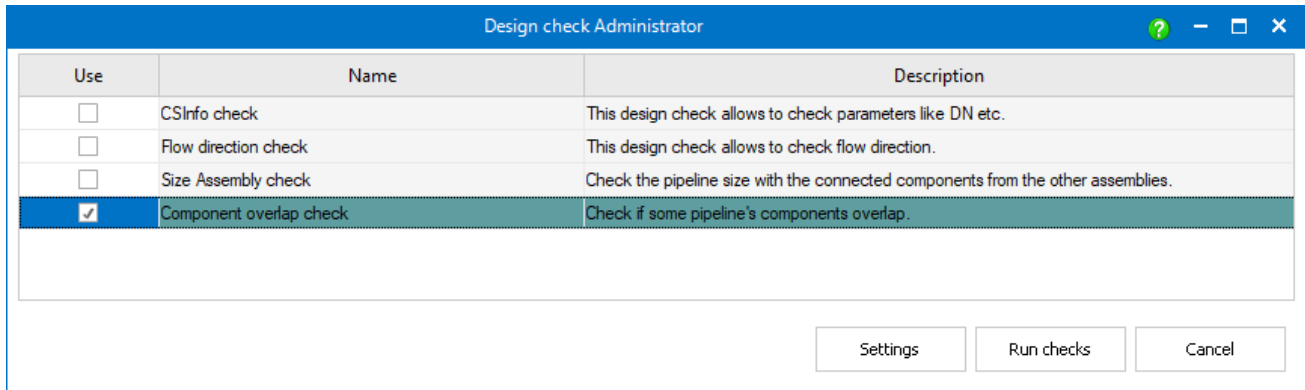
When running this design check, the coordinate systems (INPUT/OUTPUT) contained in the fittings are checked for overlap, not their solid model.
If overlaps are found, the affected components are displayed individually in a result dialog.

By clicking on the respective entry, the component is highlighted in the CAD work window.
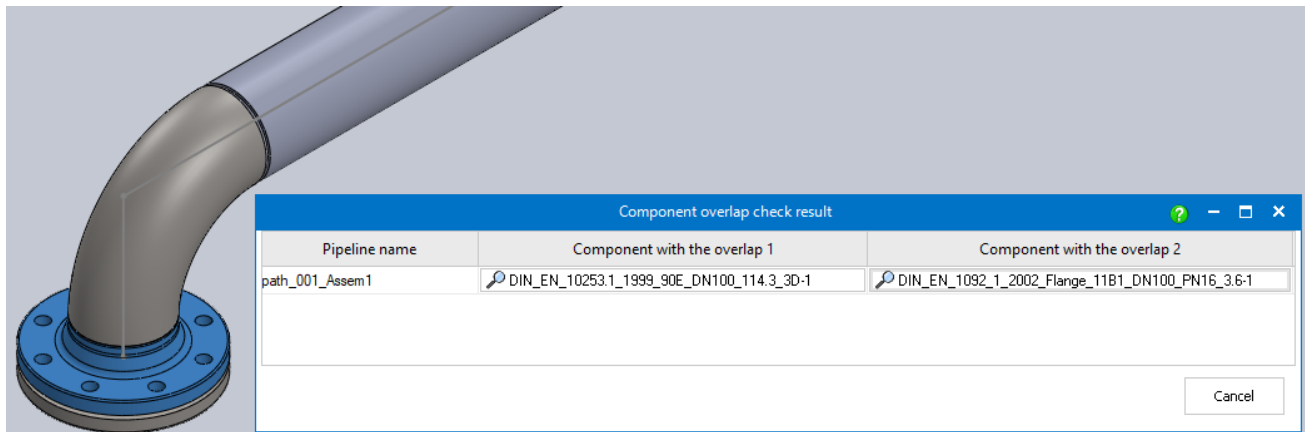
New button/function "Refresh list of all piping subassemblies"

In the dialogs of the functions **Refresh all subassemblies** and **Switch all to detail engineering** 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** and **Switch all to detail engineering** functions.
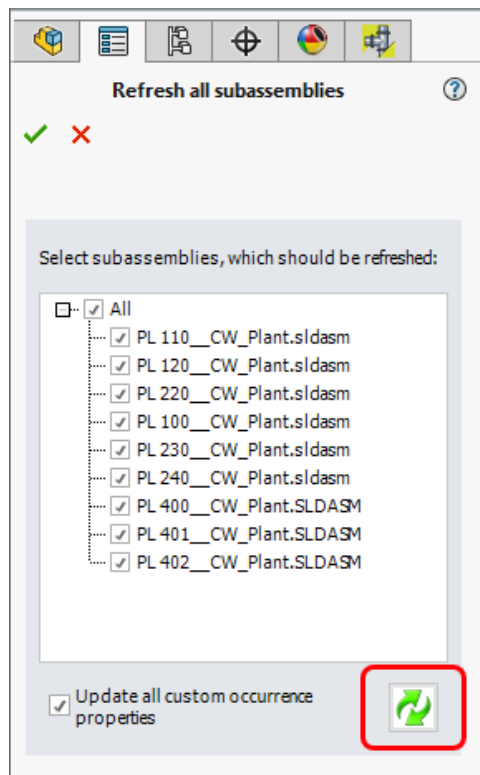
New option "Sort pipeline part names alphanumerical"

This new option is in the **Options** of Smap3D Piping application.
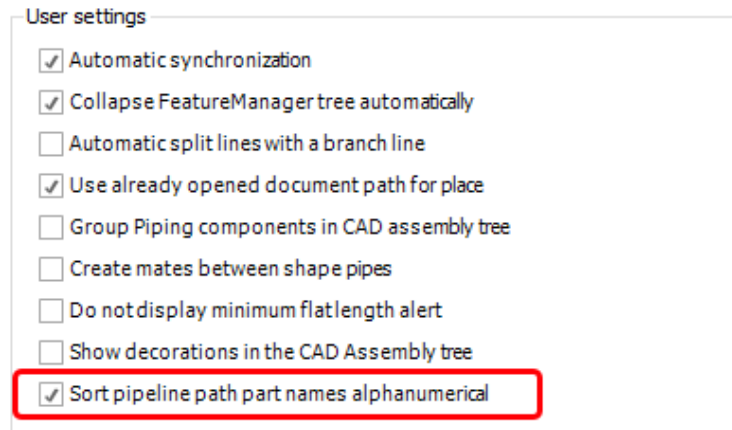
If this is activated, all pipeline paths are displayed with their display names sorted alphanumerically in the Piping structure tree.

The Piping structure tree reacts to this sorting setting, each time the application is loaded and when the **Refresh view** function is executed if changes have been made to the piping path names in the meantime.
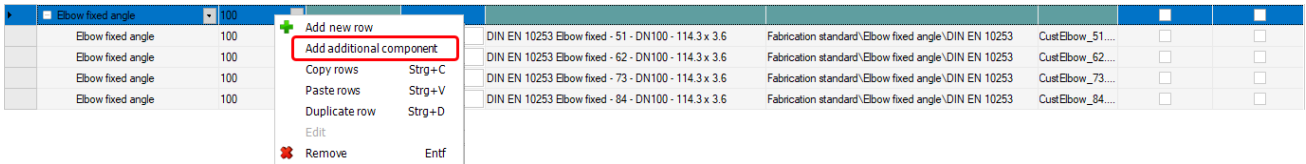
New in Pipe specification editor

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.

ℹ️ *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.

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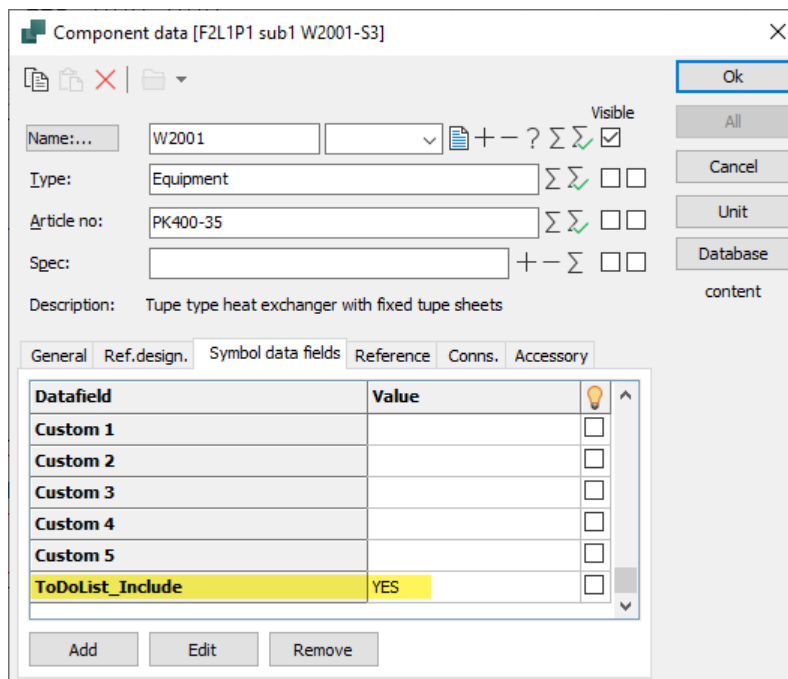
The insertion of this symbol data field can be done already in the symbol library (by an administrator or Key User) for all library symbols, so that the P&ID user only must enter or change the required value **Yes** or **No** when using a symbol.

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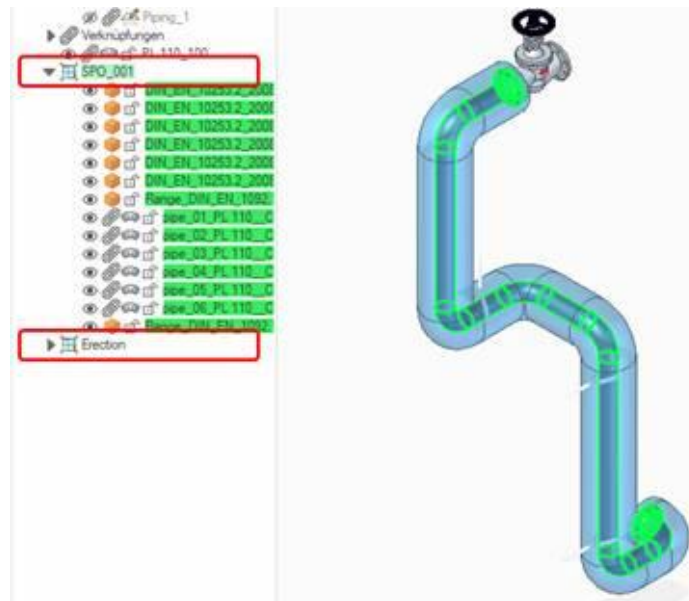
As a result, a user will at least get one display group for a spool (with the defined spool numbering from the Spool Analyzer spool style), as well as a group with the name **Erection**.

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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